IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

U- Clamp Structural Analysis

Name of 1st Author-Pradyumn Krishnat Jadhav, Name of 2nd Author- Dr.Balu Ashok Phugate

Designation of 1st Author- Student, [2]Designation of 2nd Author- Assistant Professor

Name of organization of 1st Author-Savitribai Phule Pune University, Pune, INDIA

Abstract: U-clamps are widely used mechanical components for supporting and securing pipes, tubes, and cylindrical structures in various engineering applications. Despite their simple geometry, they are subjected to complex loading conditions that may lead to stress concentration, deformation, and possible failure. Therefore, a detailed structural investigation is necessary to ensure safe and efficient performance.

In this study, a U-clamp is analysed using the Finite Element Method to evaluate its structural behaviour under different loading conditions. The analysis includes static structural analysis to determine stress distribution and deformation, along with modal analysis to study the vibration characteristics of the component. Two different cases are considered by varying geometric parameters such as diameter and mass to observe their effect on structural response.

The results indicate that maximum stress occurs at the curved region of the U-clamp due to bending effects, while deformation increases with higher applied loads and lower stiffness. In modal analysis, the natural frequency is found to be dependent on the ratio of stiffness to mass, where an increase in stiffness raises the natural frequency, and an increase in mass reduces it. The mode shapes represent relative vibration patterns rather than actual displacement under loading.

Overall, the study provides a clear understanding of the mechanical behaviour of U-clamps and highlights the importance of proper design selection. The findings can be used to improve structural strength, reduce failure risk, and optimize the design for better performance in real-world applications.

METHODOLOGY

The structural analysis of the U-clamp was carried out using the Finite Element Analysis (FEA) approach. The following systematic procedure was adopted:

◈ Geometry Creation A 3D model of the U-clamp (20 mm and 30 mm diameter) along with the I-beam was created using CAD software (SolidWorks/Creo/ANSYS Design Modeler). Accurate dimensions were used to represent real industrial geometry.

◈Material Properties Material selected: Mild Steel (Fe250) Properties assigned: Young’s Modulus = 210 GPa Poisson’s Ratio = 0.3

◈ Meshing The model was discretized into finite elements. Fine mesh was applied at: U-bend region Contact areas Coarser mesh used in less critical regions to reduce computation time.

◈ Boundary Conditions the I-beam was fixed at its ends to simulate real support conditions. The U-clamp was properly constrained to avoid rigid body motion.

◈ Load Application A vertical downward load of 10,000 N (1 ton) was applied on the U-clamp. Load direction and magnitude were chosen based on practical conditions.

◈ Contact Definition Contact interaction between: U-clamp and beam Appropriate contact type (frictional / bonded) was defined to simulate realistic behavior.

◈ Solution & Analysis The model was solved using a static structural solver. The following results were obtained: Total deformation (displacement) von Mises stress distribution

◈ Result Evaluation Results were analyzed and compared with: Material yield strength Acceptable deformation limits Factor of Safety (FOS) was calculated to check design safety.

◈ Comparison Study Both 20 mm and 30 mm U-clamps were analyzed under the same load. Performance comparison was done based on: Stress, Strain & Displacement

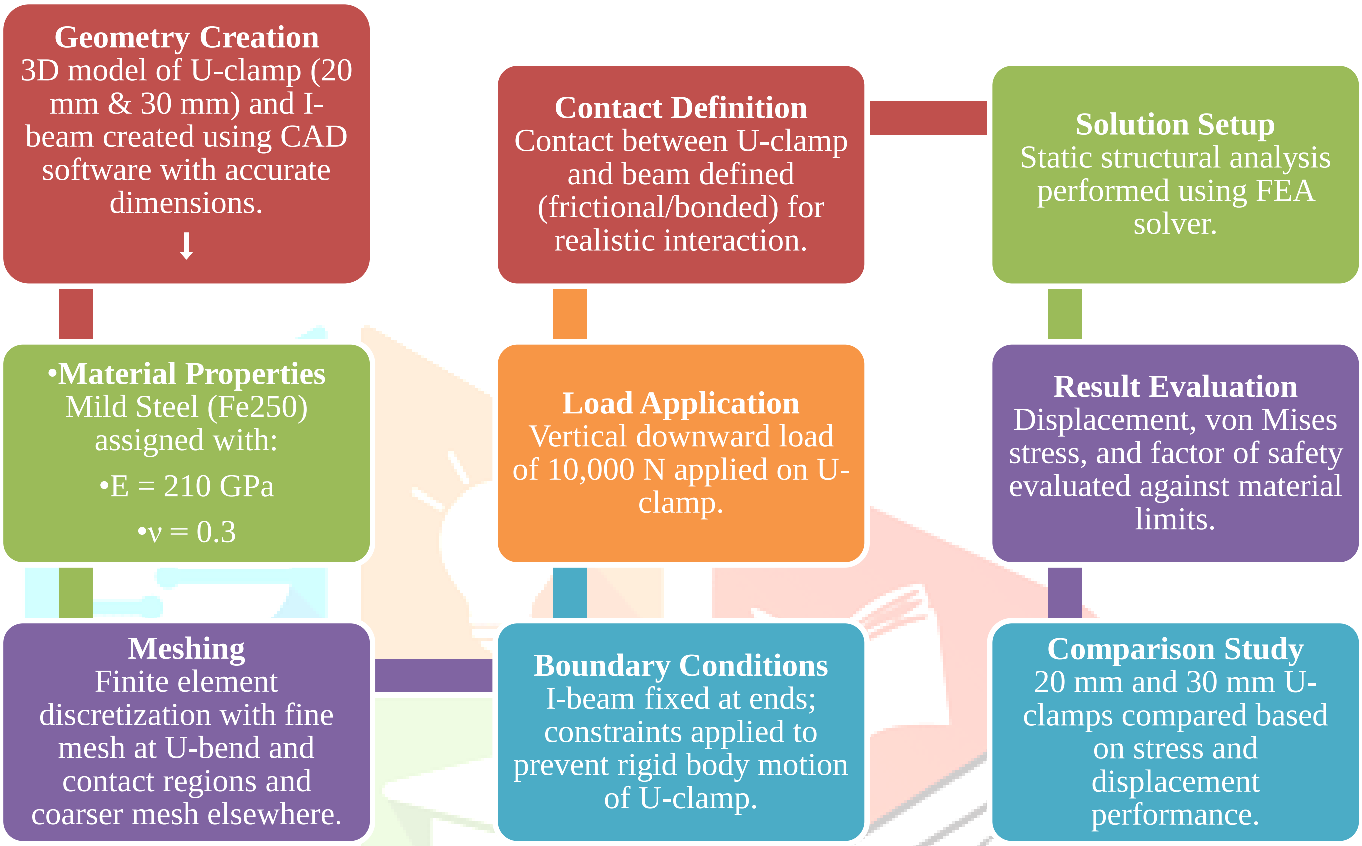


Fig1.Methods

MODELING

5.1.1. Software based FEA
Pre-Processing

Import Step File

Meshing in Hyper mesh

Applied Load and Constrain conditions

In the first preprocess stage, Import the cad data information and confirm the center of the cad document. Take a look at all surfaces checking and intersections check of the U- Clamp. First Start the meshing as per criteria in line with the excellent standards. Discretizing means divide the model into a small quantity of factors the usage of the finite element's method technique. After finishing meshing take a look check mesh at the best of the elements. Then follow the residences and substances. After completed mesh use of loading conditions and boundary condition, test wherein the version is fixe and where the loads is carried out. Building up the version to recognize the specified solution. Following that, simulation software program is used to perform the simulation.

Table 3. Material properties:

Materials	Modulus of Elasticity (N/mm2)	Poisson ratio	Density of Material (Ton/mm3)
1.STEEL	2.1e5	0.3	$7.85e^{-09}$

TABLE NO.3

Post Processing

The post processing is the checking of the results simulations for modal analysis and displacement stress. In post-processing, checking the output result and take the specified step action to improve the consequences. Hyper-view software using for the submit processing results. In HyperView checking Import the simulation output document and checking the end result.

Hyper mesh

Hyper mesh is the mostly pre-processing meshing modeling software used to construct the version. In hyper mesh, first import the CAD part data geometry, then after completing meshing, assigning property and materials of parts, and later exporting the whole element. Within the hyper mesh software, first as require to outline which meshing is use: 1D beam, 2d shell, or 3D hex We employed 1D and 3D Hex mesh as required. Because the machine can handily manage a certain number of factors, it's miles vital to keep in mind of the amount.

Hyper mesh software use for model: • Hex Mesh. • 3D Meshing.

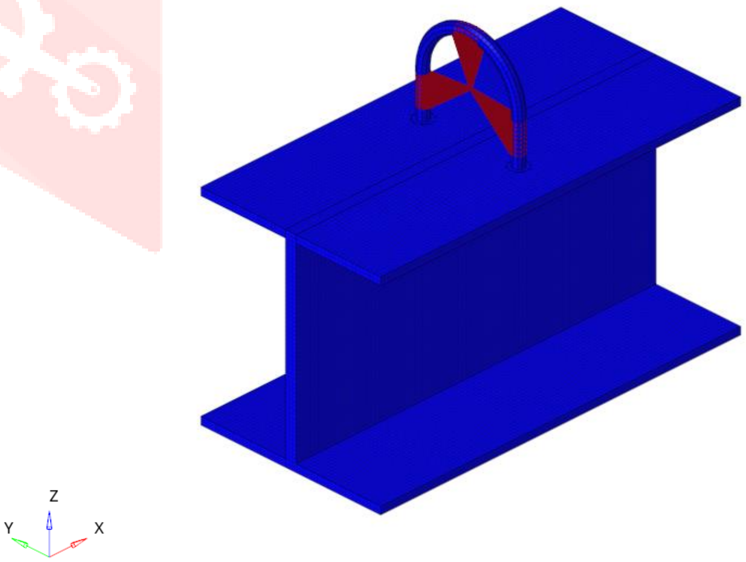


Fig.5 Modeling of U-clamp

Criteria selection of U-Clamp

- Warpages: Warpage means that the node deviates from the plane so that one node goes outside to the remaining three nodes on the outside of the plane. Warpages 25 mm is used.
- Skewness: Skewness measure how much a hexa (or any element) deviates from ideal shape. Skew 60mm used
- Aspects Ratio: This is the ratio of max. length and min. length. The aspect ratio is always below its minimum limit. The aspect ratio is used to be 7 mm.
- Jacobian: The change of the shape of an element to the global coordination system and local coordination system this is a Jacobian. Jacobian is use at 0.3 mm.
- Table 4. Quality index uses for meshing U-Clamp.

Mesh Quality parameter	
Warpage of element in degree	25 mm
Aspect ratio of element	7 mm
Skew in degree of element	60 mm
Jacobian of element	0.3 mm

TABLE NO.4

Element quality is used in U-Clamp

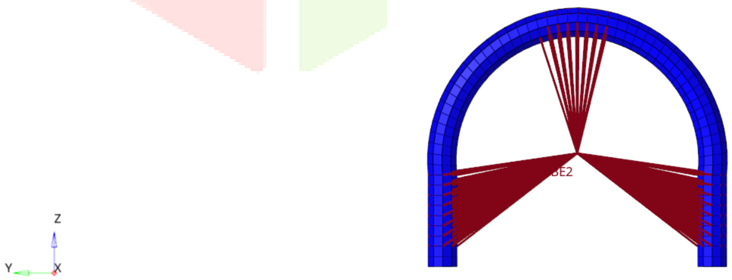


Fig.6 Rigid Element

Represent Rigid body elements (Two-dimensional element):

Importance of rigids element:

a) The Rigid of the 1Dimension element, is using to mention the bolt connect and connect to different components.

b) The Rigid elements have six degrees of freedom, which is can be released as needed.

c) Move into three directions three translations and three rotations.

d) The property of a Rigid element is that there will be no displacement or stress to the region of the element in that region.

e) The Rigid component will be adding stiffness to the main structure, whereas not in RBE2.

f) Regardless of where force or moment is applied, the RBE2 element distributes it equally throughout all connected nodes.

g) The RBE3 element is a constraint equation for distributing forces and moments by distance. Classified of the rigid element:

a) RB2.

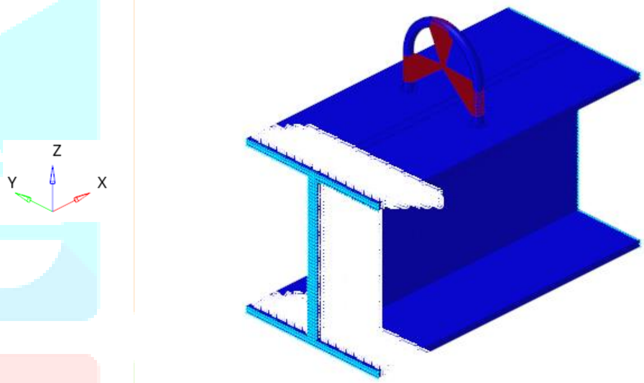
b) RB3.

c) Rigid Bars element.

d) Rigid Beams element.

e) Rigid Rods element.

f) 1D Springs element.



5.1.2. Constraints for U- Clamp

Fig 7: U-Clamp Boundary Condition using SPC

Single Point Constraint (SPC) is a boundary condition used in finite element analysis (FEA) to restrict the movement of a node or a specific degree of freedom (DOF). It is applied when certain parts of a structure are fixed or constrained in real-life conditions. In simple terms, SPC defines where and how the model is held or supported so that the simulation behaves like an actual physical system.

- Translation along X-axis
- Translation along Y-axis
- Translation along Z-axis
- Rotation about X-axis
- Rotation about Y-axis
- Rotation about Z-axis

SPC allows control over these movements by fixing one or more DOFs.

Static analysis

Static analysis is used to evaluate how a structure behaves when subjected to steady (non-changing) loads. In this type of analysis, the applied forces remain constant over time, and the response of the structure is measured in terms of stress, strain, and deformation. For a U-clamp, static analysis helps determine whether the clamp can safely hold a pipe or component without permanent deformation or failure.

Static analysis Equation:

Static structural behavior is governed by the equation:

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

Where:

- [K] = Stiffness matrix
- {u} = Displacement vector
- {F} = Applied force vector

RESULTS & CONCLUSION

6.1. Static analysis U-Clamp

In this work, the structural performance of a U-clamp is examined under two different static loading conditions to understand its displacement behavior and overall mechanical response. Static analysis is carried out to evaluate how the clamp reacts to constant loads without considering time-dependent effects.

Case-1 U Clamp Analysis

For diameter 20 mm how, it behaves and check the stress, strain & displacement with help of Static analysis.

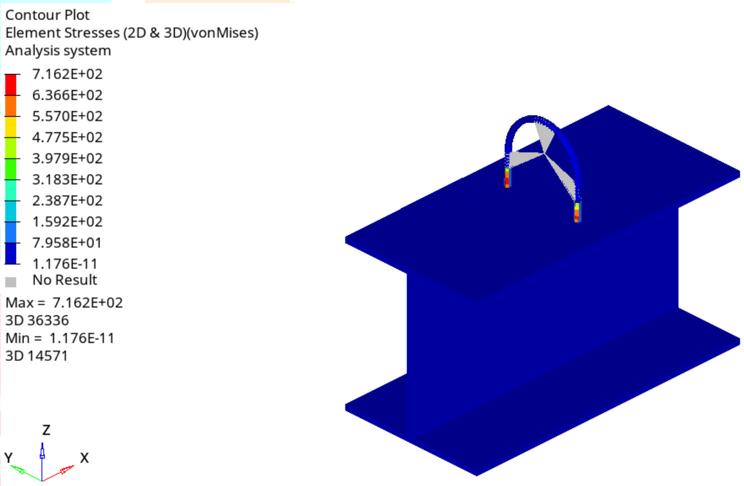
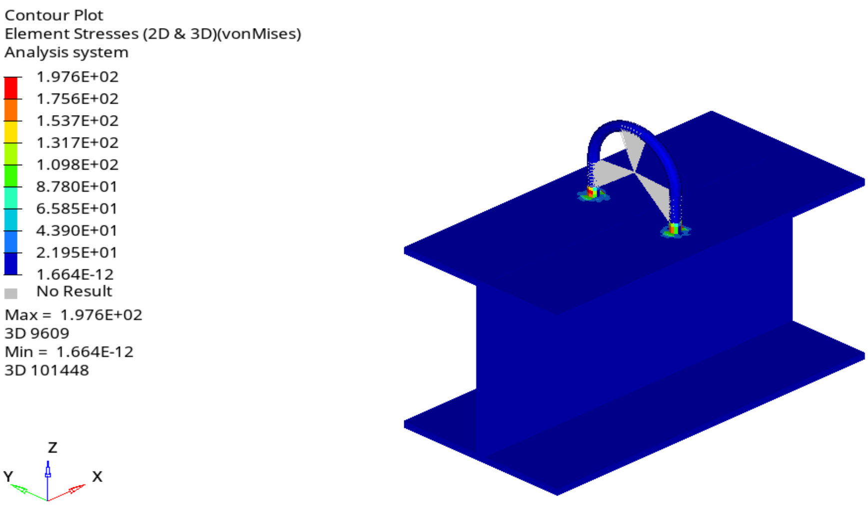


Fig 15: Stresses on U- Clamp

Case-2 U Clamp Analysis

For diameter 30 mm how, it behaves and check the stress, strain & displacement with help of Static analysis.



Sr.no	Load	Behaviour
1	0-2 ton	Fully Safed
2	2-5 ton	Small deformation
3	5-7 ton	Yielding starts
4	6-10 ton	Failure zone
5	>10 ton	Complete fracture

Fig 18: Stress of U- Clamp

6.3 Behavior vs Load

Result charts For (Stress)

Sr.no	Diameter	Stress
1	20 mm	716 MPA
2	30 mm	197.6 MPA

- The structural analysis of the 20 mm diameter U-clamp was performed under an applied load of 10,000 N (≈1 ton) using finite element analysis.
- The maximum von Mises stress obtained is 716 MPa. This value significantly exceeds the yield strength of mild steel (≈250 MPa). Therefore, the U-clamp does not operate within the elastic region under the given loading condition. Instead, the material will undergo plastic deformation, leading to permanent distortion and potential structural failure.
- The maximum von Mises stress obtained is 716 MPa, which is above the yield strength of mild steel (250 MPa). This indicates that the U-clamp operates within the elastic region and does not undergo permanent deformation under the given loading condition.
- These results indicate that the current design is unsafe for the specified load and requires modification, such as increasing the cross-sectional diameter, selecting a higher-strength material, or optimizing the geometry to reduce stress concentration.
- After reinforcement, the 30 mm diameter. The maximum von Mises stress is 197.2 MPa, which is below the yield strength of mild steel (≈250 MPa). This indicates that the U-clamp operates within the elastic region under the applied load and does not undergo permanent deformation. Therefore, the reinforced design is structurally safe for the given loading condition.

7.1. FUTURE SCOPE

U-clamp design can be extended in several important directions to improve its performance and reliability. Dynamic and fatigue analysis should be conducted to evaluate the behavior of the clamp under cyclic loading and vibrations, which are common in real-world applications such as automotive and pipeline systems. This will help in predicting the service life and failure due to fatigue. Material optimization can also be explored by analyzing alternative materials such as alloy steels (EN8, EN19), stainless steel, and composite materials to enhance strength, corrosion resistance, and durability. Furthermore, design optimization can be carried out by modifying geometric parameters such as increasing the diameter, adjusting the bend radius, and improving thread design to achieve an optimal balance between weight and strength.

In addition, nonlinear analysis, including plastic deformation and large deformation finite element analysis, can be performed to more accurately predict the behavior of the clamp near failure conditions. The inclusion of contact interactions and friction effects between the clamp and the beam will further improve the realism and accuracy of the simulation. Experimental validation through physical testing and comparison with finite element results is also essential to ensure the reliability of the analysis. Moreover, future studies can investigate multi-clamp systems to understand load distribution among multiple clamps in practical industrial setups. Finally, thermal and environmental effects such as temperature variations, thermal stresses, and corrosion should be considered to evaluate the performance of the U-clamp under varying operating conditions.