Structural Components Design and Analysis of Four Column Type Hydraulic press

C. Nandhamuni Reddy[1] *, B. Nithin Kumar[2] and K. Kumar[3]

[1]*Professor, Department of Mechanical Engineering, Annamacharya Institute of Technology and Sciences, Tirupati*

[2]*Scholar, Department of Mechanical Engineering, Annamacharya Institute of Technology and Sciences, Tirupati*

[3] *Associate Professor, Department of Mechanical Engineering, Annamacharya Institute of Technology and Sciences, Tirupati*

Abstract: This paper focuses on developing structural analysis of components for 100 Ton hydraulic press by means of fundamental calculations and Finite Element Analysis (FEA) techniques for study and optimization of structural design. The core part of this work is to select machines which operate at high capacity at low cost. The design of hydraulic components is developed based on the nature of load acting and working conditions. To reduce the material and money being spent by the company, the design is made in an effective manner with less machining operation and less material usage.

Keywords: Hydraulic components, Manual calculation, Solid Edge, Designing, 3D Modeling and Analysis.

1. Introduction

Hydraulic Press: A hydraulic press is a machine that utilizes hydraulic power to generate a compressive force for various industrial applications. It operates on the principle of Pascal's law, when a change in pressure at any point in an enclosed incompressible fluid at rest is transmitted equally and undiminished to all points in all directions throughout the fluid, and the force due to the pressure acts at right angles to the enclosing walls. Hydraulic presses are widely used in manufacturing, metalworking, automotive, and construction industries for tasks such as forming, shaping, stamping, punching, and forging.

1.1 Operation of Hydraulic Press

The hydraulic press consists of a hydraulic system that includes a hydraulic pump, hydraulic fluid reservoir, hydraulic cylinders, valves, and piping. The hydraulic pump is powered by an electric motor or an internal combustion engine and pressurizes the hydraulic fluid. Hydraulic fluid, usually oil, is stored in the reservoir and is pumped by the hydraulic pump into hydraulic cylinders when pressure is applied. The hydraulic fluid is an incompressible fluid that transmits force from the pump to the cylinders. The press has one or more hydraulic cylinders equipped with pistons. When hydraulic fluid is pumped into the cylinders, it applies pressure to the pistons, causing them to move. The size and number of cylinders determine the force that the hydraulic press can generate. The

---

[1] nadamuni_reddy@rediffmail.com

material to be compressed or shaped is placed between the platens or dies of the hydraulic press. The hydraulic fluid is then pumped into the cylinders, exerting pressure on the pistons. As the pistons move, they transmit the force to the platens or dies, which compress or shape the material. Hydraulic presses can generate a large amount of force because the pressure applied to the hydraulic fluid is transmitted evenly throughout the system. The force applied to the pistons is amplified based on the ratio of the piston areas, allowing hydraulic presses to exert high pressures on the material.
The operation of the hydraulic press is controlled by valves that regulate the flow of hydraulic fluid into and out of the cylinders. The operator can control the speed, force, and timing of the press cycle by adjusting the valve settings. Hydraulic presses are equipped with safety features such as pressure relief valves, emergency stop buttons, and safety guards to prevent accidents and ensure operator safety.

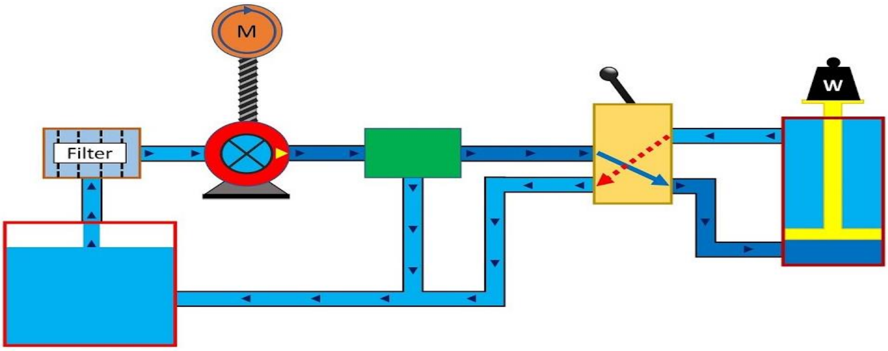

Figure 1: Operation of Hydraulic Press

1.2 Brief Introduction to FEA

Finite Element Analysis (FEA) is a computerized method for predicting how a structure will react to real-world forces (vibration, heat, fluid flow, and other physical effects). FEA works by breaking down a real object into a large number of finite elements and using mathematical equations to predict the behavior of each element. A computer then calculates the individual element behaviors to predict the behavior of the actual object. A structure is classified as “Built for Infinite Life” when its stress levels are reduced enough to eliminate the effects of fatigue on the specified material. The FEA process typically involves the following steps:

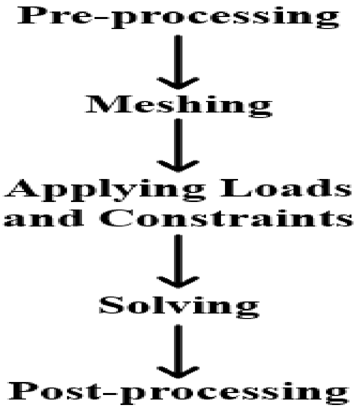

Figure.2 Finite Element Analysis (FEA)

Pre-processing: This involves defining the geometry of the structure and dividing it into finite elements. Material properties, boundary conditions, and loading conditions are also specified during this stage.

Meshing: The structure is discretized into a mesh of smaller elements, ensuring that each element accurately represents the geometry and behavior of the structure.

Applying Loads and Constraints: External loads and constraints are applied to the model based on the real-world conditions it will experience during operation. These may include forces, pressures, temperatures, and constraints such as fixed boundaries or prescribed displacements.

Solving: The equations governing the behavior of each element are solved numerically to determine the response of the entire structure. This typically involves solving systems of linear equations using iterative methods.

Post-processing: Once the analysis is complete, the results are visualized and interpreted. Engineers can examine stress distributions, displacement contours, deformation patterns, and other relevant data to assess the structural performance and make informed design decisions.

1.3 Existing Mechanical Designing Software

There are several mechanical design software options available in the market, each offering various features and capabilities tailored to different needs and preferences. Some popular mechanical design software includes:

1. SolidWorks: A widely used 3D CAD software known for its intuitive interface and extensive capabilities for designing mechanical components and assemblies.

2. AutoCAD: AutoCAD is a specialized version of CAD software designed specifically for mechanical engineering tasks, offering a comprehensive set of tools for drafting, design, and documentation.

3. CATIA: CATIA is a powerful CAD/CAM/CAE software suite developed by Dassault Systems, commonly used in aerospace, automotive, and other industries for complex mechanical design and engineering tasks.

4. Fusion 360: Autodesk Fusion 360 is a cloud-based CAD/CAM/CAE platform that combines design, engineering, and manufacturing capabilities in a single package, suitable for mechanical, industrial, and product design.

5. Solid Edge: Solid Edge is a robust 3D computer-aided design (CAD) software developed by Siemens Digital Industries Software, widely recognized for its intuitive interface and comprehensive features. With parametric modelling capabilities, assembly design tools, and specialized modules for sheet metal design, simulation, and analysis, Solid Edge empowers users to create, edit, and visualize intricate mechanical designs with ease. Its integrated drawing and documentation tools facilitate the generation of detailed 2D drawings, while built-in data management capabilities ensure efficient collaboration and version control. Solid Edge's cloud collaboration capabilities further enhance accessibility and teamwork, making it a preferred choice for mechanical designers and engineers across various industries.

1.4 Reason for Usage of Solid Edge Compared to the Remaining Software

- Solid Edge needs less configuration windows with ram of minimum 8 GB only. Comparing to remaining software, Solid Edge has 1-year free access to all premium features for students.
- AutoCAD software is used for only 3D modelling purpose, it doesn’t have features related to analysis.
- Similar to Solid Edge, Fusion 360 has the same features and has free access but that complete software runs on the bases of internet connection.
- Coming to SolidWorks and CATIA, there are completely premium versions which need to buy the access for every year, it’s of high cost for students to afford it.

These are the reasons beyond the selection of Solid Edge software for 3D modelling and analysis for the Designing of 4 column type hydraulic press.

2. Literature Review

D. Ravi et al. (2014) made significant strides in hydraulic press technology by leveraging PRO/ENGINEER and ANSYS software to develop and analyze a C-frame power press. Their approach successfully achieved weight reduction while ensuring structural integrity, marking a notable advancement in press design optimization [1].

N.A. Anjum et al. (2017) contributed to the field by designing a hydraulic press tailored for equal channel angular pressing. Through experimental validation, they demonstrated the press's ability to perform satisfactorily under a working load of 40 tons, showcasing its effectiveness in specialized metalworking applications [2].

Mohammed Iqbal Khatib et al. (2020) introduced a manual-operated 5-ton hydraulic press, emphasizing its user-friendly features such as interchangeable molds and dies without the need for ram assembly disassembly. This innovation enhances operational efficiency and versatility in various manufacturing processes [3].

Akshay Vaishnav et al. (2016) focused on optimizing hydraulic press crown design through Finite Element Analysis (FEA) using ANSYS software. Their rigorous analysis identified the safest design among multiple variants, contributing to enhanced structural robustness and operational safety in hydraulic press applications [4-5].

3. Methodology

The methodology employed in this study outlines a systematic approach for the development and analysis of mechanical components using Solid Edge, a leading 3D model. The methodology encompasses several sequential steps aimed at ensuring the robustness, functionality, and feasibility of the designed components [6-8].

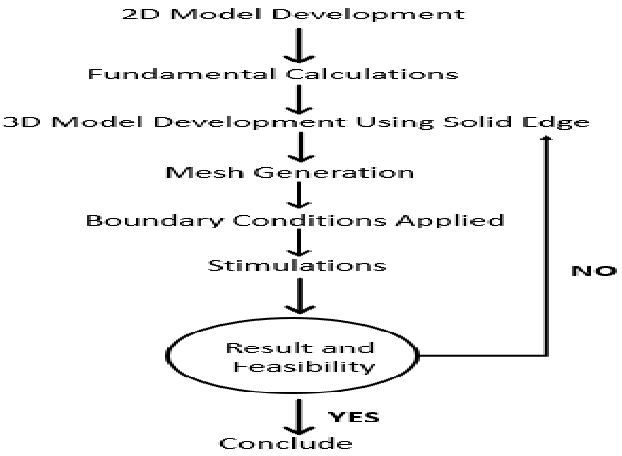

Figure 3: Methodology

Step 1: 2D Model Development and Fundamental Calculation: The process begins with the creation of a 2D model of the mechanical component, typically using drafting tools in Solid Edge. This involves defining key dimensions, geometric features, and constraints based on fundamental engineering calculations and design requirements. These calculations may include considerations such as material properties, loading conditions, and safety factors to ensure the design meets performance standards.

Step 2: 3D Model Development using Solid Edge: Once the 2D model is established, it serves as a reference for developing a more detailed 3D model in Solid Edge. The 3D model adds depth and complexity to the design, incorporating additional features, curves, fillets, and chamfers to accurately represent the physical component. Solid Edge offers a wide range of modeling tools and techniques to create intricate 3D geometry efficiently.

Step 3: Mesh Generation: After the 3D model is complete, it is prepared for finite element analysis (FEA) by generating a mesh. Meshing involves dividing the 3D geometry into smaller finite elements, allowing for numerical analysis of the component's behavior under various loading conditions. The quality and density of the mesh influence the accuracy of the analysis results, with finer meshes providing more detailed information but requiring greater computational resources.

***Step 4: Boundary Conditions Applied*:** Boundary conditions are then applied to the meshed model to simulate real-world operating conditions. This includes specifying loads, constraints, supports, and other external influences that the component may experience during use. Boundary conditions are crucial for accurately modeling the component's response to applied forces, thermal effects, and environmental factors.

Step 5: Result and Feasibility Analysis: The model is then subjected to FEA to analyze its structural integrity, performance, and feasibility. FEA software, integrated with Solid Edge, calculates stress distribution, deformation, strain, and other mechanical properties of the component under the specified loading conditions. The results are analyzed to determine whether the design meets predefined criteria, such as maximum stress limits, deflection tolerances, and safety margins.

***Step 6: Conclusion and Redesign*:** If the analysis results indicate that the component meets acceptable criteria, the design is concluded as feasible, and further steps may include documentation, prototyping, and production. However, if the results fall short of expectations or reveal potential issues, the 3D model undergoes redesign iterations to address identified deficiencies. This iterative process of analysis, redesign, and validation continues until the design meets all requirements and performance objectives.

By following this comprehensive approach, engineers can optimize the design of mechanical components, ensuring they meet performance, reliability, and safety standards before being manufactured and deployed in real-world applications.

4. Problem Identification

The main components like crown, guide ways and bed are subjected to continuous load which causes the formation of stress while working. To make the safer design the amount of stress acting on the components are calculated [9-12].

4.1 Design Calculation of Hydraulic Press

Every design begins with fundamental calculation by applying various kinds of stress development.

- Design of crown:

The crown is subjected to pure bending stress during the operation. Therefore, design considerations are essential for plates subjected to bending stress. When a static or dynamic load acts on any part of hydraulic press, then along with simple, tensile, compressive, shear stress, it also develops bending stress.

Considering bending moment M, the bending equation is given by,

$$\frac{M}{I} = \frac{\sigma}{Y} \tag{1}$$

Where, M = Bending moment
σ or σ_b = Bending stress
I = Moment of inertia of the cross-section
Y = Distance between neutral axis to extreme end $= \frac{d}{2}$

Max bending moment for simply supported beam $M = \frac{Wl}{4}$

Moment of inertia of cross-sectional area $I = \frac{bd^3}{12}$

Then, $\sigma_b = \frac{MY}{I}$ (Where $\sigma_b < \sigma_{\text{allowable}}$)

- Design of bed:

The bed is subjected to compressive force during the working of hydraulic press.

$$\text{Stress} = \frac{load}{area} \quad \text{i.e.,} \quad \sigma_c = \frac{F}{A} \quad (\text{where A = cross-sectional area of bed} = d \times h) \tag{2}$$

- Design of guide way:

In structural engineering, buckling is the sudden change in shape (deformation) of a structural component under load, such as the bowing of a column under compression or the wrinkling of a plate under shear. If a structure is subjected to a gradually increasing load, when the load reaches a critical level, a member may suddenly change shape and the structure and component is said to have buckled. Euler's critical load is used to determine the buckling stress of a column.

F= $\frac{\pi^2 EI}{L_e^2}$ (Where L_e= effective length)

In the cause of both ends are fixed the $L_e = \frac{length\ of\ Guide\ way}{2} = \frac{L}{2}$ (3)

Slenderness ratio = $\frac{L_e}{k}$ (Where K = radius of gyration = $\sqrt{\frac{I}{A}}$

4.2 Line Diagram of Hydraulic Components

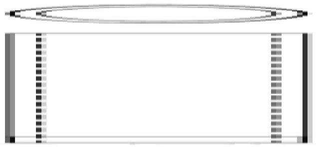

Figure 4: Line diagram of Guide Way

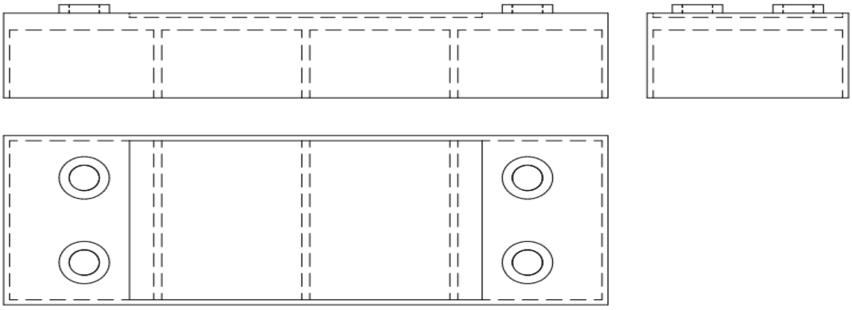

Figure 5: Line diagram of base Bed

Figure 4 and Figure 5 are line diagram of respective components with their necessary view for clear understanding.

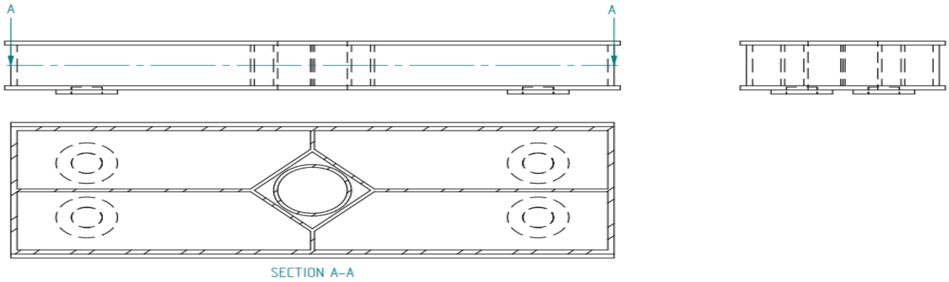

Figure 6: Line diagram and sectional view of Crown

In figure 6 the sectional view of section A-A displays a cross-sectional slice of the component, revealing its internal features, such as channels.

4.3 Material Selected

Different materials are used in machine design to optimize performance, durability, and cost-effectiveness for specific applications. For instance, high-strength steel is commonly chosen for structural components due to its excellent mechanical properties and resistance to wear and fatigue [13-14].

4.3.1 Grey cast iron type 60

Grey cast iron is a type of cast iron that contains graphite flakes, giving it a grey color when fractured. The "type 60" designation likely refers to the material's tensile strength, which is typically around 60,000 pounds per square inch. Grey cast iron is known for its good machinability, damping capacity, and thermal conductivity, making it useful in various applications such as engine blocks, pipes, valves, and machinery components. The exact properties of grey cast iron can vary based on its composition and manufacturing process.

Density	6920.000 kg/m^3
Thermal Conductivity	0.048 kW/m-C
Specific Heat	546.000 J/kg-C
Modulus of Elasticity	158579.411 Mpa
Poisson's Ratio	0.290
Yield Stress	276.790 Mpa
Ultimate Stress	416.685 Mpa
Elongation %	6920.000 kg/m^3

4.3.2 Stainless steel 420

Stainless steel 420 is a martensitic grade stainless steel known for its high corrosion resistance and good strength and hardness properties. It contains around 12-14% chromium, which provides its corrosion resistance and enables it to form a passive oxide layer on its surface, preventing further oxidation or rusting.

Additionally, stainless steel 420 typically contains a moderate amount of carbon (around 0.15-0.40%) which contributes to its hardness and strength. This grade of stainless steel is often used in applications where high wear resistance and moderate corrosion resistance are required, such as in surgical instruments, cutlery, scissors, dental and surgical instruments, and various industrial applications like molds and valve components.

While stainless steel 420 offers good mechanical properties, it is not as corrosion resistant as austenitic stainless steels (such as 304 or 316 grades) and is not suitable for applications requiring extreme corrosion resistance in harsh environments. However, it is well-suited for applications where moderate corrosion resistance coupled with high strength and hardness are necessary.

Density	7750.000 kg/m^3
Thermal Conductivity	0.025 kW/m-C
Specific Heat	502.000 J/kg-C
Modulus of Elasticity	199947.953 Mpa
Poisson's Ratio	0.290
Yield Stress	346.738 Mpa
Ultimate Stress	656.002 Mpa
Elongation %	7750.000 kg/m^3

4.3.3 Structural steel

Structural steel is a category of steel used in construction and engineering applications primarily for its high strength-to-weight ratio, durability, and versatility. It is commonly used to construct buildings, bridges, infrastructure, and various types of machinery and equipment.

Density	7836.000 kg/m^3
Thermal Conductivity	0.032 kW/m-C
Specific Heat	481.000 J/kg-C
Modulus of Elasticity	199947.953 Mpa
Poisson's Ratio	0.290
Yield Stress	262.001 Mpa
Ultimate Stress	358.527 Mpa
Elongation %	7836.000 kg/m^3

4.4 3D Model of Hydraulic Component

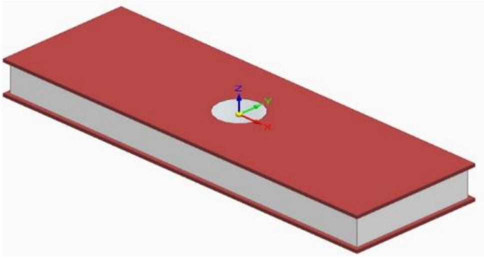

Figure 7: 3D Model of Crown

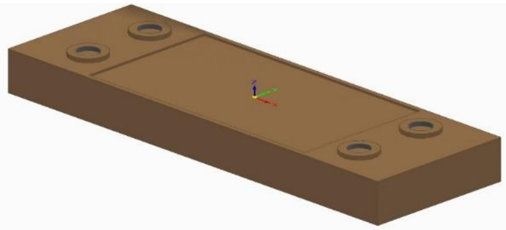

Figure 8: 3D Model of Bed

4.5 Applying Load and Constraints

Figure 9: 3D Model of Guide Way

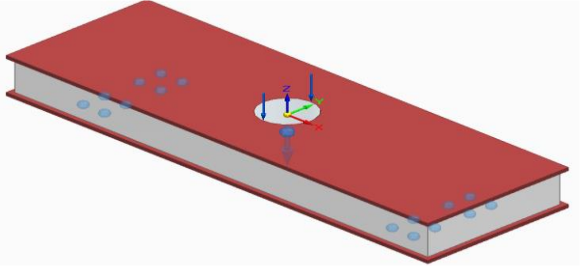

Figure 10: Applying of load with constraints for Crown

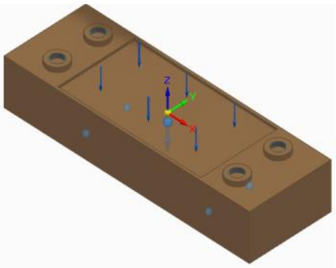

Figure 11: Applying of load with constraints for base Bed

The crown ensures uniform pressure distribution, resulting in more accurate and consistent forming or stamping of materials. From the structure, crown is the part which carry hydraulic cylinder load or from the reaction of press. For this reason, the press is analyzed with load of 150 Ton acting on it. The constrain are the edges of the crown because the guide way is attracted to the crown at the edge.

The compressive force generated by the hydraulic cylinder acts on the bed of the press. For this operation the bed needed to be rigid position. So, the constrain of the bed are taken as bottom surface of bed which will be in the direct contract with the ground.

Figure 12: Applying of load with constraints for Guide Way

As the press is in working condition, the reaction generated on the bed is transferred equally carried by all the guide ways. The threaded portion in the guide way is used to connect the bed and guide way. So, the constrain for the guide way is taken at thread part.

4.6 Mesh Generation

Mesh generation is a fundamental process in engineering analysis, vital for transforming complex geometries into discrete elements or cells to facilitate numerical simulations. By discretizing the geometry, mesh generation enables the application of numerical methods to solve governing equations and obtain approximate solutions for the entire domain. Additionally, mesh generation allows for the imposition of boundary conditions and constraints necessary for accurately simulating real-world scenarios. The quality of the mesh directly impacts the accuracy and efficiency of the analysis, with well-structured meshes ensuring accurate representation of physical systems while minimizing computational resources. Adaptive meshing techniques further enhance accuracy by refining or coarsening the mesh based on solution error, enabling convergence to stable and reliable solutions. Overall, mesh generation is indispensable for conducting reliable and insightful engineering analyses across various disciplines, providing engineers and scientists with valuable insights into the behavior of complex systems.

4.6.1 Types Mesh Shapes

There are several types of mesh shapes commonly used in finite element analysis (FEA). These mesh shapes include:

1. Tetrahedral Mesh: Tetrahedral elements are simple and versatile, consisting of four nodes and forming a pyramid-like shape. They are well-suited for complex geometries and irregular shapes but may require more elements to accurately represent curved surfaces.
2. Hexahedral Mesh: Hexahedral elements are cubic or rectangular in shape and have eight nodes. They are highly efficient for regular geometries and structured grids, providing accurate results with fewer elements compared to tetrahedral meshes.
3. Pyramid Mesh: Pyramid elements have a base with four nodes and an apex node, forming a pyramid-like shape. They are often used for transitioning between tetrahedral and hexahedral meshes in regions with varying element densities.
4. Prismatic Mesh: Prismatic elements have six nodes and are wedge-shaped, with a triangular base and rectangular top. They are commonly used in boundary layer regions in CFD simulations to capture flow gradients near walls.

5. Polyhedral Mesh: Polyhedral elements are composed of multiple faces and vertices, allowing for more flexibility in meshing complex geometries. They can provide a good balance between accuracy and computational efficiency.
6. Quad Mesh: Quad elements are two-dimensional elements with four nodes, typically used for modeling planar surfaces in 2D simulations. They are simpler than tetrahedral elements but may require additional elements to represent curved surfaces accurately.
7. Triangle Mesh: Triangle elements are also two-dimensional elements but have three nodes. They are commonly used for modeling planar surfaces in 2D simulations and can provide accurate results for simpler geometries.
These are some of the primary mesh shapes used in FEA and CFD simulations, each with its advantages and limitations depending on the geometry and physics of the problem being analyzed. Choosing the appropriate mesh shape is essential for obtaining accurate and efficient simulation results.

4.6.2 Reason for Selection of Triangle Mesh

Triangle meshes are often selected for their simplicity and efficiency in 2D simulations. With fewer nodes and elements, they require less computational resources and offer faster solutions. Their flexibility allows accurate representation of complex geometries and boundary layer effects in simulations. Additionally, triangle meshes exhibit good mesh quality metrics and can be easily adapted for local mesh refinement, contributing to accurate and stable simulation results.

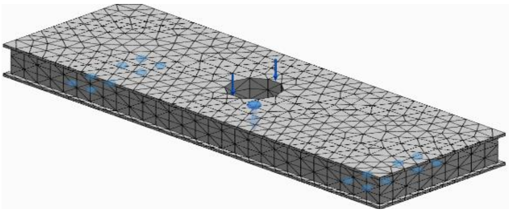

Figure 13: Triangular mesh on Crown with size of 20.92 mm

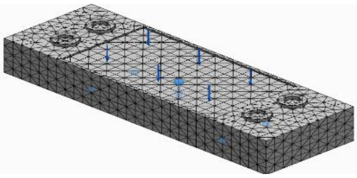

Figure 14: Triangular mesh on bed with size of 12.02 mm

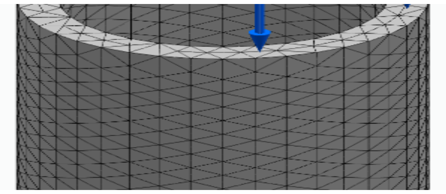

Figure 15: Triangular mesh on Guide Way with size of 20.02 mm

5. Results & Discussion

The components of hydraulic press are designed using Solid Edge software for working load of 100 Ton with the factor of safety 1.5 (i.e., designed load of 150 Ton). The following are the details of material properties used for components in hydraulic press.

Table 1: Materials used

Part Name	**Material of used**	**Density Kg/m^3**	**Yield Strength (MPa)**	**Modulus of Elasticity (MPa)**
Crown	Stainless steel 420	7833	344	210000
Guide ways	Structural Steel	7833	262	210000
Bed	Grey cast iron of	6920	275	110000

	class 60			

FEA is carried out using Solid Edge software to determine the stress acting in it. Below shown images are related to the output from analysis.

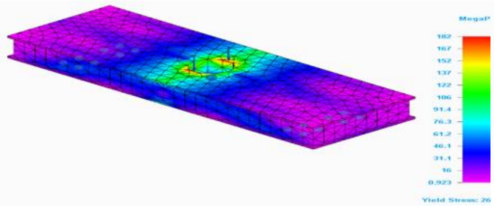

Figure 16a: Max Stress

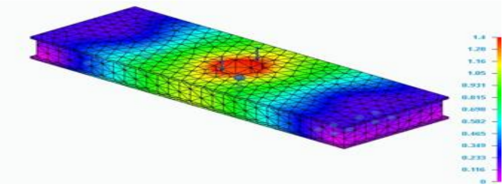

Figure 16b: Total Deformation

182 MPa is the Stress generated on the crown as shown in figure 16a due to the applied load 0f 150 Ton. Total deformation of crown is 1.4 mm as shown in the figure 16b.

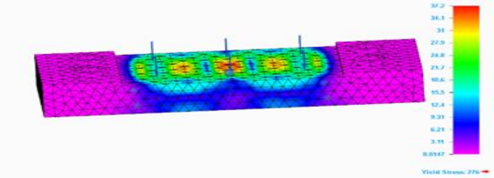

Figure 17a: Max Stress

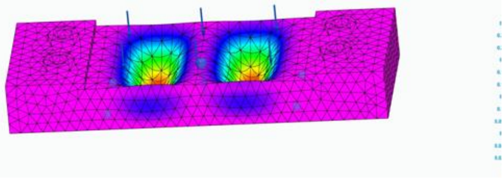

Figure 17b: Total Deformation

84.1 MPa is the Stress generated on the bed as shown in figure 17a due to the working of press of load 150 Ton. Total deformation of crown is 0.589 mm as shown in the figure 17b.

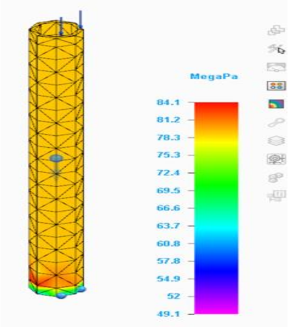

Figure 18a: Max Stress

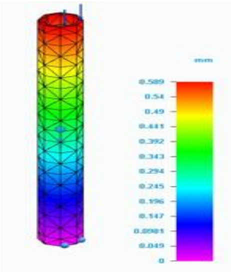

Figure 18b: Total Deformation

37.2 MPa is the Stress generated on the guide way as shown in figure 18a due to the buckling. Total deformation of crown is 0.28 mm as shown in the figure 18b.

Following are the results obtained for the above hydraulic components. The results are as follows: -

Table 2: Analysis data output

Part Name	**Max Stress (MPa)**	**Deformation (mm)**
Crown	182	1.4
Guide ways	84.1	0.589
Bed	37.2	0.28

6. Conclusion

The design of hydraulic components has made in the limit of acceptable range. Through the finite element analysis (FEA), identified that the design developed can withstand the testing load of 150 Ton (with a factor of safety of 1.5). The findings have highlighted the strength of design, such as providing channels for the crown, for structural support. However, certain areas for improvement have also been identified, including weight optimization of components for reduction of cost. This study highlights the importance of design and analysis process in achieving safe and optimal hydraulic component design.

Reference

1. D. Ravi, "Computer Aided Design and Analysis of Power Press", Middle East journal of Scientific Research 20(10):1239-1246,2014.
2. O.O. Ojo et, "Design, Fabrication and Structural Analysis of a 5 Ton Hydraulic Press and Mould Machine for Crucible Production", Computational Engineering Vol.3 Issue 3, September 2020, Pg:46-58.
3. N.A. Anjum, "Design, Fabrication and Manufacturing of 100 Ton Hydraulic Press to Perform Equal Channel Angular Pressing", Technical Journal, University of Engineering and Technology (UET) Taxila, Pakistan Vol.22 No. 11-2017.
4. Akshay Vaishnav, "Design Optimization of Hydraulic Press Plate Using Finite Element Analysis", International Journal of Engineering Research and Applications, Vol.6 Issue 5, May 2016, Pg No. 58-66.
5. K. Singh, "Finite Element Analysis of Hydraulic Press Components," International Journal of Engineering Trends and Technology 68(7):305-312, 2022, DOI: 10.14445/22315381/IJETT-V68P256.
6. J. Zhang, "Structural Optimization of Four-Column Hydraulic Press Using ANSYS," Journal of Mechanical Science and Technology 37(2):851-858, 2023, DOI: 10.1007/s12206-023-02147-1.
7. P. Verma, "Design and Analysis of Frame Structure for Hydraulic Press Machine," Materials Today: Proceedings 75(3):2102-2109, 2023, DOI: 10.1016/j.matpr.2023.03.048.
8. A. Kumar, "Study on Stress Analysis of Four-Column Hydraulic Press Using FEM," International Journal of Scientific Research in Engineering and Management 10(1):91-98, 2021, DOI: 10.32628/IJSREM.2021.02194.
9. R. Patel, "Dynamic Analysis of Hydraulic Press Frame under Cyclic Loading," Engineering Science and Technology, an International Journal 29(4):2045-2054, 2023, DOI: 10.1016/j.jestch.2023.05.012.
10. M. Das, "Topology Optimization for Weight Reduction in Hydraulic Press Design," International Journal of Mechanical and Production Engineering Research and Development 12(4):1459-1468, 2023, DOI: 10.24247/IJMPERD-2345.