

Study on post buckling behavior of plate girder with web opening

Submitted in partial fulfilment of the requirements for the award of

Master of Engineering Degree in Structural Engineering

by

AKASH NITHISH K M (39380001)



**DEPARTMENT OF CIVIL ENGINEERING
SCHOOL OF BUILDING AND ENVIRONMENT**

SATHYABAMA

**INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)**

Accredited with Grade "A" by NAAC

JEPPIAAR NAGAR, RAJIV GANDHI SALAI, CHENNAI - 600 119

APRIL 2021



SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY) Accredited with "A" grade by NA
Jeppiaar Nagar, Rajiv Gandhi Salai, Chennai – 600 119
www.sathyabama.ac.in



DEPARTMENT OF CIVIL ENGINEERING

BONAFIDE CERTIFICATE

This is to certify that this Project Report is the bonafide work of AKASHNITHISH K M (39380001) who carried out the project entitled **“Study on post buckling behavior of plate girder with web opening”** under my supervision from November 2020 to April 2021

Internal Guide

(Mrs. R. Nirmala)

Head of the Department

Submitted for Viva voce Examination held on 16/04/2021

Internal Examiner

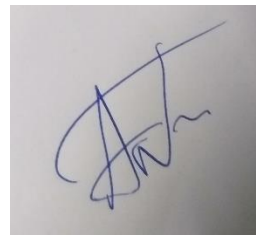
External Examiner

DECLARATION

I **AKASHNITHISH K M** hereby declare that the Project Report entitled “**Study on post buckling behavior of plate girder with web opening**” done by me under the guidance of **Mrs. R Nirmala** at **SATHYABAMA INSTITUTE OF SCIENCE AND TECHNOLOGY** is submitted in partial fulfilment of the requirements for the award of Masters of Engineering degree in Structural Engineering.

DATE: 12/04/2021

PLACE: CHENNAI



SIGNATURE OF THE CANDIDATE

ACKNOWLEDGEMENT

I am pleased to acknowledge my sincere thanks to Board of Management of **SATHYABAMA** for their kind encouragement in doing this project and for completing it successfully. I am grateful to them.

I convey my thanks to **Dr. DEVYANI GANGOPADHYAY., B.Arch., M.T.P., Ph.D.,** Dean, School of Building and Environment and **Dr. S. Packialakshmi,, M.E., Ph.D.** and **Dr. R. PADMAPRIYA, M.E., Ph.D.** Head of the Department, Dept. of Civil for providing me necessary support and details at the right time during the progressive reviews.

I would like to express my sincere and deep sense of gratitude to my Project Guide **Mrs. R. Nirmala., M.E., (Ph.D)** for his valuable guidance, suggestions and constant encouragement paved way for the successful completion of my project work.

I wish to express my thanks to all Teaching and Non-teaching staff members of the Department of Civil who were helpful in many ways for the completion of the project.

ABSTRACT

The conventional I beam, which is made from different structural steel plates / parts instead of a single rolled steel cross section, is welded or bolted or rotated together so that the vertical mesh and horizontal edges of the beam are called plate girders. Plate blocks are commonly used as a structural element in the construction of buildings, bridges, ships and marine utilities, which help to run services such as pipes and tubes beneath the web panel. Reducing the load-bearing capacity of web openings on steel plate edges, redistributing stress and suspending the structure.

This study presents a final strength analysis of web holes in plate girders subject to shear and curvature, as well as testing and numerical analysis for post-buckling behavior. Two plate ropes 1.5 m long. The first girder section measures 500 mm x 120 mm, while the second girder section measures 400 mm x 120 mm.

Two specimens of each plate were tested to determine the mechanical properties of the plate curvature components. Two plate ropes were examined in the laboratory under concentrated loads applied at medium intervals to assess the final shear power. The steel behaviour of the girders under the control of loads was analysed using the software application, which resulted in a large number of plate girders being checked (ANSYS). Finally, the results of the ANSYS Workbench were used to evaluate the numerical analysis failure mode, including load-deflection curves, strength reliability, and analysis results, and compare them to the results of the specified element analysis.

TABLE OF CONTENTS

CHAPTER NO	TITLE	PAGE NO
	Abstract	v
	List of Tables	viii
	List of Figures	ix
1	Introduction	1
	1.1 General	1
	1.2 Elements of Plate Girder	2
	1.3 Depth of Plate Girder	2
	1.4 Providing Stability against Web Buckling	3
	1.5 Plate Girder with Web Openings	3
	1.6 Failure modes plate girder with web opening	4
	1.7 Advantages plate girder	5
2	Literature Review	6
	2.1 General	6
	2.2 Review of Literature	6
	2.3 Scope of Study	9
	2.4 Objective	9
	2.5 Methodology	10
3	Numerical Investigation & Experimental Investigation	11
	3.1 General	11

	3.2 Finite Element Model Development	11
	3.3 Brief Description of Ansys And Its Features	13
	3.4 Graphical User Interface Window	16
	3.5 Description of Element Selected for The Present Work	17
	3.6 Material Property	22
	3.7 Modelling and Meshing	23
	3.8 Static Analysis	23
	3.9 Preparation and Testing of Specimen	24
	3.10 Test Set-Up	25
4	Results and Discussions	30
	4.1 General	30
	4.2 Numerical Results	30
	4.3 Experimental Results	36
	4.4 Comparison of experimental results with analytical results	38
5	Conclusions	41
	References	42

LIST OF TABLES

TABLE NO	TITLE	PAGE NO
3.1	Common Elements used in normal static analysis	16
3.2	Details of the specimens tested	25
4.1	Tabulation for load-deflection values for Ansys results	34
4.2	Tabulation for load-deflection values for experimental results	37

LIST OF FIGURES

FIGURE NO	TITLE	PAGE NO
1.1	Elements of Plate Girder	3
1.2	Common types of web openings	4
3.1	SHELL181 Geometry	18
3.2	Layered properties of shell 181	19
3.3	Material Property window for Linear Isotropic	22
3.4	Modelling and the meshing of the plate girder with opening	23
3.5	Plate Girder Loading	24
3.6	Details of the specimen's configuration	26
3.7	Fabricated specimen of PG1 and PG2 for single hole with geo-location tagged	27
3.8	Testing specimen of PG1 and PG2 for single hole with geo-location tagged	29
4.1	Specimen PG1 – max stress (single hole, double hole, triple hole)	31
4.2	Specimen PG1 – max deflection (single hole, double hole, triple hole)	32
4.3	Specimen PG2 – max stress (single hole, double hole, triple hole)	33
4.4	Specimen PG2 – max deflection (single hole, double hole, triple hole)	34
4.5	Specimen PG1 - Stress vs Strain plot	35
4.6	Specimen PG1 - Load vs Deflection plot	35

4.7	Specimen PG2 - Stress vs Strain plot	36
4.8	Specimen PG2 - Load vs Deflection plot	36
4.9	Graph of PG1 & PG2 - Load vs Deflection plot using experimental results	38
4.10	Comparison of the load–deflection curves obtained using ANSYS and experimental results	40

CHAPTER 1

INTRODUCTION

1.1 General

A plate circumference is a deep-structured beam made up of plates (or plates and angles) that can support heavy loads over wide gaps. Even with cover plates, heavily loaded ropes of single I-sections with wide gaps do not provide the necessary moment of resistance and shear resistance. The rolled steel beam could be used with gaps of up to 15 meters. 15 m to 20 m I-section with cover plates can be feasible. For wide spaces, such arrangements cannot be adequate, and the most feasible alternative is a plate diameter.

Plate girders are currently made by welding three plates together: two flange plates and an I-section to a web fit. (However, since plate edges were previously formed by riveting or bolting, angles must be used to connect the web to the joints.) Blade cutters are a cost-effective solution for broad span beams (over 20 m). The flexural strength of a plate is highest along its circumference. The resistance to bending and cutting can be increased by increasing the distance between the edges. Thin web plate is provided at the support and load points to protect the hook and thin web from buckling stiffeners. Welded plate girders, the most common form of plate girders, are structural steel members that consist of flange plates welded to a web plate with fillet welds.

They are used to support loads at long intervals and to support structural loads that are too large to be supported by rolled steel forms. They are used as transfer ropes in building structures to support columns above areas where there is no large column. Plate ropes can be used to rehabilitate existing building structures that require non-columns and existing columns must be cut below a certain ground level or removed. Plate girders are used as crane support girders in heavy industrial structures with long gaps. The web plates or edge plates are connected as an assembly by appropriate angles. Also, since the plate circumference is expected to carry large loads, horizontal and vertical stiffeners are used in places where concentrated loads are operated to avoid excessive buckling. To increase the moment of resistance, a flange plates can be inserted at the flange angles. The

bending moment to be resisted is reduced and the plates can be lowered where the plates are not needed.

1.2 Elements of Plate Girder

- The vertical plate of the G plate circumference is called the web plate.
- The angles at which the top and bottom of the plate are attached are called the flange angle.
- The horizontal plates attached to the ang flange angles are called flange plates or cover plates.

1.3 Depth of Plate Girder

- The depth between the outer surfaces of the edges is called the total depth (d_o) or depth of the plate circumference.
- General In general, the depth of the plate circumference is kept at intervals of $1/10$ to $1/12$. The distance between the C.G or C.G of the compression edge. The tension is called the effective depth of the flange plate circumference (d_e).
- And the distance between the vertical legs of the up and down flange angles is called the apparent depth (d) of the plate circumference.
- When the depth of the plate circumference is less than 750 mm, such girders are called 'shallow plate girders'.
- When the depth of the plate plate circumference is more than 750 mm, such ropes are called 'deep plate curters'.

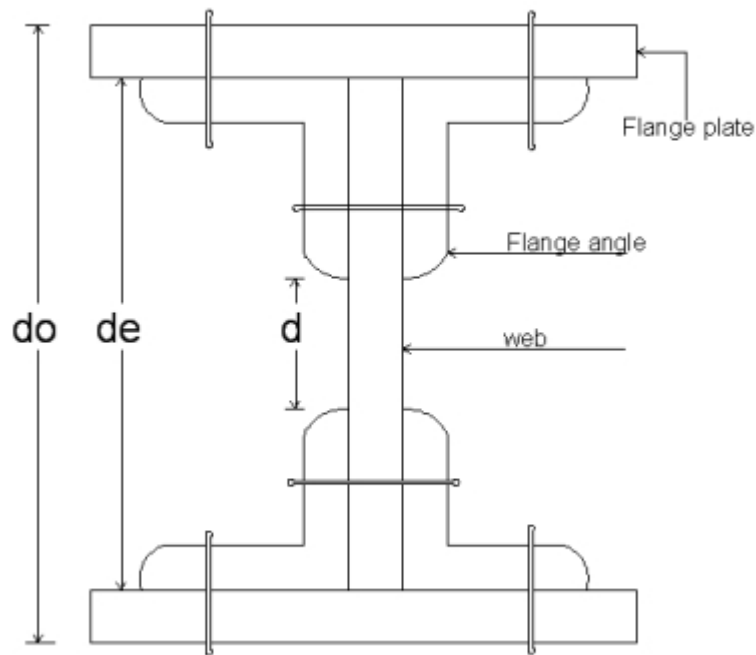


Fig 1.1 Elements of Plate Girder

1.4 Providing Stability against Web Buckling

Since the rolled parts are so proportionate, there's typically no risk of web buckling under regular loads, so local stability might not be a good idea. But, the web is deep and thin around the perimeter of a plate and is responsible for the hook. Buckling is neglected by providing stiffeners.

1.5 Plate Girder with Web Openings

The responsibility of a structural engineer is not only to design the structure with safety and serviceability in mind, but also to consider the operational requirements based on the application intended for the structure. When designing a power station system or multi-storey building, the traditional structural steel structure consists of ropes with solid nets. These prevent the supply of ducts and air conditioning pipes required for the satisfactory operation of the structure. Often, the service engineer on display will have to adjust the air conditioning hoses in place after the configuration is complete. Redirecting utilities (or raising the floor height at the construction stage to accommodate them) incurs extra costs and is usually unacceptably inefficient. Providing web openings in ropes has become standard

engineering procedure, removing the risk of a service engineer making holes in the wrong places.

The shape of web openings is determined by the designer's preference as well as the intent of the opening. The shapes of openings are not dictated by any hard and quick rules. Openings in standard shapes (such as oval or rectangular) are normally chosen for the designer's ease. The stiffness of the beams is reduced as holes are introduced in the web, resulting in greater variations than with strong webs. Plastic deformations caused by both the moment and the cut in the openings should be used to control the power of beams with openings.

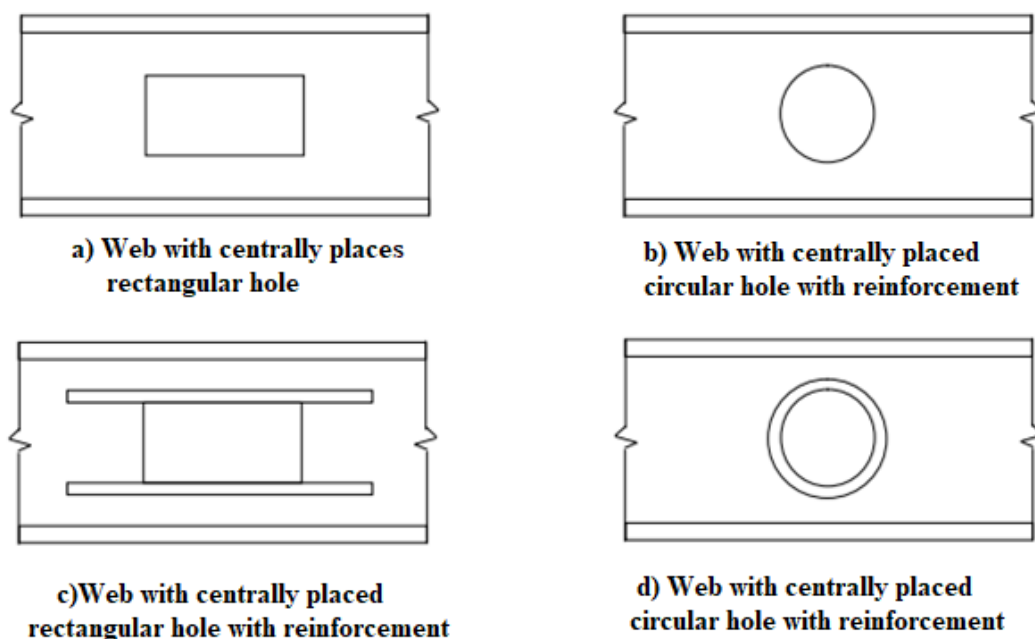


Fig 1.2 Common types of web openings

1.6 Failure modes plate girder with web opening

In general, a plate girder can fail in one of three ways:

- i. Yield of the tension flange
- ii. Abstract flange buckling, such as online buckling, territorial buckling, and dorsal buckling
- iii. Web Buckling

Excessive web cutting can result in extreme diagonal tensile and compressive stresses close to the supports. The thin web, which can survive diagonal tensile stresses but not diagonal compression stresses, is responsible for the hook.

1.7 Advantages plate girder

- Structure is easy to have structural features
- The section of the G environment is specifically designed for the desired purpose.
- Plate The strength is very high for the weight ratio of a plate environment.
- An alternating interval, loading capacity.
- Stability and long fatigue life.

CHAPTER 2

LITERATURE REVIEW

2.1 General

Over the past decades, major researchers have been openly studying steel plate blocks, and the purpose of the literature study is to understand the basic concepts and behaviors of steel plate blocks. Experimental and analytical studies of a steel plate ropes with openings The literature related to strength is discussed in this chapter.

2.2 Review of Literature

Sruthi et.al, Conducted to examine the strength behavior of trapezoidal curved corrugated beams with different radii of curvature, web thickness and edge thickness. The purpose is to compare the buckling strength of plate ropes without corrugated and trapezoidally curved corrugations. Corrugated web beams advantage is to increase the buckling resistance without the need to weld stiffeners to the web. The result is reduced beam weight without compromising on strength, which can reduce costs by up to 30%. The use of corrugated nets in bridge construction will minimise the need for intermediate diaphragms for cross-load movement.

Sung et.al, the revised inverse analyzes are subjected to a clean cut of inversely hardened plate perimeter web panels (without elongated stiffness) on three-dimensionally defined element specimens, which includes the initial flattening effects. Currently. The American Association of State Highway and Transportation Officers (ASHTO) and the American Institute of Steel Construction (AISC) have developed standards that differentiate between resilient bucking strength and post-buckling strength and incorporate these resisting capacities. Using the aspect ratio of web panels as a guide. While the calculations in these requirements accurately predict overall shear strength, they often underestimate re-shear buckling strength, which in turn underestimates the stiffness at the flange-web junction and the post-buckling strength of certain web panels. As a function of the lack of bending stresses outside of the plane. Based on the effects of the parameter analysis of the figure. To assess the final cutting capabilities of web tables, new architecture equations have been

suggested. Examine the following calculations. The final shear strengths measured using the equations are compared to test data already available.

Jonathan et.al, In the final article, we looked at an overview method for determining buckling cut power. To begin, the dynamics of plate cutting buckling are studied using an experimentally validated finite element model. The solution is similar to axis compression plates, but the axis compression works on a diagonal in this case. To estimate the shear strength of a plate, a body model and structure based on the compressive strength of a plate are created. This abstract approach yields strengths that are similar to theoretical and finite element effects for the general design parameters of most bridge and building systems than the strongest and widely known formula based on stress field behaviour. Overall, the findings of this analysis suggest that an abbreviated approach to calculating plate post-buckling shear performance is well reflected by an honest representation of shear buckling dynamics and detailed test data, where advanced associations with stress field action formulas are found in many situations.

Donald et.al, In contrast, the 12 most promising versions for hardened steel I-curters are evaluated for precision and ease of use. Many well-established models of civil engineering practise, as well as many newly proposed models, are considered. Since the model built in Posler's seed study is currently the selective approach in US practise, this article focuses on the benefits and drawbacks of alternative models shared with Posler. Statistical analysis is conducted on 129 test cutting tests, including 30 hybrid and 11 horizontally curved i-curters, using new data from 129 test cutting tests. In comparison to the AASHTO LRFD bridge design requirements and the 2005 AISC standard for structural steel buildings, the Posler model, which was adopted in 2004, offers the best balance of precision and simplicity in measuring the shear resistance of hardened steel I-girders.

Kim et.al, AASHTO (2007) and AISC (2007) adapted various specifics of the research conducted to establish cross-sectional standards for steel I-girders (2010). The development of revised rigid parameters for perimeter cut buckling and postbuckling (tension-field action) strengths is detailed. As a result, the AASHTO (2007) and AISC (2010) equations are simple to use and offer the advanced essence of the demands on intermediate cross-sectional stiffeners. The correct application of

static standards is discussed. To demonstrate the implementation of cross stiffness criteria, a design illustration is given.

Shady Gomaa et.al, Three-blade girder web panel Analyzed non-finite components in three-dimensional models representing steel blade cutters with and without horizontal stiffeners to study the behavior of thin sections. To achieve this, a finite element model simulating the geometric and non-linear nature of the problem has been developed. Verification is provided with previously published test data. The effect of the presence of horizontal stiffener in the web group and the increase in its aspect ratio are presented in this study. The numerical results of the parameter analysis for thin beams with different dimensions show a significant effect on the failure load of the horizontal stiffeners.

Rolando Chacon et.al, In cross-section hardened steel ropes, the action of the interlocking cut and connection loading mechanisms is defined. The static intelligence depicted in this paper depicts the web thickness and flange yield strength and pressure distribution, as well as the important loads and the influence of cross-sectional steps in the equilibrium direction of its specific loading. The previously validated number model is used as a modelling technique in a standardised manner. To predict and idealise the most useful features of the depicted machine model, the stress, strain, strength, and displacement fields are used.

Shahabian et.al, Based on current hypotheses and formulas, this paper presents theoretical predictions for the ultimate resistance of thin plate ropes for the shear load used, which demonstrate a strong connection with the test results. When a plate diameter is exposed to a bending moment in addition to the cut, determining the final resistance becomes more difficult. A touch formula for the ultimate resistance of thin plate ropes for combined bending and shear loading is presented here, which shows a satisfactory connection with existing concepts and is suitable for practical applications. Internet panel aspect ratios, $b_w = d_w$, 1 to 2, and thin ratios, $d_w = t_w$, 150 to 300, are used in the proposed contact equation.

Roberts et.al, Thin plate rope web panels are often exposed to a combination of bending, cutting, and localised compression edge or patch loading. While contact formulas for the final resistance to the combination of the two in the three types of

loading have been suggested and tested by test results, it does not appear that any contact formula has been proposed for the three types of loading relation. Test results for integrated curvature, cut, and connection loading are used to validate an integrated formula for the final resistance of thin plate perimeter web panels.

2.3 Scope of Study

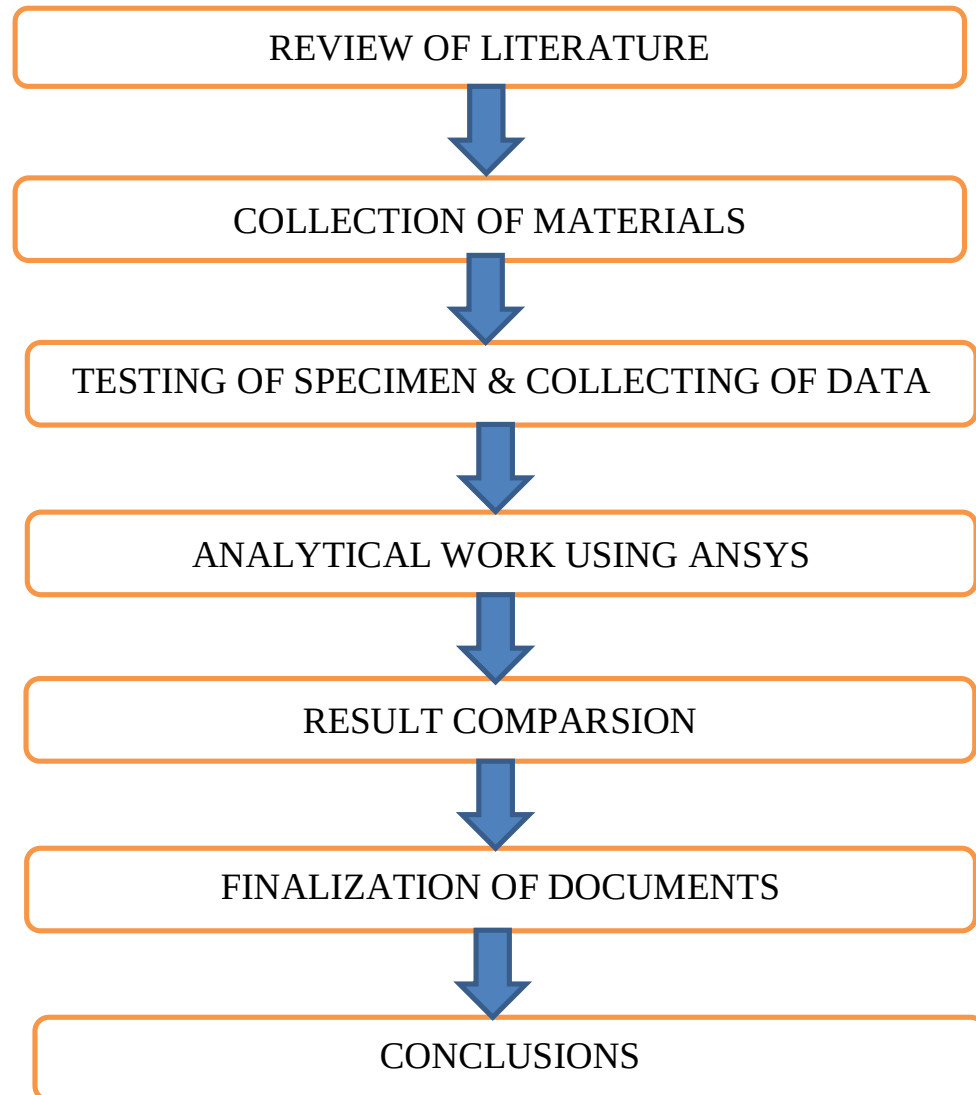
In this study, the constant behavior of steel plate girders is focused on the opening, the opening being made to detect its structural performance of the beam which is simply supported with curvature and shear. The failure mode, load-deflection curves, and strength potential of fixed performance steel plate curvatures with openings are addressed in detail based on the test results. Finally, using the ANSYS Workbench data, numerical analysis was done, and the test results were compared to the specified element analysis results.

2.4 Objective

The main objectives of this project are,

- To study the static behaviour of Steel Plate Girders with opening.
- To determine how well the hole in the web does.
- Analyzing and comparing the test behavior results of steel steel plate girders with the opening
- Compare mode failure mode, load-deflection curves and strength capability

2.5 Methodology



CHAPTER 3

NUMERICAL INVESTIGATION & EXPERIMENTAL INVESTIGATION

3.1 General

Number analysis is performed using ANSYS, a finite element software. The beam element is modeled considering the material properties of the steel for analysis. Numerical analysis is carried out for the model element taking into account the various parameters affecting the strength, and this chapter examines the strength properties of the blade curlers with the opening details of the test trials carried out on the test specimens.

3.2 Finite Element Model Development

Various limited element software packages include ATENA, ABAQUS, Hyper-Mesh, Nostron, ANSYS.

ANSYS (Analysis System) is one of the most efficient finite component packages available for nonlinear analysis. ANSYS uses computers to do simulated testing of systems, which is a common practise in the field of research and development. The mathematical representation of a physical structure that contains an assembly (sample), material properties, and relevant boundary conditions is referred to as a combined pre-processing, the solution of that mathematical representation, and the interpretation of the effects of that solution, according to ANSYS Tutorial Limited Element Analysis. Simple patterns and problems can exist, and they are often solved by hand. ANSYS V17 was used to build a 3D non-linear finite element model in this analysis to simulate steel plate curbs with the same openings as the single-point load test. The effect of the different depths of the beam is taken to be 500 mm × 120 mm and 400 mm 120 mm, as well as different initial circular hole sizes of 150 mm and 75 mm per hole, two holes and three holes are studied and compared. Two of the best models from these six models at ANSYS are being tested for work. The best model is selected based on the load capacity and deviation value from the ANSYS software.

The steel plate girders are 1500 mm in thickness. To study the effect of different depths of steel plate girders with different sizes of web opening of the web. Further information on ANSYS related theory and finite components currently implemented in ANSYS is discussed. The graphical user interface in ANSYS offers a quick and easy way to solve a variety of linear and nonlinear analysis problems. Shell components may reflect steel plate material.

All of the steps necessary to build these models in ANSYS are detailed. The analysis discusses the failure load and the process used to generate the associated failure stresses. The finite element model's predictions were compared to the test results.

3.2.1 Pre-Processing

FEA software usually uses the CAD representation of the physical model for pre-processing and splits it down into smaller bits known as finite "components." The term for this procedure is "machining." The statistical representation of the physical model is easier when the mesh (collection of elements) is of higher quality. The primary aim of an element is to link nodes using predictable mathematical equations based on node stiffness; the form of element used is also determined by the problem to be solved. Each element's action is immediately well known. Using the simple "closed form" formulas used in traditional engineering textbooks, one can forecast the actions of nonsensical types by comparing the behaviours of each variable simultaneously using equations.

3.2.2 Post-Processing (*Interpretation of Results*)

Post-processing of finite element data usually requires additional software to streamline the output, which means that it is easy to understand whether the construction is acceptable or not. The yield of an integrated thinker is usually quite substantial if the source data. This amount of source data is difficult and difficult to explain without the graphical representation commonly referred to as data sorting and post-processing.

It's used to make graphical displays that demonstrate how stresses, strains, deformations, temperatures, and other facets of the model are distributed. The approach to finding useful knowledge regarding possible issue, material waste, or

other sample output characteristics is to interpret these processed post-results (thermal and sample).

The most important thinking occurs during the FEA process, as the consumer examines the results and compares them to the predicted outcomes. Engineering should explain results using "common sense," but it cannot claim that the consumer must decide if the results are relevant. If the findings vary from what was predicted, one can look for an interpretation before believing the results fully.

Tools for viewing sample descriptions or vector layers in samples in order to provide results summaries (such as min/max values and locations). Cutting techniques that are both powerful and intuitive help the user to get more accurate results than given areas of geometry.

3.3 Brief Description of Ansys And Its Features

ANSIS is FEA software used in the engineering field to analyze a newly developed product or product. It is used as a tool and solution to detect the behavior of materials for extreme conditions such as high loads, temperature variation, and fatigue.

Here are some analyzes that can be done using ANSYS.

1. Standard analysis
2. Sample analysis
3. Harmonic analysis
4. Unstable analysis
5. Spectrum analysis
6. Buckling analysis
7. Ancillary structural analysis

3.3.1 Static Analysis

A static analysis involves time-varying loads that can be calculated as constant depression loads (like gravity and rotational speed) and constant equal loads (such as fixed equivalent wind and seismic loads, commonly defined in many building codes). Static analysis is used to calculate displacements, pressures, strains, and forces in systems or materials induced by loads that do not cause major recession or density consequences. The load and response of the structure are assumed to change progressively over time under fixed loading and response conditions. The following are the different methods of loading that can be used in normal analysis:

- Make use of external forces and stresses
- Powers of fixed-phase recession (such as gravity or rotational speed)
- Displacements that are not zero
- Fluids and temperature (for heat strain) (for nuclear inflammation)

3.3.2 Modal Analysis

When designing a mechanical component or determining the vibration characteristics of a structure (natural frequencies and mode patterns), sample analysis is used. It may also be used to kick off a more in-depth dynamic analysis, such as a transient dynamic analysis, harmonic analysis, or spectrum analysis.

3.3.3 Harmonic Analysis

When designing the vibration characteristics (natural frequencies and mode patterns) of a structure or a mechanical device, sample analysis is useful. It may also be used to kick off a more in-depth dynamic analysis, such as a transient dynamic analysis, harmonic analysis, or spectrum analysis.

3.3.4 Transient Analysis

Intermediate dynamic analysis (also known as time-history analysis) is a technique for determining a structure's dynamic response to some common time-dependent load. This method of analysis can be used to calculate time-varying

displacements, strains, stresses, and pressures in a system that is subjected to a variety of rigid, unstable, and flexible loads. For recession or density impact, loading time is critical.

The equation of motion solved using transient dynamic analysis is (M)
 $\ddot{U} + (C) \dot{U} + (K) U = F. (D)$

When and where:

(M) is the mass matrix.

(C) denotes a mitigation team.

(Q) = Matrix of stiffness

($\ddot{U}$) = vector of nodal acceleration

($\dot{U}$) = vector of nodal velocity

(U) = vector of nodal displacement

F (t) denotes Load vector

3.3.5 Spectrum Analysis

For the known distribution of the results of a sample analysis, a spectrum analysis is used to measure the displacements and stresses in a sample. It is used to assess the response of structures to random factors such as earthquakes, wind loads, ocean wave loads, jet engine propulsion, and rocket motor movements in areas where time-historical research is done.

3.3.6 Buckling Analysis

It's a method for calculating bucking loads (loads under which a structure becomes unstable) and hook mode patterns.

3.3.7 Sub structuring Analysis

It's a method of suppressing a set of finite elements by using a single element called a squad. A super element refers to a single-team element. The super element, like every other ANSYS element category, can be used in an analysis.

Element Type for Structural Analysis:

The elements are categorised into categories based on their similarities. The first section of the element name is made up of these group names (PLANE, SOLID, SHELL, etc.). The second part of the element name is the number. The interpolation of an element into the interior of the nodal effects is referred to as element series. It defines how outcomes differ over an aspect and is particularly important if the gradient is steep.

Table 3.1 Common Elements used in normal static analysis

Element Order	2D Solid	3D Solid	3D Shell	Line Elements
Linear	 PLANE42 PLANE182	 SOLID45 SOLID185	 SHELL63 SHELL181	 BEAM3/44  BEAM188
Quadratic	 PLANE82/183  PLANE2	 SOLID95/186  SOLID92/187	 SHELL93	 BEAM189

3.4 Graphical User Interface Window

The GUI allows multi-level customization, from a simple change in the sizes of the GUI and its parts to a more complex change in the menu sequence and dialog boxes design. GUI properties that you can change include these items:

The size of the U GUI and the areas within it

Colors and fonts

Menus displayed at U GUI startup

Focus mouse and keyboard

Menu Menu Sequence and dialog boxes.

By default, key areas of the GUI (Application menu, Main menu, Fixed toolbar, ANSYS Toolbar, Input window, Graphics window, and Output window) are displayed when you enable the GUI. Ansys has a graphical user interface that is simple to use and reduces the amount of time spent modelling and analysing a product or products.

3.5 Description of Element Selected for the Present Work

Choosing the right factor for the right analysis, properties that are consistent with the substance or product model, is crucial in FEM analysis.

3.5.1 Shell Element Description

Shell 181 is a steel plate material that can be used to make steel items like steel sections. It's an eight-tip structural shell element for studying thin and relatively thick shell structures. There are eight nodes in the element, each with six degrees of freedom: transformations on the x, y, and z axes, as well as rotations on the x, y, and z-axes. (Only the sum of translation per element is free when using the membrane option.)

It's better for applications that need linear motion, large rotation, and/or large non-strain. In nonlinear analyses, the change in shell thickness is measured. Full and minimised integration schemes are provided in the element region. The factor measures the distributed stress follower (load stiffness) results.

It may be used in plastic shell or sandwich structure simulation applications. The first-order shear-deformation theorem governs accuracy in modelling composite tiles (commonly referred to as the Mindlin-Risner shell theory). Logarithmic tension and real pressure action are used to shape the elements. Restricted membrane strains are possible due to elemental dynamics (stretching). Curve variations over time, on the other hand, are considered minor.

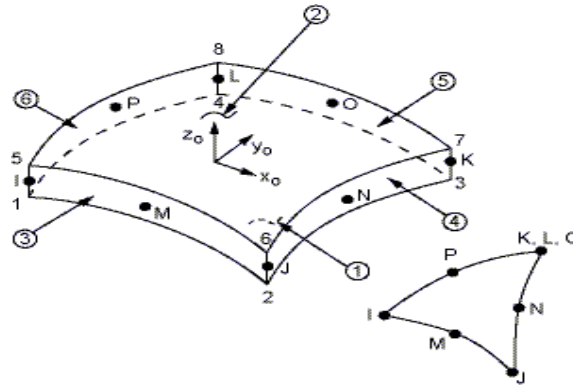


Fig 3.1 SHELL181 Geometry

The idea of a single layer shell segment gives you a lot of choices. You may, for example, determine the number of coordination points used and the direction of the piece. Layer shell specification is possible with multi-layer shell segment commands. The thickness of the layers may be used to determine the thickness, material, direction, and number of coordination points.

The results of cross-sectional decay are covered in SHELL181. The element's cross section is a 2x2 matrix, as seen below:

$$E = \begin{bmatrix} E_{11} & E_{12} \\ \text{sym} & E_{22} \end{bmatrix}$$

The default cross-sectional stiffness for a single layer shell made of isotropic material is:

$$E = \begin{bmatrix} kGh & 0 \\ 0 & kGh \end{bmatrix}$$

In the above matrix, $k = 5/6$, G = the cutting modulus, and h = the shell thickness.

It employs a sophisticated shell algorithm to precisely combine the initial curve results. Both shell-membrane and thickness strains are included in the equation for successful shell curvature adjustment.

3.5.2 SHELL181 Input Summary

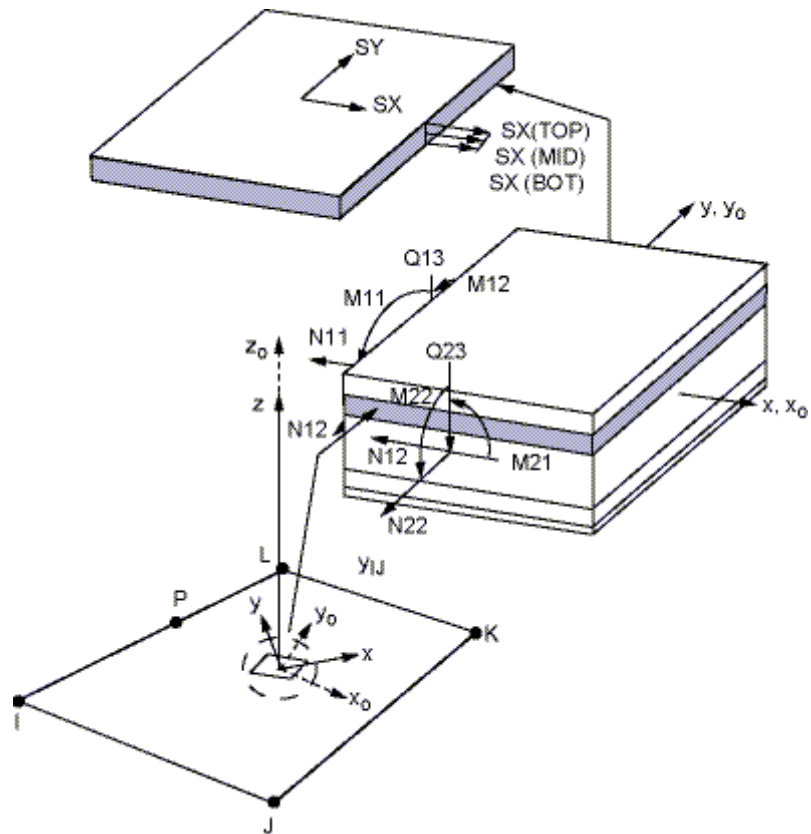


Fig 3.2 Layered properties of shell 181

Degrees of Independence for Nodes I, J, K, L, M, N, O, and B

If KEYOPT (1) = 0, then UX, UY, UZ, ROTX, ROTY, ROTZ are all valid options.

If KEYOPT (1) is 1, then UX, UY, and UZ are used.

Material characteristics

EX, EY, EZ (PRXY, PRYZ, PRXZ or NUXY, NUYZ, NUXZ), NUXY, NUYZ, NUXZ, NUXY, NUYZ, NUXZ, NUXY, NUYZ, NUXZ, NUXY

THSX, THSY, THSZ (CTEX, CTEY, CTEZ or THSX, THSY, THSZ), ALPX, ALPY,
ALPZ (CTEX, CTEY, CTEZ or THSX, THSY, THSZ)

ALPD, DENS, GXY, GYZ, GXZ

Just one BETD per variable can be defined, and it can be allocated on a layered basis.

Pressures on the surface of the loads - Face 1 (I-J-K-L) (bottom, + N direction),

2nd Face (I-J-K-L) (top, -N direction),

Faces 3 (J-I), 4 (K-J), 5 (L-K), and 6 (L-K) (I-L)

Physical exertion

Thermometer

When KEYOPT (1) = 0 (bending and membrane stiffness), the following results are obtained:

T1, T2, T3, T4 (below layer 1), T5, T6, T7, T8 (under layers 1-2); similarly, the layer ends with NL (4 * (NL + 1) max) temperature between the next layers. As a result, eight temperatures are used for one layer elements.

1 = T1, T2, T3, T4 for layer 1, T5, T6, T7, T8 for layer 2, equally for all layers (4 * NL max) for KEYOPT (1) (only membrane stiffness). As a result, four temperatures are used for one layer elements.

Specifications

Shell 281 offers element functionality and features.

Plasticity is the ability to change (PLASTIC, BISO, NLISO, BKIN, MISO, MKIN, KINH, HILL CHABOCHE).

Hyper-Flexibility is a term used to describe a person's ability (AHYPER, HYPER, BB, CDM)

Viscosity is a measure of how thick something is (PRONY, SHIFT)

Creep / viscoplasticity (creep, rate)

Adaptability (ANEL, ELASTIC)

Additional information (USER, SDAMP)

It is more difficult when you are stressed.

Excessive variation

a lot of pressure

the first point

Stabilization that isn't linear

Technology for element collection is chosen automatically.

Linear chaos (section description for shell shells and pre-integrated shell sections for homogeneous section stiffness input) and birth and death (section definition for shell shells and pre-integrated shell sections for homogeneous section stiffness input).

3.6 Material Property

The experimental analysis provided the criteria needed to characterise the material samples. Steel material is designed in layers; Each layer is assigned with different material blocks that define the material properties. The plate element represents shell 181. The shell 181 element requires linear isotropic and many linear orthotropic material properties. VanMyses uses failure criteria to define multiple linear orthotropic material plate failure.

EX - Flexibility Modulus or Young Modulus (c)

PRXY - Poison rate (ν)

Properties of Outline Row 3: Structural Steel			
	A	B	C
1	Property	Value	Unit
2	 Density	7850	kg m ⁻³
3	 Isotropic Secant Coefficient of Thermal Expansion		
6	 Isotropic Elasticity		
12	 Alternating Stress Mean Stress	 Tabular	
16	 Strain-Life Parameters		
24	 Tensile Yield Strength	2.5E+08	Pa
25	 Compressive Yield Strength	2.5E+08	Pa
26	 Tensile Ultimate Strength	4.6E+08	Pa
27	 Compressive Ultimate Strength	0	Pa
28	 Isotropic Thermal Conductivity	60.5	W m ⁻¹ C ⁻¹
29	 Specific Heat	434	J kg ⁻¹ C ⁻¹
30	 Isotropic Relative Permeability	10000	
31	 Isotropic Resistivity	1.7E-07	ohm m

Fig 3.3 Material Property window for Linear Isotropic

The given material properties, as shown in Figure 3.3, should be equal to the total number of layers that make up the material volume, and each material model represents the property for each layer of steel material.

3.7 Modelling and Meshing

Using shell lay-up option in ANSYS for the plate material could be modelled, which intends the number of layers and thickness of the layer. Refer fig 3.1 shows the meshing for one hole in the plate girder. Similarly, two holes and three holes are done respectively.

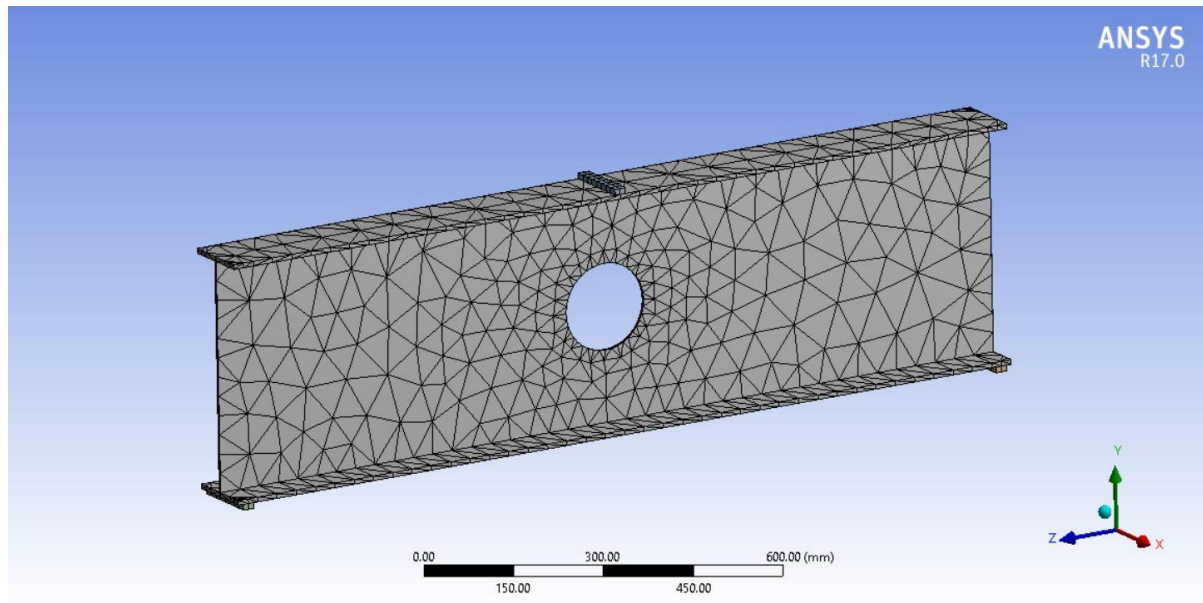


Fig 3.4 Modelling and the meshing of the plate girder with opening

3.8 Static Analysis

A configuration is considered to be in continuous equilibrium in linear static analysis. The structure is considered to revert to its initial, undeformed state until the used load is withdrawn. The arrangement, however, tends to deform under a specific set of loads without increasing the loading capacity. The configuration has been dysfunctional or warped in this situation. The material is believed to be non-linear in elastic or linear deformation analysis, and the orientation of the applied loads does not change. The effect of differences that exist, such as the practical high-order strain displacement relationship of geometry, element form, and application, is decomposed. Differential stiffness refers to the linear approximation of softening to a compressive axial load and hardening to a tensile axial load from a physical perspective. Partial equation problems are converted into a series of linear algebraic equations using the finite element scheme.

Where Q - Erection Matrix = $F [K] q = F$

q is the vector of nodal displacement.

Nodal vector force (F)

3.8.1 Loading and Boundary Conditions

Boundary conditions should be used where support and loading points are located to ensure that the model works similarly to the test model. In the middle of the beam the load is applied. As the whole sample was analyzed, the support conditions were similar to the test condition. All DOFs were controlled at the other end of the beam section similar to the test conducted. Analysis was performed using an indirect function, linear static analysis. Loads were applied to the beam section. The results of the FE model developed on the basis of normal stress across the beam are obtained. See Fig. 3.5 showing that the plate is mounted to a hole in the perimeter. Similarly, two holes and three holes are made, respectively.

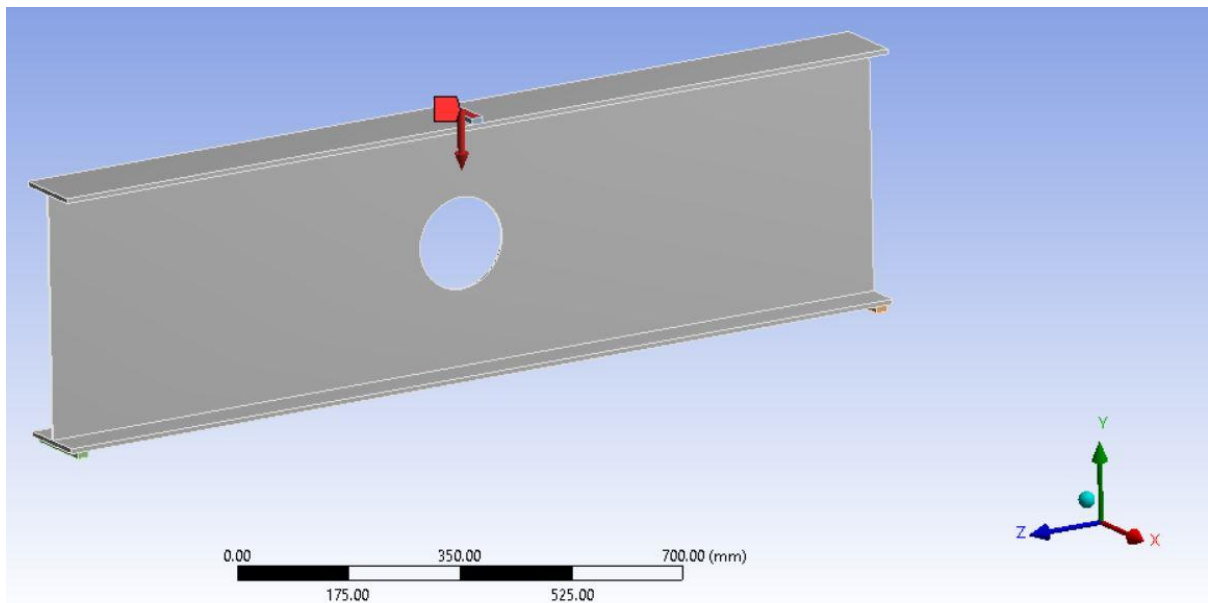


Fig 3.5 Plate Girder Loading

3.9 Preparation and Testing of Specimen

The test specimens consist of two plate ropes 1.5 m long. Although two blade cutters were tested in each series, the first girder section was 500 mm x 120 mm and the second girder section was 400 mm x 120 mm. 4 blade cutters were discovered and checked for inspection. The aperture of the first model is 150 mm

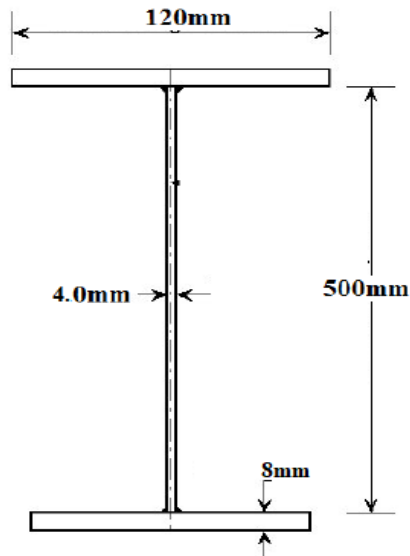
and the second model is 75 mm. The steel used has a yield strength of 250 N/mm². The web depth is the only distinction between the two girders. By welding the vertical mesh layer to the horizontal edge plates, each enclosure made of three plates functions as a unit (one upper and one lower edge).

Table 3.2 Details of the specimens tested

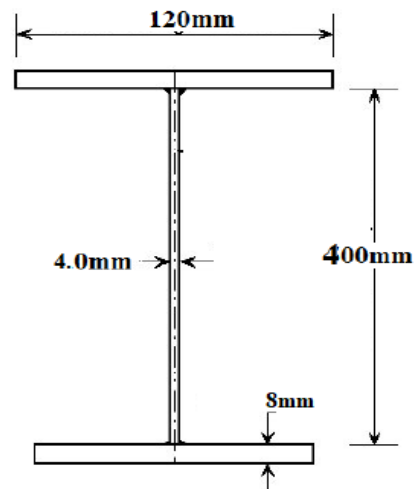
SPECIMEN NAME	WEB HEIGHT (mm)	WEB THICKNESS (mm)	FLANGE WIDTH (mm)	FLANGE THICKNESS (mm)
PG1	500	4	120	8
PG2	400	4	120	8

3.10 Test Set-Up

On a loading machine with a hydraulic jack with a maximum capacity of 1500 kN, a test was performed. Under single-point loading, both versions were tested for flexibility. I-section girders were simply assisted and added to the centre of the load plate girders before being checked under concentrated loads. The applied load is automatically reflected on the test machine's big dial on the indicator cabinet. The two girders' deviations were calculated using a dial of 0.01 mm measurements. The dial gauge is mounted under the flange below the girder's centre. At each 2 kN / min load rise, the deviation was measured. The test was called off as the ropes began to sag and a significant deformation occurred, and there was no way to raise the load on the web panel and edges. The load was withdrawn after the girders collapsed.



Girder(1)



Girder (2)

Fig. 3.6 Details of the specimen's configuration

The fabrication work and testing work in the loading frame apparatus of specimen PG1 and PG2 with geo-tagged location is shown in fig 3.7 & 3.8.





Fig. 3.7 Fabricated specimen of PG1 and PG2 for single hole with geo-location tagged







Fig. 3.8 testing specimen of PG1 and PG2 for single hole with geo-location tagged

CHAPTER 4

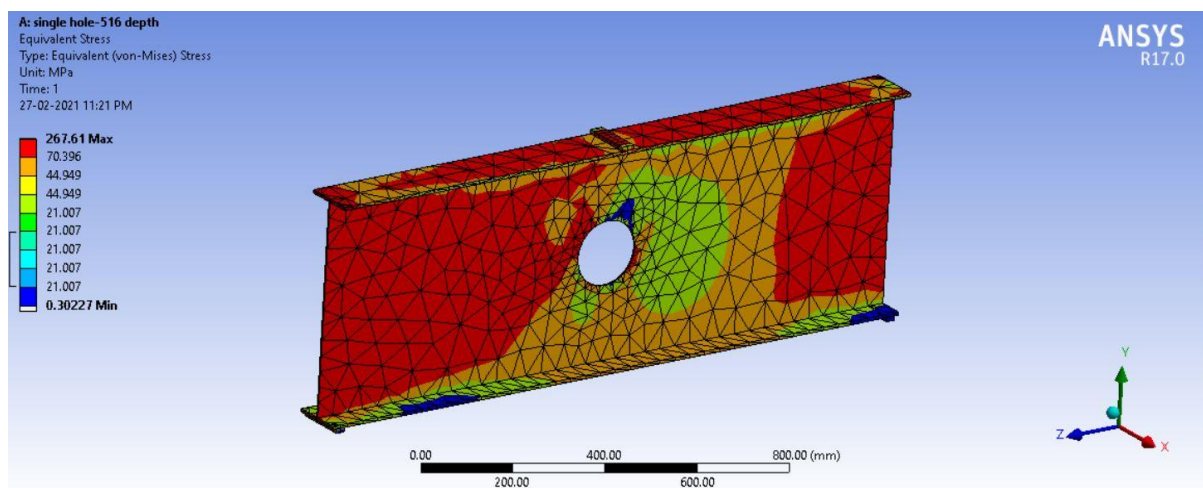
RESULTS AND DISCUSSIONS

4.1 General

Numerical simulation and experimental trial analysis are carried out considering various parameters in the plate blocks, with the opening open to detect its structural performance. The results of the parameter study are discussed in this chapter.

4.2 Numerical Results

The numerical part of the analysis is done with ANSYS. We looked at two steel plate girders that were bent and cut. The entire elastoplastic activity of the material was taken into account when modelling the girders. For example, the pressure definition of plate girders with openings is shown in Figs. 4.1 & 4.3 and the maximum deflection of the plate ropes with openings is shown in fig. 4.2 & 4.4. Beams Table 4.1 shows the load-deviation values obtained using ANSYS results for the PG1 and PG2 models.



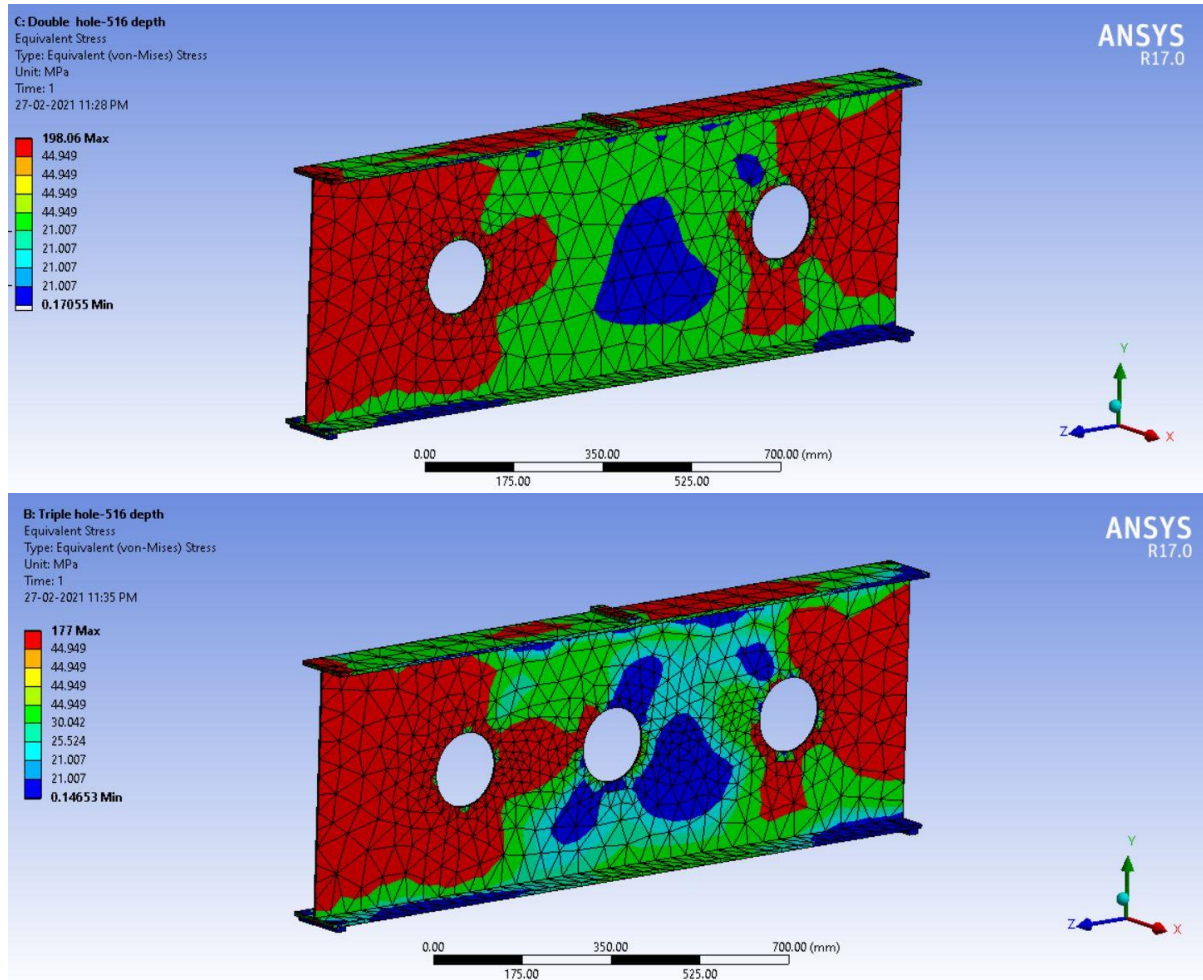
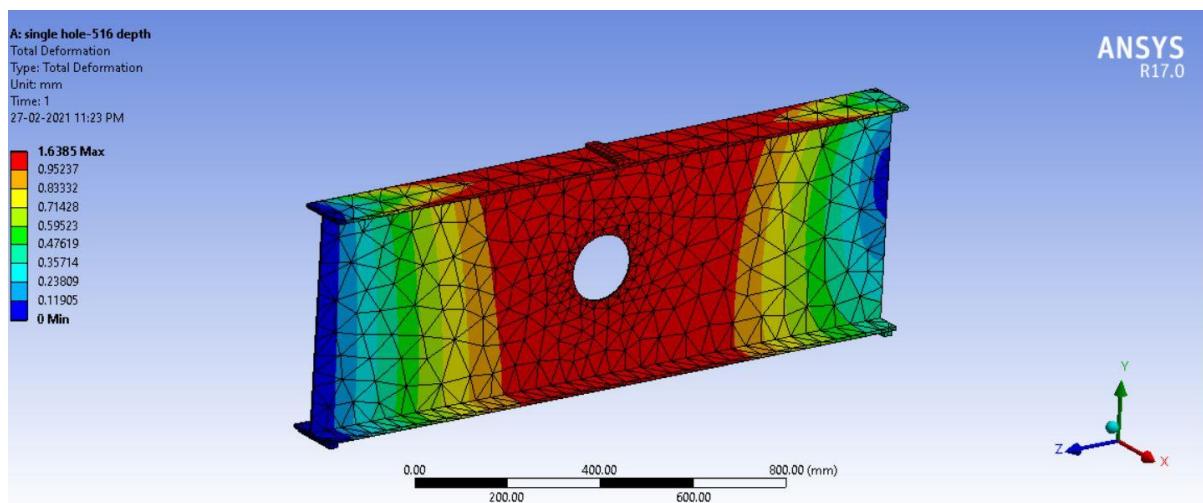


Fig.4.1 Specimen PG1 – max stress (single hole, double hole, triple hole)



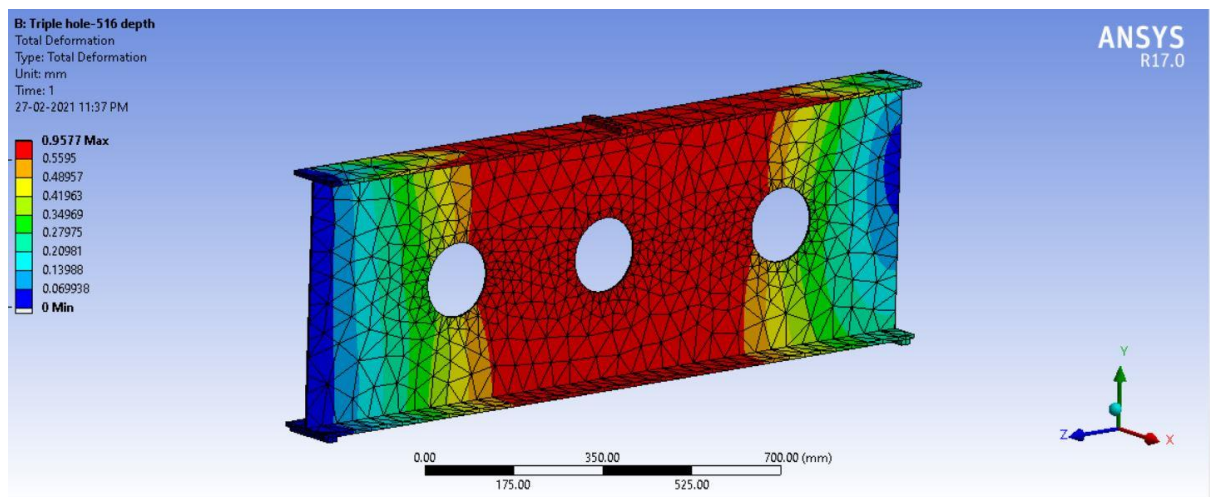
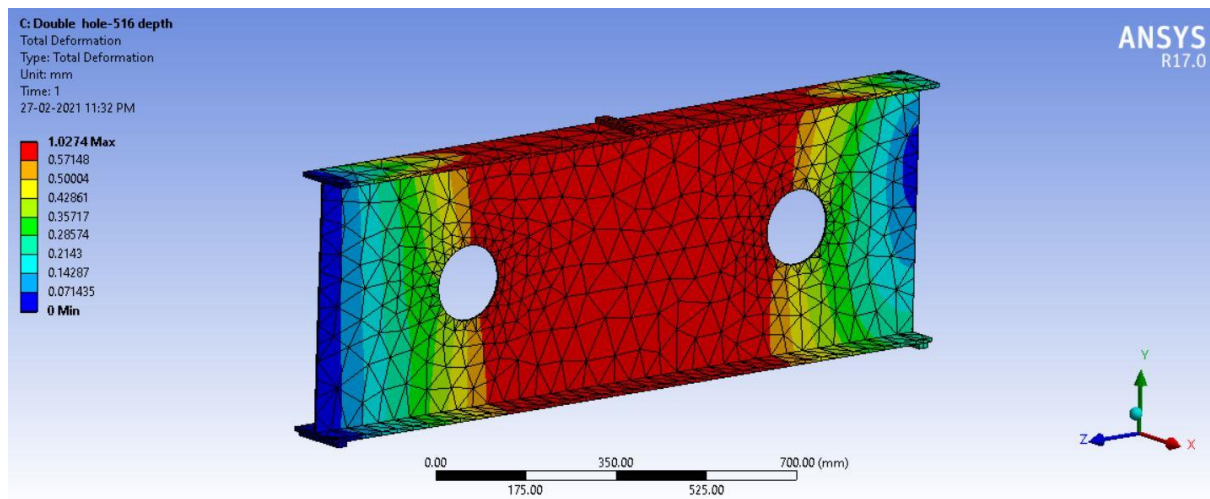
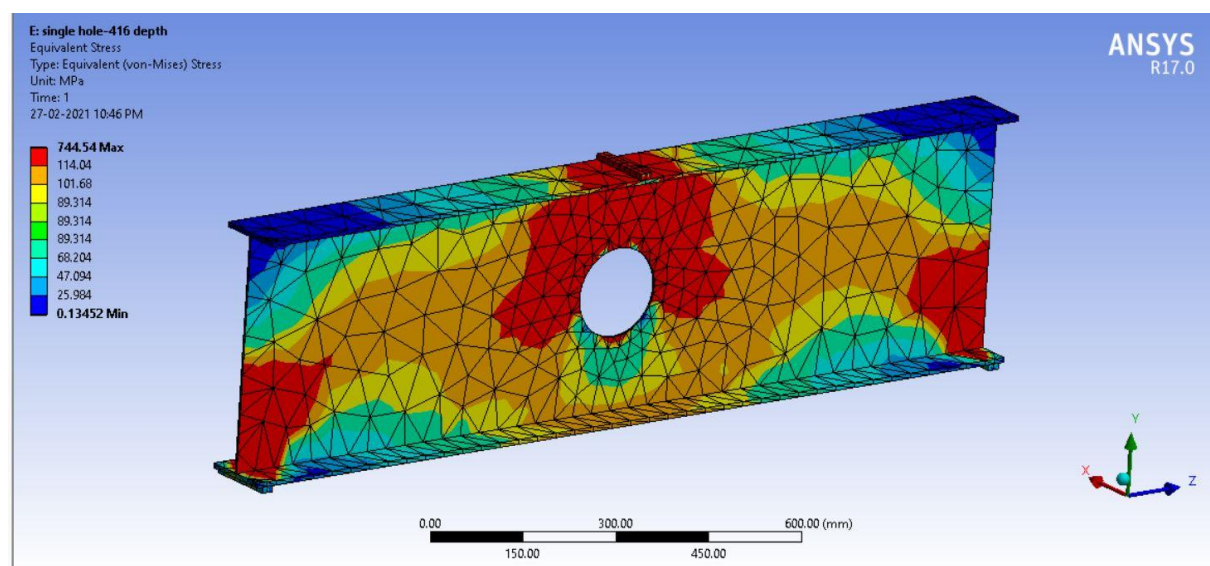


Fig.4.2 Specimen PG1 – max deflection (single hole, double hole, triple hole)



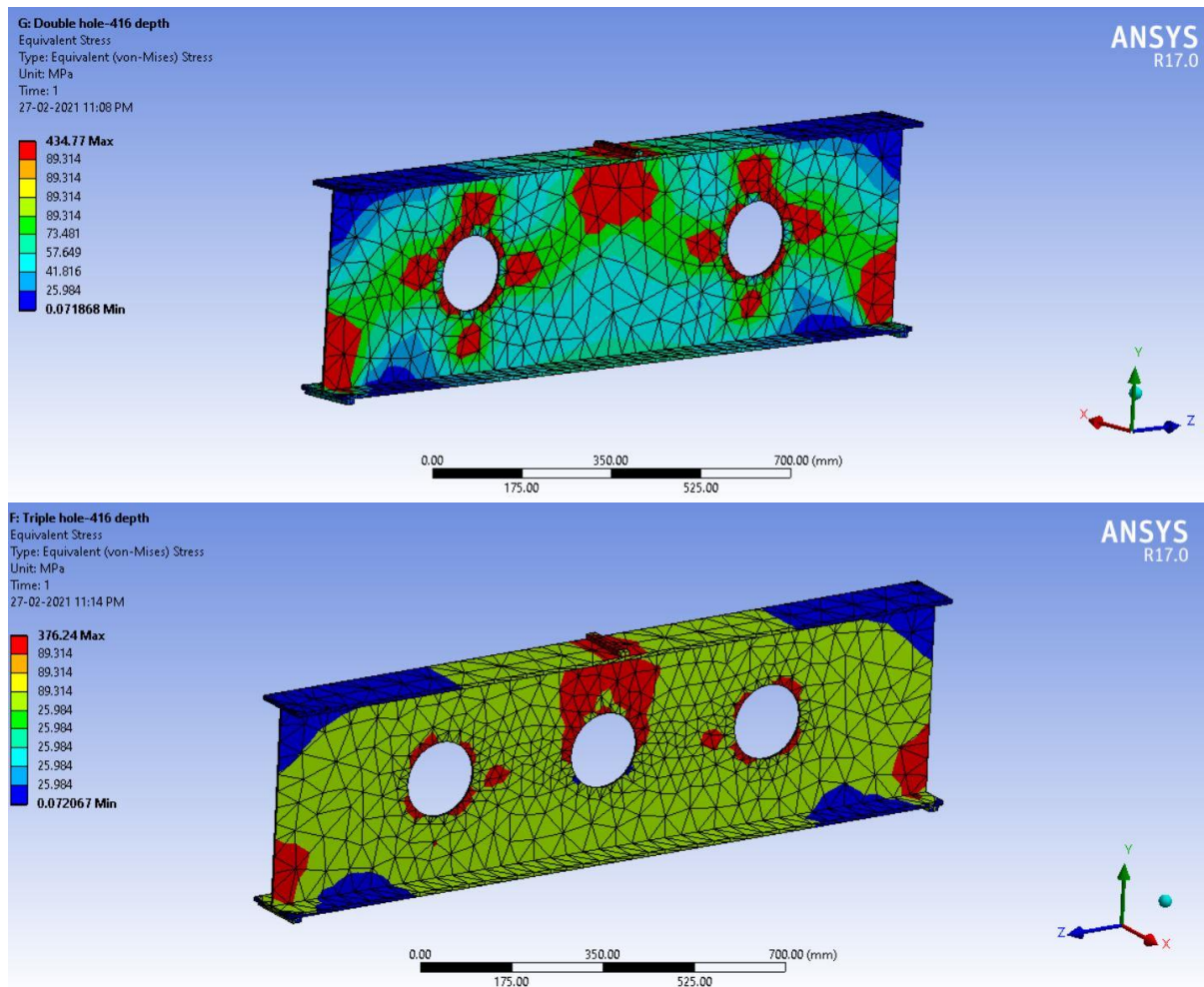
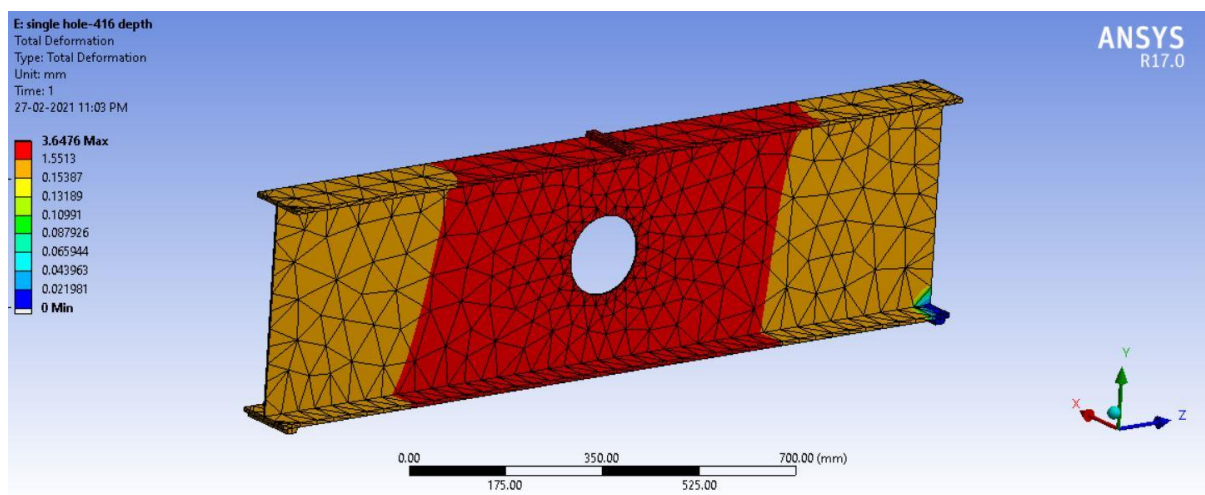


Fig.4.3 Specimen PG3 – max stress (single hole, double hole, triple hole)



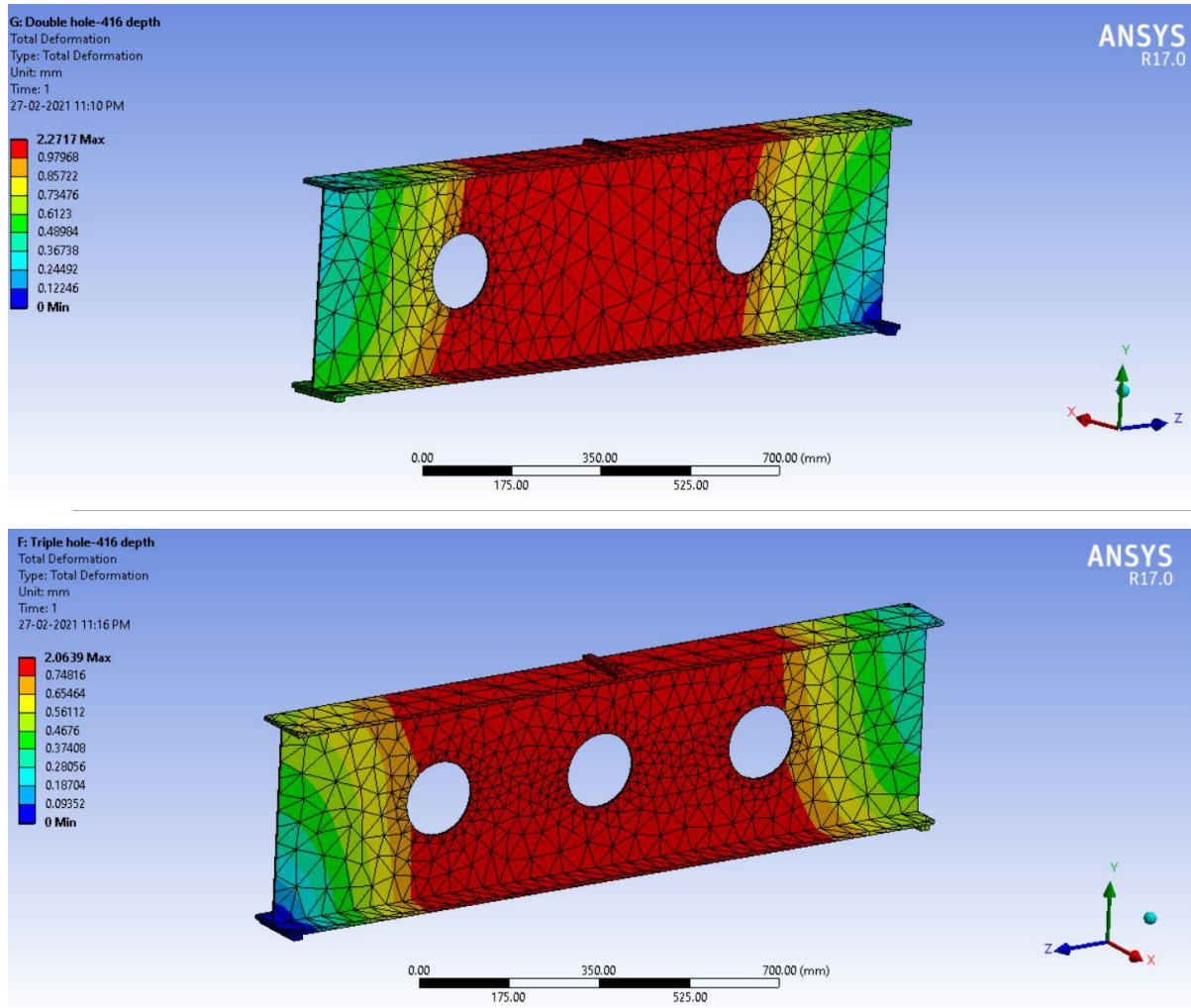


Fig.4.4 Specimen PG2 – max deflection (single hole, double hole, triple hole)

Table.4.1 Tabulation for load-deflection values for Ansys results

PG1	LOAD(KN)	DEFLECTION(mm)
SINGLE HOLE	379.45	1.6385
DOUBLE HOLE	217.8	1.0274
TRIPLE HOLE	195.69	0.9577

PG2	LOAD(KN)	DEFLECTION(mm)
SINGLE HOLE	170.94	3.6476
DOUBLE HOLE	101.55	2.2717
TRIPLE HOLE	88.029	2.0639

The Von-Mises yield criterion was used to calculate the object linearity. Results The final load value of the first girder (379.45KN, 217.8KN, 195.69KN for SH, DH & TH, respectively) and the second circumference (170.94KN, 101.55KN, 88.029KN for SH, DH & TH, respectively). The pressure vs strain plot and load vs deviation results of each plate environment are shown in Fig.4.5, 4.6, 4.7 & 4.8.

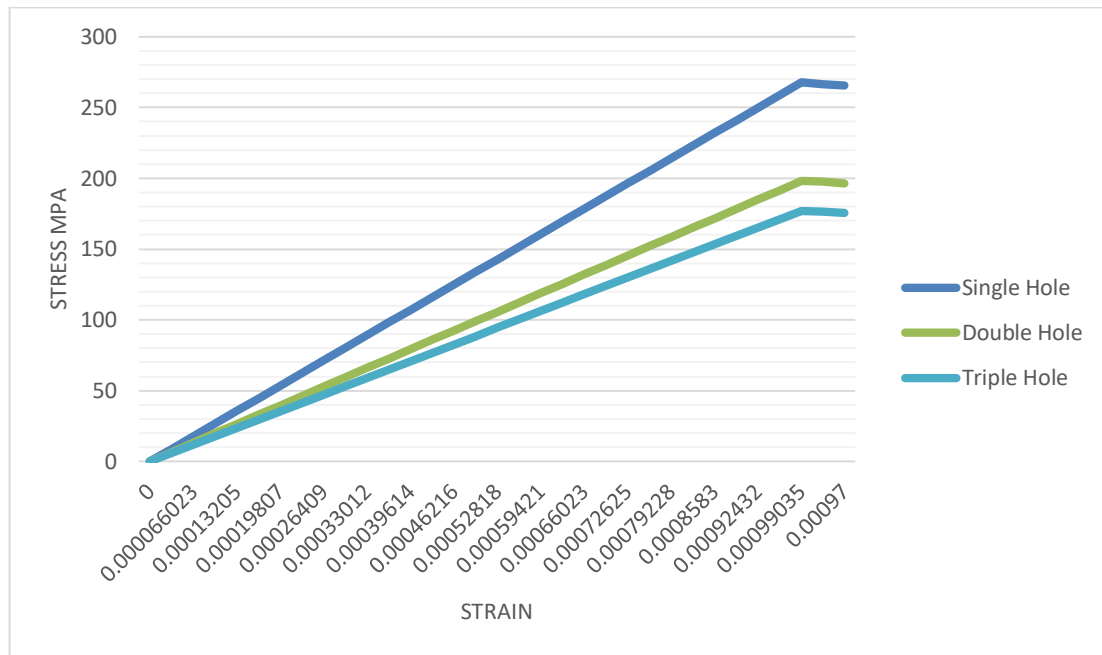


Fig.4.5 Specimen PG1 - Stress vs Strain plot

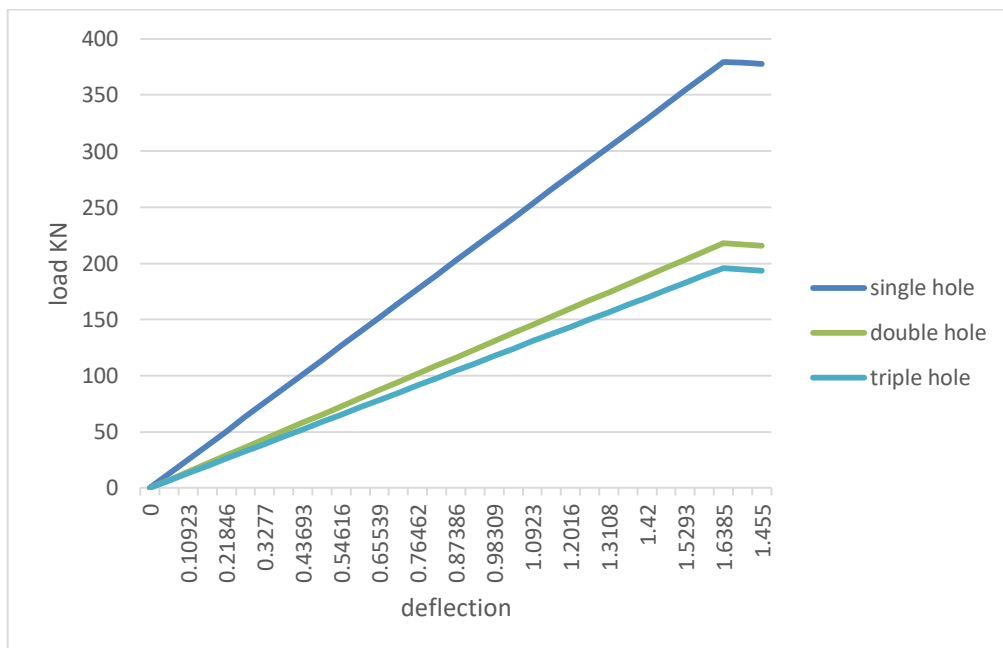


Fig.4.6 Specimen PG1 - Load vs Deflection plot

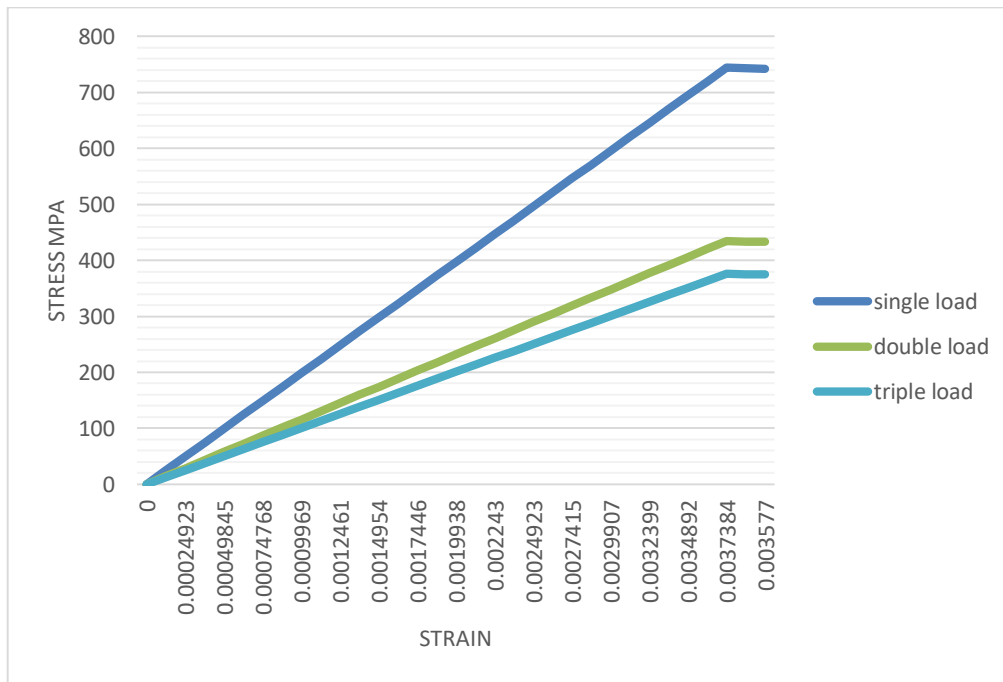


Fig.4.7 Specimen PG2 - Stress vs Strain plot

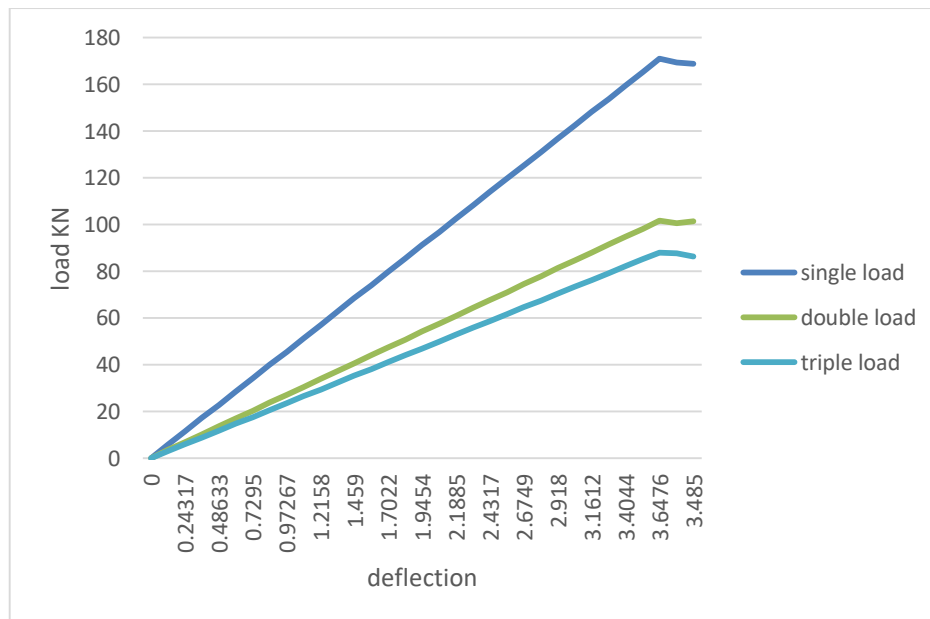


Fig.4.8 Specimen PG2 - Load vs Deflection plot

4.3 Experimental Results

Two plate ropes with a single hole were inspected using gradual loads until failure was achieved. The first circuit's final load value is (341.45 KN), and the second circuit's diameter is (158.7 KN). Figure 4.9 shows the load-deflection curves

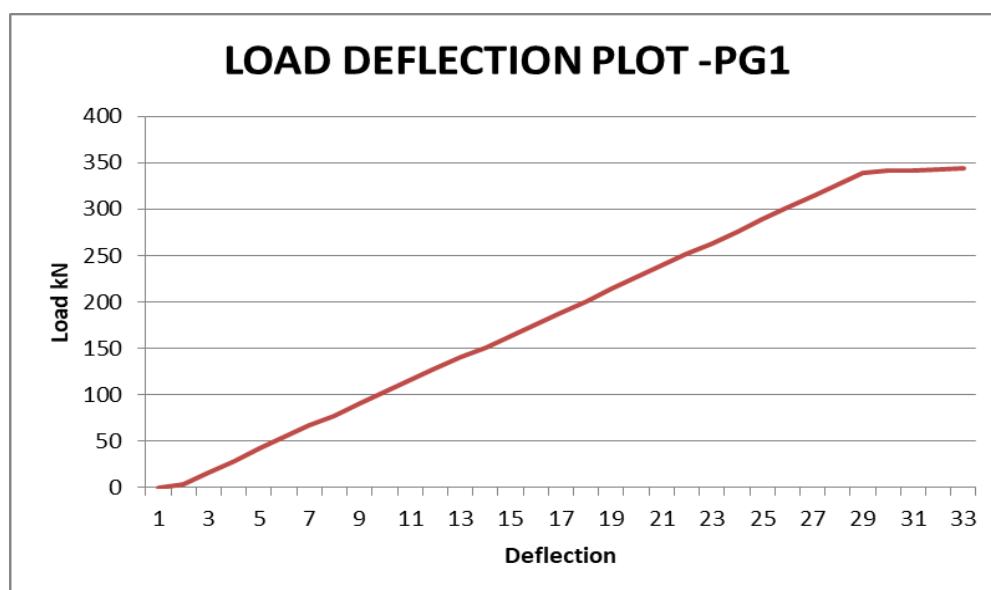
for the girders. The relationship was almost straight for the majority of the climb, but as the curve's web board deforms, the curve starts to bend slightly. All stages of deformation of steel ropes were specified, first, the web was bent (local buckling), and then the entire beam was hooked (global buckling).

Note: For test results, two of the six ANSYS results are sampled and followed for testing. (I.e.) Single hole with 500 mmx 120 mm and 400 mmx 120 mm is taken.

Table 4.2 shows the final load and overall deflection of the plate circuit for a single opening. The diagram of this table is shown in Fig. 4.9.

Table.4.2 Tabulation for load-deflection values for experimental results

SI.NO	SPECIMEN	LOAD(KN)	DEFLECTION(mm)
1	PG1	341.45	1.474
2	PG2	158.7	2.98



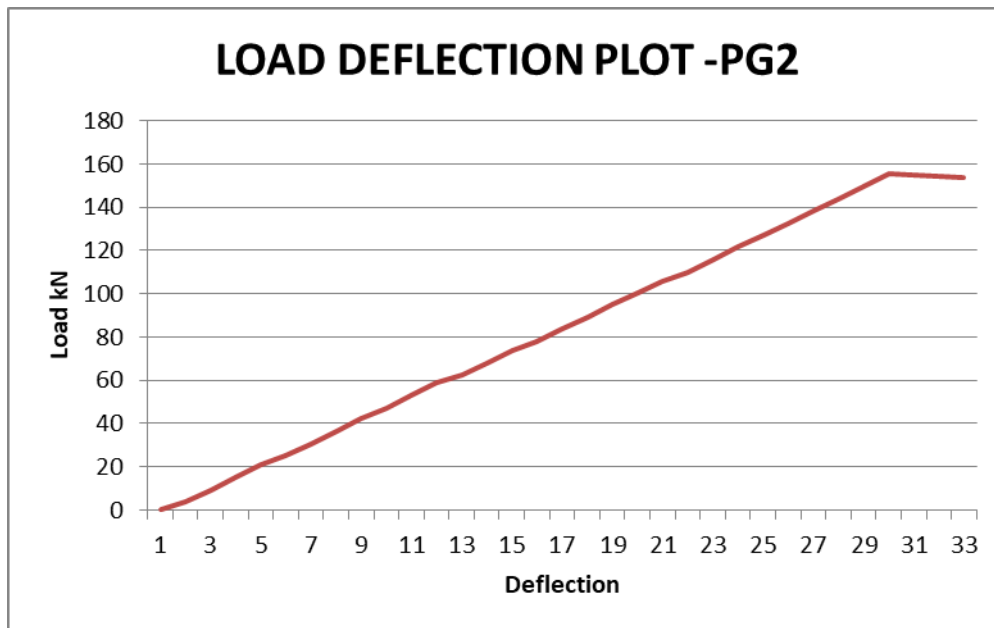
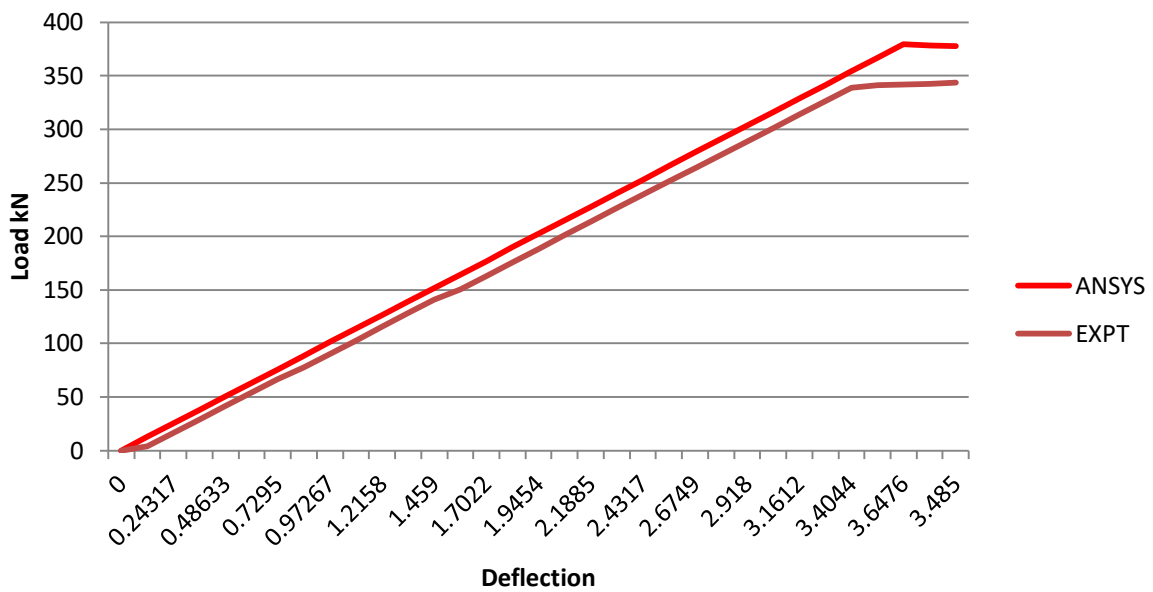


Fig.4.9 Graph of PG1 & PG2 - Load vs Deflection plot using experimental results (for single hole)

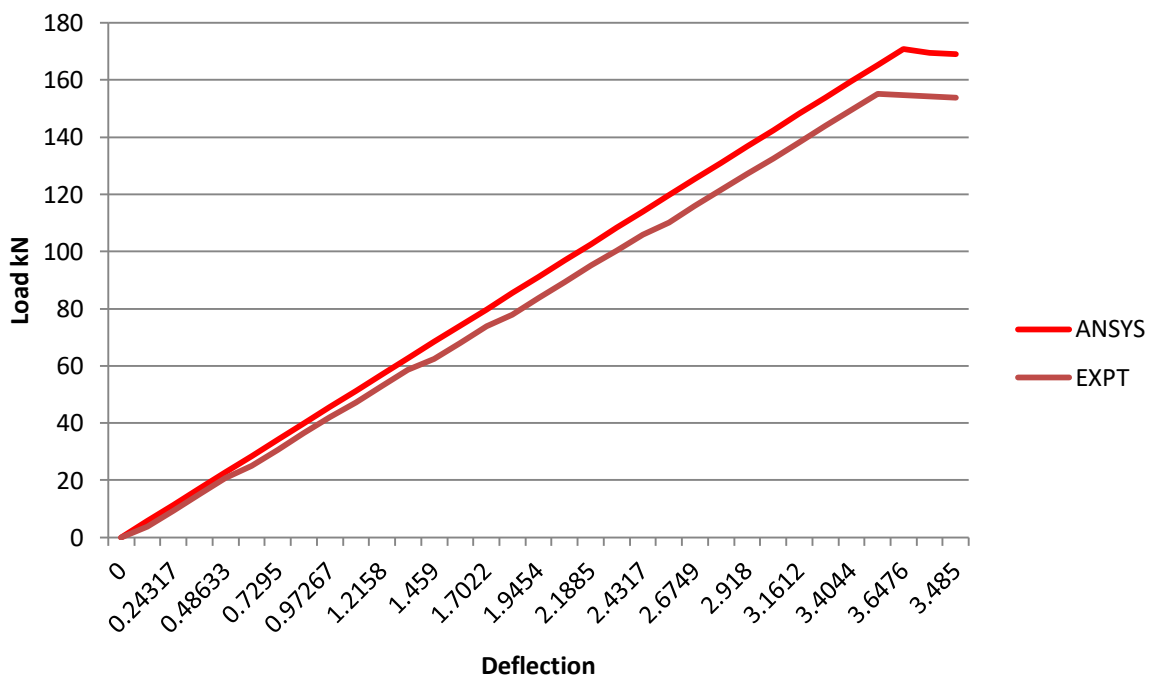
4.4 Comparison of experimental results with analytical results

Two separate distinctions were made, the first of which was between test outcomes and numerical analysis results. Tables 4.1 and 4.2 provide the data. For blade cutters with single hole holes, Figure 4.10 indicates a comparison of load-deflection curves obtained using ANSYS and experimentally. The Ansys finite element analysis results are almost identical to the test results, as can be shown. From this comparison (comparable in both cases) the ratio $\text{LoadEXP} / \text{LoadANS}$ shows a reasonable agreement (7 to 10%).

LOAD DEFLECTION PLOT -PG1 COMPARISON



LOAD DEFLECTION PLOT -PG2 COMPARISON



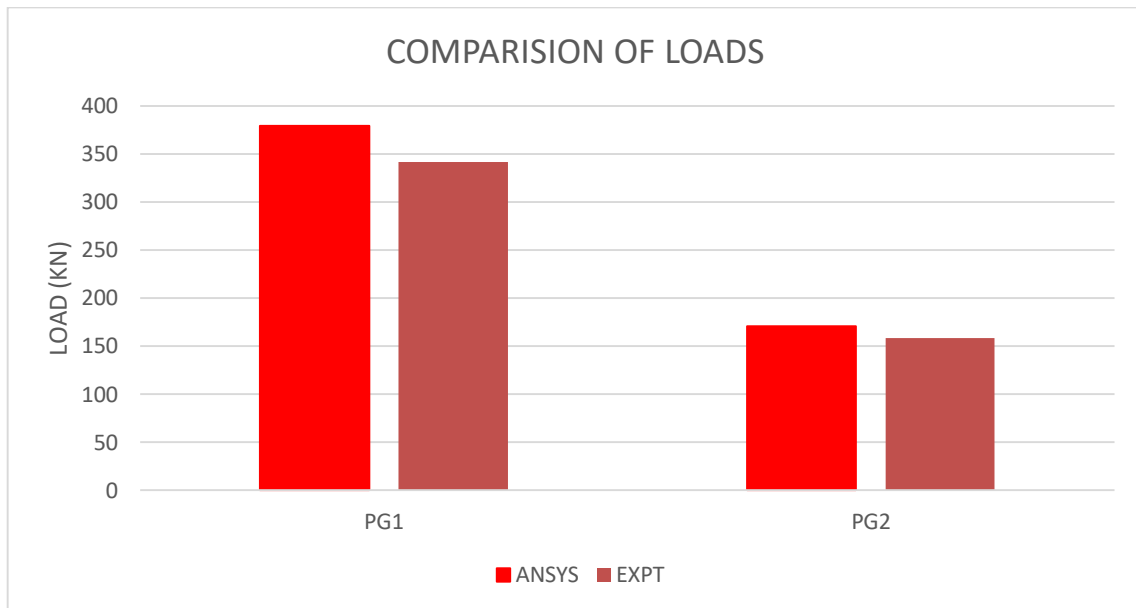


Fig.4.10 Comparison of the load–deflection curves obtained using ANSYS and experimental results (for single hole)

CHAPTER 5

CONCLUSIONS

From various theoretical activities and experimental studies on the structural action of plate ropes with openings, the following conclusions can be taken.

- To research the actions of shear and shear strength and buckling, two plate ropes were tested until they collapsed, taking into account the joint effect of stretching and bending. The findings were derived from test and numerical data.
- The findings revealed that the scale of the site and the distance / depth ratio had a substantial impact on the final cutting power of the panel web panel. The models were effective in simulating the web team's action and produced outstanding results.
- The results of the evaluation and the numerical interpretation are generally satisfactory and in fair agreement.
- The existence of web openings, as well as the size of these openings, results in a reduction in final load space. As the S web thickness and agility stiffness rates increase, the final load decreases.
- Plate girders with open opening failed by web cut bucking. In models with a depth of 500 mm, the initial failure was due to local flange bucking.
- The test results vary by 7 to 10% compared to the analysis using AN ANSYS. This disparity will cause the test beams to fail more slowly before completely collapsing. These algorithms do not exist in limited element models.

REFERENCES

1. Chai Yoo.H, Sung Lee.C (1998), "Strength of plate girder web panels under pure shear" journal of structural engineering, Vol. 124, pp.184–194.
2. Elgaaly.M, Hamilton.R.W, Seshadri.A (1996), Shear strength of beams with corrugated webs, J. Struct. Eng. ASCE Vol. 122 (4), pp. 390–398.
3. Jian Jiang, Qi Zhu, Guo-Qiang Li (2015), "Local buckling of compression flanges of H-beam with corrugated webs" journal of constructional Research Vol. 112, pp. 69-79.
4. Jonathan Glassman.D, Maria Moreyra Garlock.E (2016), "A compression model for ultimate post buckling shear strength" Thin -walled structures, Vol. 102, pp. 258 -272.
5. Kim.Y.D, Donald White.W (2014), "Transverse Stiffener Requirements to Develop Shear-Buckling and Post buckling Resistance of Steel-Girders" Journal of structural engineering, Vol. 140, pp. 198–208.
6. Magnucka.G (2014), "Mechanical Behavior of the Shear-Patch Loading Interaction on Transversally Stiffened Steel Plate Girders", Latin American Journal of Solids and Structures, Vol. 39, pp.1721-1743.
7. Samadhan Morkhade.G, Rutuja Lokhande.S, Umesh Gund.D, Ajinkya Divate.B, Saurav Deosarkar.S, Mrunmayi Chavan.U, "Structural behaviour of castellated steel beams with reinforced web openings" Asian Journal of Civil Engineering, (2020).
8. Shady Gomaa.G (2014), "Behaviour Analysis of Stiffened Slender Plate Girders" Global Thinking in Structural Engineering) ISSN: 2278-0181, Vol.3 Issue 5, pp.1644-1648.
9. Shahabian.C (2008), "Behaviour of Plate Girders Subjected to Combined Bending and Shear Loading" -Scientia Iranica, Vol.15, pp.16-20.
10. Sruthi.V, Athira.B, Krishnan, Sruthi Das (2019), "Finite Element Investigation on Buckling Behaviour of Corrugated Web Beams-Ansys Workbench", International Research Journal of Engineering and Technology (IRJET), Vol. 06, Issue 05, pp. 6599–6604.
11. Zhang.W, Li.Y, Zhou.Q, Qi.X, Widerra.G.E.O (2000), Optimization of the structure of an H-beam with either a flat or a corrugated web. Part 3. Development and research on H-beams with wholly corrugated webs, J. Mater. Process. Technol., Vol. 101 (1), pp. 119–123.