International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 12, December-2016

Modal Analysis of Gait Frequency Structure on ANSYS 15.0 based on FEA

Ruchita S Gawande[1],
[1]Student, Mechanical Dept,
RSCOE, Pune,
Maharashtra, India

S N Khan[2]
[2]Professor, Mechanical Dept,
RSCOE, Pune,
Maharashtra, India

***Abstract*— A gait frequency structure is one which is excited by the human gait frequency. This structure is used for emergency rescue training. The runner runs over the platform of the structure inducing an excitation frequency. In this paper we are carrying out modal analysis of a gait frequency structure. Resonance being a critical phenomenon modal analysis is important. The given structure is required to respond to human gait frequency which can vary between the range of 6-15 Hz. At the same time the structure should not fail. The modes of the structure are kept in this range. The modal analysis of the structure is done on ANSYS 15.0.**

Key words - FEA, Gait frequency, Mode shape, Optimization .

I. LITERATURE REVIEW

Resonance is a critical factor for the failure of a structure. So efforts are made and measures are taken to avoid it. Getting a modal analysis done is has an equal importance as that of structural analysis. Modal analysis helps in getting the required data for design optimization of structure, reduce the time required for design and attaining the required frequency. The durability of the structure can be predicted by data obtained from modal analysis. Modal analysis helps us calculate service life of a structure which promises a certain period of time for the structure[2].

Optimization is a crucial and beneficial factor design of a structure. For optimization, objective functions and design variables are decided. Range of the variables and its variance is finalized. The data obtained for each trial is compared with the next one till we get the optimized design. The data is tabulated for each trial. The parameters which are important are included in the table. According to the prioritization of the parameters, the optimized design is finalized. Evolutionary structural optimization is the method used for the design[1].

II. INTRODUCTION

From a manufacturers point of view, the need and capacity to produce high quality products in short time with prior development of design, is of main interest.

Finite Element Analysis is one of several numerical methods that can be used to solve complex problems and is the dominant method used today. As the name implies, it takes a complex problem and breaks it down into a finite number of simple problems. A continuous structure theoretically has an infinite number of simple problems, but finite element analysis approximates the behaviour of a continuous structure by analysing a finite number of simple problems.

Dr. John Swanson founded FEA software, inc. in 1970 with vision to commercialize the concept of computer simulated engineering, establishing himself as one of the pioneers of finite element analysis (FEA). FEA Software supports the ongoing development of innovative technology and delivers flexible, enterprise – wide engineering systems that enable companies to solve the full range of analysis problem, maximizing three existing investments in software, hardware and industry. FEA Software continues its role as a technological innovator. It also supports a process centric approach to design and manufacturing, allowing users to avoid expensive and time consuming “build and break” cycles. FEA Software analysis and simulation tools give customers ease of use, data compatibility, and multi-platform support and coupled field multi physics capabilities[2].

. The design and optimization of acquired structures can be done using simulation software. One being used for this study is ANSYS 15.0.

III. DESIGN

The design of given structure is developed in ANSYS 15.0.The process of designing undergoes three stages as follows:

A. Geometry
B. Mesh generation
C. Constraint Definitions
D. Load Definitions
E. Modal Analysis in ANSYS 15.0

A. Geometry

The conceptual design is as defined by term is the original concept or a drawing. Figure 1 shows the conceptual design of a beam structure.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 12, December-2016

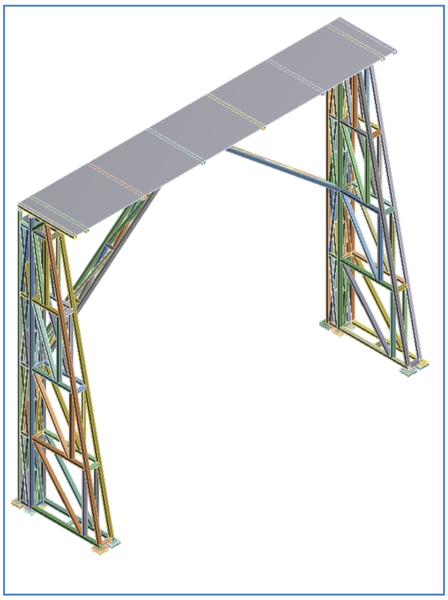

Figure 1: Design of a Beam Structure

The design of this structure is made up of hollow square cross section of the dimension 91.5 × 91.5 × 4.8 mm. The overall dimensions of the structure are 10 × 8 × 2 m. It is by welded down for assembly at the site of erecting. The material used is structural steel beams.

B. Mesh Generation

Method us for meshing is hex-dominant as the structure consists majorly beams of squared cross sections. The statistics of the mesh are tabulated as follows .

Element Size	
Beams	520 mm
Platform	80 mm
Base Plates and Bolts	40 mm

Table 1: Element Size

Mesh Statistics	
No. of Elements	200421
No. of Nodes	58955

Table 1: Mesh Statistics

Figure below shows the mesh generated in Model.

Figure 2: Mesh generation in model

Enlarged view of the mesh generated is shown in the below figure.

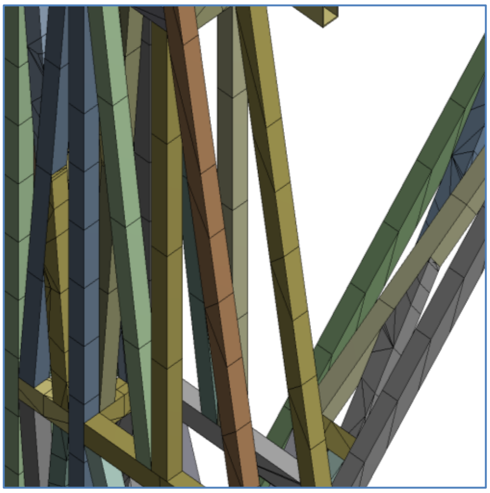

Figure 3: Mesh generation in model

C. Constraint Definition

The structure is based on a concrete foundation with the set of lower base plates buried inside the foundation in a frictionless contact and the upper set of base plates on the ground.

Frictionless support is applied to the lower faces of the base plates. Fixed support is applied to the curved faces of the bolts.

A. Frictionless Support: Lower faces of base plates

B. Fixed Support: Bolts

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 12, December-2016

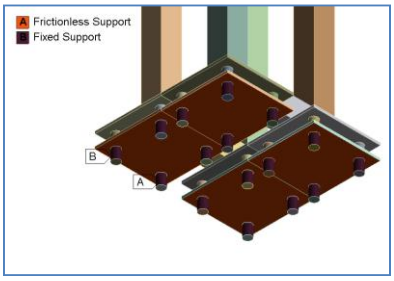

Figure 4: Boundary Conditions

D. Load Definition

Modal analysis on the given structure is carried out under gravitational load. For the basic modal analysis no external force is acting on the structure.

E. Modal Analysis

Modal analysis is done on the above meshed geometry under the gravity. The following figure shows the first three modes of vibration.

1. First mode of Vibration

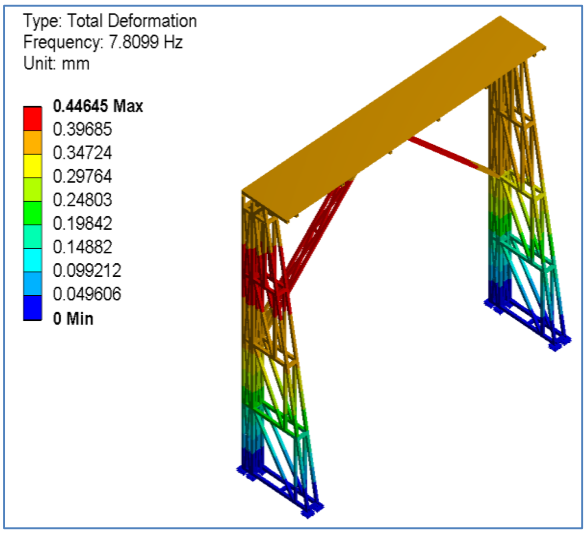

Figure 5: First mode of vibration

2. Second mode of Vibration

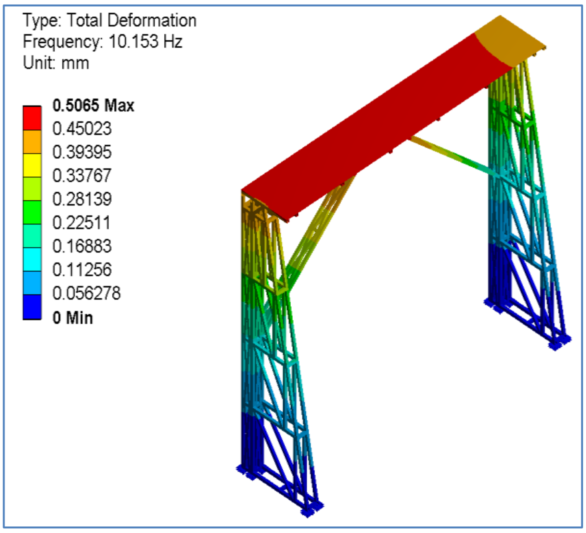

Figure 6: Second mode of vibration

3. Third mode of Vibration

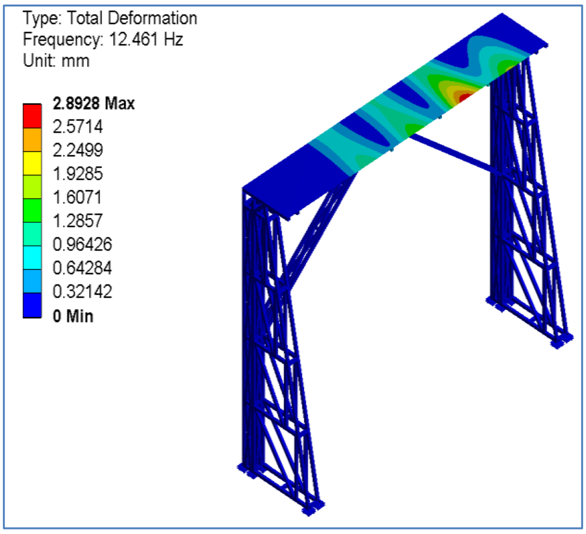

Figure 6: Third mode of vibration

IV. RESULTS

The design is optimized and verified for final design with the suggestion of all the fellow designers. Fundamental mode frequency and respective deformation is calculated. The results are tabulated as under.

The result table shows that the first three modes of the structure which are in the range of 6-15 Hz. The maximum deformation ranges below few 3m which indicates shows the design is safe.

The result is tabulated as follows:

Sr No	Mode Shape No	Frequency in Hz	Maximum Deformation in mm
1	1St Mode shape	7.8009	0.4464
2	2nd Mode shape	10.153	0.5065
3	3rd Mode shape	12.461	2.8928

Table 1: Mode shapes and maximum deformation

V. DISCUSSION

The structure dynamics are dependent on the stiffness of the design. Lowering the stiffness of any structure leads to a decreasing fundamental frequency and further deteriorating the dynamic performance . Lowering the fundamental mode of frequency gets the structure close to any kind of harmonic or rhythmic responses.

VI. CONCLUSION

The main purpose of the structure to give instability training is met with as the fundamental mode frequency of the structure is in the range of human gait frequency.

REFERENCES

[1] Optimization design for beam structures of rail weld CNC fine milling machine based on Ansys Workbench Mao HuajieShu Min Li Chao and Zhang Baojun, Applied Mechanics and Materials Vols 716-717 (2015) pp 817-824 Submitted: 2014-11-13 Trans Tech Publications, 2014-11-13doi:10.4028/www.scientific.net/AMM.716-717.817

[2] Finite Element Analysis and Structural Optimization of the Box on the ANSYS Workbench Kun Cheng Advanced Materials Research Vols 211-212 (2011) pp 434-439 Online: 2011-02-21 Trans Tech Publications, Switzerland doi:10.4028/www.scientific.net/AMR.211-212.434

[3] Ansys Workbench case is proficient in engineering application [M]. Beijing: electronic industry press, Chen Yanxia, Chen Lei. 2012:96

[4] ANSYS ADVACED ANALYSIS TECHNIQUES GUIDE, AnsysInc, 1998