Static and Vibration Analysis of External Rotor Radar Stabilized Platform Using Ansys

Mukkudupalli Atchi Raju[1], Yalamanchili Suresh[2]

Atchiraju1994@gmail.com[1] , yalamanchilisuresh324@gmail.com[2].

Gonna Institute of Information Technology And Sciences,VSKP

ABSTRACT

The radar stabilized platform is used to isolate the loader movement and keep the radar relatively stable. The modal analysis of stabilized platform could provide proof for modification and optimization of structure dynamics. The 3D model of novel external rotor stabilized platform is built by CREO. The finite element model is built by Ansys after the 3D model is simplified. The modal of platform is analyzed through finite element theory and Ansys software. And the natural frequency and vibration catalog of the eighth modal for stabilized platform is acquired. The modal of different materials are compared. It provides references for like analysis. And it also provides proof for optimization of stabilized platform. The program is designed to improve dynamic performance of stabilized platform.

INTRODUCTION

With development of modern science and technology, radar is wide applied in defense field. And the demand of lightness, modularization and intelligence is much higher. As supporting apparatus of antennal, study on the lightness, minimization, higher accuracy, larger rigidity, better dynamic performance and manufacturing of radar stabilized platform becomes more and more importantfl].

The modal analysis is the modern method to study the dynamic character of structure. It provides powerful tool for structure design and performance evaluation for all kinds of products. So it is the base of structure designfl].

The 3D model of novel stabilized platform based on external rotor is built by Pro-E. The finite element model is built by Ansys after the 3D model is simplified. The modal of platform is analyzed through finite element theory and Ansys software. And the natural frequency and vibration catalog of the eighth modal for stabilized platform is acquired. The modal and weight of different grids and different materials are compared. It provides reliable proof for further optimization.

DIGITAL MODEL BUILDING FOR STRUCTURE OF EXTERNAL ROTOR RADAR STABILIZED PLATFORM

The designed external stabilized platform is composed of radar load stage, stator of pitch axis external rotor moment electric engine, rotor of pitch axis external rotor moment electric engine, pitch axis shell, azimuth axis shell, rotor of azimuth axis external rotor moment electric engine, stator of azimuth axis external rotor moment electric engine, base, shaft bearing, encoder, load stage, radar antenna, screw fastener and optical fiber gyro The digital model for structure of external rotor stabilized platform is built by CREO 3.0.

Dynamic Model of the Press Frame Based on the theory of vibration, the dynamic response of system is composed of two parts: one is transient decay free vibration depends only on its initial conditions, and the other is stable forced vibration also determined by harmonic excitation. Harmonic response analysis only calculated the stable forced vibration. Motion equation of forced vibration is:

Mu+Cu+Ku=F(t) ……….(1)

Where: M--mass matrix of press frame structure.

C-damping matrix of press frame structure.

K-- Stiffness matrix of press frame structure.

F(t)--exciting force vector.

it , it ,U --acceleration vector, velocity vector and displacement vector of structure respectively. Under the assumption that frequency and phase of each hannonic excitation were same on the multidegree of freedom system, the exciting force can be expressed by the complex number:

F(t) = (F; +iF2)eJOJI ………. (2)

Where OJ --Harmonic excitation frequency,

F \--real exciting force vector of structure,

F2--virtual exciting force vector of structure. Displacement can be expressed as:

u=(u1+iu2)eJOJI …….. (3)

Where

uj --Real displacement vector,

u2 --Virtual displacement vector.

Therefore, motion equation of the dynamic response analysis:

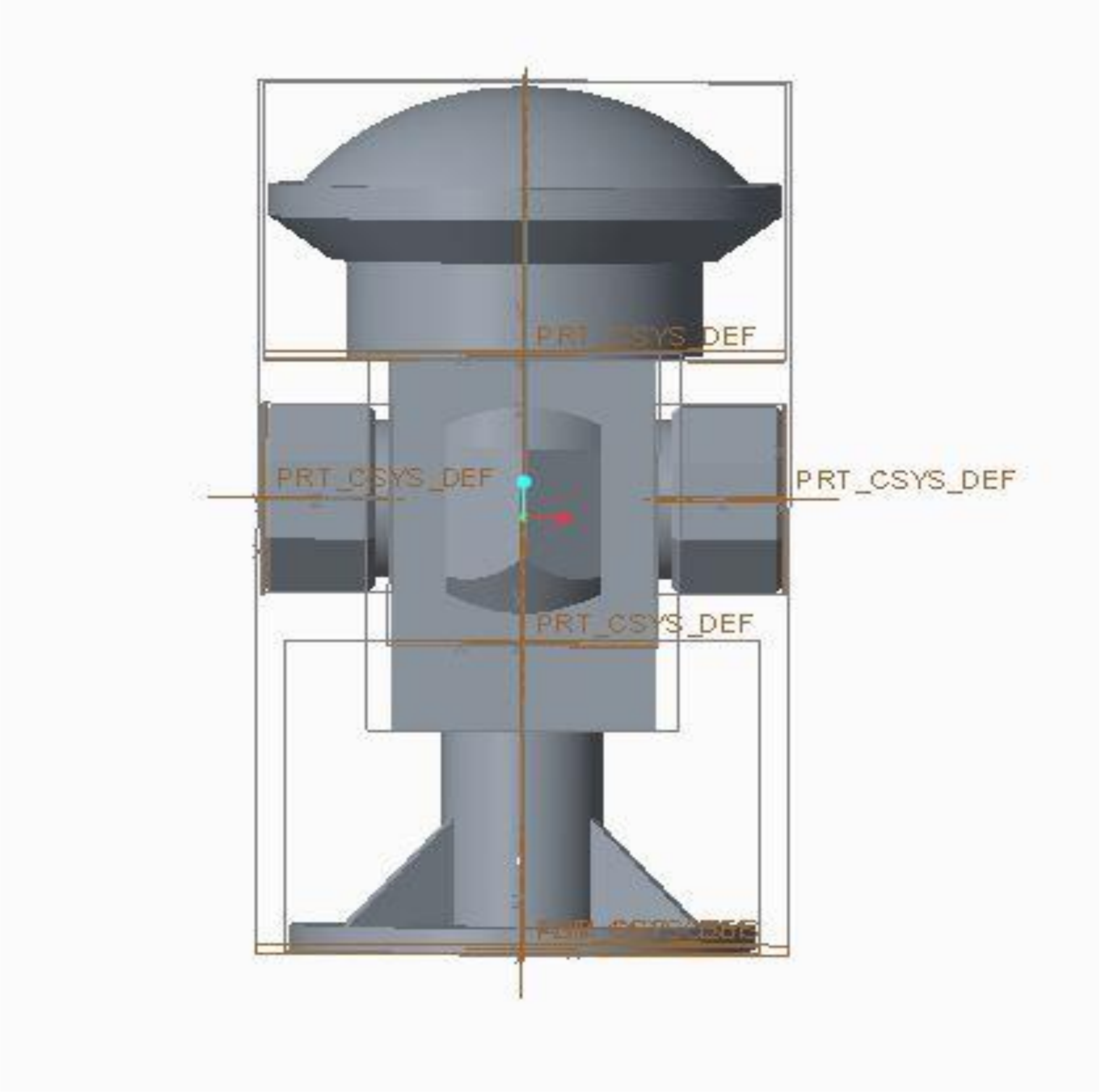
- oi M +ioC +(uj +iu2)K = F; +iF2…… (4)

Simplify model of the Structure Inner structure of the under the top-presses frame was designed very complex to meets process requirement.

Therefore, establishment of finite element analysis model needs to be appropriately simplified:

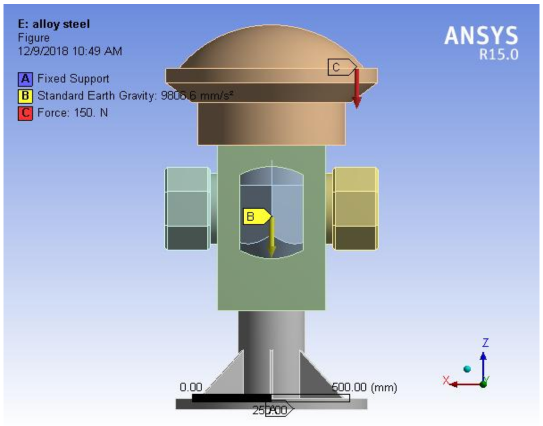
I) The structural characteristics, such as the small holes, fillets, chamfers etc. of the frame have a little effect on its natural characteristic, so its effect could be neglected.

2) The geometric non-linearity has a little effect on the dynamic characteristics of the press, so the linear performance of the structure was only considered.

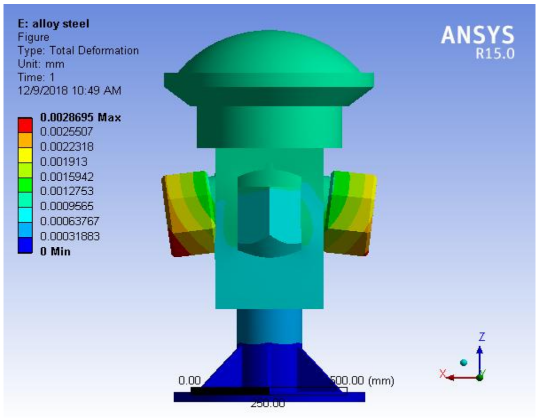


GEOMETRY

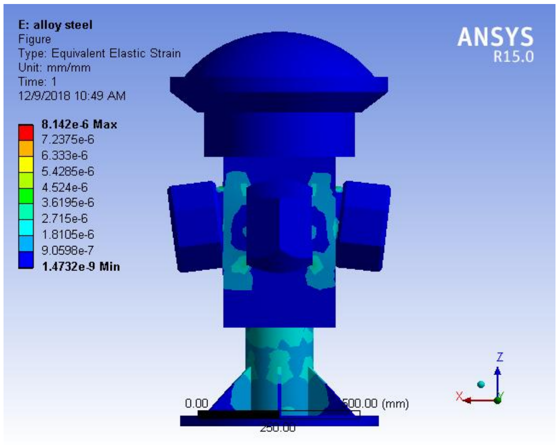
BOUNDARY C/NDITIONS APPLIED ON PLATFORM



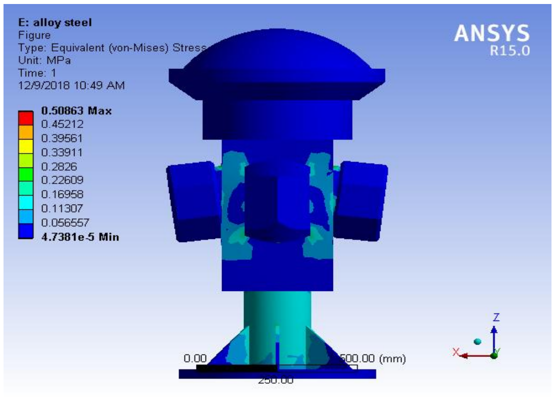
RESULTS AND DISCUSSION



TOTAL DEFORMATION



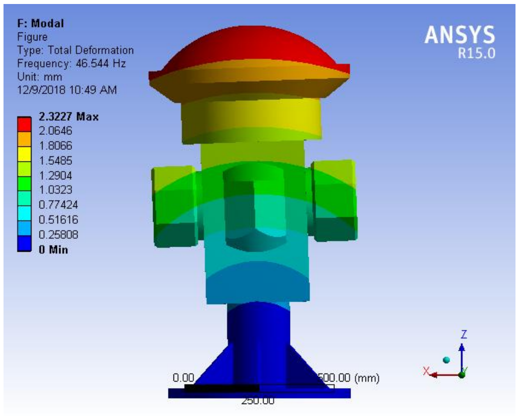
EQUIVALENT STRAIN



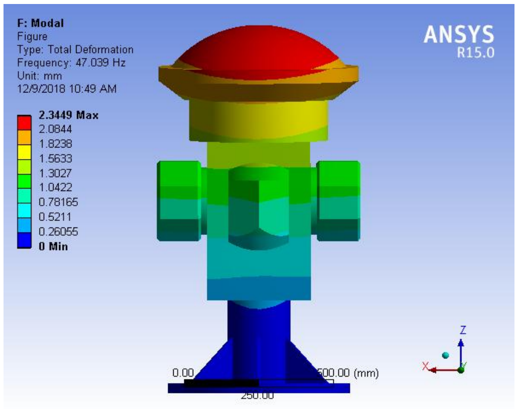
EQUIVALENT STRESS

MODEL ANALYSIS RESULTS

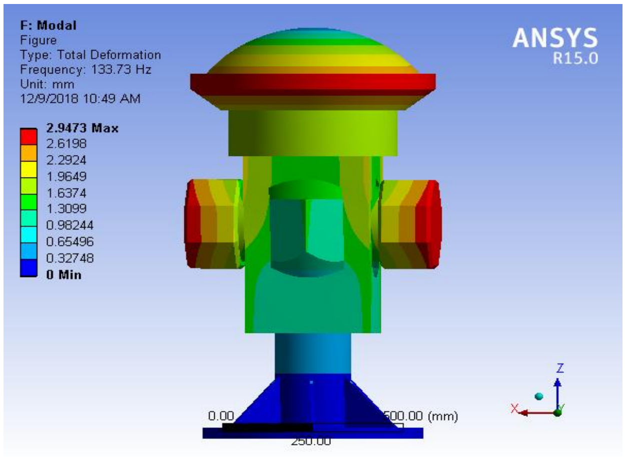
1st mode of deformation



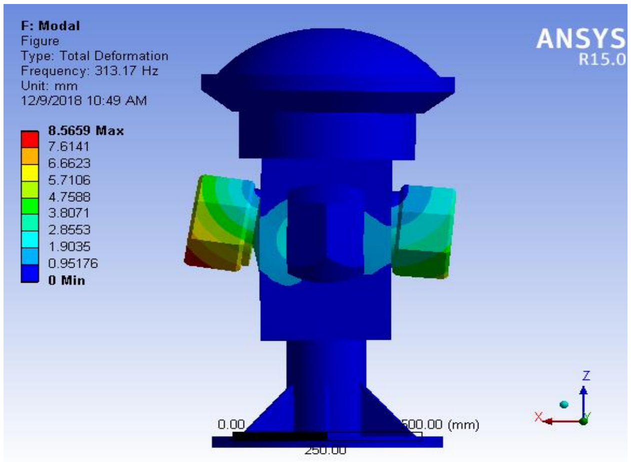
2nd mode of deformation



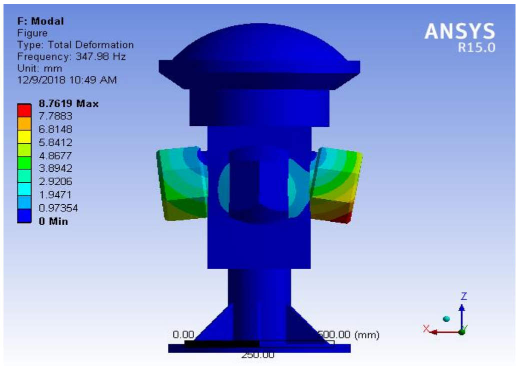
3rd mode of deformation



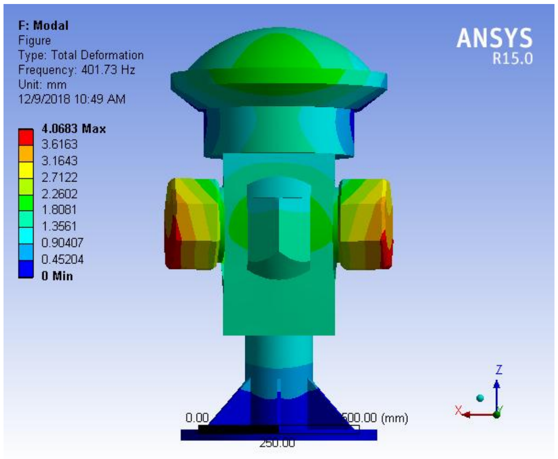
4th mode of deformation



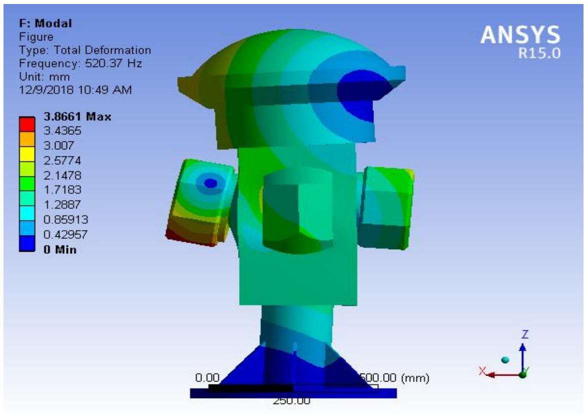
5th mode of deformation



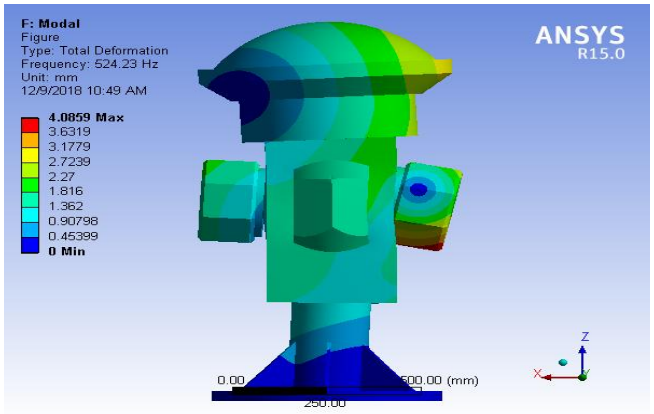
6th mode of deformation



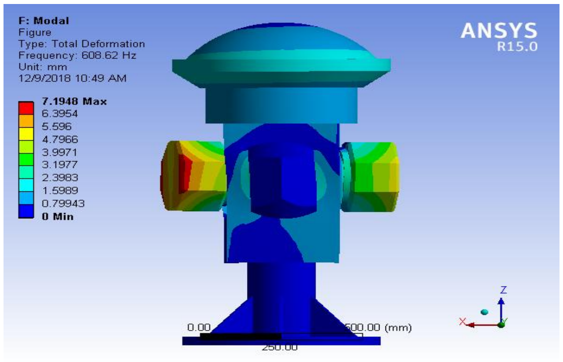
7th mode of deformation



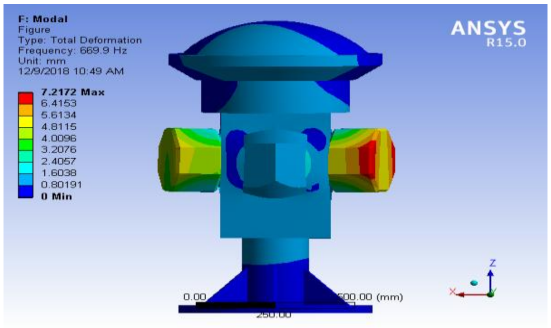
8th mode of deformation



9th mode of deformation

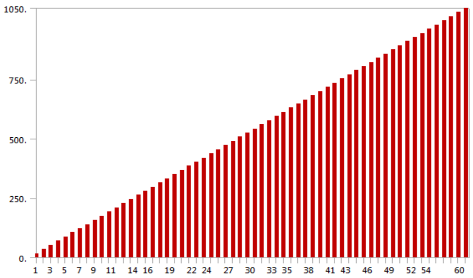


10th mode of deformation



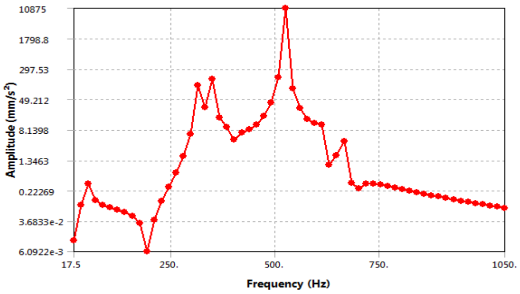
HARMONIC RESULTS

Harmonic Response Analysis by ANSYS Any sustained periodic load will have a sustained periodic response in the structural system, this is the hannonic response. Hannonic response analysis can help people prediction of sustained dynamic characteristics of structural, and the design whether it could be overcome the resonance, fatigue and other harmful effects caused by forced vibration are verified by designers. According to the finite element model as shown in Figure I, vibration at different frequencies in dangerous point of the under the top-presses could be obtained by using the full method (Full method) harmonic response analysis module of ANSYS software. Parameter setting: because the vibration in one direction of upper load platform and on top of leg were almost same, therefore, the midpoint of lower surface of the upper beam was select as the research object and amplitude of 150N sinusoidal excitation force was applied, its exciting frequency width is 0 to 1050Hz, loading method was stepped. According to the previous set of the parameters and the finite element model, harmonic response curve of X, Y, Z three different directions in a cartesian coordinate system could be obtained by using ANSYS software

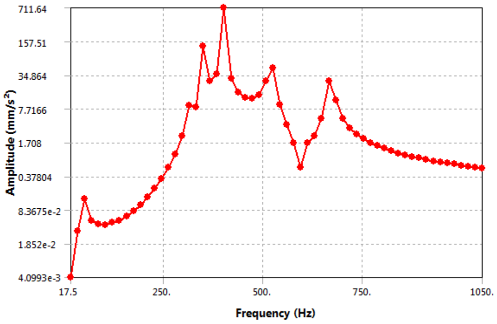


Definition						
Type	Directional Acceleration			Directional Velocity		
Orientation	X Axis	Y Axis	Z Axis	X Axis	Y Axis	Z Axis
Suppressed	No					
Options						
Frequency Range	Use Parent					
Minimum Frequency	0. Hz					
Maximum Frequency	1050. Hz					
Display	Bode					
Results						
Frequency FOR DURALUMIN	525. Hz	402.5 Hz	525. Hz		402.5 Hz	525. Hz
FREEQUENCY FOR STEEL	516.67 Hz	400. Hz	516.67 Hz		400. Hz	516.67 Hz

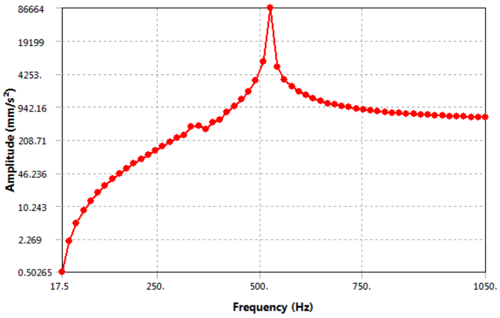
Frequency Response



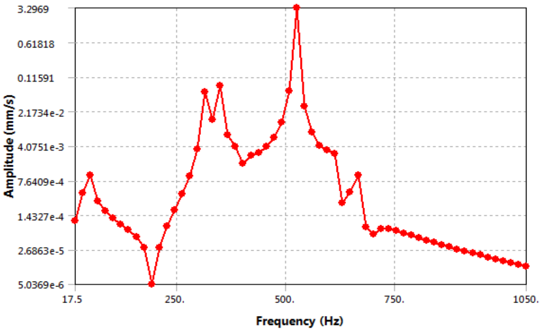
Frequency Response 2



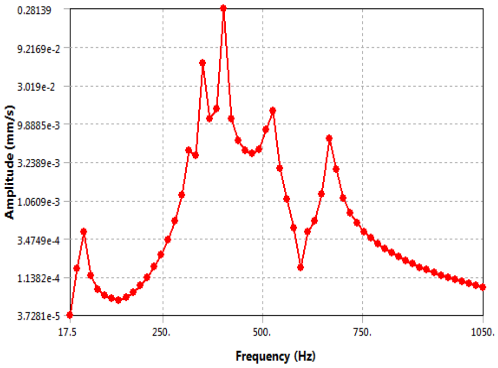
Frequency Response 3



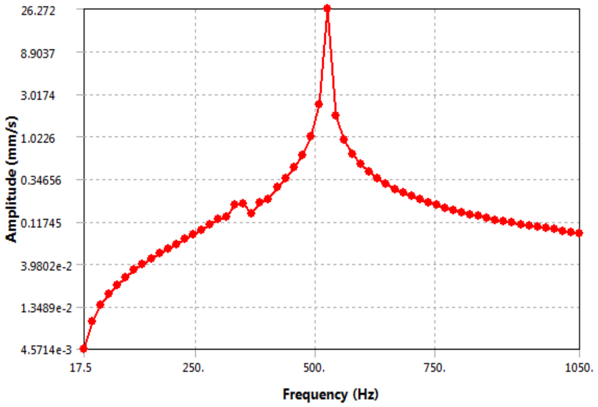
Frequency Response 4



Frequency Response 5



Frequency Response 6



Mode	Frequency [Hz] FOR DURALUMIN	Frequency [Hz] FOR DURALUMIN
1.	46.544	45.981
2.	47.039	46.469
3.	133.73	133.3
4.	313.17	309.75
5.	347.98	344.17
6.	401.73	397.59
7.	520.37	514.31
8.	524.23	518.21
9.	608.62	602.41
10.	669.9	661.76

CONCLUSION

1. From the results of model analysis, we can say that the model is safe from dynamic point of view. as the first mode of defomation is well within the safe limits.

2. for both the materials the stress value is well below the yield strength of the materials in case of dead loads.

3. Upon comparing the peekload freequency from HARMONIC analysis and free vibrating freequencies of model analysis, there is no coincidence in the freequencies. therefore we can say the structure is safe from dynamic point of view as well since there will not be occurance of resonance for this particular load value.

4. from table, /we can say that the platform made of steel gives the best stability from dynamic stand point. So, its recommended to use steel platform.

FUTURE SCOPE:

only dead loads have been considered so far in our study to determin the stability of the frame. But in real condition, so many other factors such as wind loads, sesimic loads etc. would influence the stability of the structure. So, we need to consider the vigarous loads to estimate the stabilty of the structure. and the strength of the frame can also be enhanced by using several alloys of steel. so, there is a scope for the research from materials front as well.

REFERENCES

- Duan Yongjun, Gu Jifeng, Ping Lihao,.modal analysis and experimental study on radar antenna stage[J]. mech- anical design and fabrication, 2010,2(2):214-216.

- James Callum Russell, A Control System and Application Program Interface (API) for an Elevation Over Azimuth Tracking Pedestal, A dissertation submitted to the Department of Electrical Engineering,University of Cape Town, in partial fulfilment of the requirements for the degree of Bachelor of Science in Electrical and Computer Engineering.20 October 2008.

- Yang Xiuli, Jia Hongguang, Feng Changyou. Finite Element Modal Analysis of the Gyroscope-Stab ilized Pla tform[J] . Computer simulation,2008,25(9): 70-73.

- Chen Guodong, Han Xu, Liu Guiping.Dynamic character analysis and optimizaion of some heavy duty commecial vehicle operator cabin

- LI Jian, XU Zhi-wei. Manufacture Process of a Satellite-borne Active Fixing-board [J]. Electro-Mechanical Engineering, 2012, 28(3):62-64.

- Wang Jin-long, Wang Qing-ming, Wang Wei-zhang. The Finite Element Method and based on ANSYS 12. 0 [M]. Beijing: Machinery Industry Press, 2010.

- LI Chun-you. Modality calculation and experimental analysis on large scaled double layered nonlinear vibration screen [J]. JOURNAL OF MACHINE DESIGN, 2009, 26(5): 28-30.

- Ward Heylen, Stefan Lammens, Paul Sas. Modal analysis theory and testing [M]. Katholieke
- Universiteit Levven-Departement werktvi gkunde, 1997.

- LIU Zheng-shi, GAO Rong-hui, CHEN En-wei. Mechanical Dynamical Foundation [M].
- Beijing: Higher Education Press, 2010.

- DUAN Yong-jun, GU Ji-feng, PING Li-hao, LIU Ji-cheng. Mode analysis and experiment
- research of radar antenna pedestal [J]. Machinery Design & Manufacture, 2010(2): 214-216.

- YANG Jin-song. Modal Correlation Analysis of Frame Structure of a Space Camera [J].
- Computer Simulation, 2006, 26(1), 274-276.

- XIE Shao-wang, LI Qing-fen. Modal calculation and correlation analysis of the frame of a
- platform compass [J]. Journal of Harbin Engineering University, 2009, 30(2): 183-186.

- Kang Guidong, Huang Yingyun, Liu Peng. Investigation of Experimental Modal Analysis of
- TBD234V6 Diesel Engine Crankshaft [J]. Small Internal Combustion Engine and Motorcycle,
- 2011, 40(4): 31-34.

- LI Hui, DING Hua. PROGRESS IN MODEL UPDATING FOR STRUCTURAL DYNAMICS