e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
Volume:03/Issue:07/July-2021 Impact Factor- 5.354 www.irjmets.com

AN INVESTIGATION FOR STRENGTH CHARACTERISTICS OF SUPPORT STRUCTURES USING DIFFERENT BOLT PATTERN CONFIGURATION

Snehal S. Devkar*1, Prof.A.S. Bharule*2

*1PG Student, Mechanical Department, Ssgmc, Shegaon, Maharashtra, India.

*2Assistant Professor, Mechanical Department, Ssgmc, Shegaon, Maharashtra, India.

ABSTRACT

This study involves the development of an analytical model for understanding the behaviour of the extended. During a visit to the pristine engineering solutions, Pune it was revealed that there is a need of designing a bolted structure for pressure vessel application. Currently welded connections are used in structural connections for the pressure vessel however there is the probability to tend to buckle because of their rigid nature. So, it is not preferred in the case of extended connections. Welded connection shows a brittle fracture in column webs, column flanges, welds, and shear tabs. Most bottoms of the beam flange damaged during the use of the welded connection. An alternative method to avoid such types of failures is the use of bolted connections. Bolted connections offer flexibility and can withstand the local buckling. This work presents the comparison study of the T-stub, bolted together in different patterns. Finite element analysis will be done to verify the best pattern among line, rectangular and circular patterns using CAD and CAE software.

Results from the FE mathematical model are verified with results from the ANSYS computer program as well as with the test results. Significant geometry and force-related variables are introduced; and by varying the geometric variables of the connections within a practical range, a matrix of test cases is obtained. Finally, salient features of the optimized Artificial Neural Network (ANN) via Genetic Algorithm (GA) analysis are introduced and implemented to predict the overall behaviour of the connection.

I. INTRODUCTION

1.1 Bolt Pattern

Structural connections of high-pressure cookware can be carried out mainly by welding joints; but its resistance, strong influence is also historical and they are likely to be formed by its harsh nature. Why they can't be used in a portable head either. Brittle failure of columns, sheets, tables, column flanges, welding and cutting protrusions has been reported. The most significant damage was organized in the lower part of the beam flange. Cracks are initiated by the welding process, roots, and then spreads to the fabric and / or flange. The tragedy that the test initiative appeared was also studying the reaction of the connection of a steel beam with a column. But the bolted connections of the end plates provide such comfort and a response to local bending. You can also use them, not that pressure is exerted, because they can be removed. When it comes to bolt-on design, the most important aspect of design is bolt painting. This design directly affects the strength of the structure. The purpose of this paper is to establish a set of guidelines, or a set of phrases, that will serve as a guide for bolting the bolted pattern of pressure vessel end plates. The main degrees of similar behavior are between T-shaped plugs and flange tension at the end of the plate for threaded leaning, and in the past researchers have used T-shaped plugs to facilitate coupling behaviour.

1.2 Bolt Pattern Configuration:

The bolt template is selected from the correct geometric shapes to simplify the design and manufacturing process. As shown in the figure below, the presence of correct geometric shapes needs to be done, which is suggested and checked in the study. Figure (a) shows the bolt diagram configure all bolts located at the same distance relative to the barrel. Name, name "Line", given this, vida configuration. Figure (b) is shown in the context of the traditional square bolt pattern. This model is widely used in industry, various payments and pitch distance. As follows (and so on), it is indicated that the proposed circular bolt scheme.

The main degrees of similar behavior between N-characters and tension in the flange of the end plate of a bolted joint, many people in the past researchers have used T-stumps to facilitate behaviors, relationships.

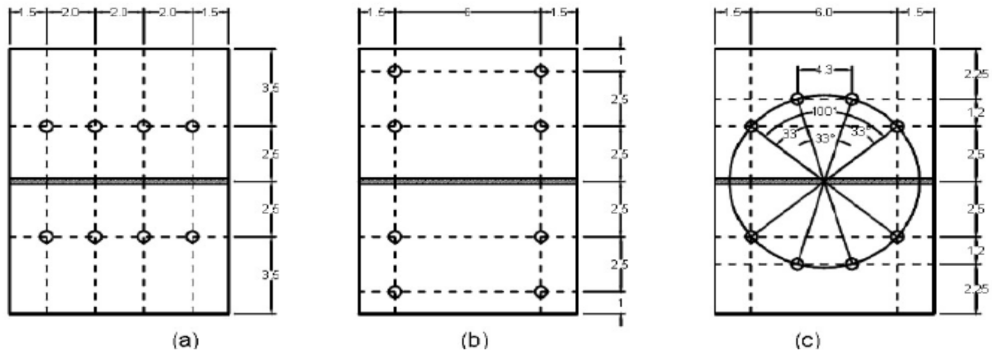

Figure 1.1: Bolt Pattern Configurations, (a) Line, (b) Square and (c) Circular

The distribution of bolt forces and the resulting reaction of the bolt group text T is a function of many parameters, including the diameter of the bolt and the preload, the caliber and the screw pitch and the number of rows of bolts, flexibility, detailed elements, the position of the contact surface, etc.

Saddle Angles and also the end plate in the yard work was done on the reconstruction of the environment, soft pre-tensioned bolts," influencing the tensile, bending and Flexural strength of the overall behavior of T-shaped connection. Even if the bolts are designed to transmit a tensile force, the design elements, these features can be reinforced by expanding the activity. But there is a bending of the screw, which causes an additional bending force on the bolt handle to increase the tensile force in the bolt. When the joists are pulled together, the posts with the lateral component are laterally forced and the bolt acts. However, in many cases, the cohesive sliding force is assumed to be the friction of the compression part of the bond.

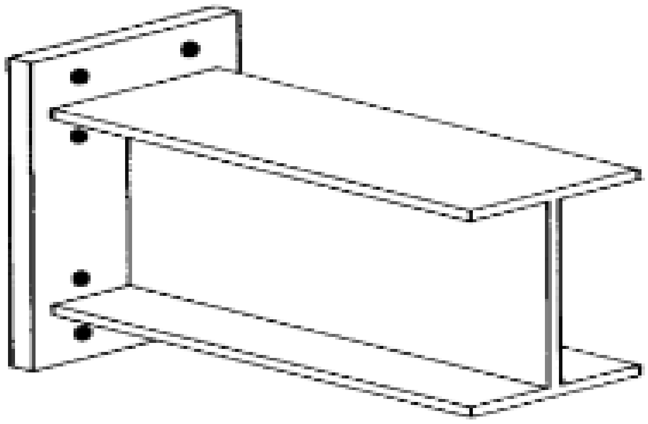

Figure 1.2: Typical configuration of end Plate connection

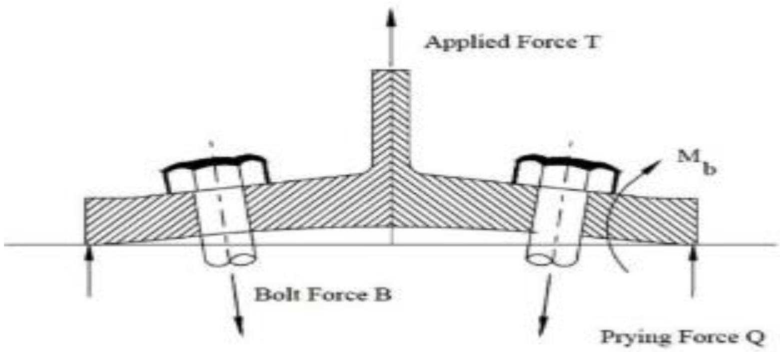

Figure 1.3: Typical configuration of end Plate connection

Tension: When the contact is turned, the bolt is the space that will affect the overall response, the external features of the story, but also the tensile strength that it affects. By an external force that defies the claim of forces, the body, bolt and connected layer work as one, which leads to an overall increase in rigidity. After the claim must be overcome (plate separation), the bolt force is only equal to the external load depending on the amount of pre-tension of the bolts.

Prying Action: Bolts of the high-voltage group components are exposed to foreign influence. It is used so that the thrust does not have to be transmitted through the connection of two rows of flags, which are shown in the photo.2. The bending moment that occurs in connection with the flange must be eliminated by the part whose size is Q=q (Mb) of the flange edge. This power developed based on the bending of the flanges is called the curious effect. If the figure shows that the bolt acts as a rallying force, then enter it with forces that make up the applied load and additional forces set outside it. For bolts acting to resist the applied load, H is given as follows

Flexural Rigidity: If the tree stumps in the flange are largely rigid during bending, then the deformation resulting in this way reduces the flange moments to a minimum. But, if the lip is flexible enough, the flexural deformation of the flange will lead to very and now, so the value will not be place.

Pretension: The joint location is a flange tear that occurs when an applied force overcomes the isca from the island. But as the Mb-required flanking member states evolve, the void value should remain invisible, and the float should remain undeformed. Therefore, if we assume that their lips are very flexible, then such reductions as space, cash bonuses of steps will lead to a decrease in the amount of external load. Test results show that the primary clip force does not affect the amount of clamping forces these limitations may apply.

II. METHODOLOGY

2.1 Objectives:

I'm in on it. For the purpose of studying communication parameters, via; Stud size, metal thickness in size increments Experimental Design Technique (DOE), using software, Mini-TAB.

ii. In order to study the correct design of the bolt for vascular diseases, high pressure transportation using the application, selected connections, settings) through external economic activities and experimental experiments in order to understand the bolt connection profile.

2.2 Scope of the project:

The initial form of the technical solution, as well as various types and sizes of vessels, must be made. These vessels are usually connected by welding. But this is in the welding process, the application is very difficult, and in some issues (what are the available options for welding /bolt"). In this project, the bolts and associated design will suggest that during the requirements of the welding process. It is a flexible alternative than the original software.

2.3 Research Methodology:

I. with a statement of approval.

ii.Literature review.

iii. The design is a calculation of the forces of the joints.

iv. GAZELLE, since this combination is chosen sample.

c. CAD modeling of the bottom of the plate with various bolts, the structure offered by the DOW.

vi. Analysis of FE - ANALYSIS constructs.

vii. Analysis and modification of the design, if necessary.

viii. Complete the Design

ix. Results and discussion

III. FINITE ELEMENT ANALYSIS

3.1 Introduction of Finite Element Analysis

Of course, element analysis(CEA) is an approach to solving practical (technical) problems. Problems that are first transformed into a series form a partial differential equation. As a result, to get a solution to the problems, you need to solve both differential and integral partial differential equations. The volume of solved equations is often so large that it can be said to be impossible to obtain a solution without using computers. And that is why there is a need for different packages, FE. Many spa packages are available for different applications. Some of our most popular feature packages are Pro-Mechanical, ANSYS, NASTRAN, etc.

Last element mathematical analysis (fea makets) is a numerical method for finding short-range solutions to external problems for partial differential equations. Use the allocation of an entire domain of the problem to simpler parts, called finite elements and variational methods, one of the calculus of variations to solve the problem by minimizing function-related errors. Similarly, the idea is that a very small pass can directly approximate a large circle. This FUNCTION covers methods for passing very simple, elementary equations over very small subdomains called finite elements, to approximate more complex equations over a large section.

Of course, element analysis (CEA) is a useful and powerful method for determining the stress-strain state of a structure or its components, but it is very difficult to analyze with serious analytical methods. It is a technique, composition, or device that breaks very small pieces (a limited number of elements) into a wide variety of types, shapes, and sizes. Such factors are also assumed to be a simplified picture of linear and quadratic deformation, etc.) due to "nodes" usually located at the corners or edges of the element. Elements are established in mathematics using the basic rules of mathematics, i.e. equilibrium and stability of motion, which leads to a large system of simultaneous equations. To solve these large, simultaneous equations of the system, a computer can be used to obtain a deformed shape, structure, or component under load. Based on this, stresses and deformations can be calculated. Of course, element analysis (FEA) is probably the most universal method for calculating stress intensity coefficients. This approach primarily estimates displacements in nodal parts of the body, which were idealized in a system of factors related to the arena and nodal point. Foreign economic activity has become a powerful tool for numerically solving a wide range of engineering problems. FEA is widely used to solve problems with scope numbers and difficulties in modals.

3.2 Steps of Finite Element Analysis

Solving a technical public service problem, such as finding tension and stress, design requires a three-step process:

1. Primary processing
2. Solution
3. post-production

Brief description of each step all rights reserved below

Step 1: Surface preparation

Using software, design, CAD, that, is provided by any INSURANCE software or 3D CAD model, that is, Pro-E, Catia, solid Edge, etc. is provided by the software service provider and the structure is modeled. The last FEATURE of the model consists of several parts that together make up the entire structure. They simply represent segments of the structure, but also that allows models of mechanical behavior and properties.

In areas where the geometry is complex (curves, cutouts, holes, etc.), more elements are required to accurately represent the shape, and areas with simple geometry can be delivered in a rough grid (not elements). Choosing the right elements that require initial experience in foreign economic activity, knowledge of the structure, behavior, and elements, availability of the program and its functions, and so on. Elements are connected by a node or shared things. At the stage of primary processing, combined with the geometry of the structure, certain constraints, pressure and mechanical properties of the structure are determined. So, during the initial processing, the entire structure is completely determined by the model geometry. The structure appears to be that of nodes and elements, called a "grid".

Step 2: Solution

At this stage, the geometry constraints, mechanical properties, and loads will be used to generate a matrix of equations for the elements, and can then be put together to create a global matrix structure equation. In the form of a single equation, as well as structural equations, always

$\{F\} = [K]\{u\}$

Where

$\{F\}$ = external force of the matrix,

$[K]$ = Global stiffness matrix,

$\{u\}$ = Offset matrix.

Then the equation is the solution for the transformation. Use deflection values you can calculate, stress management, stress management and response. All results are saved and can be used to create graphs, graphs and charts, analyze the post.

Step 3: After treatment

This is the last step of analyzing the last elements. The results of step 2 are usually presented as raw data and are difficult to interpret. After the analysis, CAD programs are used to manipulate the data in order to generate the rejected shape, design, stress graphics, animations, etc., the presented graphical appearance of the results is of great importance for understanding the behavior of the design.

3.3 FE Model

Numerical simulations were not performed using the general-purpose universal finite element program standard ANSYSTM 14.5.

Universal finite element - general-purpose software ANSYSTM 14.5 is used for modeling, analysis, and post-processing in recent models with round, triangular, square, and rectangular shapes.

3.3.1 Preference

In present study, for 2-D plate with cut out prefer static structural analysis.

3.3.2 Preprocessor

3.2.1 Element

For the analysis of plate, 8 NODE 183 SOLID Element is considered shown in fig.

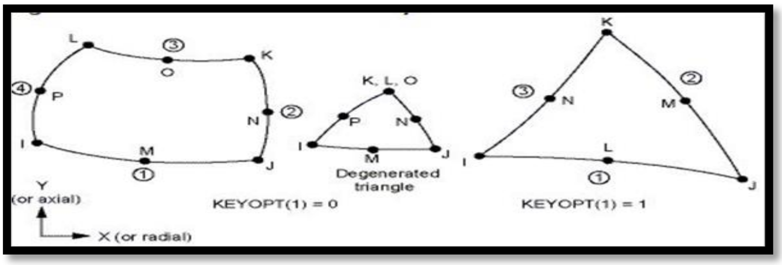

Figure 3.1: PLANE183 Geometry

This element is defined at 8 nodes with two degrees of freedom at each node: carefully act by observing in the nodal x and y directions. This element can be used as a straight element (plane stress, plane strain, and generalized plane strain) or an axisymmetric element.

This element has plasticity, hyper elasticity, creep, tenacity when stressed, large deflection and large deformation capabilities are present.

Then selected 8-node 183 to select a flat stress element with thickness in the" settings " section of the element type.

3.3.2.2. Real money

In the real world, the thickness of the model is constantly changing. Here, in this study, the thickness of 5 mm is

3.3.2.3 Material

The simulation uses solid-state elements with eight nodes. To study the stress distribution in an elastic series, NO is modeled as a linear-elastic material. For comparison, we note that the materials used during plate modeling in ANSYSTM 14.5) was the same as that of the experimental work, which is shown in Fig.3.2.

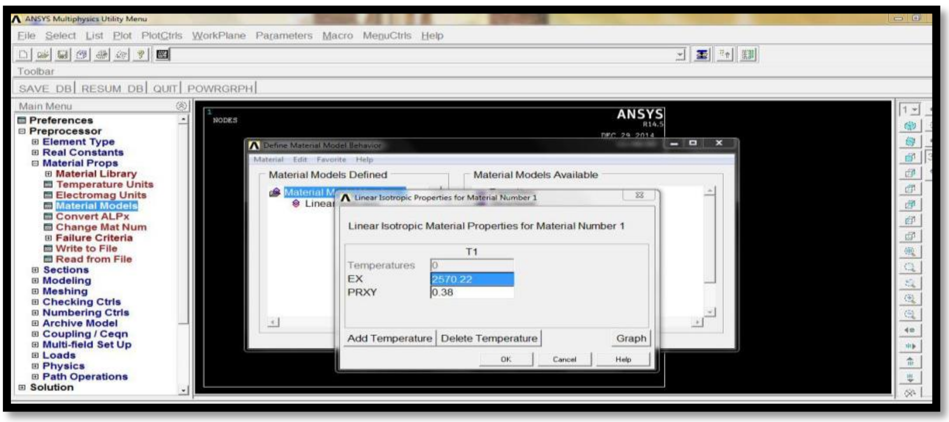

Figure 3.2: Material Properties in ANSYSTM 14.5

3.4 Finite Element Analysis

Of course, element analysis (CEA) is a useful and powerful method for determining the stress-strain state of a structure or its components, but it is very difficult to analyze with serious analytical methods. It is a technique, composition, or device that breaks very small pieces (a limited number of elements) into a wide variety of types, shapes, and sizes. Such factors are also assumed to be a simplified picture of linear and quadratic deformation, etc.) due to "nodes" usually located at the corners or edges of the element. Elements are established in mathematics using the basic rules of mathematics, i.e. equilibrium and stability of motion, which leads to a large system of simultaneous equations. To solve these large, simultaneous equations of the system, a computer can be used to obtain a deformed shape, structure, or component under load. Based on this, stresses and deformations can be calculated. Of course, element analysis (FEA) is probably the most universal method for calculating stress intensity coefficients. This approach primarily estimates displacements in nodal parts of the body, which were idealized in a system of factors related to the arena and nodal point. Foreign economic activity has become a powerful tool for numerically solving a wide range of engineering problems. FEA is widely used to solve problems with scope numbers and difficulties in modals.

3.5. basic concepts

Splitting the domain into simpler parts has a number of advantages:

I'm in on it. It is difficult to imagine exactly geometry

ii. Integration of various material properties.

iii. Simple general solution ideas

iv. Capture local effects.

A typical method of operation is to divide the scopes into a set of subdomains, with each subdomain represented by a set of equation elements, the original assignments, and later a system recombination of all sets of equation elements of the global system of equations for the final calculation. It is a global system of equations that has known methods for solving, and can be computed from the primary value of the original problem to obtain a numerical answer.

In the first stage of the above equations, there are elements of simple equations that locally approximate more complex equations to be studied, where the source of equations are often partial differential equations (PDEs). To explain the convergence in this process, FEM is usually posed as a special case of the Galerkin method. The process in the language of mathematics is to create the inner products of the remainder and weight functions used, and set to zero. Simply put, this procedure reduces the approximation error by setting the PDE trial function. The discrepancy is the error created by the trial functions; the body mass of the function is the polynomial approximation characteristics projected by the discrepancy. This process eliminates all spatial derivatives that are PDEs, thus approximating local PDEs to a set of algebraic equations for stationary state problems and a set of ordinary differential equations for transient problems.

These sets of equations are equations of elements. They are linear if the underlying PDE is a line, and vice versa. Sets of algebraic equations arising from urban state problems are solved by numerical linear algebra methods, and sets of ordinary differential equations arising in transition problems are solved by creation and general integration using standard methods like the Euler method or the Runge-Kutta method.

In the next step, the above global system of equations is created for the element equations by changing the coordinates from local nodes to the lower alan global nodes domain. This is a space that includes the corresponding orientation adjustments applied to the base coordinate system. This process is often carried out in FIVE programs that, with their help, coordinate information created in subdomains.

Well, this can be understood by its practical application, known as last element analysis (fea makets). Foreign economic activity is applied in mechanical engineering from the calculation of a tool for conducting engineering analysis. This includes using methods that create a grid to divide a complex problem into smaller components, as well as using a software product encoded using FIVE algorithms. When applying fea makets, complex problems usually represent physical systems beyond its underlying physics, so the Euler-Bernoulli beam equation, heat equation or Navier-Stokes equations, explicit or PDE, or integral equations, and, divided into small elements, complex problems represent different realms of a physical system.

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
Volume:03/Issue:07/July-2021 Impact Factor- 5.354 www.irjmets.com

FEA is the best choice for complex area analysis tasks (for example, automobile and oil pipelines), area changes (as in solid reaction with traffic abroad), when the required accuracy varies across the entire area or the solution has not. Foreign economic activity modeling is the most valuable resource, as it overcome many cases where creating and testing a rigid prototype for various high-precision cases. For example, when simulating a pre-collision, you can increase the accurate forecast for "important" areas, like the front of the car, and reduce it for the rear (and thus reduce the cost of modeling). Another example might be numerical weather forecasting, where more important than accurate forecasts are the development of strong non-linear events (such as tropical cyclone conditions or ocean eddies), rather than relatively calm areas.

IV. RESULTS AND DISCUSSION

4.1 ANALYSES OF RESULTS

4.1.1 Effect of Bolt Diameter

Studied the effect of different bolt diameters on primary stability, last-moment, and rotational ability, which were found to be characteristic, due to the three values of end plate thickness of 16, 20, and 25 mm, it should be noted that the growth of bolt diameter and plate cover thickness leads to an increase in primary stability, last-moment, and rotational ability.

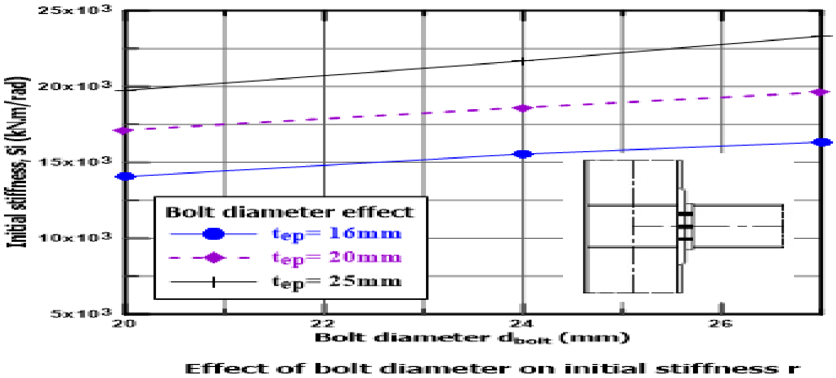

Figure 4.1: Effect of bolt diameter

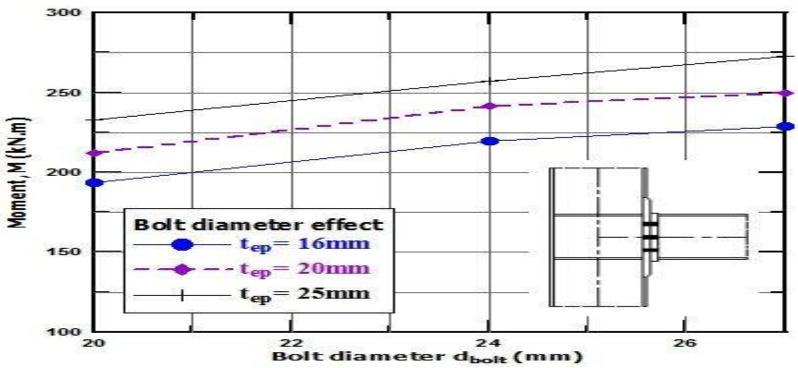

Figure 4.2: Effect on ultimate moment

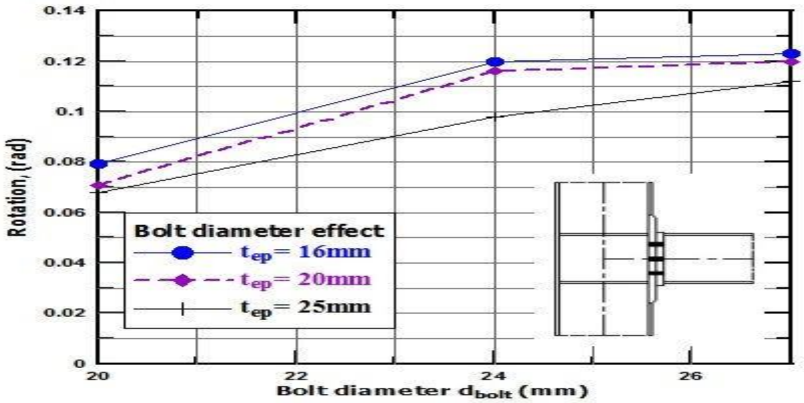

Figure 4.3: Effect on rotation capacity

The effect on bolt diameter is on primary stability, last-minute movement, and rotational ability. When the smallest initial stiffness that limits the moment and power of rotation is, in the event that a bolt with a diameter of 20 mm is used. in this case, the fracture occurs because the bolt breaks apart based on bolt preparation, causing brittle fracture of the joints and damage caused by bending resistance. Precisely, the end plate, with an average thickness of 20 mm, increases the bolt diameter from 20 to 24 mm, which increases the primary stability, last-minute, and rotational capacity by approximately 8.7%, 14%, and 63%, respectively. But to increase the diameter of the bolts from 24 to 27 mm, you need to increase the early strength, maximum torque and rotational power by about 5.4%, 3.5% and 3.5%, respectively.

4.1.2 effect on plates

with Photo thickness. 10 shows the effect of changing the thickness of the end plate on the primary stability, at the last moment, and rotational ability to show the transition. Three values of plate thickness are considered, after all, 16, 20 and 25 mm, the results for calculating the bolt diameter together, 20, 24, 27 mm, in general, it seems that the initial strength increases significantly with increasing plate thickness, and it increases with increasing bolt diameter. In addition, it can be noted that, in the diameter of one bolt, and at the end of the thickness of the plate, has a serious impact on the service life and rotational ability.

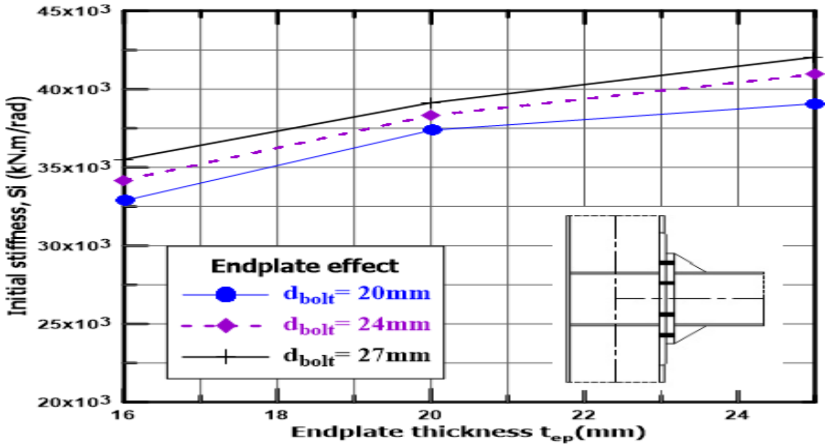

Figure 4.4: Effect on initial stiffness

This may be due to an increase in the thickness of the end plate, giving more traction to the screw, which increases the force, but this growth is not so great, and the reason will be brittle, failure of bolts, nuts, small rotational ability of the joint. It is assumed that, in the end, the plate needs to increase the thickness of the end plate to prevent brittle destruction of the bolt and to get plastic contacts for the possibility of return occurs at the end of the album. But, in the end, the plate must have the necessary thickness to prevent brittle destruction of the most end plate.

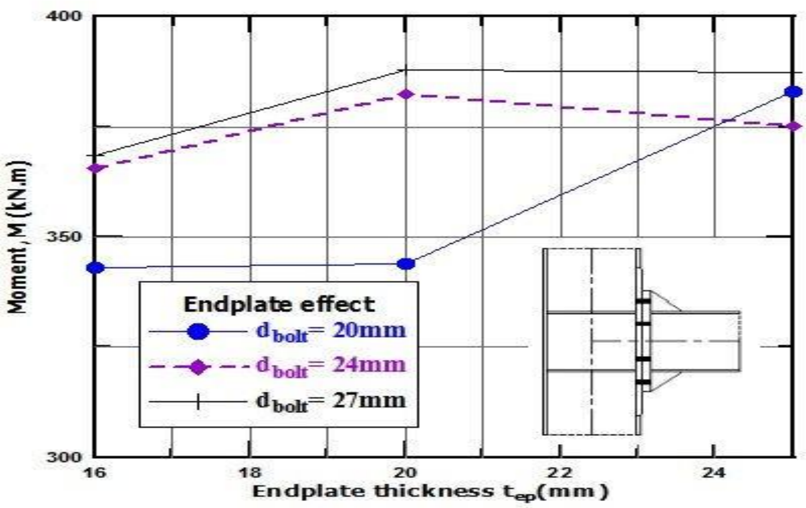

Figure 4.5: Effect on ultimate moment

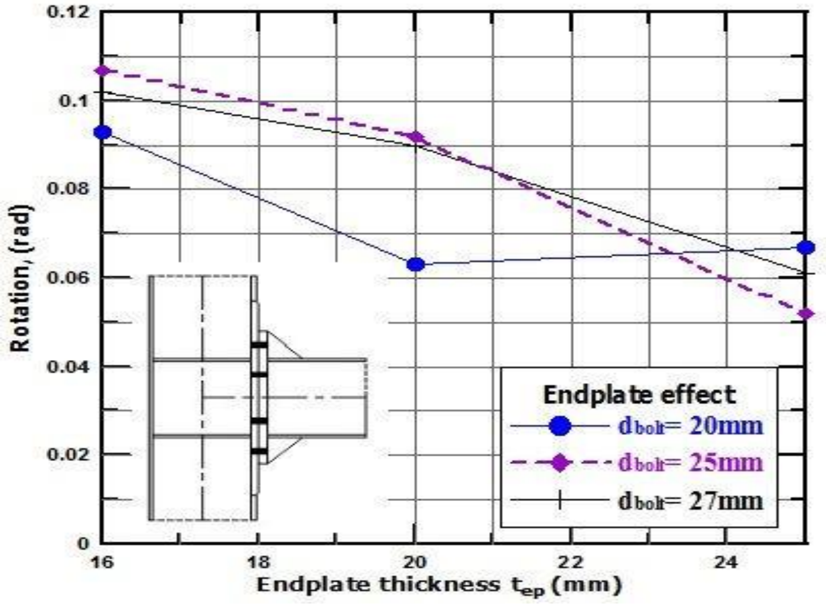

Figure 4.6: Effect on rotation capacity

4.1.3 Effect of Column Tension Stiffener

In order to study the influence of column reinforcement stress behavior in communication, we analyzed four models with different values in the column reinforcement stress at height, " but the column does not say,

hstiff= 1/2 (- ht − 2tf)hstiff = 3/4 (- ht − 2tf) and hstiff = (ht − 2tf)". Contact with the tension of the column, the tenacity representing the flush nature of the connection with the end plate. Curve moment - rotation of the entire photo. 11. It should be noted that with the column, the column reinforcement increases the limiting present time and rotational capacity, small size, 3%, within. It can be noted that the initial strength of the joint slightly increases with increasing product size. Because in this column, but it is rigid, its flange is easily bent if the intersection point between the canvas column and the flange.

The presence of a panel zone, the stress of the stiffening ribs, reduces the size of deformation in the panel connection zone, although it does not seriously affect the an-rotational ability. For end joints of plates, their rotation consists of shear deformation of the panel area, as well as rotation and rotation, at the end of the album. This result indicates that deformations of the panel area are more important as a source of end plate-end plate coupling during rotation of the uncompressed panel, rather than territorial deformations that are characteristic of stability. How can the length of time for the deformation of the panel area cause the connection of the closing plate to be in semi-rigid

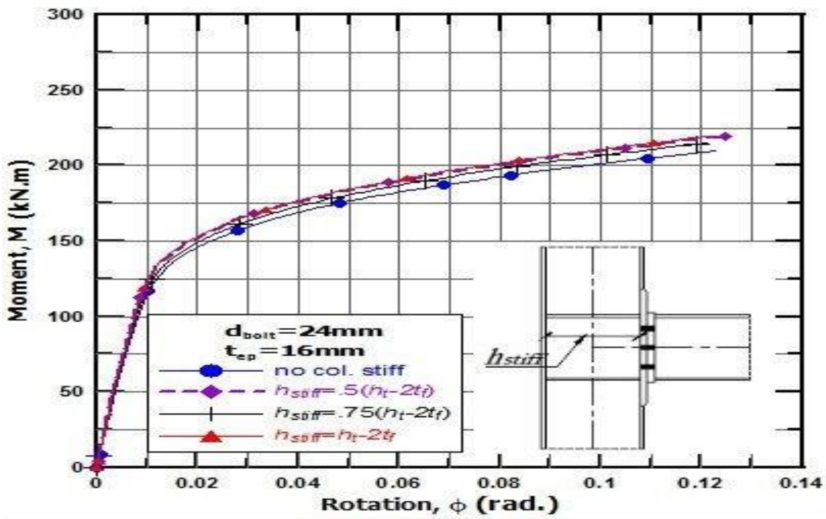

Figure 4.7: Variation of M- curve for different length of column stiffener.

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
Volume:03/Issue:07/July-2021 Impact Factor- 5.354 www.irjmets.com

4.1.4 Effect of Extended End-Plate and Angle of Rib Stiffener

In this section, we investigate the effect of an elongated end plate with a rib reinforcement on the behavior of the ratio under consideration. Four models were analyzed. The model is not in rib reinforcement to represent the colors of the end plate connection type. All three other models had an extended end plate reinforced with ribbed fabric. Three important stiffener angle parallel widths of 30°, 45° and 60°will be considered. Curve moment-rotation of the entire photo. 12, a bolt with a diameter of 20 mm, and, in the end, a plate with a thickness of 16 mm compared to the flush plate, the disk type of primary stability, at the last moment, and the rotational ability of the elongated end joint plate is significantly higher, and this indicates that the elongated end number, as well as its ribs and piece, significantly contribute to improving the behavior in the connection of the end plate. By using the extended end plate, instead of emptying the last moment growth of 60 to ~ 90 %. It is the increase in the stiffener angle from 30° to 45° and 45° to 60° that causes an increase in the torque limit, with an increase of approximately 16% and 4%, respectively.

Deformation of a unit shape equivalent to plastic deformations is shown in Fig. 13 and extended joints of end plates of various stiffener angles. It should be noted that the rotation of the buried nature of the connection between the end plate mainly contributes to the relative deformation between the end plate and the column flange. Conversely, when the end joints of the plates are stretched, the rotation comes from shear deformation of the panel area, as shown in Fig. 13. It should also be noted that the difference between the end lövhəli and the flange-column affects the stiffener angles. After that, increase the angle of the stiffener to reduce the habitus to deformation. This may be due to an increase in the size of the arc reinforcement ribs, which increases the tensile strength of the plate, which can delay reaching the income level and limiting moments.

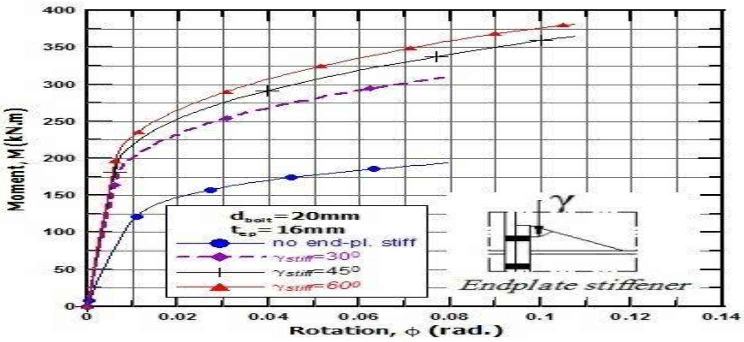

Figure 4.8: Variation of M-ϕ curves for different connection type

Rib reinforcement has a serious impact on the behavior of elongated end coupling, and the above results show that very rigid joints, the higher the last, but not currently, tenacity, because reinforcement can change the load switching mechanism and develop a pent-up state.

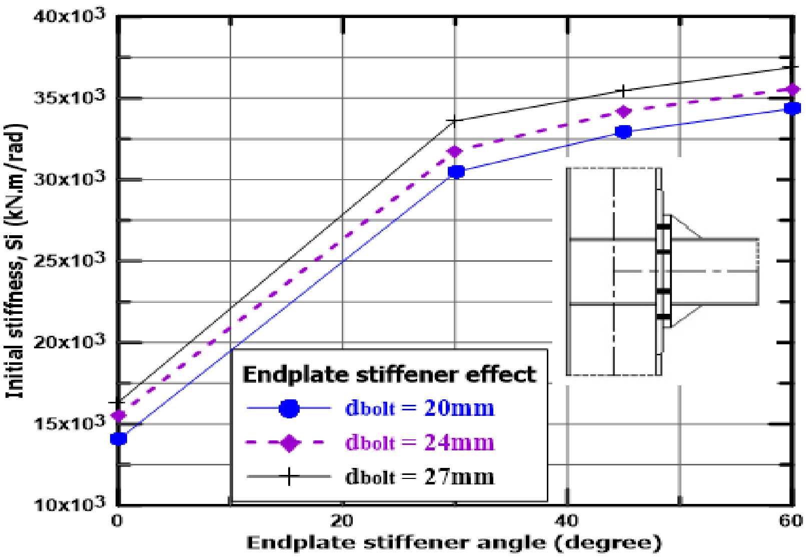

Figure 4.9: Effect on initial stiffness

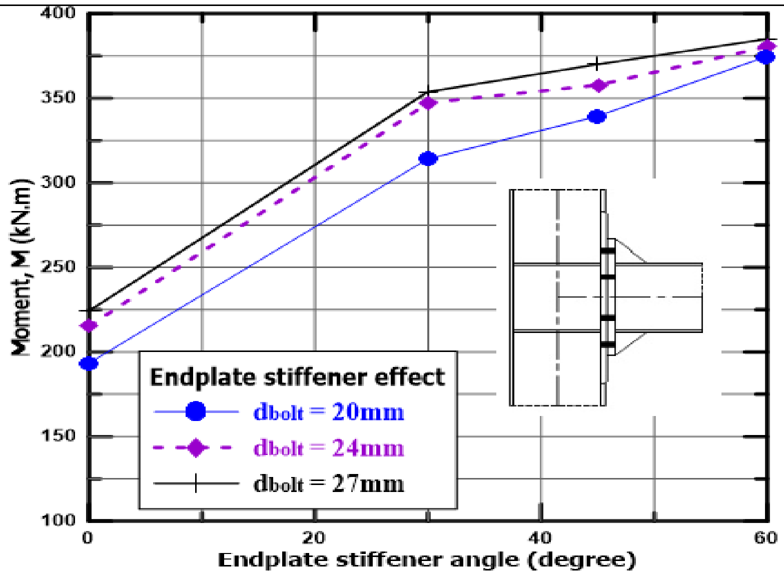

Figure 4.10: Effect on ultimate moment

The results are presented: the thickness of the end plate, in 16 mm, in general, it seems that h (considerations), growing edges-iron corners, to all values, culture. After that, during rotation, the capacity increase in case the bolt diameter is greater than or equal to 24 mm, namely the use of an elongated end plate with ribbed reinforcement can increase at the last moment by 60 ~ 90%, and the initial strength is about 115~140% more than in the flush connector. Then, when the main stiffeners increase from 30° to 60°, the early strength increases, and at the last moment about 9 ~ 12% and from 8 to 19% - up, respectively.

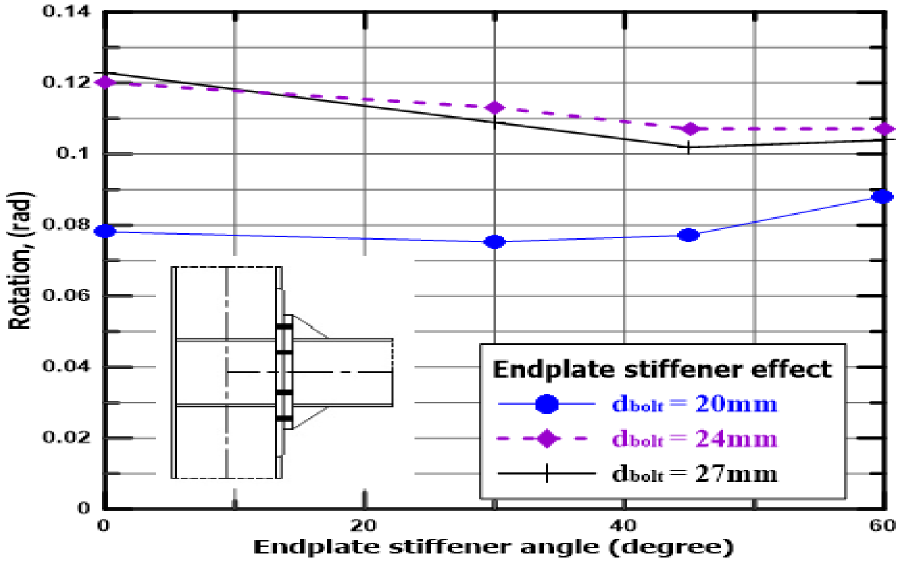

Figure 4.11: Effect on rotation capacity

The results obtained for the static structural analysis of vida oil by issuing from the boundary conditions, vibration characteristics of the machines and its weight, the analysis of the machines that were, already represents a finite element analysis. Thus, verification of these results takes place in the creation and, in general, by the method of the ANSYS software package.

V. CONCLUSION

5.1 Conclusions

In this paper, we present a three-dimensional finite element model for analyzing joints, sheets subjected to monotonous loading, with various types of data, as well as for studying the influence of various parameters on the behavior of joints. Cases in which this study provides the following main results:

The proposed finite-element model allows that, in fact, both for non-linear behavior of end plates, but also for steel contact. The pre-tension force in the bolts and the connection between the end lövhəli and the columns is carried out smoothly. First, an eigenvalue analysis is performed, in order to go to the first mode of stability loss, the corresponding scales are used, because the first one works in nonlinear analysis.

Comparison of the experimental results with the results obtained using the Finite Element model shows that at the moment the numerical model is able to model and predict quite well the accuracy of the behavior of the bolt connection at the end of the album. The comparison includes the moment-rotation curve and the failure location.

The communication feature makes it possible to simulate the real behavior of low-cost, and in a relatively short period of time compared to the one established during the experimental sınaqlarda. ABAQUS, the program provides a formal and rational study of the behavior of joints, end plates. The model is experimental evidence for the court, any other requests can be used.

Numerical analysis shows that the good behavior of the pre-tensioned bolt is consistent with theoretical and experimental results.

5.2 Summary

I deal with structural relationships and related types of research.

ii) Discussed in various bolt model layouts.

iii) construction, bolt, nuts, together with the forces acting on the bolt that is the area under study.

iv) the importance of the pattern and bolt and link is analyzed.

VI. REFERENCES

[1] Douty, R. T., and McGuire, W. (1964). “High Strength Bolted Moment Connections with application of plastic design,” Journal of the Structural Division, ASCE, 91(2), 101-128.

[2] A.R. Kukreti, T.M. Murray and A. Abolmaali (1987). “End-plate connection moment-rotation relationship”, Journal of Constructional Steel ResearchVolume 8, 1987, Pages 137-157.

[3] Murray T. M.. AISC Design Guide Series 4 “Extended End-Plate Moment Connections”, American Institute of Steel Construction, Chicago 2015.

[4] Mehdi Ghassemieh, Mohsen Nasseri, “Evaluation of Stiffened End-Plate Moment Connection through Optimized Artificial Neural Network” at Journal of Software Engineering and Applications, 2012, 5, 156-167.

[5] Kuldeep Kaushik, Avadesh K. Sharma, Rishi Kumar “A Review on Finite Element Analysis of Beam to Column Endplate Bolted Connection” IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume8, Issue 1 (Jul. -Aug. 2013), PP 97-103.

[6] Roxana Balc, AlexandruChira, NicolaeChira, “Finite element analysis of beam to column end plate bolted connection” at ActaTechnicaNapocensis: Civil Engineering & Architecture Vol. 55 No.1 (2012).

[7] R. E. S. Ismail, A.S. Fahmy, A. M. Khalifa, Y. M. Mohamed , “Numerical Study on Ultimate Behavior of Bolted End-Plate Steel Connections”, Latin American journal of solids and structures - DOI: http://dx.doi.org/10.1590/1679-78251579.

[8] Kennedy, D.J. and Hafez, M.A. (1984). “A Study of End-Plate Connections for Steel Beams,” Canadian Journal of Civil Engineering, 11(2), 139-149.

[9] Aggarwal, A.K. and Coates, R.C. (1987). “Strength Criteria for Bolted Beam-Column Connections,” Journal of Constructional Steel Research, Elsevier Applied Science, 7(3), 213-22.

[10] Murray, T.M. (1988). “Recent Development for the design of Moment End-Plate Connections,” Steel Beam-to-Column Building Connections, W.F. Chen, ed., Elsevier Applied Science, New York, 133- 162.

[11] AISC (1989). Manual of Steel Construction, Allowable Stress Design, American Institute of Steel Construction, Chicago.

[12] Murray T. M., (1990). AISC Design Guide Series 4, Extended End-Plate Moment Connections, American Institute of Steel Construction, Chicago.

[13] Krishnamurthy, N. (1978). “A Fresh Look at Bolted End-Plate Behavior and Design,” Engineering Journal, AISC, 15(2), 39-49.

[14] Ghassemieh, M., Kukreti, A.R. and Murray, T.M. (1983). “Inelastic Finite Element Analysis of Stiffened End-Plate Moment Connections,” Research Report No. FSEL/MBMA 83-02, Fears Structural

Engineering Laboratory, School of Civil Engineering and Environmental Science, University of Oklahoma, Norman, Oklahoma.

[15] Murray, T.M., Kurkreti, A.R. (1988). "Design of 8-Bolt Stiffened Moment End-Plates," Engineering Journal, AISC, Second Quarter, 1988, 74-78.

[16] Ghobareh A, Osman A, Korol RM. (1990), "Behavior of Extended End-Plate Connections under Cyclic Loading," Engineering Structure, 12(1), 15-27.

[17] Ghobarah A, Korol RM, Osman A. (1992), "Cyclic Behavior of Extended End-Plate Joints," Journal of Structural Engineering, 118(5), 1333-53.

[18] Graham, J. (1993). "Observation from the Behavior of Bolted Beam to Unstiffened Column Rigid Connections," The Structural Engineering, Institution of Structural Engineers, 71(6), 99-105.

[19] Borgsmiller, J.T. (1995), "Simplified Method for Design of Moment End-Plate Connections," M.S. Thesis, Department of Civil Engineering, Virginia Polytechnic Institute and State University, Blacksburg, Virginia.

[20] Adey, B.T., Grondin, G. Y., and Cheng, J.J.R. (2000). "Cyclic Loading of End Plate Moment Connections," Canadian Journal of Civil Engineering, National Research Council of Canada, 27(4), 683-701.

[21] Yorgun, C., Bayramoglu, G., (2001). "Cyclic Test for Welded-Plate Sections with End-Plate Connections," Journal of Constructional Steel Research, 57(12), 1309-20.

[22] P. Ganesh Prabhu, D. Ambika, "Study on Behaviour of Workers in Construction Industry to Improve Production Efficiency", International Journal of Civil, Structural, Environmental and Infrastructure Engineering Research and Development (IJCSEIERD), Vol. 3, Issue 1, Mar 2013, 59-66.