University of Southern Queensland
Faculty of Engineering & Surveying

Stress Analysis of Axially Loaded Perforated Steel Plates by Experimental and Finite Element Methods

A dissertation submitted by

Benjamin J Field

in fulfilment of the requirements of

ENG4112 Research Project

towards the degree of

Bachelor of Engineering (Mech)

Submitted: October, 2005

Abstract

In this project thin steel plates containing elliptical holes were subjected to uniaxial tension and the strains produced at various locations around the plates measured using strain gauges. The loading applied was great enough for a significant amount of plastic deformation to occur across the plate section.

The mechanical properties of the steel were determined by conducting a series of tensile tests on standard "dogbone" samples.

Plane stress, finite element models were created using the student editions of both ANSYS v5.5 and ABAQUS V6.4.

Comparisons between the experimental and model results of the strains in the plates were made.

University of Southern Queensland
Faculty of Engineering and Surveying

ENG4111/2 *Research Project*

Limitations of Use

Prof G Baker
Dean
Faculty of Engineering and Surveying

Certification of Dissertation

I certify that the ideas, designs and experimental work, results, analyses and conclusions set out in this dissertation are entirely my own effort, except where otherwise indicated and acknowledged.
I further certify that the work is original and has not been previously submitted for assessment in any other course or institution, except where specifically stated.

Benjamin J Field

0050027030

Signature

Date

Contents

List of Figures

List of Tables

Nomenclature

UYS	Upper Yield Strength for materials such as mild steels
LYS	Lower Yield Strength for materials such as mild steels
YPE	Yield Point Elongation for materials such as mild steels
σ	Engineering stress, based on initial dimensions
σ_{true}	True Stress, based on instantaneous dimensions
σ_y	Yield stress
σ_u	Ultimate stress
ε	Engineering strain, based on initial dimensions
ε_{true}	True (logarithmic) strain, based on instantaneous dimensions
E	Elastic Modulus of the steel
ν	Poisson's ratio of the steel
GF	Gauge factor of particular strain gauges
R	Resistance of strain gauge
ΔR	Change in resistance of strain gauge due to strain
L	Gauge length of strain gauge
ΔL	Change in length of strain gauge

Chapter 1

Introduction

Finite Element Analysis (FEA) is commonly used in industry today for a wide range of stress analysis applications. As a result of this popularity there are many different commercial software packages available each offering various analysis possibilities. Results obtained from finite element models however often need to be qualified before confidence in accepting their predictions can be established. A common activity to verify the results from a model of a component or structure is by making comparisons with the experimentally measured response of a real component when subjected to similar conditions. Such "benchmarking" tests can provide valuable insight about not only the performance of the model but also the qualification of any assumptions made during the modelling process.

Undergraduate programmes in Mechanical Engineering offered by the University of Southern Queensland contain basic courses in stress analysis using both analytical and numerical methods. Students get an introduction into the use of a commercial finite element modelling software package, currently ANSYS, by analysing some simple systems. In all cases a very limited amount of experimental stress analysis is performed to verify theortical results by actually measuring the response of a real system, and in general the stress analysis problems encountered largely focus on the assumption of linear, elastic material properties.

This project was undertaken to gain further insight into the methodology required to, obtain measurements of the responses of real systems, model real systems using commercial finite element modelling software, and assess the methods used by comparing model and experimental results.

In this project it was proposed that the response of mild steel plates containing single elliptical holes and loaded in tension be analysed in both the elastic and plastic ranges of the steel. A comparison between the strains measured, using strain gauges, and those obtained by FEA would be undertaken at various positions on the plate surface. To obtain a wider understanding of the model building process and evaluate the differences between commercial software packages the student editions of both ABAQUS (version 6.4) and ANSYS (version 5.5) were to be used.

Chapter 2

General Project Theory and Literature Review

2.1 Mechanical Properties of Mild Steel

2.1.1 Elastic Behaviour

To better understand the behaviour of a mild steel plate under the influence of an applied axial load it is beneficial to review the concepts of stress and strain. When a material is subjected to a load a stress is created within the material. This stress acts to balance the influence of the applied loading keeping the system in equilibrium. For a body subjected to uniaxial loading the stress state of the material making up the body is simplified to a case of uniaxial stress. The direction of the principle stress is located along the same plane as the loading is applied. For states of uniaxial stress the magnitude of the principal stress can be expressed as the magnitude of the load divided by the sectional area over which it is applied. It should be noted that these relationships only apply in cases where the loading is uniformly distributed over the entire cross section. The loading of a body also gives rise to deformation of the material, as the loading is increased this deformation also increases. The ratio of the deformation of a unit length of the material under load to a unit length of the material in its unloaded state is refered to as the strain. Between a range of stress levels steel can be said to behave elastically. Any deformation caused by the applied loading will completely disappear when the loading is removed.

2.1.2 Plastic Behaviour

At a certain level of stress a defined change in the linear relationship between stress and strain occurs. The largest value of stress at which the steel still behaves elastically is referred as its upper yield strength (UYS). A large amount amount of strain can now be produced by a near constant and significantly lower stress. The Stress level at which a large increase in strain can be acheived with little or no required increase in stress is known as the lower yield strength (LYS). The strain that can be produced before a further increase in loading is needed is known as the yield point elongation (YPE), this can be observed in Figure 2.1. Sometimes if the strain rate of the material is low the upper yield point characteristic will be suppressed (Davis 2004). In general the yield stress (σ_y) of the material is taken to be equal to the LYS. After a period of YPE the material begins to work harden and a further increase in stress is required to cause increased strains. The maximum stress that the material can withstand before failing is known as its ultimate strength (σ_u). Mild steel at normal temperatures, as encountered in this project, can be said to be a ductile material, under the influence of loading a significant amount of deformation can occur before the ultimate failure of the material.

2.1.3 Stress-Strain Relationships

One standard method for determining the mechanical properties of a material is to plot the engineering stress-strain relationship of a specimen as it is loaded uniaxially to failure. Determination of the engineering stress and strain make use of the original undeformed cross sectional area and length of the test specimen. When a sample of low carbon mild steel, such as AISI 1010, is subjected to such a unidirectional load a characteristic behaviour between the engineering stress and engineering strain can be observed, as in Figure 2.1.

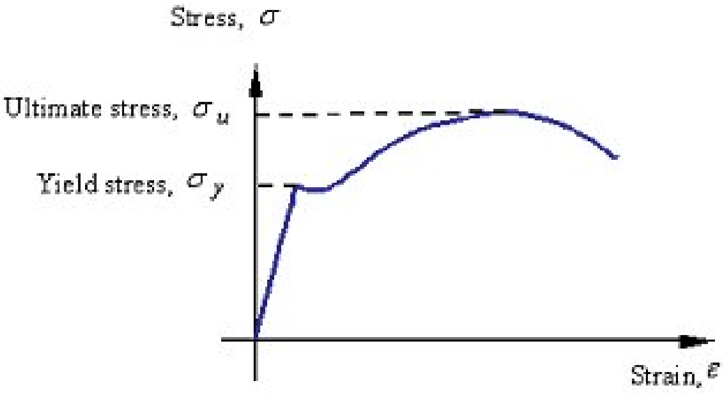


Figure 2.1: Typical Stress-Strain Diagram for a Mild Steel.

Whenever a material is elongated to many times its maximum elastic strain the engineering stress-strain relation becomes somewhat fictitious because it is based on a cross sectional area that is different than that which actually exists. This is due to the reduction in area as a result of Poisson's ratio induced strains. In this case a more accurate representation can be obtained by determining the true stress (σ_{true}). The true stress can be found from the engineering stress (σ) and engineering strain (ε) by,

$$\sigma_{true} = \sigma(1 + \varepsilon)$$

Similarly the engineering strain is not a realistic measure when large strains are involved. In these cases it is more appropriate to use true strain (ε_{true}) values. Again the true strain values can be determined from the engineering strain values by the relationship,

$$\varepsilon_{true} = ln(1 + \varepsilon)$$

The elastic modulus or Youngs modulus (E) of steel is defined as the ratio of the engineering stress to engineering strain in the linear elastic region of the materials response. The elastic modulus however can also be defined with negligible error as the ratio of true stress to true strain due to the small strains generally encountered at the yield point.

2.1.4 Poisson's Ratio

For a plate subjected to an axial load a certain amount of strain will be produced in the direction of the applied loading, this however is not the only direction in which strains are produced. Strains are also produced in directions perpendicular to the direction of the loading. These lateral strains are smaller in magnitude and are related to the axial strain by a relationship known as Poissons ratio (ν). Poissons ratio is defined as the ratio of lateral strain to axial strain. Thus for a plate subjected to an axial load as the length of the plate increases the width and thickness will decrease.

2.1.5 Yield Criteria

For most practical applications failure of the material can be said to have occurred with the onset of yielding, therefore a method is required that can predict the initiation of yielding in the material. Firstly a consideration of the stress state of the material should be taken. In this case as the plate is loaded in one direction a state of uniaxial stress is created. The value of stress likely to cause yielding in this case will be equal to the yield stress found from testing a sample of the same material in a tensile test machine, as the stress states are identical. Although this simple criterion is sufficient for simple calculations for uniaxial stress states a more general approach is required for cases of biaxial or triaxial stress. As finite element analysis programs are generally concerned with these more general stress states a further discussion on general failure criteria is warranted.

There are two commonly used theories to predict yielding in ductile materials, these being the Maximum Shearing Stress Criterion and the Maximum Distortion Energy Criterion, which is also known as the Von Mises criterion. The maximum shearing stress criterion states that a given material will yield only when the maximum shear stress in the material is greater than the maximum shear stress in a tensile test specimen of the same material at its yield point. The maximum distortion energy criterion predicts that failure by yielding will occur when the distortion energy per unit volume in the material is greater than the distortion energy in a tensile test specimen at the

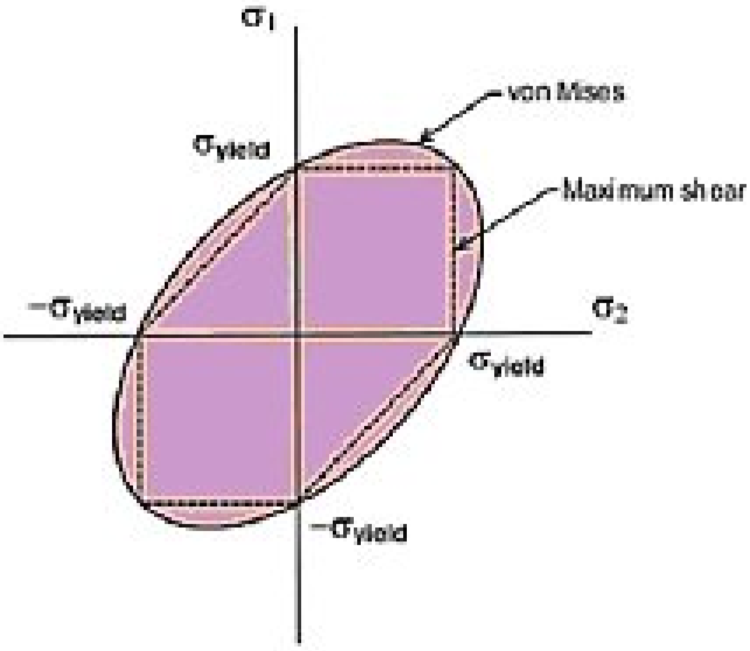


Figure 2.2: Failure Criterion For Mild Steel.

point of yielding. A comparison of both these criterion can be seen for a case of plane stress in Figure 2.2. It should be pointed out that the material would yield if the stress state falls outside bounded region for each criterion. It has been observed that the Von Mises Criterion corelates better with experimental results (Spencer 1968) however in the case of uniaxial stress it should also be observed that both theories would provide identical results.

2.1.6 Work Hardening

No load reversals will be applied in this project however the properties of steel when subjected to yielding in repeated load reversals also needs to be considered as these properties are needed for input into the finite element models. For isotropic, ductile materials yielding is assumed to occur at the same stress in both tension and compression. Therfore if a material has become work hardened by the application of a tensile force the yield strength of the material when subjected to a compressive force should also exhibit the same increase. This characteristic is known as Isotropic Hardening.

However in practice the yield strength of steel when subjected to a load reversal is less than that of the previous loading. This is known as the Bauschinger effect. Materials that exhibit this behaviour are said to display kinematic hardening. As the plates in this project will not be subjected to reversed loading hardening will not need to be accounted for.

2.2 Determination of Steel Properties

To be able to make good comparisons between experimental and modelled results it is necessary that the mechanical properties of the material be accurately determined. For mild steels the elastic modulus, poisson's ratio, and yield strength are generally well known and can be found within manufacturers catalogues or from material specification handbooks. Even though these properties can be easily found from these sources they are generally quoted as a range of values subjected to the manufacturing processes such as hot, cold rolled, or annealed and do not give an indication of the materials post yield characteristics. To obtain an accurate representation of the specific material tensile tests should be conducted on samples cut from the same plate as the experimental test specimens. By obtaining the stress-strain characteristics of the material an accurate material model can be formed for eventual FEA use.

Tensile testing of specimens should be carried out in accordance with Australian Standards. In this case AS1391, Methods for the Tensile Testing of Metals. This standard stipulates the dimensions of standard testpieces and outlines testing procedures and methodologies for the determination of the mechanical properties.

2.3 Stress Concentrations

When a plate containing a hole is loaded uniaxially the stress, and strain, distribution across a section passing through the hole will not be uniform. Instead highly localised stresses will occur around the outside edges of the hole that have a magnitude considerably greater than that given by just dividing the load magnitude by the cross sectional area. An area where this occurs is known as a stress concentration. If the loading on the plate is continually increased the material in this area of higher stress will be the first to yield. As the material yields the load is redistributed to the surrounding material and the plastic zone, where yield takes place, will continue to expand.

2.4 Strain Measurement

2.4.1 General Gauge Characteristics & Requirements

It is an essential component of this project to be able to measure the strain caused by loading. A common method of measuring strains on the surface of components and one that is the most suitable in the majority of cases (BSSM 1979) is by using foil strain gauges. Foil strain gauges have the following characteristics, sensitivity and temperature coefficient of resistance. The sensitivity of a foil strain gauge is termed the gauge factor(GF) and is the ratio between the fractional change of resistance and the strain producing it,

$$GF = (\Delta R/R)/(\Delta L/L)$$

where ,R, is the resistance of the gauge and ,L, is the grid length. One major undesireable characteristic of foil strain gauges is that the resistance of the grid will change with temperature as well as elongation, this is why the temperature coefficient of resistance is important. Unless this effect is accounted for large errors can be introduced into the strain measurements. For guages designed for use at room temperatures however this variation in the gauge factor will be negligible as long as the tests are conducted at room temperature.

The advantages of using foil gauges include, relatively easy installation, the ability for direct strain readout using the correct associated circuitry, and good precision and accuracy. The main criteria for selecting a suitable gauge type for measuring the strain in the plates are,

- Grid configuration
- Grid size

- Maximum elongation
- Grid resistance

For a uniaxial stress state a single grid gauge is suitable for measurements and should be aligned parallel to the axis of the principle stress, which is in turn will be in plane with the applied loading. These strain measurements can be then applied directly with the relationship to the elastic modulus to determine the principal stress value in this plane. This relationship is only valid however in the linear, elastic region.

The grid size must be sized in response to the strain gradients likely to be encountered. High strain gradients are expected around the stress concentration therefore selection of a small grid size for these areas will increase the accuracy of measurements by eliminating some of the strain averaging effects that are characteristic of foil gauges. This effect can be better observed by viewing Figure 2.3.

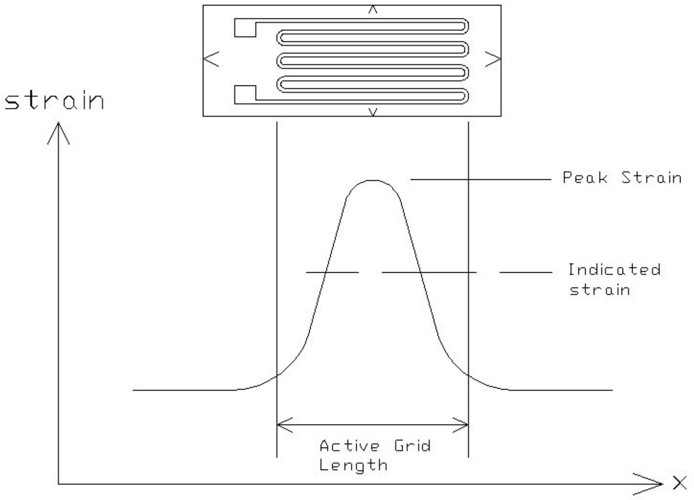


Figure 2.3: Strain averaging effect by gauge measurement.

The disadvantages of selecting a small gauge for accuracy is however offset by the reduction in the maximum elongation possible. Typically the smaller the grid length the lower the maximum permissible elongation therefore a gauge length must be chosen that optimizes this relationship. In practice the choice of gauge length will also be determined by gauge cost and availability.

2.4.2 Ancillary Strain Measurement Requirements

This project makes use of an automated strain measurement and recording system produced by Vishay Measurements Group. The use of this system considerably expedites strain data collection by allowing the user to measure and record strains from multiple strain gauges during the testing time period. This data can then be exported by the system in a standard Microsoft Excel format. This system allows the use of both standard 120Ω and 350Ω resistance strain gauges. The Connection of the strain gauges to the system input card must be through standard 9 pin, male, D-shell connectors.

2.4.3 Strain Gauge Bonding

For accurate strain measurements to be taken the gauges must be bonded to the plates adequately. This requires carefull preparation of the bonding surfaces. The purpose of this is to produce a chemically clean surface having a roughness suitable for the gauge installation requirements, a neutral surface alkalinity, and visible gauge layout lines. There are five basic operations that need to be performed to prepare the bond surface. These are, in order of execution,

- Solvent degreasing
- Surface abrading
- Application of gauge layout lines
- Surface Conditioning
- Surface neutralizing

Degreasing is performed to remove oils, greases and other soluble chemical residues. Degreasing can be accomplished by the application of aerosol degreasing agents. The surface must then be abraded to remove any loosely bonded surface adherents, such as rust or scale, and to develop a suitable surface texture. This can be achieved by grit blasting or by using silicon carbide paper. The area must be finished using silicon carbide paper of the appropriate grit size to provide the required surface roughness value. For general stress analysis work a surface roughness value between 1.6-3.2um RMS is recommended, this can be achieved using a 320 grit paper (BSSM 1979). The area for guage location can then be marked. It is appropriate to burnish the surface rather than score it. This can be acheived using a fine, round pointed object. The area should be marked with perpendicular intersecting lines so that they can be aligned with the gauge backing aligning markings. For this project it will be necessary that these markings be made parrallel to both edges of the plate so that alignment with the principle stress can be achieved. It is recommended that the surface now be conditioned using a conditioner recommended by the gauge manufacturer. The purpose of this is to remove all the fine traces of contaminants from the bond surface. The procedure for this involves wetting the surface with the conditioning agent and then cleaning the area using cotton tipped applicators untill the tips no longer become discoloured. The conditioner must then be cleaned off using a clean gauze sponge. The final step involves bringing the surface back to a near neutral alkalinity. A neutralizing agent can be applied to the surface then again cleaned with cottom tipped applicators and once again dried using a clean gauze sponge. The surface is now ready for strain gauge bonding. It is recommended that bonding take place within 45 minutes of surface prearation (BSSM 1979). The gauges are bonded using an adhesive agent. Strain measurement with bonded resistance strain gauges relies upon the assumption that the surface strain is passed through the adhesive layer to the gauge. The selection of a suitable adhesive is important as the wrong selection can influence the gauge characteristics. Different gauge manufacturers supply suitable adhesives for bonding their gauges to different materials and there rcommendations should be heeded (BSSM 1979). To bond the gauges to the plate a thin layer of adhesive can be applied to both surfaces, the gauge can then be placed in position and a pressure applied untill the adhesive has cured. The time to cure is dependant upon adhesive type and should be found from the manufacturers information.

2.5 Finite Element Analysis

2.5.1 Modelling Approach

The geometry and loading of the plate allows a range of possible modelling approaches. The fact that the plate is loaded uniaxially and has a constant cross section allows the use of plane stress modelling using two dimensional element types. The plate could also be sucessfully modelled using shell elements or even solid brick elements however it is recomended that the simpler plane stress modeling techniques be used where it is appropriate as simpler models are generally more accurate, both from a solution standpoint and due to the fact there is less chance of input error (Adams & Askenazi 1999). The use of geometrical symmetry can also be used in this problem. It is reccomended that if symmetry exists in a problem then it should be used as it will result in shorter run times, more accurate boundary conditions, and greater solution accuracy (Adams & Askenazi 1999). The plates have two planes of symmetry both passing through the centre of the elliptical hole. By using plane stress modelling techniques combined with the use of symmetry only a quarter of the cross section will need to be modelled. This has the advantage of the model requiring a much lower number of nodes and elements, which is of vital importance when using the student editions of both Abaqus and Ansys as they are limited to model sizes not exceeding 1000 nodes.

2.5.2 Loading & Boundary Conditions

The loading and boundary conditions used must accurately model those applied during the tensile tests. Loading and boundary conditions can either be applied onto the solid model or directly to the elements or nodes of the meshed model. The former approach is the best option as it automatically assigns the equivalent condition to each individual node or element face which has the advantage of allowing the mesh to be modified without loss of this data thus increasing the speed of any subsequent mesh modifications.

The axial force applied to the plate can be approximated by applying a uniformly distributed, negative pressure to the end of the model edge. In reality the force is applied by contact between the grips and plate and due to these practicalities it is likely that the pressure distribution will not be completely uniform however due to St Venant's theorem this approximation should still provide an accurate representation of the load distribution at the measurement points as they are at a distance from the load application.

The magnitude of the pressure load can be calculated simply by applying the relationship, P = F/A, where, P, is the pressure, F is the applied force, and A is the cross sectional area at the clamped ends of the plate. It should be noted that this pressure value is uniform over the entire section and therefore will remain the same magnitude even though quarter symmetry is being used.

As symmetry will be used to decrease the model size it is necessary that this condition be accounted for. To achieve this, symmetry boundary conditions can simply be applied to the solid model edges that lie on symmetry planes. This will act to constrain the displacement of nodes along these edges in the direction perpendicular to the symmetry planes.

2.5.3 Assignment of a Material Model

It is commonly assumed that mild steel is a homogeneous and isotropic material. When plasticity is to be expected the material model used must take into account both the elastic and plastic characteristics of the mild steel. Material properties used to build FEA material models generally require that the true stress and true strain properties be used and this is indeed the case for both the software packages used (ABAQUS Inc. 2003)(ANSYS Inc. 1998). For low strains, less than several times the elastic limit, the engineering and true stress and strain values are nearly identical and therefore the engineering values can be used. For analysis of large strains however it is vital that the true stress and strain values be used for material model input.

A common material model used for steel is the elastic-perfectly plastic model. This bilinear model assumes that the material will behave elastically, with a proportionality

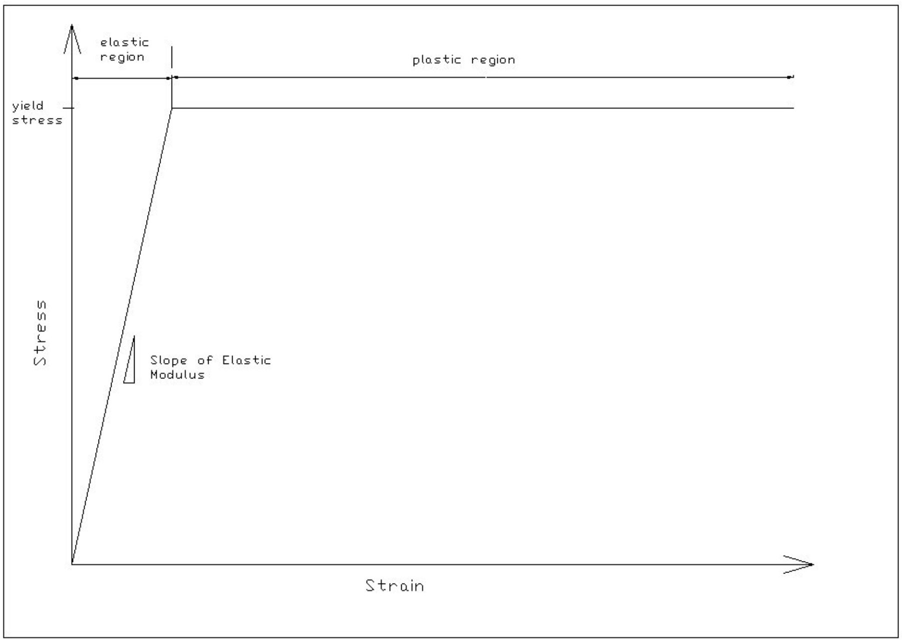


Figure 2.4: Elastic-Perfectly Plastic material model for analysis of plastic strains.

constant of the elastic modulus, up to the yield stress of the material where it will then model the plastic behaviour by allowing continued strain at the constant yield stress. This relationship can be better observed by viewing Figure 2.4. This model is an idealization of the actual properties of mild steel however it can provide a sufficiently accurate representation if some restrictions for its use are not exceeded. The Elastic-Perfectly Plastic model assumes that the material does not strain harden, for low carbon steels that experience YPE this approximation is adequate where the magnitude of plastic strains are not expected to reach far into the strain hardening range.

If the loading on the plate is increased high enough the limitations of the bilinear model will be exceeded and can result in a loss of accuracy or divergence of the solution (Adams & Askenazi 1999). To model the response of the plates at high loads and subsequent large plastic strains it may become necessary to account for the strain hardening range of the steel. In this case Adams & Askenazi (1999) suggests that a multilinear material model may be required that takes into account both elastic, plastic and also hardening regions by linking together a series of linear segments. A typical multilinear material model can be seen in Figure 2.5. The ABAQUS Inc. (2003) analysis guides also show this approach for typical problems involving plasticity for steels.

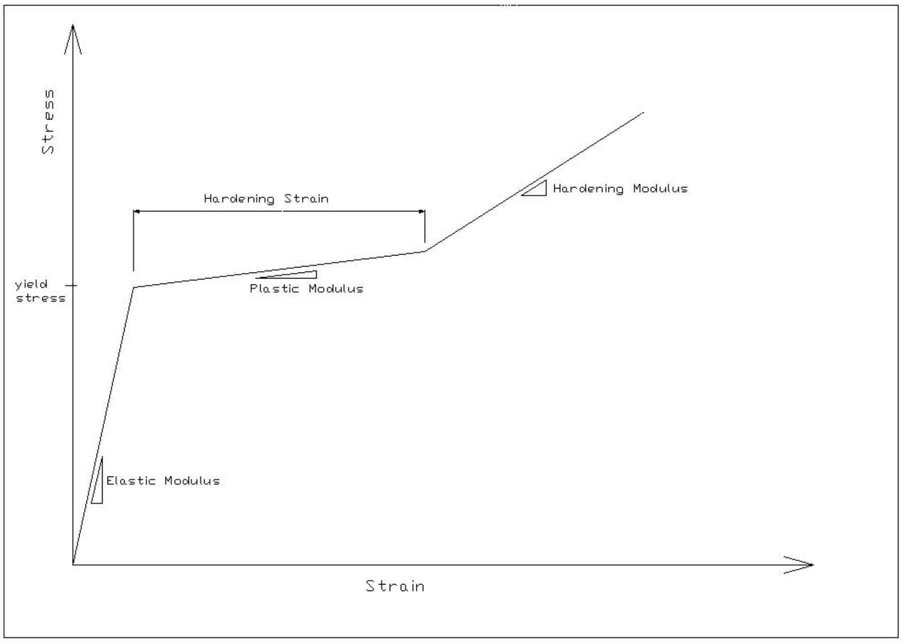


Figure 2.5: Multilinear material model for analysis of large plastic strains.

The yield criteria for use in the model must also be selected. From discussion in previous sections it has been shown that the Von-Mises criterion has provided the most accurate comparisons with experimental results. Both ANSYS and ABAQUS use this criteria as the default setting and this will be the criterion chosen for modelling. Allthough there will be no load reversals both software packages require that a hardening rule be chosen to fully characterize the material. From previous discussion it can been seen that either kinematic or isotropic hardening could be chosen.

2.5.4 Element Selection & Meshing

A variety of two dimensional structural elements are available in both packages. Both 1st and 2nd order, plane stress, isoparametric element types are available in ANSYS and ABAQUS. These are available in both triangular and quadrilateral form. The 8 node elements give better nodal results and have a greater ability to model curved boundaries than other element types(Moaveni 2003, pg312). For this reason they will be used exclusively for modelling. In ANSYS this 8 node element type is named PLANE82. This element uses a 4 point gaussian integration rule. In ABAQUS the 8 noded plane stress quadrilateral is available with full(9 point rule) or reduced(4 point rule) integration and are named CPS8 and CPS8R respectively. It has been suggested that, as the number of integration points used for element formulation is increased the accuracy of the elements decreases, generally erring by being too stiff (Cook 1995). On the other hand it has been suggested in the ANSYS user's manual that to obtain better accuracy when modelling plasticity it is beneficial to use more integration points.

So that a quality comparison can be made between software packages a mapped meshing technique will be used. This will ensure that identical meshes are obtained for each model removing a source of possible differences in results obtained. It will be beneficial to refine the mesh around the position of expected high stress gradients so that a more accurate solution can be obtained in these areas. The use of a mapped mesh has benefits in this case as it allows these areas to be meshed with consistently sized and well shaped elements also providing a smooth transition in element size over the plate length. Another benefit of using a mapped mesh is that it allows the element size to be refined in a systematic manner while determining the convergence of the numerical solution.

Chapter 3

Determination of the Mechanical Properties of the Plate Material

To determine accurate material properties some standard tensile tests were conducted. Following the appropriate Australian Standard, AS 1391, two test specimens were manufactured from the same 3mm thick plate as that of the experimental specimens. The orientation of these test specimens in the parent plate matched that of the experimental specimens so that any anisotropic effects of the rolling process could be matched. This can be better observed by viewing Figure 3.1. The dimensions of the tensile test specimens were determined using the standard and can be viewed in Appendix B.

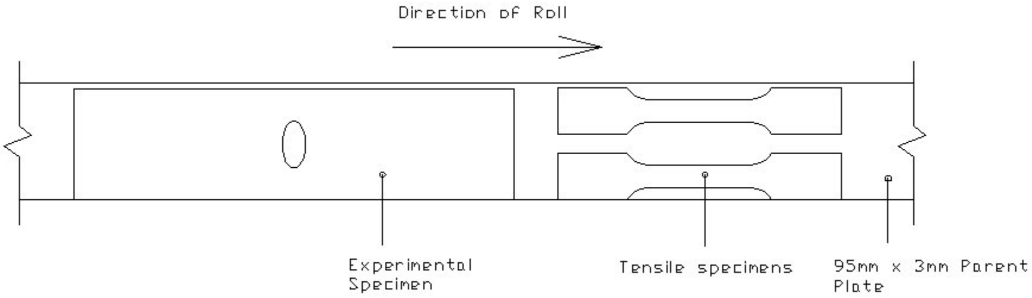


Figure 3.1: Orientation of Tensile Test Specimens in Parent Plate.

The testing of the specimens was undertaken on an automated testing machine. The load and extension of the samples could be monitored via sensors integral to the machine however due to the variation in cross sectional area of the specimens the strain measurements were made by fixing an extonsometer to the specimens gauge length, as shown in Figure 3.2.

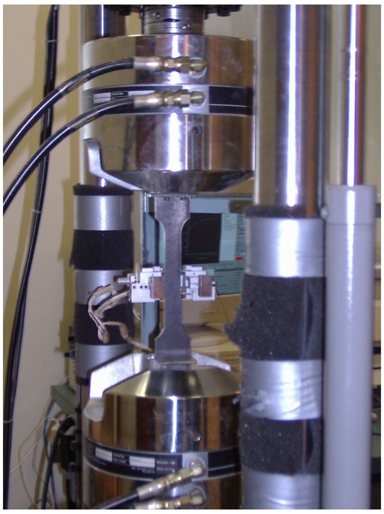


Figure 3.2: Tensile Test Setup

Due to damage to the testing labarotoy's high extension extensometer a substitute device had to be used. Limitations existed however with using this extonsometer. The maximum permissible strain that could be measured was 0.3%. This limitation severely restricted the ability to determine the steels characteristics accurately at strains in excess of the limit. The materials elastic modulus, upper and lower yield strengths could however be determined from this strain range.

To obtain a more accurate representation of the material properties, and decrease the chances of including an uncharacteristic sample, two samples were tested. The stress-strain diagrams produced, along with the calculated properties, can be seen in Figures 3.3 and 3.4.

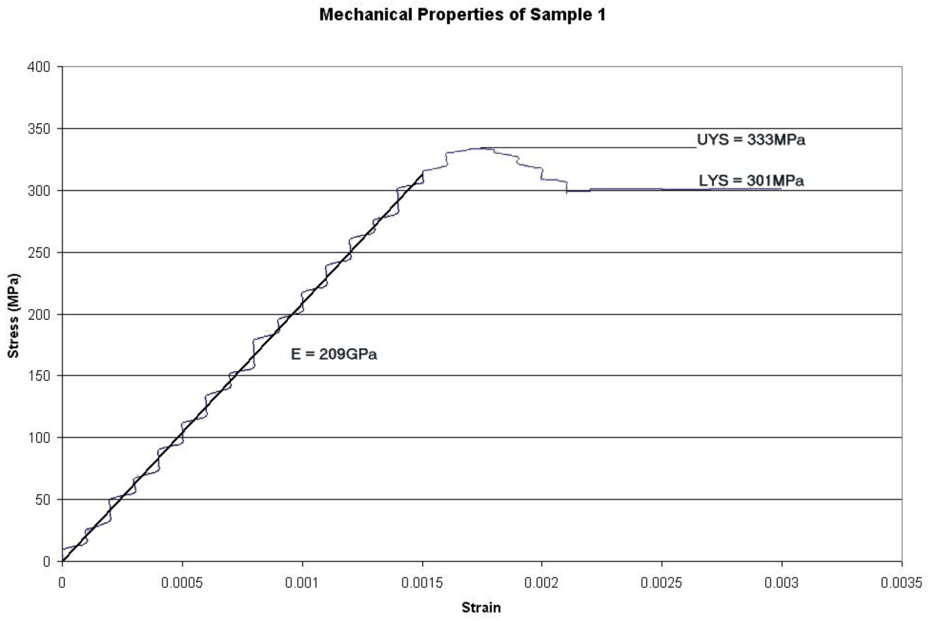


Figure 3.3: Stress-Strain Diagram for Test 1

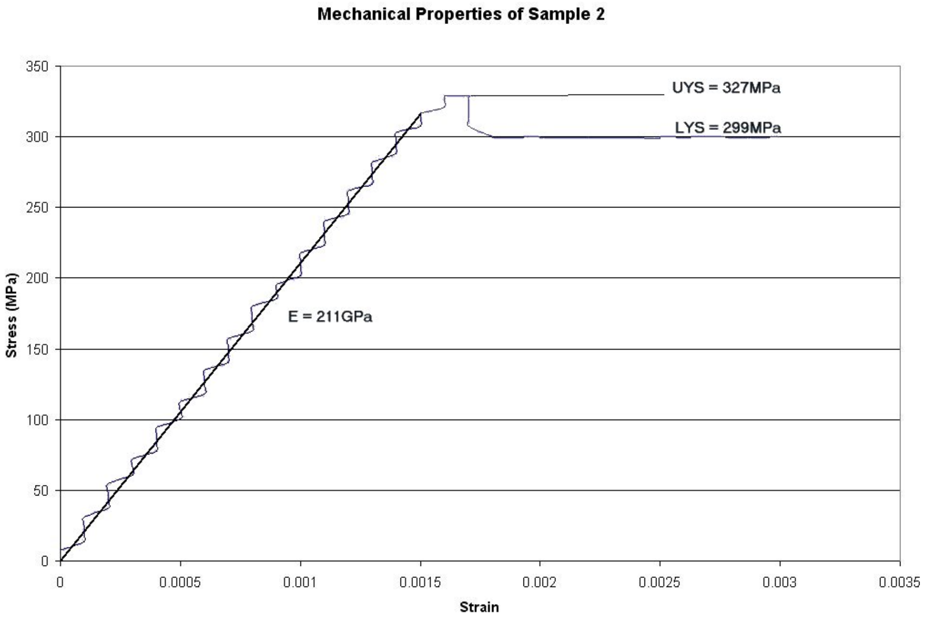


Figure 3.4: Stress Strain Diagram for Test 2

It can be seen that the variation in the mechanical properties between samples was

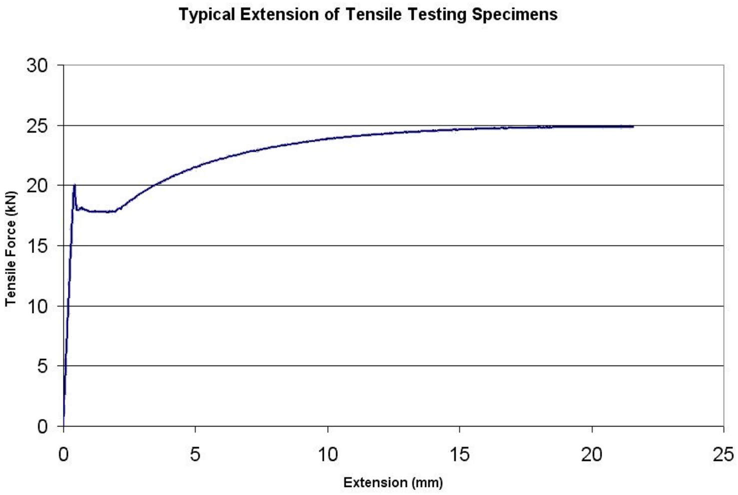


Figure 3.5: Extension of test specimen during testing.

small and corresponded well to the recognised range of values for AISI 1010 steel as found from material property databases. By taking a rounded average of the properties of each sample the mechanical properties found can be seen in Table 3.1.

	Sample 1	Sample 2	Overall
E	209GPa	211GPa	210GPa
UYS	333MPa	327MPa	330MPa
LYS	301MPa	299MPa	300MPa

Table 3.1: Determined Mechanical Properties

Allthough the strain data restrictions limit the determination of accurate data above 0.3% the general behaviour of the material can be observed by viewing a typical force-extension diagram, as shown in Figure 3.5, which shows the typical response for mild steel up to approximately its ultimate strength. It can be seen that the steel does strain harden after a period of constant load yielding.

Poisson's Ratio was not determined during these tests and thus a recognised value for mild steel of 0.3 (Beer & Johnson 1992)(Juvinall & Marshek 2000) can be assumed.

Chapter 4

Measurement of Strains in Perforated Plates

The first step in conducting the experiment was to determine the dimensions and material specifications of the experimental test plates. The design of the plates needed consideration of factors such as,

- Testing machine limits.
- Manufacturing limits.
- Material availability.
- Strain gauge placement requirements.
- Provision of a near uniform stress distribution adjacent to the clamp face.

The 500kN tensile testing machine in the University of Southern Queensland's Toowoomba campus engineering materials testing laboratory was selected for use due to it's ability to provide automatic readout of extension and force values. This machine has mechanical wedge lock jaws that can accomodate up to 100mm wide plates providing the thicknesses is greater than 1.5mm. By taking these limits into account it was proposed that that the plate width be 95mm so that a small clearance between the edges of the jaw could be maintained ensuring even loading be applied across the entire plate width.

With the plate width set at 95mm the dimensions of the elliptical holes could now be decided. The major criteria for selection of ellipse dimensions was to allow sufficient space for multiple strain gauges to be mounted between the ellipse centreline and the plate edge. On preliminary inspection of easily available strain gauge sizes it was determined that the major radius of the ellipse could be 20mm and thus as the ratio of major to minor radius was to be 2 the minor radius was set as 10mm. Manufacturing of the ellipse would require a NC slot drilling operation which was deemed possible using the available USQ workshop machine. The length of the plate was made upon consideration of the clamping area required by the tensile testing machine and test length required so that a near uniform stress distribution could be maintained adjacent to the clamp jaws. This length was approximated by conducting some preliminary FEA using a linear elastic model with approximate material properties. The FEA methodology used is described further in Chapter 5. From this preliminary analysis an approximate test length of 200mm would be needed. The length required for clamping was 90mm at each end thus the total plate length required was to be 380mm. Workshop material availability led to the use of 3mm thick AISI 1010 carbon steel plate. Fully dimensioned drawings of plate dimensions can be found in Appendix B.

The aim of the project was to make a comparison between FEA and experimental results. There were no particular locations that must be measured but rather there existed areas on the plate where it was of interest to make comparisons. The obvious choice for one such measurement is on the outside edge of the elliptical holes. To gain insight into the approximate strain distribution in the plate some preliminary linear, elastic FEA models were created. it was observed that the path of increasing strain led from the edge of the hole to the outside edge at roughly 45 degrees. 4 strain gauges were available for use on each test plate so it was proposed that two of these gauges could be used to measure strains at locations around the stress concentration(positions 1 and 2), another measurement could be taken around the edge of the plate (position 3), and another to capture the averaged stress at a location far from the stress concentration(position 4). The results of the preliminary studies can be seen in Figure 4.1 which also shows the proposed gauge measurement positions.

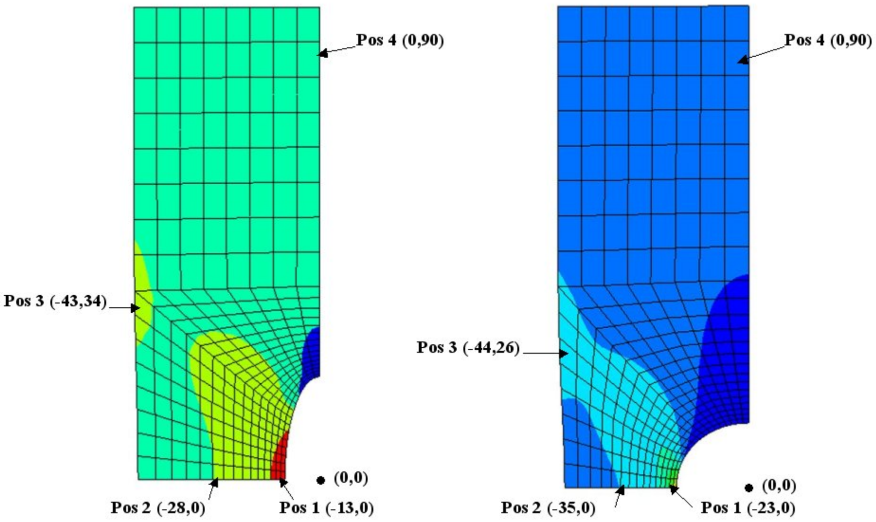


Figure 4.1: Axial strain distributions determined by preliminary FEA showing proposed strain gauge positions.

With the measurement positions determined the strain gauges could now be bonded to the surface. The gauges chosen were of a uniaxial grid configuration each having a gauge length of 3.18mm, gauge factor of 2.08, and a maximum elongation of around 1.5-2%. The gauges were bonded and connected to the measurement system as discussed in Section 2.4. Both plates were tested and strain data gathered untill a significant amount of plastic deformation was observed. The relationship between the axial force and the strains produced can be observed in Figures 4 and 4.3.

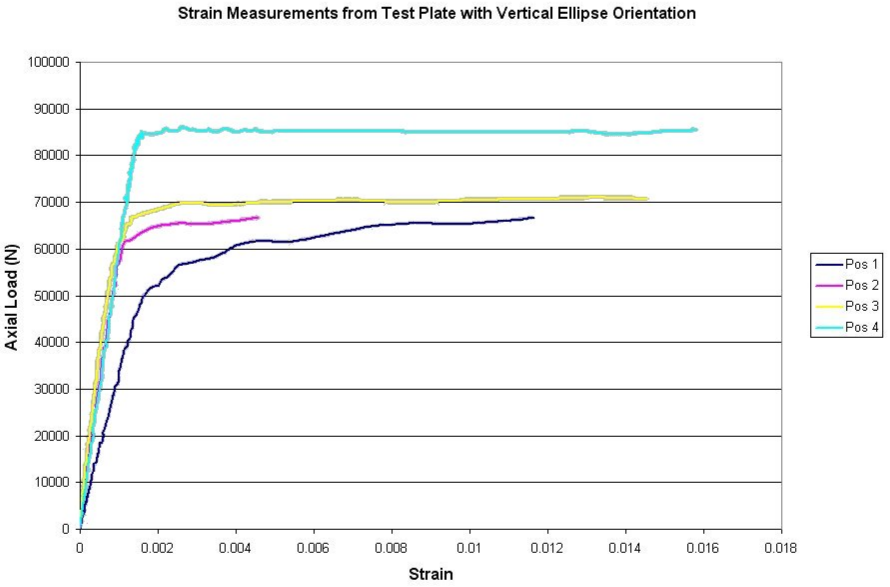


Figure 4.2: Strain data gathered from testing plate containing vertically oriented elliptical hole.

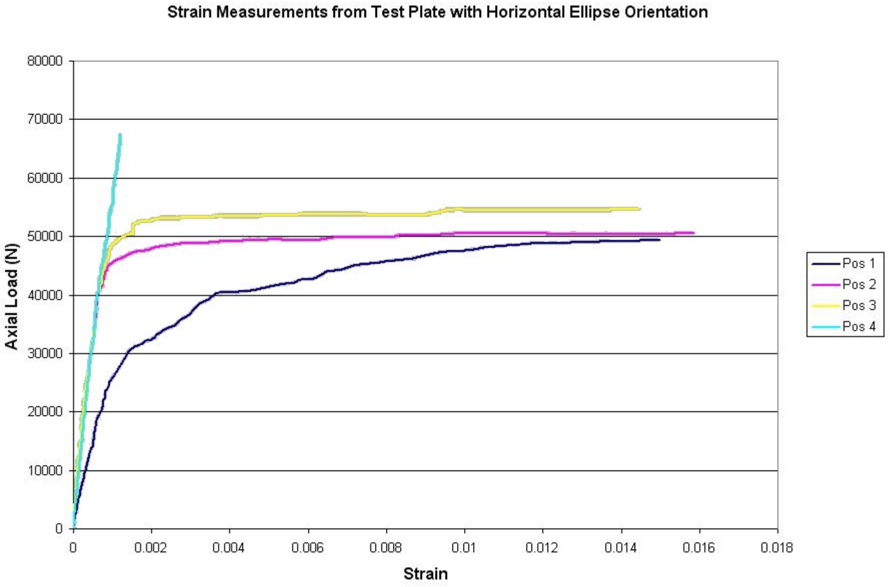


Figure 4.3: Strain data gathered from testing plate containing horizontally oriented elliptical hole.

The strain results show the characteristics expected with the data nearest the holes showing a significantly higher strain than at the other positions due to the stress concentration effects. Positions 1 on both plates don't exhibit the distinct linear elastic to plastic transition as displayed in the other positions but instead show a curving transition which is indicative of the large strain gradients in this area and the averaging effects of the strain gauge measurement. Positions 2-4 on both plates show an initial linear response. This can be expected as these positions are not subject to the high strain gradients caused by the discontinuity. It can be observed that as the loading is increased the localised region of high strains expands toward position 2 and then outward to position 3 with yielding occuring in this order.

A further observation of the strain data for position 4 in Figure 4 indicates that it is possible to determine the steel's mechanical properties if the assumption of a uniform stress distribution across a section near the clamps is taken. An excellent correlation between the stress-strain diagrams as found by testing the "dogbone" specimens and that from the strain gauge measuring point exists which suggests that the limitations of the extonsometer can be neglected and that the yield point elongation is at least 1.6%. It should be noted here that the distinct UYS shown by tensile testing the dogbone specimens is not shown in the strain gauge data. This is most likely due to the fact that this test was conducted a slower strain rate which can act to suppress this effect, as discussed in Section 2.1. This comparison can be observed in Figure 4.4.

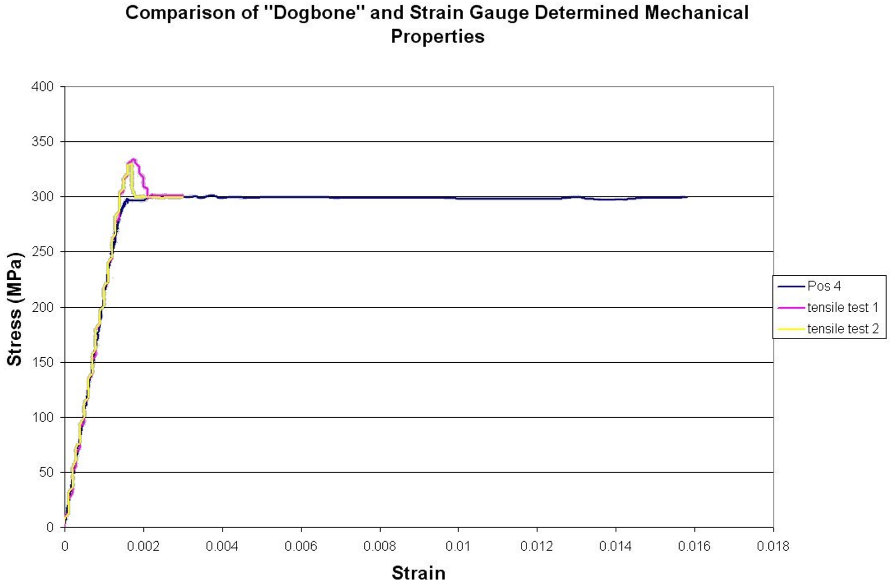


Figure 4.4: Comparison between "dogbone" and testplate determined mechanical properties.

It was also found that the actual maximum extension of the strain gauges was around 1.5% however in some cases such as position 2 in Figure 4 a lower maximum strain reading was obtained. This was most likely a result of poor bonding between the plate and gauge causing relative motion between the surfaces.

Chapter 5

Finite Element Analysis

5.1 Model Creation

A basically identical process was used to build the plane stress models in both analysis packages. In both cases only a quarter of the plates geometry was required to be modelled. Only the portion of each test plate between opposing clamps was considered for model formulation. Symmetry boundary conditions were applied to both planes of symmetry and a uniformly distributed presssure load applied to the model geometry so that an equivalent axial loading as measured in the experimental procedure be applied. The plane stress, 8 node isoparametric element types available in both packages were used. In ANSYS this element type uses a 4 point integration rule. In the ABAQUS package both full and reduced integration varieties were used which use 9 and 4 point rules respectively. The models were meshed using a mapped meshing technique so that identical models could be created in each case and a beneficial mesh density could be obtained around the hole edge. Typical models used showing the meshing technique as well as the loading and boundary conditions used can be observed in Figure 5.1.

An elastic-perfectly plastic material model was specified. The values used to define this model were the elastic modulus and LYS as shown in Table 3.1. The engineering stress value was used rather than the true stress value, the difference between them found to be negligible at such low strain values. The default flow rule and yield criteria were

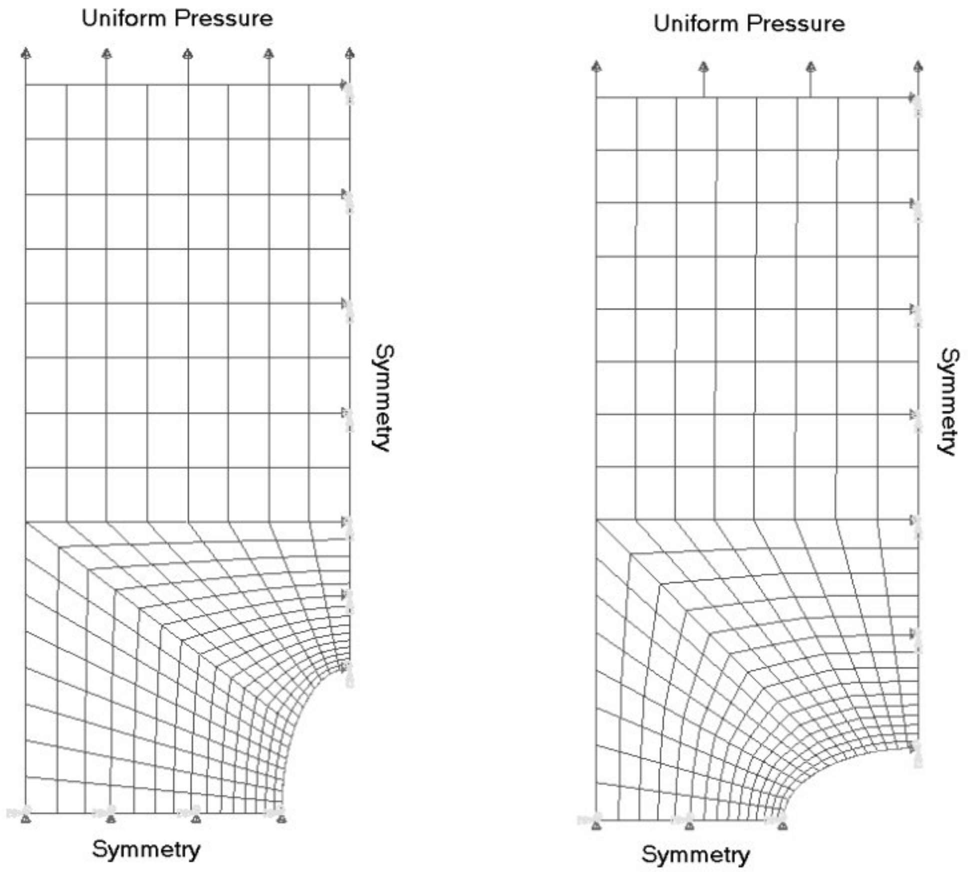


Figure 5.1: Typical models used for analysis showing loads, symmetry boundary conditions and mapped meshing technique used.

used and, allthough of no importance in this analysis, isotropic hardening specified.

5.2 Solving & Data Output

In each case results were written at 10 equal load inrements during solving. This was done so a force-strain curve could be plotted for comparison to the expermental data. It was found that the use of the elastic-perfectly plastic material model restricted the maximum load possible to be applied with excessive loads causing the solution to diverge. This can be better defined in practical terms as the load at which an entire plane through a cross section reached the elastic limit thereby, due to the assumption of perfect plasticity, allowing plastic collapse. As a result of this the loading applied in each case was kept just below this limit which was well below the actual loading applied during testing of the plates.

The axial strains determined at the centroids of elements at the same coordinates as the measurement points were used for comparison to the measured strains. Allthough the elements at the measurement points were not the same size as the gauges it was observed that the strain gradient at most measurement locations was low and therefore an averaged elemental strain was assumed to present a reasonable portrayal of the strain over the entire element. As strain gauges essentially take an averaged strain measurement over their gauge area this technique was considered reasonable. For the measurement positions close to the hole edge a high strain gradient exists and therefore caution of using this approach is warranted. In these cases however the element size closely matched the strain gauge grid size and therefore, similarly, the centroidal strains were assumed as a reasonable representation of the whole measurement point.

5.3 Results & Discussion

A comparison between the axial strains from the analysis results and the experimental results can be observed in Figures 5.2 to 5.9.

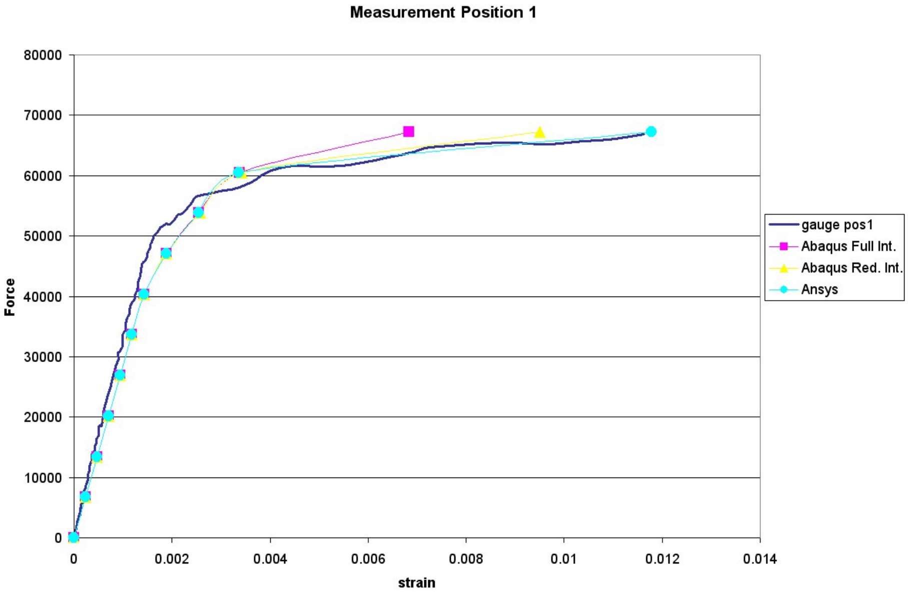


Figure 5.2: Results of FEA by ANSYS and ABAQUS for vertically oriented hole compared to experimental measurements at position 1.

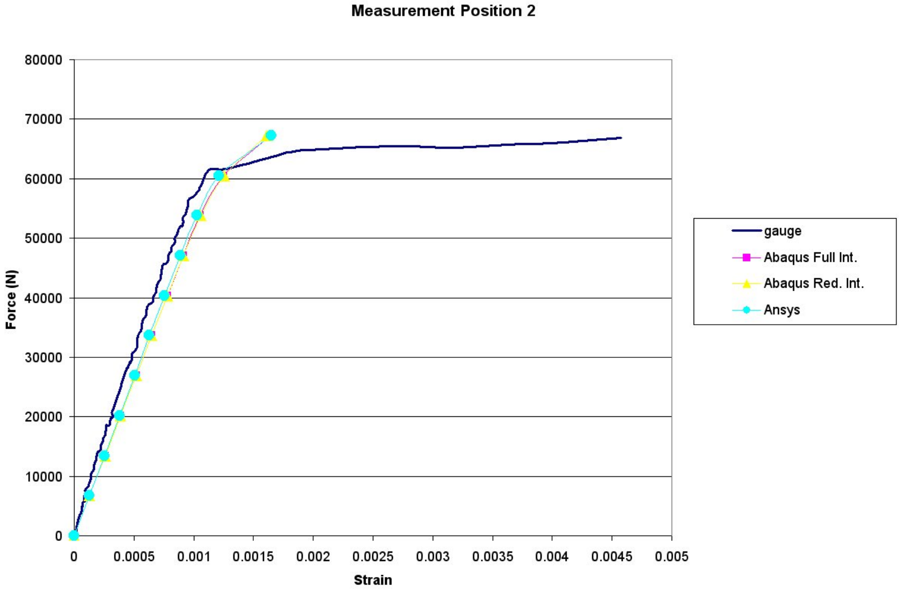


Figure 5.3: Results of FEA by ANSYS and ABAQUS for vertically oriented hole compared to experimental measurements at position 2.

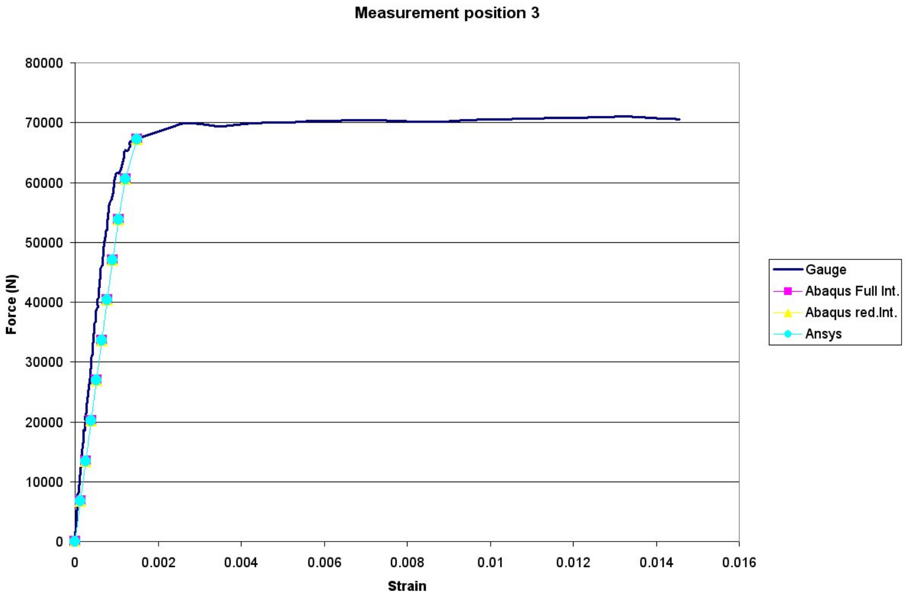


Figure 5.4: Results of FEA by ANSYS and ABAQUS for vertically oriented hole compared to experimental measurements at position 3.

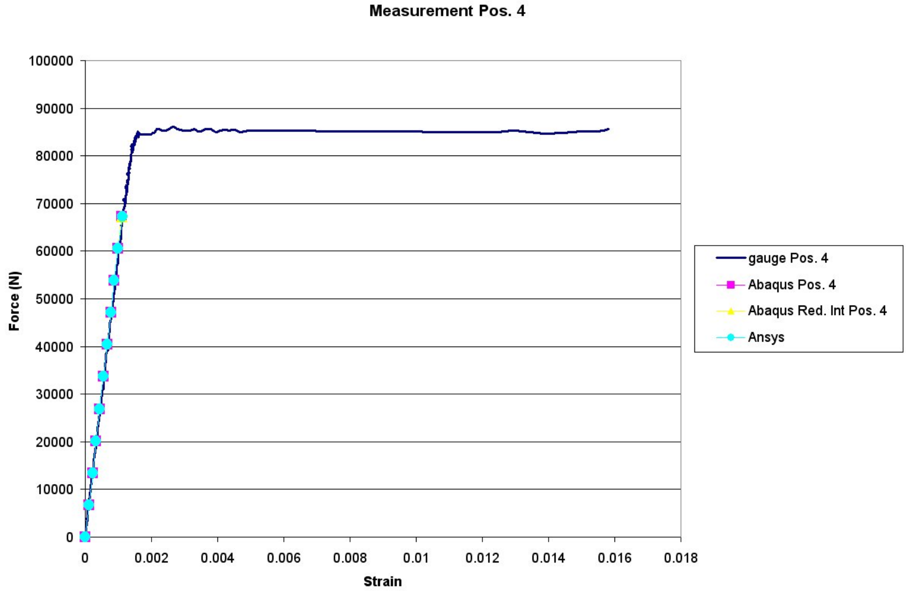


Figure 5.5: Results of FEA by ANSYS and ABAQUS for vertically oriented hole compared to experimental measurements at position 4.

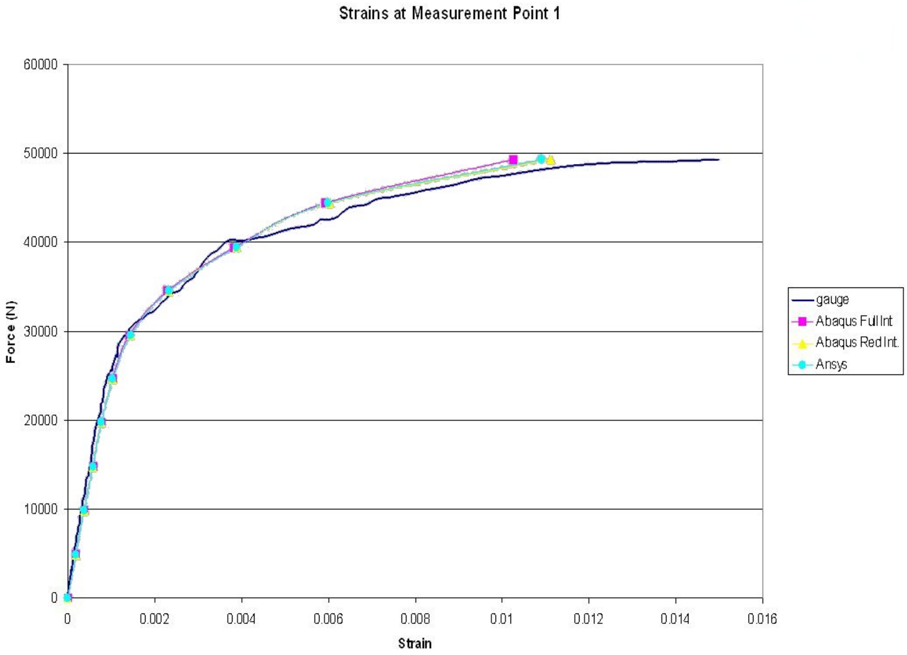


Figure 5.6: Results of FEA by ANSYS and ABAQUS for horizontally oriented hole compared to experimental measurements at position 1.

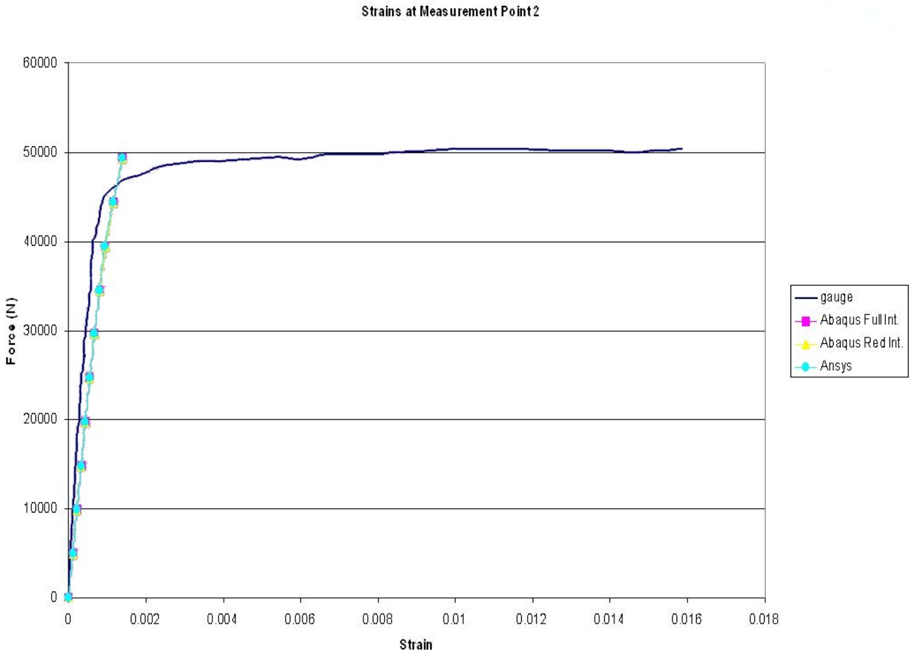


Figure 5.7: Results of FEA by ANSYS and ABAQUS for horizontally oriented hole compared to experimental measurements at position 2.

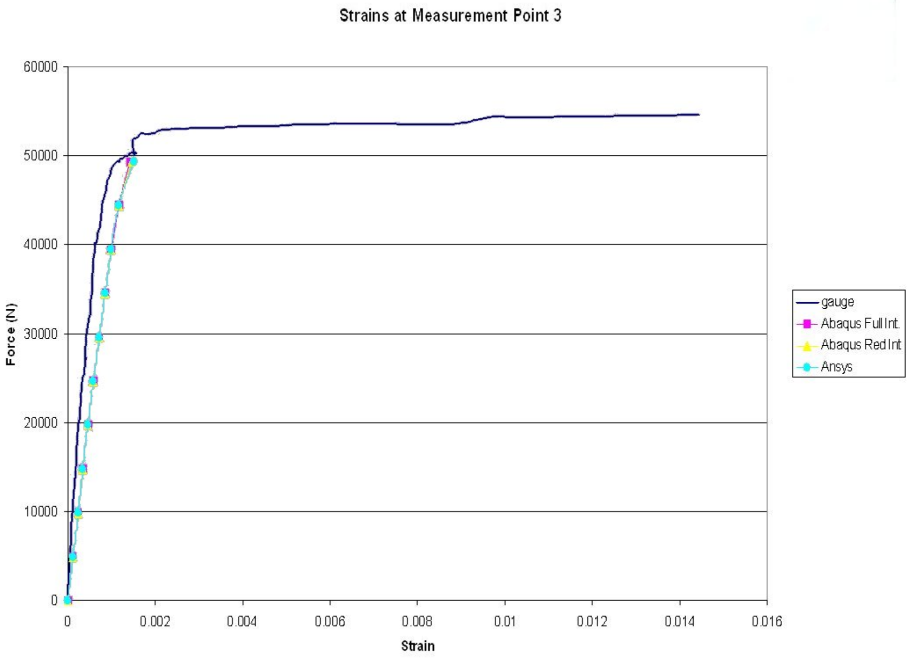


Figure 5.8: Results of FEA by ANSYS and ABAQUS for horizontally oriented hole compared to experimental measurements at position 3.

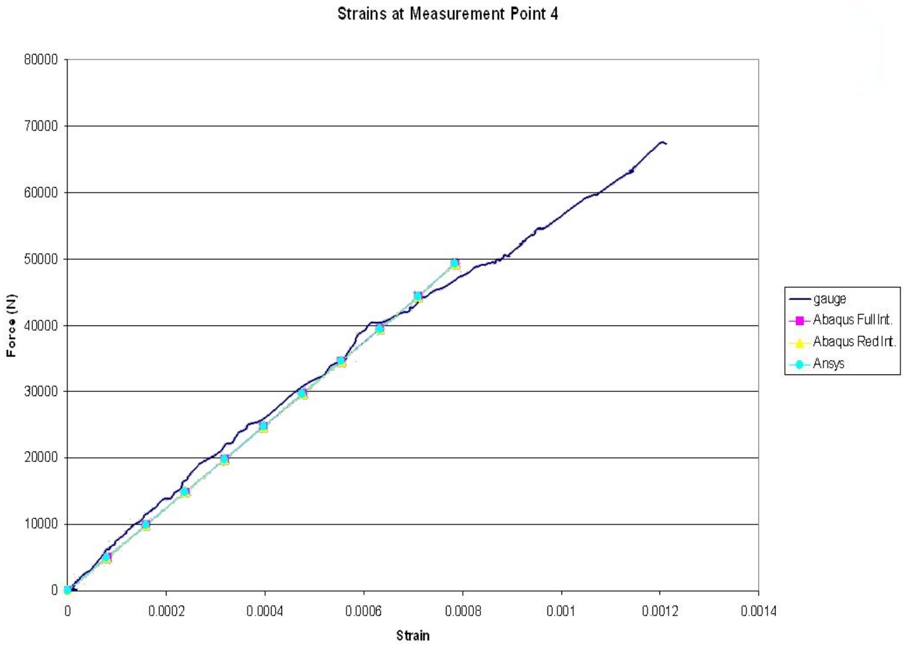


Figure 5.9: Results of FEA by ANSYS and ABAQUS for horizontally oriented hole compared to experimental measurements at position 4.

It can be observed that results obtained by ANSYS and both element types in ABAQUS were essentially identical during most of the load range. A variation in the results however can be observed at higher strains as observed in figures 5.2 and 5.6. It can be noted that the full integration element type in ABAQUS tended to present stiffer results than the reduced integration variety which was expected and discussed in Section 2.5.

In most cases a good correlation existed between the experimental model data. It can be seen that, excluding positions 1, the modelled strains were well under the maximum strain limits of the gauges and were not large enough to cause yielding at all measurement positions which limited the ability to make a good comparison around the yield points.

There are many factors that would influence the accuracy of both the experimental and modelled data. This includes the inherent approximation characteristic of the finite element method. Some factors that could have produced discrepancies in the experimental results include,

- Innacuracies in the force measurement equipment
- Innacuracies due to strain gauge and measurement system tolerances
- Misalignment of strain gauges to plate axis
- Misalignment of plate axis to the plane of loading
- Slippage of gauges relative to plates due to poor bonding

The method used to collect strain data from the model for comparison was not ideal and a potential source of error. This could be reduced by decreasing the element size around the measurement points. The accuracy of the model, in general could likely be improved by using a smaller mesh size, unfortunately this could not be achieved during this project due to being restricted to the use of only 1000 nodes.

Potential errors could have also been created by modelling the applied load by assuming a uniform pressure distribution across the plate end face. In reality the pressure distribution accross the plate ends is not expected to be uniform but vary somewhat due to the stiffness variations within the plate caused by the holes and the method of load application through contact by taper lock serrated jaws. As the ratio of the hole dimensions and the distance to the endfaces was relatively large the assumption of a uniform pressure distribution was likely however to cause insignificant error due to the application of St Venant's Principle.

Further assumptions made regarding the steels properties could also be a source of errors. In reality it is possible that rolled steel plate would not be isotropic and homogeneous due to inclusions and anisotropic effects caused by the rolling process. The mechanical properties of the steel are also likely to vary throughout the plate section. Given the accurate testing of the material properties performed during this project, and the small size of the plates, these effects would most likely be negligible.

To extend the loading that can be applied the use of a multilinear material model that includes more plastic data points incorporating the strain hardening characteristics of the steel could be used. A multi-linear material model was not used in this project. This was largely due to the fact that accurate strain hardening data could not be determined due to the limitations of the extensometer. It is expected however that by using a more comprehensive material model a significantly higher loading could be applied to the model and subsequently a broader range of strain results obtained for comparison.

Chapter 6

Conclusions and Further Work

6.1 Achievement of Project Objectives

The project objectives were achieved and a comparison between the experimentally measured strains in perforated steel plates under tension and those obtained from finite element analysis made. By using a plane stress modelling approach using 8 node quadrilateral elements and by making some simplifying assumptions regarding material behaviour and loading a good correlation between the results was obtained. By reviewing the plastic behaviour of steel a model was formulated that succesfully predicted the yielding in the plates at low plastic strains. Both ANSYS v5.5 and ABAQUS v6.4 software packages were used to create the models and the results obtained from both packages found to compare favourably. It was found that both software packages offered similar controls over the model building, solving, and post processing process which enabled easy transition between packages. In general it was found that the ABAQUS package was easier to learn and use however the help file system available in the ANSYS package presented more information about element use and offered more insight into analysis procedures.

6.2 Further Work

It would be interesting to conduct some further work into modelling the plates at higher loads and subsequently larger strains. In this project the maximum loading that could be applied to the plates was limited by the use of an elastic-perfectly plastic material model however by conducting some additional material tests a more comprehensive material model could be defined which takes into account the extra strength of the material after yielding caused by strain hardening. Further to this it would be of interest to make comparisons with results obtained from models built using other element types such as p-element varieties.

References

ABAQUS Inc. (2003), ‘ABAQUS Version 6.4 Documentation Collection’.

Adams, V. & Askenazi, A. (1999), *Building Better Products with Finite Element Analysis*, Onword Press.

ANSYS Inc. (1998), ‘ANSYS/ED Help System Release 5.5’.

Barber, J. (2001), *Intermediate Mechanics of Materials*, McGraw Hill.

Beer, F. & Johnson, E. (1992), *Mechanics of Materials*, McGraw Hill.

BHP. (2003), ‘BHP excelerplate catalogue’.

BSSM (1979), ‘BSSM Strain Measurement Reference Book’.

Bussler, M. (2003), ‘Nonlinear fea 101’, *Machine Design* pp. 50–53.

Cook, R. (1995), *Finite Element Modelling for Stress Analysis*, John Wiley & Sons.

Dalley, J. & Riley, W. (1991), *Experimental Stress Analysis*, McGraw Hill.

Datsko, J. (1966), *Material Properties and Manufacturing Processes*, John Wiley & Sons.

Davis, J., ed. (2004), *Tensile Testing*, second edn, ASM International.

Dowling, N. (1999), *Mechanical Behaviour of Materials*, Prentice Hall.

Jarvstrat, N. (2004), ‘Constitutive model validation’, *NAFEMS Education and Training* pp. 10–13.

Juvinall, R. & Marshek, K. (2000), *Fundamentals of Machine Component Design*, John Wiley & Sons.

Kurowski, P. (2001), 'Easily made errors mar fea results', *Machine Design* pp. 69–74.

Moevani, S. (2003), *Finite element analysis Theory and application with Ansys*, Pearson Education.

Spencer, G. (1968), *Introduction to Plasticity*, Chapman & Hall.

Standards Australia (2005), 'AS1391-2005, Metalic Materials - Tensile Testing at Ambient Temperature'.

Zienkiewicz, O. & Taylor, R. (2000), *The Finite Element Method*, Butterworth Heinemann.

Appendix A

Project Specification

University of Southern Queensland
Faculty of Engineering and Surveying

ENG4111/4112 Research Project

Project Specification

For: Ben Field

Topic: Stress Analysis of Axially Loaded Steel Plates by Experimental and Finite Element Methods.

Supervisors: Amar Khennane
Chris Snook

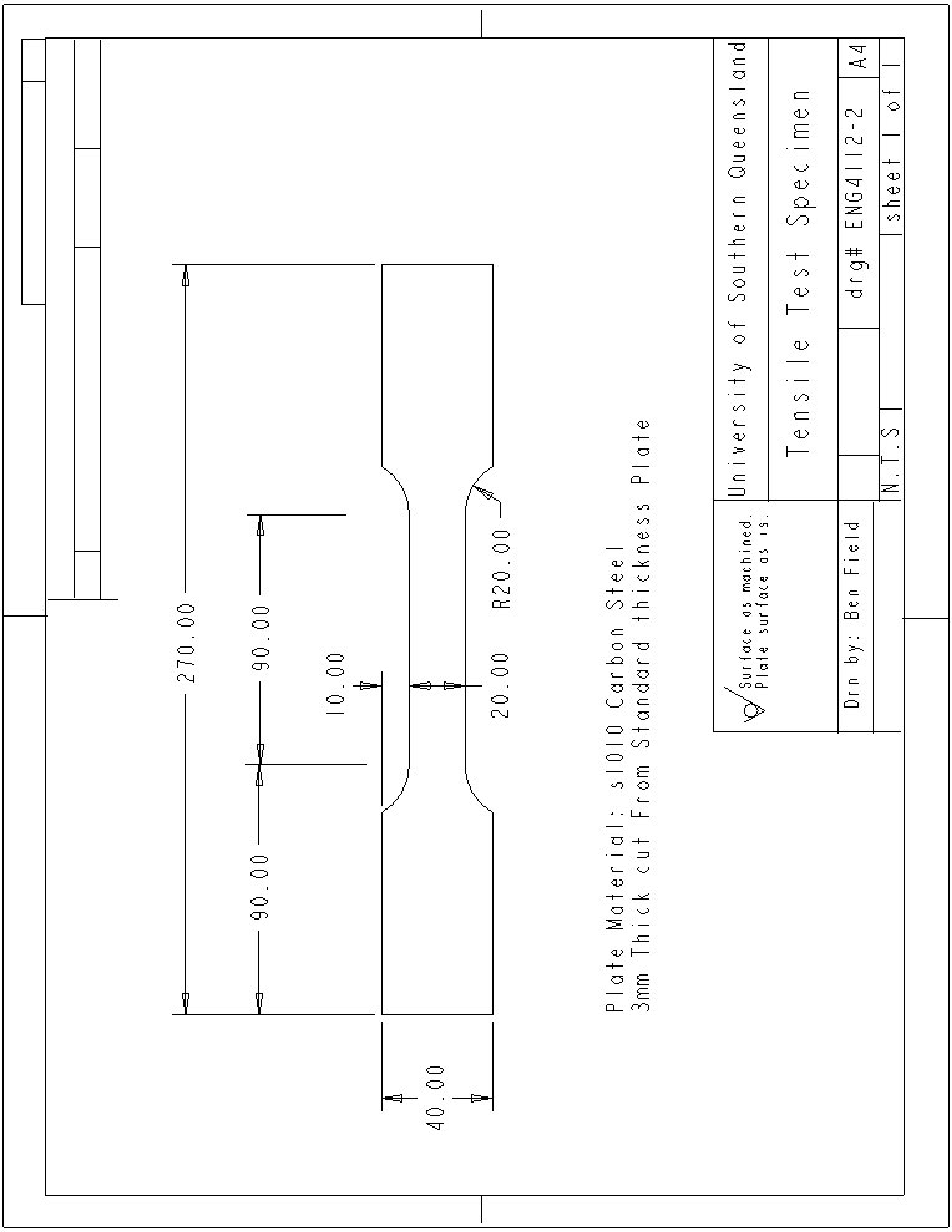
Project Aim: The project aims to compare the experimentally measured strains in perforated steel plates with those obtained from two different commercial finite element modelling software packages, namely, ABAQUS and ANSYS.

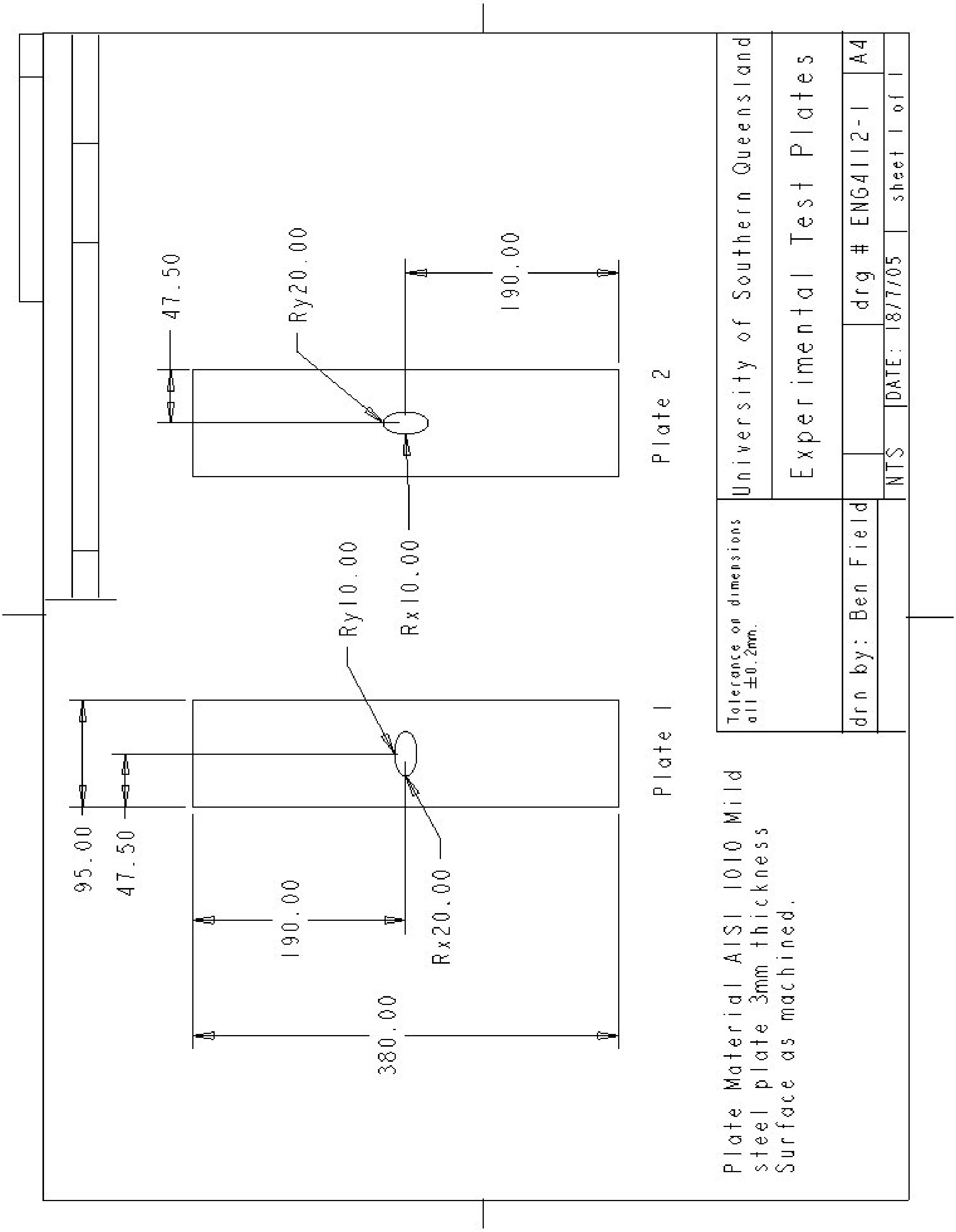
Programme: Issue A, 25th April 2005

1. Review the theory of plasticity in metals.
2. Review and select finite element analysis modelling techniques suitable to model the physical situation.
3. Design and manufacture appropriate test pieces for use in a tensile test machine.
4. Measure the strains produced in critical areas of the test piece when being subjected to a tensile load.
5. Model the experiment using both ABAQUS and ANSYS finite element modelling software packages using the suitable techniques found in (2).
6. Compare the results of the finite element models to the experimental results obtained.

Appendix B

Dimensions of Test Samples


University of Southern Queensland
Tensile Test Specimen
drg# ENG4112-2
A4
sheet 1 of 1
N.T.S
Surface as machined.
Plate surface as is.
Drn by: Ben Field
Plate Material: s1010 Carbon Steel
3mm Thick cut From Standard thickness Plate
270.00
90.00
90.00
10.00
20.00
R20.00
40.00


95.00
47.50
190.00
380.00
Rx20.00
Ry10.00
Plate 1
Rx10.00
Ry20.00
47.50
190.00
Plate 2
Plate Material AISI 1010 Mild steel plate 3mm thickness Surface as machined.
Tolerance on dimensions all ±0.2mm.
drn by: Ben Field
University of Southern Queensland
Experimental Test Plates
drg # ENG4112-1
A4
NTS
DATE: 18/7/05
sheet 1 of 1