Volume – 9, Issue – 2, June – 2023 https://doi.org/10.46593/ijaera.2023.v09i02.001

Structural Analysis of Squat Rack using Finite Element Approach

G.Sriram[1], T.Rajmohan[2*], A.Tamilarasan[3]

[1,2*,3] Department of Mechanical Engineering, Sri Chandrasekharendra Saraswathi Viswa Mahavidyalaya, Kanchipuram-631 561, Tamilnadu India
E-mail: rajmohanscsvmv@kanchiuniv.ac.in

Abstract: In this work, a Finite Element Analysis (FEA) of Squat Rack is presented for the Mild steel (AISI 1045) and Stainless steel (AISI 304) materials. The frame, top bar and J Hook of squat rack are modelled and analysed in ANSYS environment. At first, the static analysis is carried out for predicting the nodal displacements and stresses. Afterwards, the modal Analysis of the developed FE model is carried out and five mode shapes are obtained at five different natural frequencies. For both the analyses, a beam 188 element is selected. The results of static and modal analysis were plotted for the selected materials. By comparison of all the results, it is inferred that the AISI 304 stainless steel produced minimal displacements and stress values. Finally, it is observed that the performance of FE models and their solutions are accurate and robust.

Keywords: Squat rack; Mild Steel; Stainless steel; FEA; Modal analysis

I. INTRODUCTION

In body building exercises, the rest-pause technique is a resistance workout approach that manipulates the use of repetitions to failure with extremely brief inter-set rest intervals. The square rack is designed by J-hook for bearing the weightlifting device. As a result, the J-hook is an essential bearing element and the most commonly used device. Its performance has a direct impact on the safety of the squat rack operation, and if the hook is broken, it will result in casualties or significant economic losses. The strength of a J-hook is always determined by its load bearing capability. Stress analysis is an engineering field that uses a variety of techniques to determine the stresses and strains in materials and structures exposed to forces [1]. A key trait in the design of structures and artefacts is stress analysis, which is primarily a tool and not a final objective; the end goal is to build structures and artefacts that can handle a specific load while utilising the least amount of material [2-5]. The finite element methods have been utilised in recent years to calculate stress distribution.

The finite element analysis is a method of numerical analysis, generally known as FEA. Many engineering disciplines employ FEA to solve problems, including machine design, acoustics, electromagnetic, soil mechanics, fluid dynamics, and many others. As FEA is a design tool, it must be used alongside the design process. The design process should be maintained, or better still, driven. Iterations of the analysis must be completed quickly, and the results are utilized to inform design decisions that they must be dependable even with limited input. Today, most of the research efforts are being spent on FEA analysis. In this work, the static and model analysis was performed using FEA analysis. In a static analysis, the amount of applied force and the amount of displacement are proportional to each other. Stresses under load stay in the linear elastic range of the material employed. A modal analysis of the system necessitates the computation of a few lower-order frequencies. A high mode has minimal influence on the structure's dynamic properties and the low order natural frequency is likely to induce system resonance.

 ©2023, IJA-ERA - All Rights Reserved

II. STATIC ANALYSIS OF J-HOOK

A. *Finite Element Modelling*

In this approach, the entire solution domain is partitioned into discrete finite segments. The behaviour of each constituent is described by differential governing equations. All of these little parts are put together, and the prerequisites for continuity and equilibrium between adjacent elements are met. A unique solution to the overall system of linear algebraic equations can be found if the boundary conditions of the actual problem are satisfied. For analyzing the J-hook, a geometric model is needed. The accuracy of the geometric model would affect the precision of the finite element model directly and hence, the geometric model must be optimised in order to reflect the real situation. The geometric model has to be sufficiently simplified to ease modelling and to be easily examined based on the correctness of the geometric model. Then, the suitable element is preferred for performing the effective analysis. Here, Beam 188 is selected as the element. The element is a linear, quadratic, or cubic two-node beam element in 3-D. BEAM188 has six or seven degrees of freedom at each node. These include translations in the x, y, and z directions and rotations about the x, y, and z directions. A seventh degree of freedom (warping magnitude) is optional. This element is well-suited for linear, large rotation, and/or large strain nonlinear applications. The typical structure of the element is shown in Figure 1.

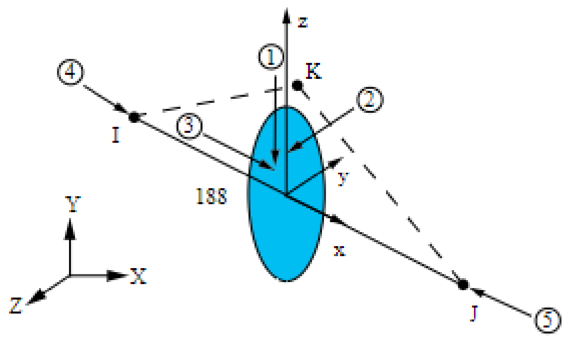


Figure 1. Beam 188 element

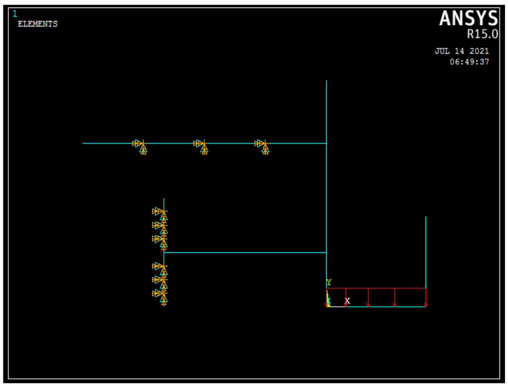
In this work, two different materials such as Mild steel (AISI 1045) and Stainless steel (AISI 304) were considered. The mechanical properties of two selected materials are given in Table 1.

Table 1. Mechanical Properties of Selected Materials

Description	Mild steel (AISI 1045)	Stainless steel (AISI 304)
Material Density	7870 kg/m^3	7900 kg/m^3
Elastic Modulus	200Gpa	193 GPa
Poisson's Ratio	0.3	0.29

Among the materials, AISI 1045 steel is characterized by good weldability, machinability, high strength and impact properties in both the normalized or hot-rolled condition. As SS 304 can withstand extremely corrosive environments, it can be shaped, machined and welded easily and it is perfect while manipulating the material. It is popular amongst a range of applications such as automotive parts, fitness and medical equipment etc. These benefits include an attractive look, ease of cleaning, a reasonably high strength to weight ratio, and widespread availability in a variety of forms. After defining the material properties, the mesh module was selected, in order to ensure the quality of elements.

 ©2023, IJA-ERA - All Rights Reserved



(a) Fabricated component (b) FEA Model

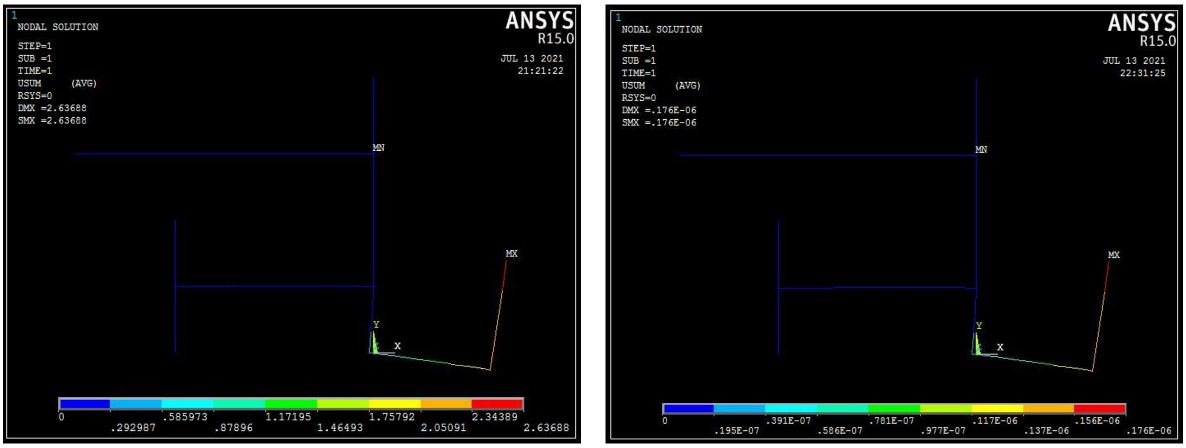
Figure 2. Fabricated component and finite element model of the J hook.

It was necessary to apply the constraints to the finite element model in order for the finite element model to be computed; otherwise, the model could not be constructed. Meanwhile, it's important to remember that constraints must be precise and represent the real circumstances if they are to yield accurate outcomes. Figure 2 depicts the fabricated component and finite element model of the J hook. According to the geometry size and various material parameters, the ANSYS finite element model is set up as clearly shown in Figure 2. In the work condition, the top roller and side wall are completely constrained with six degrees of freedom.ame. Then, the static load (150Kg) is uniformly applied in the support frame of J-hook. After applying constraints and loads, static strength analysis could be performed.

B. Results of Static Strength Analysis

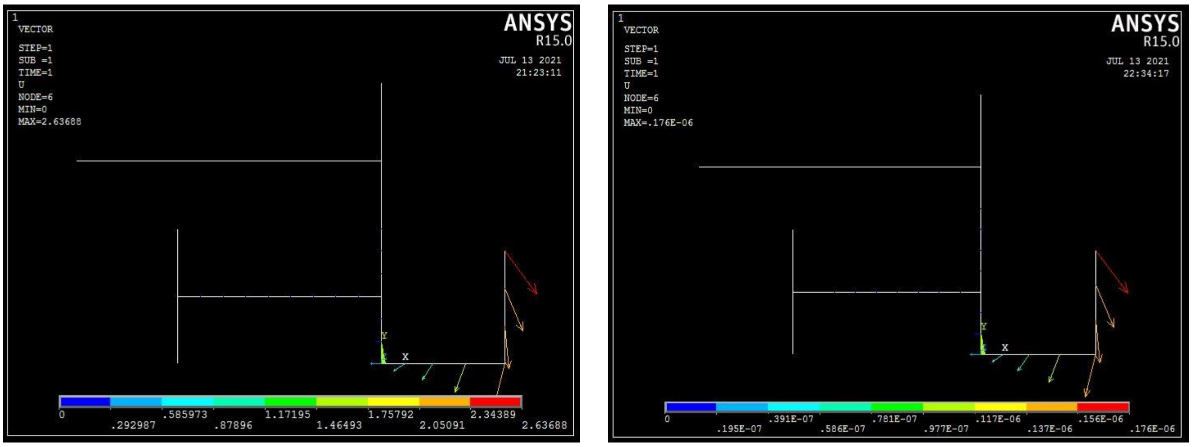
The results of nodal displacement, axial direct stress and bending stress are diametrically illustrated in Figures 3-6 for both materials. From the displacement contour plots (Figures 3(a)-(b)), the maximum displacement occurred for mild steel 2.63688mm and for stainless steel is obtained only 0.176E-6mm. Moreover, it is observed that maximum deformation at free side of J hook which evidently seen in these Figures. Also, the results of Y-displacement vector plot indicate the direction of deflection of the J hook (Figures 4(a)-(b)). The stress analysis helps identify where the Hook structures meet the expectations of desired stress existences in the component. Through carrying out these studies, it helps to create an integrity structure that can withstand the forces exerted on it. In order to analyze the stress pattern for the applied loading condition axial direct stress is simulated and shown in Figures 5(a)-(b). By comparing these Figures, stainless steel-based J hook ultimately produced maximum stress value 0.305405 N/mm^2 reached. However, the stress values for the mild 4.58108N/mm^2. The bending stress contour on hook is investigated and has been shown in Figure Figures 6(a)-(b). Realistically, the modeled J hook designed by stainless steel which produced 9.55402 N/mm^2.

 ©2023, IJA-ERA - All Rights Reserved



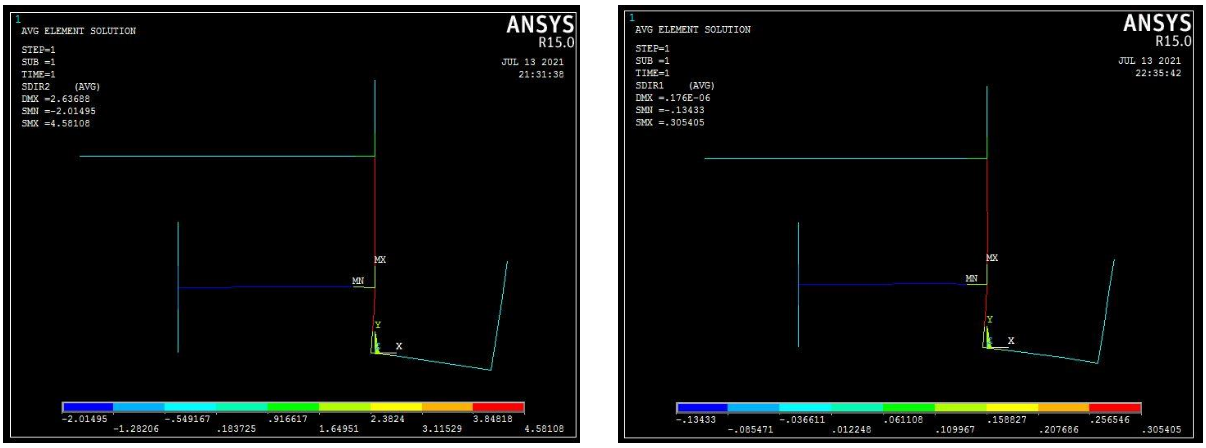
(a) Mild steel (b) Stainless steel

Figure 3. The Displacement contour of the Hook



(a) Mild steel (b) Stainless steel

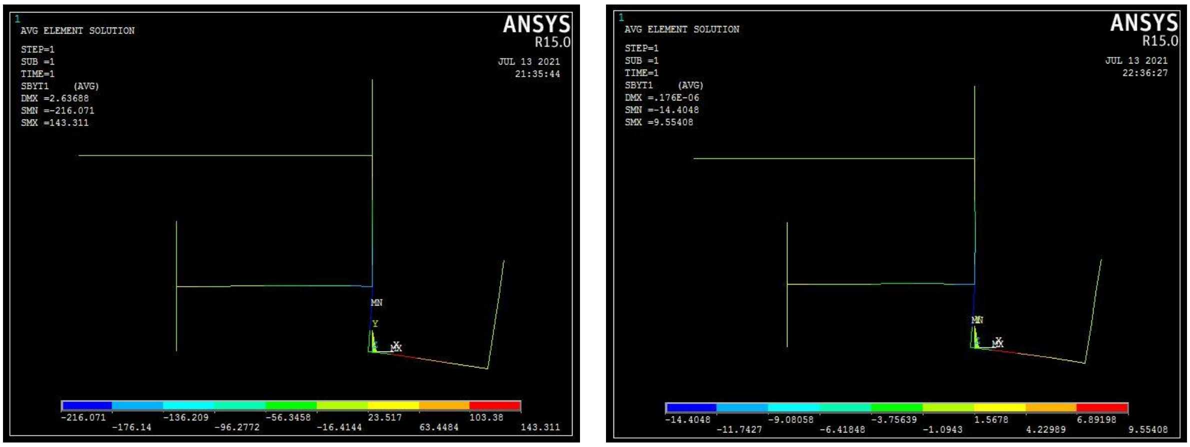
Figure 4. The results of Y-Displacement vector plot



(a) Mild steel (b) Stainless steel

Figure 5. The results of Axial direct stress

 ©2023, IJA-ERA - All Rights Reserved



(a) Mild steel (b) Stainless steel

Figure 6. The results of Bending stress

III. MODAL ANALYSIS OF J-HOOK

A. Finite Element Modelling

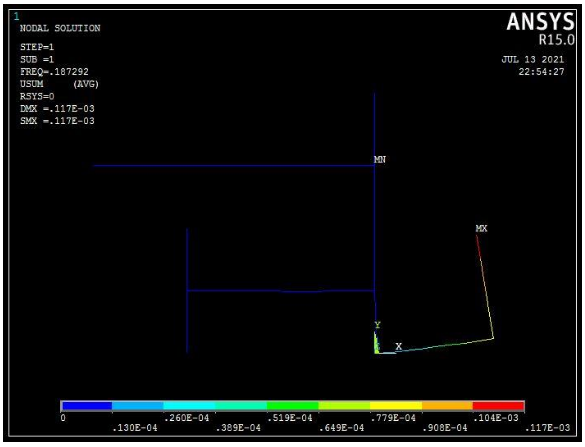
The study also extended to understand the vibrational characteristics for the presented FE model. Since, the modal analysis enables the determination of a mechanical structure's or component's vibration characteristics (natural frequencies and mode shapes) by visualising the movement of the structure are various components under dynamic loading circumstances. The natural frequencies and mode shapes of a structure are critical factors to consider when designing it under dynamic loading situations. The modal analysis consists of same three steps, pre-processing, processing and post processing. In pre-processing, the same finite element model is utilized and the same boundary conditions and loads are also considered. In the next stage of processing, the appropriate modal analysis settings such as mode extraction method (Block Lanczos), number of modes(five) were selected and assigned in the model. Then, the modal analysis is carried out within the ANSYS APDL environment, according to the specifications.

B. Results of Modal Analysis

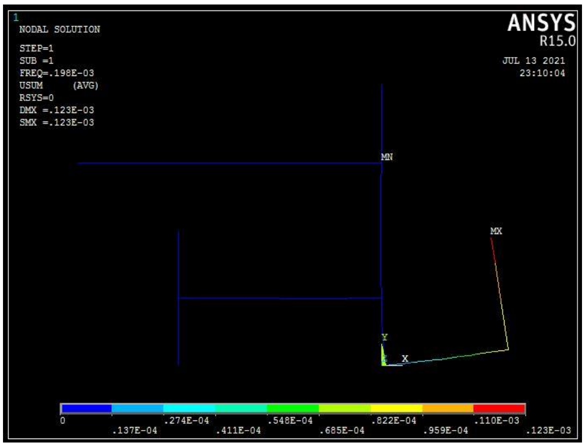
The fundamental mode shape of the hook structure is completely analyzed. There are five important mode shapes are found and Tabulated in Table 2 for the two different materials. Accordingly, Figures 7-11 show the dynamic behaviour of the first, second, third, fourth and fifth mode shape which obtained from the ANSYS APDL. All the modes are often defines the highest loads in a hook or how that hook structure will interact with the rest of the system around it when vibrating the whole system. From the obtained numerical natural frequencies (Table 2) and Figures 6-11, it is inferred that stainless steel based J hook is ultimately produced less nodal displacement along the Y-direction.

Table 2 Natural frequencies for different mode shape

Frequency(Hz)	Mild Steel	Stainless Steel
f_1	0.18792	0.198E-3
f_2	0.283646	0.299E-3
f_3	0.487092	0.514E-3
f_4	1.19593	0.001262
f_5	0.487092	0.514E-3

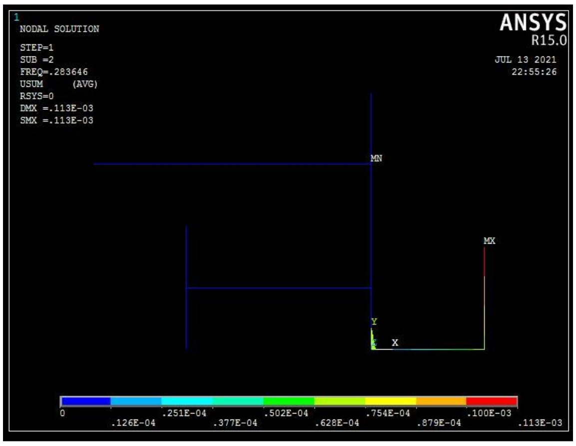


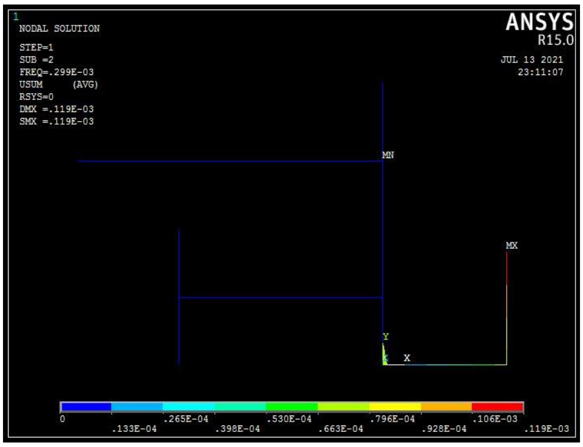
(a) Mild steel



(b) Stainless steel

Figure 7. The results of First set of mode shape

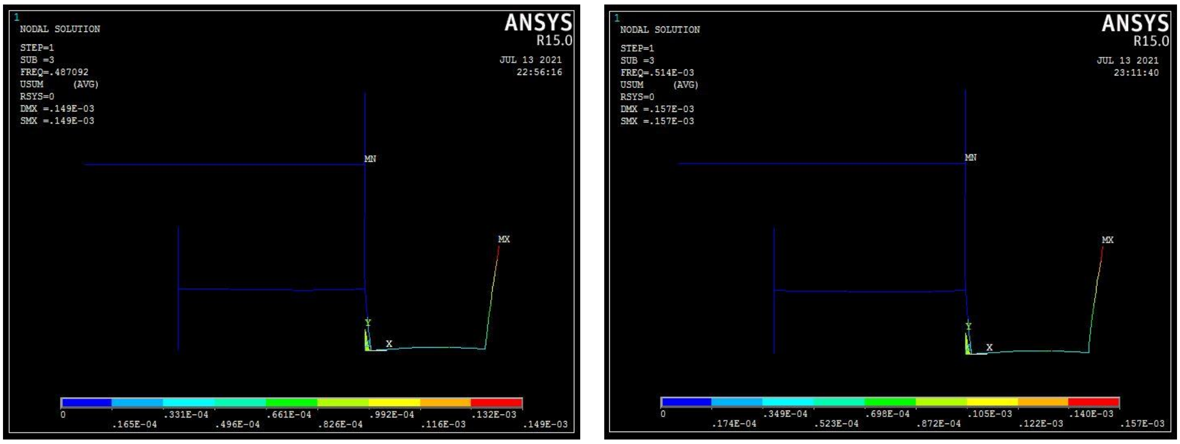




(a) Mild steel (b) Stainless steel

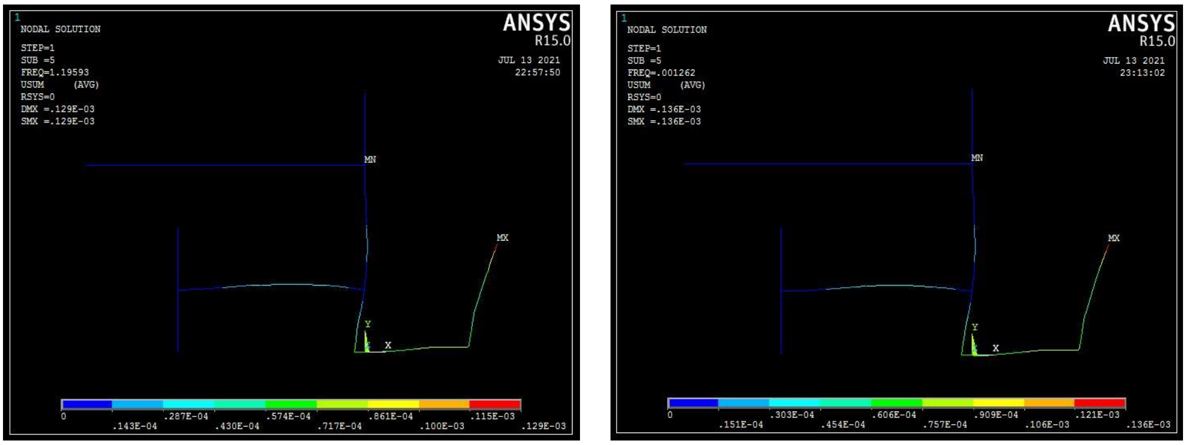
Figure 8. The results of Second set of mode shape

 ©2023, IJA-ERA - All Rights Reserved



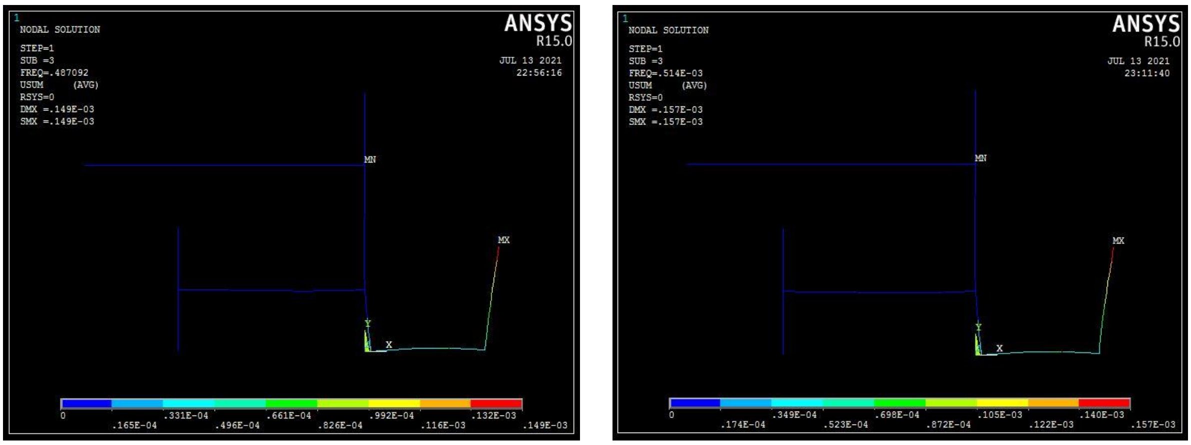
(a) Mild steel (b) Stainless steel

Figure 9. The results of Third set of mode shape



(a) Mild steel (b) Stainless steel

Figure 10. The results of Fourth set of mode shape



(a) Mild steel (b) Stainless steel

Figure 11. The results of Fifth set of mode shape

 ©2023, IJA-ERA - All Rights Reserved

IV. CONCLUSION

In this study, the squat rack J Hook is modeled and analyzed in ANSYS APDL environment. In static analysis of the hook was performed and the axial and bending stresses were calculated. Then, through the Modal analysis, the five mode shapes were successfully obtained for both materials. Based on the obtained FEA results, the overall design of the Squat Rack J hook component is reasonable. The structure is in a balanced state, the working condition was safe and reliable, the stress variation is even, and the stress distribution is also regular. Therefore, the designed Squat Rack to satisfy both the ultimate and the serviceability limit.

ACKNOWLEDGMENT

Authors are very grateful to the officials of Sri Chandrasekharendra Saraswathi Viswa Mahavidyalaya, Kanchipuram, Tamilnadu, India, for providing financial grant to this research work.

Conflict of interest: The author declares that he has no conflict of interest.

Ethical statement: The author declares that he has followed ethical responsibilities.

REFERENCES

[1] Guojian H, Jin L, Min C, Feng, PQ (2015) The Finite Element Analysis of Hook Based on Workbench. Proceedings of the 2015 4th International Conference on Sustainable Energy and Environmental engineering 864-868.
[2] Maneengam A, Saisirirat P, Suwankan, P (2017) Hook Design Loading by The Optimization Method With Weighted Factors Rating Method. Energy Procedia 138: 337–342.
[3] Zhenyu H, Richard, LJY (2016) Numerical studies of steel-concrete-steel sandwich walls with J-hook connectors subjected to axial loads. Steel and composite Structures 21(3):461-477.
[4] Vinodh S, Ravikumar R (2012) Application of probabilistic finite element analysis for crane hook design. Journal of Engineering, Design and Technology, 10(2): 255 - 275.
[5] Girisha MMR, Deshpande M, Babu S and Shivananda DC (2020) Modelling and simulation of below-the-hook lifting device balanced C-hook for load to investigate the static and model analysis for various grades of steels by numerical method. Journal of Modelling and Simulation of Materials 3(1):61-69.

 ©2023, IJA-ERA - All Rights Reserved