# Finite Element Analysis and Optimization of Swing Jaw Plate of A Jaw Crusher

**R Santhi[1], K Venkatesh[2]**
[1, 2] Dept of Mechanical Engineering
[1, 2] AVN Institute of Engineering & Technology, Hyderabad, India

***Abstract-*** *It is required to reduce the large solid particles into smaller particles. For this reason jaw crushers are used. All mechanical and metallurgical applications require the usage of crushers. Different types of jaw crushers are available depending on the capacity ranging from 0.2 ton/hr to 50 ton/hr. They are differentiated based on product size and mechanism used. Jaw crushers are mainly of 3 types. Cone crusher, Jaw crusher and Impact crusher. The crushing process used in the crushers is either pressure based or impact based and sometimes the combination of both.*

*The present project is aimed at design and analysis of swing jaw plate of a jaw crusher. During the part of project a static analysis of swing jaw plate was carried out using finite element analysis package. The 3 dimensional model of the swing jaw plate will be designed using NX-CAD. Then the 3-D model will be imported into ANSYS using the parasolid format. The analysis shall be performed in static condition.*

***Keywords***- Jaw Crushers, NX-CAD, ANSYS.

## I. INTRODUCTION

A jaw crusher is a machine component used in land mining equipment which is designed in a way to reduce the size large solid particles of raw material into smaller particles. These machines make themselves as a leader which is used by mechanical, metallurgical and allied industries. According to the different applications in industry it is available in different sizes and capacities from 0.2 ton/hr to 50 ton/hr and also considering different parameters like product size and mechanism used. According to present scenario mechanism these are available in three model namely cone crusher, Jaw crusher and impact crusher.

The main aim of the crushers is to crush the huge stone particles and reduce their size to smaller one. Milder metals, similar to placer stores of tin, gold, mineral sands and so on don't require such treatment. Huge scale smashing operations are by and large performed by mechanically worked gear like jaw crushers, gyratory crusher and move crushers. For substantial rock pieces that are too enormous for getting containers of mechanically determined crushers, percussion shake breakers or comparable apparatuses are utilized to separate them to estimate. The instrument of pulverizing is either by applying sways power, weight or a mix of both. The jaw crusher is fundamentally a pressure crusher while the others work principally by the use of impact.

Jaw crushers are the primary crushers in a mine or ore processing plant. The size of a jaw crusher is designated by the rectangular or square opening at the top of the jaws (feed opening). For example, a 24 x 36 jaw crusher has an opening of 24" by 36", a 56 x 56 jaw crusher has an opening of 56" square. Essential jaw crushers are regularly of the square opening outline, and optional jaw crushers are of the rectangular opening plan. Nonetheless, there are numerous special cases to this general run the show.

## II. METHODOLOGY

` The present project is aimed at design and analysis of swing jaw plate of a jaw crusher. During the part of project a static analysis of swing jaw plate was carried out using finite element analysis package. The 3 dimensional model of the swing jaw plate will be designed using NX-CAD. Then the 3-D model will be imported into ANSYS using the parasolid format. The analysis will be performed in static condition. Modal analysis is also carried out on the jaw crusher. From the analysis results deflections, VonMises stress, mode shapes and frequencies are documented by using FEA software. Finally design optimization of the swing jaw plate shall be done to increase the factor of safety of the jaw crusher. NX-CAD software will be used for 3D modeling of the jaw crusher and ANSYS software will be used to do the finite element analysis of the jaw crusher.

## III. 3D MODELING OF ROTOR SHAFT

The 3D model of the Jaw plate with stiffener is created using UNIGRAPHICS NX software from the 2d drawings. UNIGRAPHICS NX is the world's leading 3D product development solution. This software enables designers and engineers to bring better products to the market faster. It takes care of the entire product definition to serviceability. NX delivers measurable value to manufacturing companies of all sizes and in all industries.

Figure.1.Shows the Isometric view of Jaw plate with stiffener.

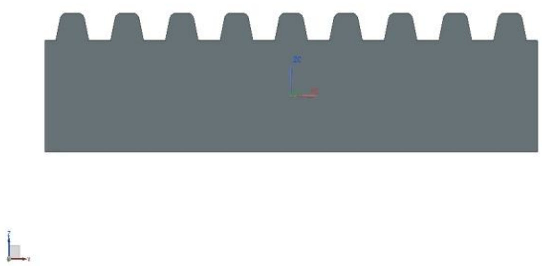

Figure.2.Front view of Jaw plate with stiffener .

## IV. STRUCTURAL ANALYSIS OF SWINGING JAW PLATE

The best CAE packages which are efficiently used for mechanical engineering analysis are Finite Element Modeling (FEM) and Finite Element Analysis (FEA). This software's are used as a most popular numerical technique for solving engineering problems. This method is generally best enough to handle any shape of geometry (problem domain) that are resulted complex structures., material properties, boundary conditions and any loading conditions. The generality of the FEM fits the analysis prerequisites of the present complex engineering systems and designs where shut form solutions are administering equilibrium equations are not available. Moreover it is an proficient design tool by which designers can perform parametric design examining different cases (different shapes, material loads etc.) analyzing them and choosing the optimum design.

## V. FINITE ELEMENT METHOD

The FEM is numerical analysis technique for obtaining approximate solutions to wide variety of engineering problems. The method originated in the aerospace industry as a tool to study stresses in complicated airframe structures. It grew out of what was called the matrix analysis method used in aircraft design. The method has gained popularity among both researchers and practitioners and after so many developments codes are developed for wide variety of problems.

## VI. METHODS OF PERFORMING STRUCTURAL ANALYSIS

To perform an accurate analysis a structural engineer must determine such information as structural loads, geometry, support conditions, and materials properties. The results of such an analysis typically include support reactions, stresses and displacements. This information is then compared to criteria that indicate the conditions of failure. Advanced structural analysis may examine dynamic response, stability and non-linear behavior. There are three approaches to the analysis: the mechanics of materials approach (also known as strength of materials), the elasticity theory approach (which is actually a special case of the more general field of continuum mechanics), and the finite element approach. The first two make use of analytical formulations which apply mostly to simple linear elastic models, lead to closed-form solutions, and can often be solved by hand. The finite element approach is actually a numerical method for solving differential equations generated by theories of mechanics such as elasticity theory and strength of materials. However, the finite-element method depends heavily on the processing power of computers and is more applicable to structures of arbitrary size and complexity.

### 1. Finite element method APPRAOCH

Finite element method models a structure as a gathering of components or segments with different types of association between them. In this manner, a constant framework, for example, a plate or shell is demonstrated as a discrete framework with a limited number of components interconnected at limited number of hubs. The conduct of individual components is portrayed by the component's solidness or adaptability connection, which inside and out prompts the framework's firmness or adaptability connection. To set up the component's solidness or adaptability connection, we can utilize the mechanics of materials approach for basic one-dimensional bar components, and the flexibility approach for more mind boggling two-and three-dimensional components. The diagnostic and computational improvement is best affected all through by methods for lattice polynomial math.

### 2. STRUCTURAL AnalStysis

Structural analysis contains the arrangement of physical laws and arithmetic required to ponder and predicts the conduct of structures. The subjects of basic investigation are designing ancient rarities whose trustworthiness is judged to a great extent in view of their capacity to withstand loads; they regularly incorporate structures, extensions, flying machine, and ships. Basic investigation fuses the fields of mechanics and progression and also the numerous disappointment hypotheses. From a hypothetical point of view the essential objective of auxiliary examination is the calculation of disfigurements, interior powers, and stresses.

## 3. STATIC ANALYSIS OF SWINGING JAW PLATE WITH STIFFENER:

A static analysis can however include steady inertia loads and time varying loads that can be approximated as static equivalent loads. The 3d model of the swinging jaw plate with stiffener is created in NX-CAD and converted into parasolid. The parasolid file is imported into ANSYS and finite element analysis is carried out using ANSYS software

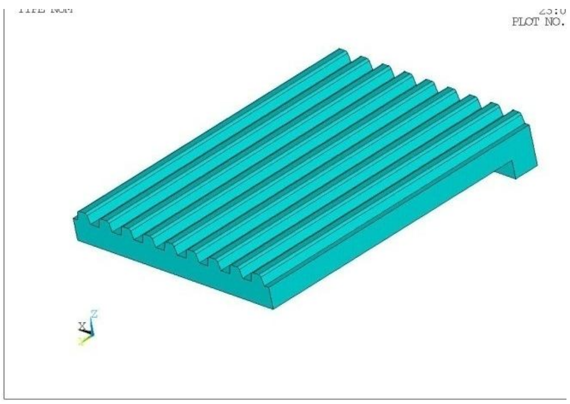


Figure.3.shows the geometric model of the Swinging jaw plate with stiffener.

## 4. MATERIAL PROPERTIES

**Material used for Swinging jaw plate with stiffener is steel:**
Young's Modulus: 200GPa
Poisson's Ratio: 0.3
Density: 7850 Kg/m3
Yield strength: 240MPa

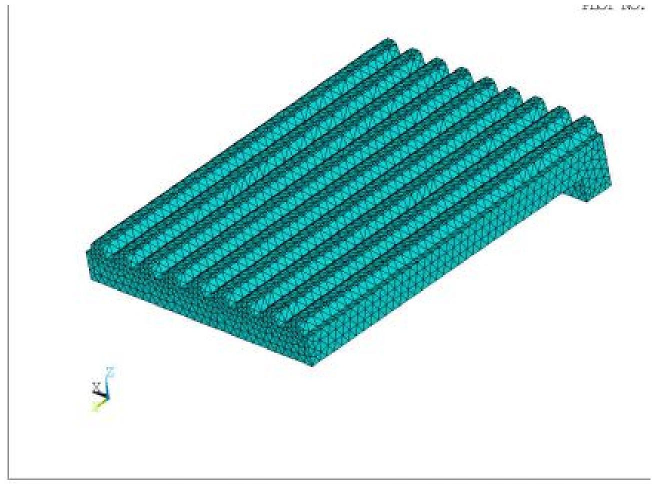

Figure.4.shows the meshed model of Swinging jaw plate with stiffener.

## 5. OUNDARY CONDITIONS:

The bottom of stiffener of jaw plate is constrained in all Dof.
A force of 10.875KN is applied on top surface of the jaw plate.

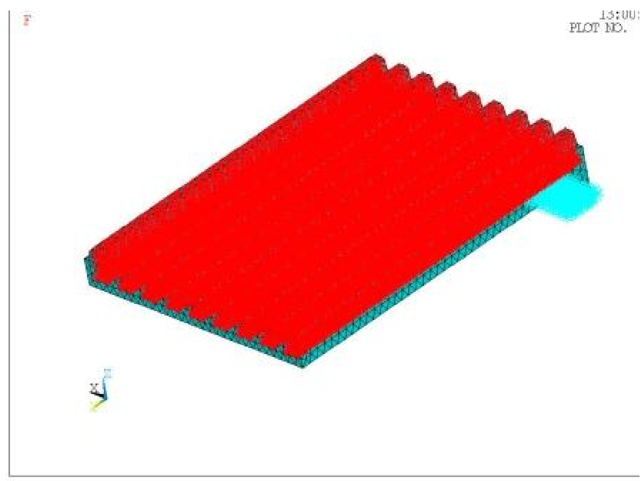


Figure.5.shows the applied boundary and loading conditions of Swinging jaw plate with stiffener.

## 6. RESULTS

The Maximum Displacement observed 0.0006mm on Swinging jaw plate with stiffener in X-direction.

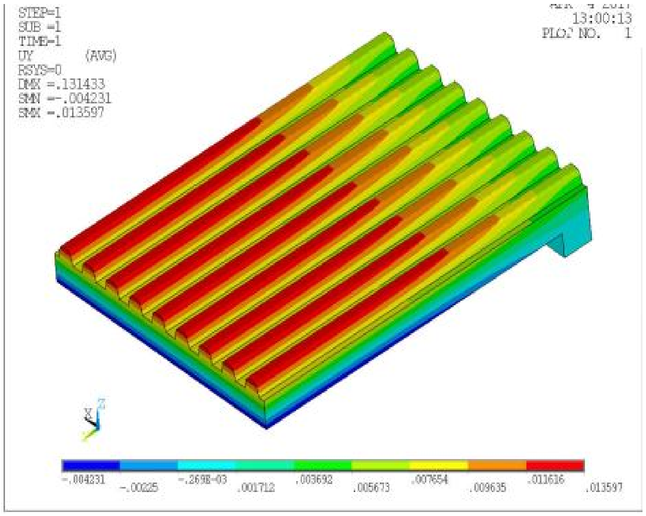


Figure.6.shows the Maximum Displacement.

From the above analysis:
The Max Deflection and the Max Avg. Von Mises Stress observed on the Swinging jaw plate with stiffener is 0.0004mm and 1.973MPa with respectively. And the Yield strength of the material steel is 240MPa.
Hence according to the Maximum Yield Stress Theory, the Von Mises stress is less than the yield strength of the material. The design of Swinging jaw plate with stiffener is safe for the above operating loads.

**7. MODAL ANALYSIS**

Modal analysis is used to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic response analysis, or a seismic analysis.

Natural frequency is the frequency at which a system naturally vibrates once it has been set into motion. In other words, natural frequency is the number of times a system will oscillate (move back and forth) between its original position and its displaced position.

**8. BOUNDARY CONDITIONS**

The bottom of stiffener of jaw plate is fixed in all Dof

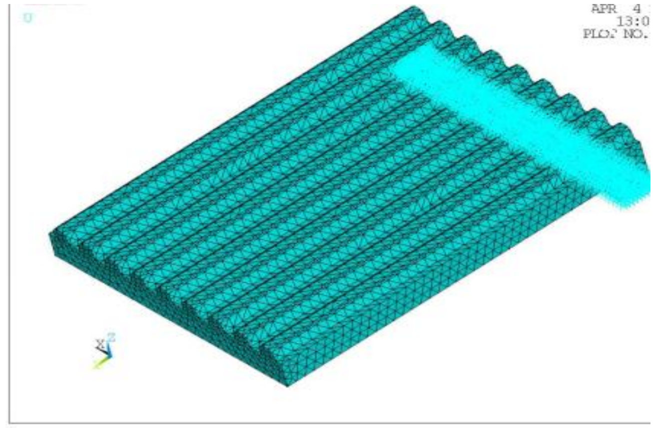

Figure.7.shows Applied Boundary conditions on Swinging jaw plate with stiffener.

The mode shapes for above frequencies are plotted below.

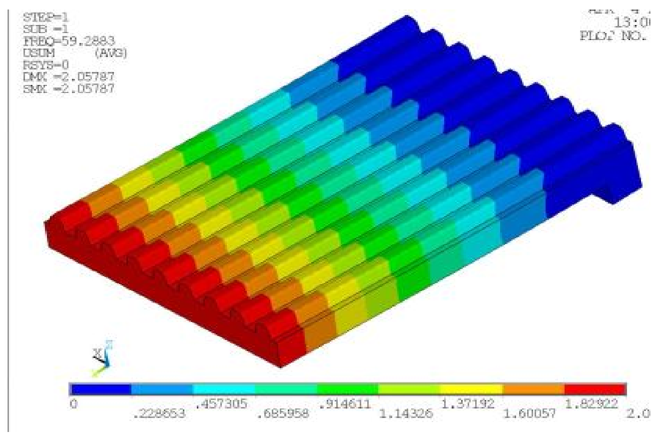


Figure.8. Shows Mode shape 1@59.2Hz for Swinging jaw plate with stiffener.

The total weight of the Swinging jaw plate with stiffener is 0.96Tone.
It is observed that the maximum mass participation of 0.54Tone in X-dir (i.e. 56% of its total weight) for the frequency of 253.7Hz.
It is observed that the maximum mass participation of 0.072Tone in Y-dir (i.e. 7.5% of its total weight) for the frequency of 368.5Hz.
It is observed that the maximum mass participation of 0.574Tone in Z-dir (i.e. 59.3% of its total weight) for the frequency of 59.2Hz.

**9. STATIC ANALYSIS OF SWINGING JAW PLATE WITH TWO STIFFENERS**

A static analysis can however include steady inertia loads and time varying loads that can be approximated as static equivalent loads. The 3d model of the swinging jaw plate with two stiffeners is created in NX-CAD and converted into parasolid. The parasolid file is imported into ANSYS and finite element analysis is carried out using ANSYS software.

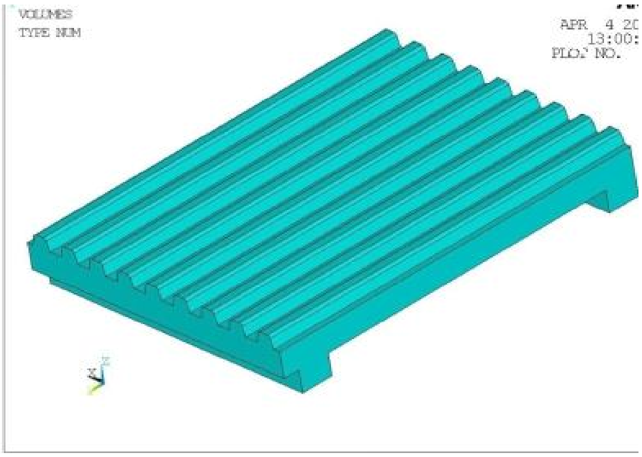


Figure.9.shows the geometric model of the Swinging jaw plate with two stiffeners.

**10. MATERIAL PROPERTIES**

Material used for Swinging jaw plate with two stiffeners is steel:

Young's Modulus: 200GPa
Poisson's Ratio: 0.3
Density: 7850 Kg/m3
Yield strength: 240MPa

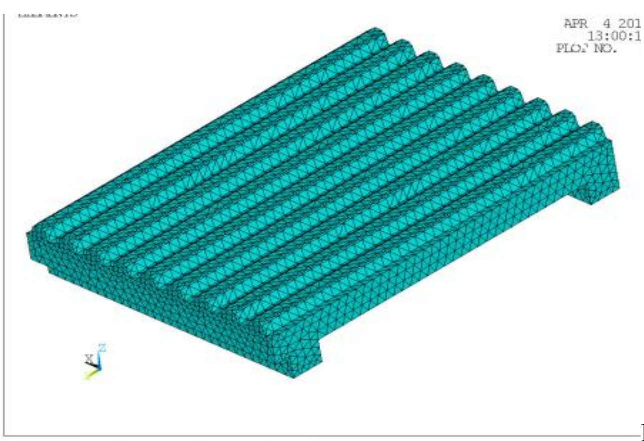


Figure.10.shows the meshed model of Swinging jaw plate with two stiffeners.

**11. BOUNDARY CONDITIONS**

The bottom of first stiffener of jaw plate is constrained in all Dof.

A force of 10.875KN is applied on top surface of the jaw plate.

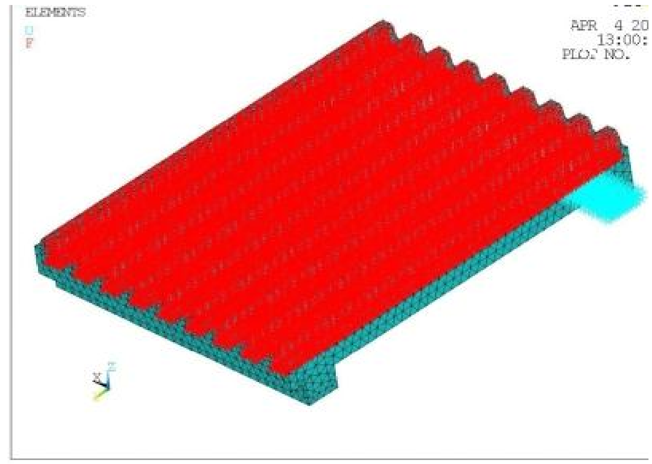


Figure.11.shows the applied boundary and loading conditions of Swinging jaw plate with two stiffeners.

**12. RESULTS**

The Maximum Deformation observed 0.0005mm on Swinging jaw plate with two stiffeners in X-direction.

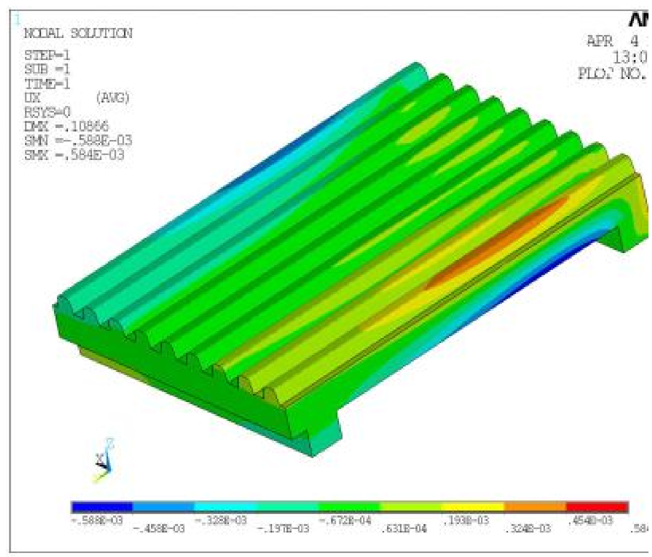


Figure.12.shows the deformation of Swinging jaw plate with two stiffeners in X-direction.

From the above analysis:

The Maximum Deflection and Von Mises Stress observed on the Swinging jaw plate with two stiffeners is 0.10mm and 7.77MPa with respectively. And the Yield strength of the material steel is 240MPa.

Hence according to the Maximum Yield Stress Theory, the Von Mises stress is less than the yield strength of the material. The design of Swinging jaw plate with two stiffeners is safe for the above operating loads.

## VII. RESULTS

**Static analysis**:

From the results, The Max Deflection and the Max Avg. Von Misses Stress observed on the Swinging jaw plate with stiffener is 0.0004mm and 1.973MPa with respectively. And the Yield strength of the material steel is 240Mpa and the Von Misses stress is less than the yield strength of the material.

## VIII. CONCLUSION

Swinging jaw plate was modeled in NX-CAD software and jaw plate was analyzed for structural analysis in ANSYS software. At first, jaw plate with one stiffener was considered. Structural analysis was performed on jaw plate and results of jaw plate with one stiffener were within design limits of material used for static analysis, but in case of harmonic analysis, the results were not in limits of the material. So, jaw plate with two stiffeners was modelled to obtain the results of harmonic analysis within limits of material used. But the results of jaw plate with two stiffeners were not within limits for harmonic analysis. So, jaw plate with three stiffeners was modelled and analysed for structural analysis. The results of both static and harmonic analysis were within the limits of material used (i.e. steel). Hence, model of jaw plate with three stiffeners was better compare to jaw plate with one stiffener and two stiffeners.

## REFERENCES

[1] Shyam Sundar.V "OPTIMUM DESIGN AND ANALYSIS OF SINGLE TOGGLE JAW CRUSHER". IJARSE, Vol. No.3, Issue No.12, December 2014.
[2] Bharule Ajay Suresh “Computer Aided Design and Analysis of Swing Jaw Plate of Jaw Crusher”.
[3] Ashish Kumar Shrivastava and Avadesh K. Sharma “Dynamic Analysis of Double Toggle Jaw Crusher Using Pro-Mechanica" ISSN: 2248-9622 www.ijera.com Vol. 2, Issue 3, May-Jun 2012, pp.1132-1135.
[4] Ashish Kumar Shrivastava and Avadesh K. Sharma “A REVIEW ON STUDY OF JAW CRUSHER". ISSN: 2249-6645,Vol.2, Issue.3, May-June 2012 pp-885-888