International Journal of Modern Engineering Research (IJMER)
www.ijmer.com Vol.2, Issue.2, Mar-Apr 2012 pp-239-244 ISSN: 2249-6645

Design Optimization of the Inner Gimbal for Dual Axis Inertially Stabilized Platform Using Finite Element Modal Analysis

Hany F. Mokbel*, Lv Qiong Ying, Amr A. Roshdy, Cao Guo Hua

College of Mechanical and Electric Engineering,
Changchun University of Science and Technology, Changchun, Jilin, China

ABSTRACT

For the purpose of stabilizing the Line Of Sight *(LOS)* of Electro-Optical *(EO)* surveillance and tracking system, a Dual Axis Inertially Stabilized Platform *(ISP)* is required to isolate the *LOS* from the carrier disturbances and also for the tracking of the moving target. In this work an Elevation over Azimuth *ISP* has been designed on the *Pro/E* software and the first five modes of the Natural Frequency were determined by the *ANSYS* software. For improving the dynamic performance of the overall system, the design was optimized based on the Finite Element Method *(FEM)* Modal Analysis. Firstly, the weight of the payload carrying frame was decreased by *35%* and the *1st* torsional mode of vibration was increased by *22%* of the original design. Secondly, the overall dynamic performance of the inner gimbal was optimized and the dynamic performance increased by 75%. Reading in the results declares that the *FEM* Modal Analysis is an effective tool to optimize the system's dynamic performance to assure the optimal working conditions far from the resonating failure.

***Keywords*- Design Optimization, Inertially Stabilized Platform (ISP), Modal Analysis.**

1. Introduction

The process of aiming, tracking and targeting on a moving target simply requires recognizing the target within the field of view, capture this target, following it, and finally giving the order to fire on the target [1]. This actually requires an optical sensor to obtain the field of view, a signal processing to recognize the target, a control system for the target tracking loop, an *Inertially Stabilized Platform (ISP)* to achieve continuous and stable *Line Of Sight (LOS)* between the optical sensor and the target, and finally the weapon system to fire on the target. The orthogonal dual axis stabilized platform is the minimum required set of *ISPs* to achieve the efficient stabilization [2]. The design process of the *ISP* is a case sensitive, which means that according to the specified function and requirements of the system, the design should be tailored.

In *Electro-Optical Tracking* system, the carrier vibration in the *Yaw (Azimuth), Pitch (Elevation) and Roll* directions induces the *LOS* sensor to rock and causes the image blurred and affects the pick-up of the target's distance, leads to failing of the tracking performance [3,4]. Therefore, the *LOS* stabilization technology must be used to isolate the LOS from carrier disturbance in order to guarantee accurate aiming and tracking of the target at the *Inertial Space*.

In our work we are seeking to design a dual-axis *ISP* that is used to stabilize the *LOS* of surveillance *CCD* Camera and a Laser Distance Meter carried on a ground vehicle moves on different terrains. The design is following the classical concept of the Elevation over Azimuth rotation. A rotary stage with ± *360º* continuous travel, Direct-drive, slotless and brushless servomotor is used for the Azimuth scanning. A massive structure is used to carry the Elevation gimbals. Another frameless, brushless servomotor is used to rotate the inner gimbal in the Elevation direction. The stall torque and peak torque are *0.2* and *0.82 N.m* respectively. The rotation limit for the Elevation direction is *-20º:+90º*. An optical encoder is used with the Azimuth rotary stage while the Elevation servo motor includes the Hall Effect sensors for the positioning feedback. The rotation speed for both directions is *90º/s*. *MEMS* gyros are used in both orthogonal directions for angular velocity measurements. Preloaded high performance axial ball bearing is used for supporting the gimbals. The Elevation gimbal consists of the Electro-Optical payload, the payload carrying frame and the front and back hoods that surrounding the inner gimbal and facing the environment. The servo robust PI controller was performed with the C^{++} and simulated on The MATLAB software. The Original design was done on the *Pro/E* software as a *3D* solid model as shown in *Fig. 1, 2*. The overall dimensions are *278* x *200* x *256 mm (L* x *W* x *H)*, and an overall mass of *10.2 Kg*.

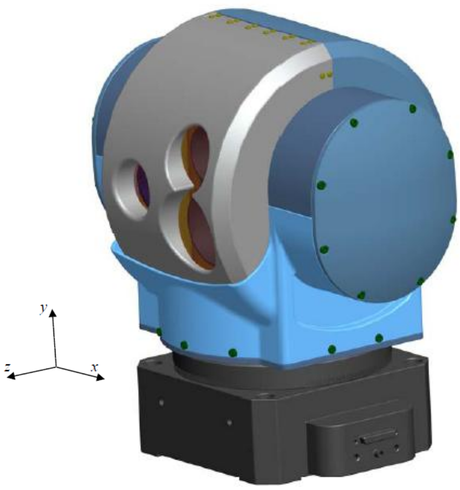


Fig. 1: *Dual Axis Inertially Stabilized Platform (ISP).*

In the actual working conditions, the *ISP* is subjected to five major loads [5], namely; *(1)* The gravity load due to the weight of its components, which is a combined loading of bending and torsion, *(2)* The fatigue load due to the cyclic rotation and loading, *(3)* Stress due to temperature changing, *(4)* Air resistance loading, and *(5)* Shock loads due to terrain's obstacles.

The design had been structurally tested for the static and fatigue loading and found to be conservative, the thermal stresses due to the change in temperature from -20º to +45º were found to be within the allowable limits for stress and deflection. The limitation of the design due to the vibration's Natural Frequencies of its components is still the main point under consideration in this work. So, the research is going to optimize the design to have the highest Natural Frequency and minimum weight within the allowable stiffness and strength.

Every mechanical structure exhibits natural modes of vibration *(dynamic response)* [6]. A structural mode can be thought of as a shape and a frequency at which the structural shape resonates. The shape and frequency of a mode are primarily a function of the structure's stiffness, damping and mass distribution. Determining the first few natural modes of vibration for a structure is of great importance to assure that the applied loads or the working conditions frequencies will never meet these Natural Frequencies to avoid resonance and failure. The *ISP* consists of many components working simultaneously. Of them the inner gimbal is the critical one. So, optimizing its dynamic performance will optimize and improve the overall system's dynamic performance.

2. Modal analysis using FEM

Finite Element Method (FEM) is the most frequently used method for computational modal analysis [7]. It is common to use the *FEM* to perform this Modal Analysis because, like other calculations using the *FEM*, the object being analyzed can have arbitrary shape and the results of the calculations are acceptable [8]. The types of equations which arise from modal analysis are those seen in Eigen-systems. The physical interpretation of the Eigen-values and Eigen-vectors those come from solving the system are representing the frequencies and the corresponding mode shapes.

The modal representation of a mechanical structure can be determined analytically if a lumped mass-spring system is concerned [9]. In the general case of a continuous structure, a numerical approximation by means of a *FEM* is made, dividing the structure into a finite number of physical coordinates. The equations of motion describing this approximated system in the time and Laplace domains are given by:

$$[M]\{\ddot{x}(t)\}+[C]\{\dot{x}(t)\}+[K]\{x(t)\}=\{f(t)\} \qquad (1)$$

$$\left|s^2[M]+s[C]+[K]\right|\{X(s)\}=\{F(s)\} \qquad (2)$$

Where, *[M]*, *[C]*, & *[K]* represents; mass, damping and stiffness matrices respectively. The solution of these equations leads to an Eigen-value problem that is solved in terms of the modal parameters. The limitations of the *FEM* approach lie in the increasing model size required to properly describe complex structures with appropriate detail. This leads to higher model construction and calculation times, but even more important, there remain inherent modeling accuracy limitations, related to the modeling of structural junctions, non-homogeneous elements, complex materials etc [10]. Modal analysis has become a major technology in the quest for determining, improving and optimizing dynamic characteristics of structures.

3. The ISP's FEM Modal Analysis Models

The limited portion of the *ISP's* design appears clearly to be the inner gimbal that rotates in the Elevation direction *(about x-axis in Fig.1)* and then rotates with its supporting frame in the Azimuth direction *(about y-axis in Fig.1)*. So, its Natural Frequency is the most important parameter in controlling the Natural Frequency of the system. The design of the inner gimbal was made to be symmetric about the axis of rotation, small counter weights were used to achieve this symmetry. The inner gimbal consists of the payload carrying frame, the Electro-optics payload, and the front and back hoods. They form about *16%*, *64%* and *20%* of the inner gimbal mass respectively. The *FEM* Modal Analysis Optimization will be carried on three steps; *(1)* Optimizing the payload carrying frame, *(2)* The payload carrying frame with the Electro-Optical payload, and *(3)* The whole inner gimbal with the front and back hoods. Again, the purpose of the design optimization process is to reduce the mass and increase the values of the natural frequencies for the first few modes of vibration *(dynamic performance)*. The *3D* solid models of these three cases are shown in *Fig.2*.

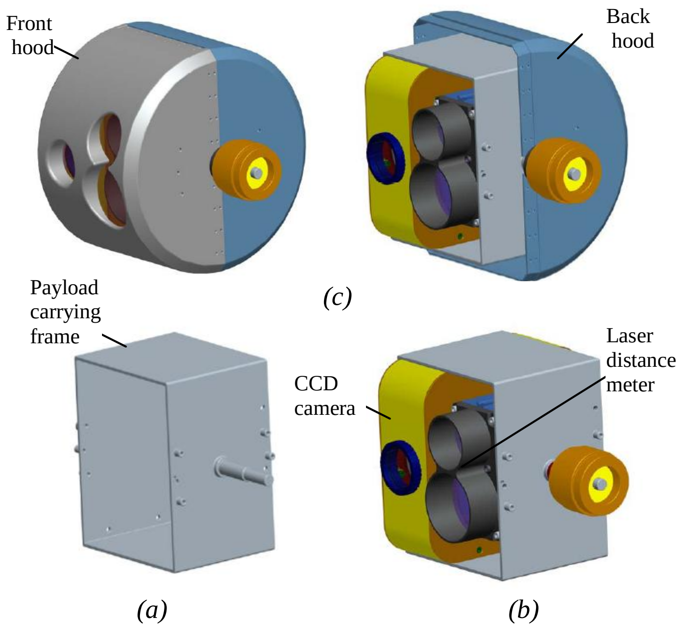
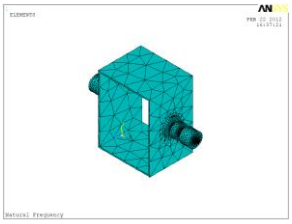


Fig. 2: *(a) the payload carrying frame, (b) the payload carrying frame with the Electro-Optical payload, and (c) The whole inner gimbal with the front and back hoods.*

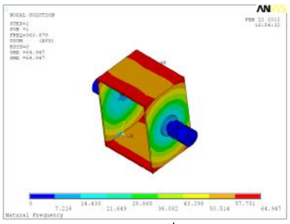
The *3D* Models of the three Cases were transformed in *IGES* format and imported into the ANSYS 12.1 software to be analyzed for the first five vibration modes *(natural frequencies)*. In ANSYS this is called Modal Analysis. Many simplifications had to be made to the *3D* model to be read by the ANSYS. Of course these simplifications are not of great importance in the analysis but they must be done to declare the model from the very fine details that couldn't be digitized by a non specialized *3D* modeling software or may take a lot of time and effort from the solver and restricts the solution ability.

3.1 The payload carrying frame FEM Modal Analysis

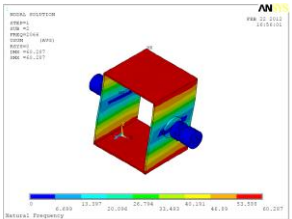
In the ANSYS software , the element type was chosen to be solid *45 (Brick 8 node)*, the material properties for the payload carrying frame was the properties of the common Aluminum alloys with density *2.79355e-9 tonne/mm*^{3}, Modulus of elasticity *73084.4 MPa* and Poisson's ratio of *0.33*. The solid model was consisted of *237* key-points, *96* lines, *92* areas and *1* volume. The finite element model, *Fig. 3.a*, was consisting of *10268* nodes and *51873* elements. A free mesh with tetrahedral shapes was applied. The Analysis type was set to modal analysis with the Block Lanczos extraction method, and normalizing the mode shapes to the mass matrix. The boundary conditions were applied to the left and right supporting area on which the bearings are mounted. The problem was solved for the first *5* modes of the natural frequencies *(Fig. 3 and Table 1)*.



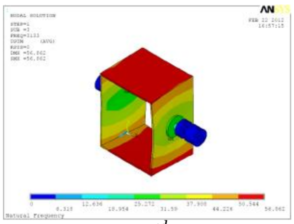
(a) The FEM model



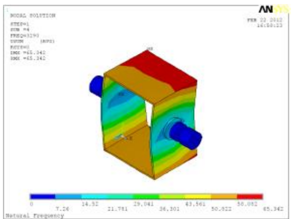
(b) The 1st mode



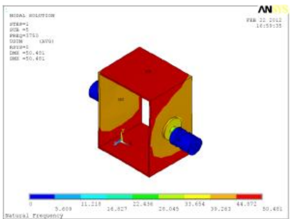
(c) The 2nd mode



(d) The 3rd mode



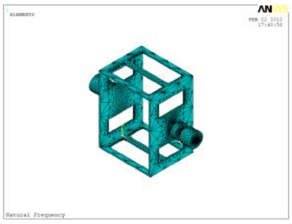
(e) The 4th mode



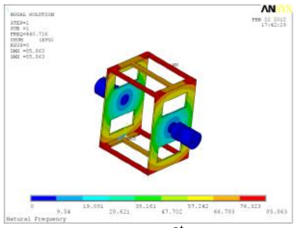
(f) The 5th mode

Fig. 3: *The FEM model and the first 5 modes of the Natural Frequency for the original payload frame.*

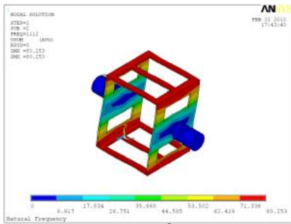
The results of the Modal Analysis shows good dynamic performance for the payload carrying frame, where the *1st* torsional mode about the axis of rotation appears at *360.68 Hz* followed by a large amplitude with the *1st* longitudinal bending in x direction at *2066 Hz*. This dynamic performance still can be improved by decreasing the mass of the payload frame by digging *4* rectangular holes *(95x35x3 mm)*, *2* in the upper and *2* in the lower side, and another *4* rectangular ones *(60x30x3 mm)*, *2* in the right and *2* in the left side. This lightning the frame about *35%* of its original mass, from *0.490 Kg* to *0.318 Kg*. The modified model's *FEM* Modal Analysis results are shown in *Fig. 4* and *Table 1*.



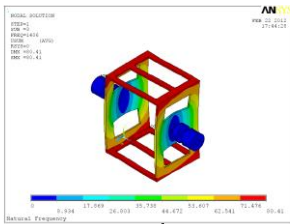
(a) The FEM model



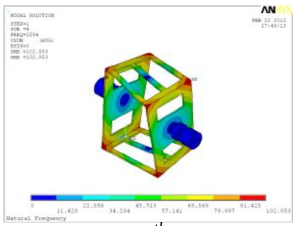
(b) The 1st mode



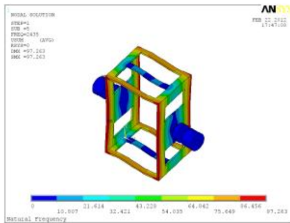
(c) The 2nd mode



(d) The 3rd mode



(e) The 4th mode



(f) The 5th mode

Fig. 4: *The FEM model and the first 5 modes of the Natural Frequency for the modified payload frame.*

Table 1: *Comparison of the original and Modified first five modes of vibration for the payload carrying frame*

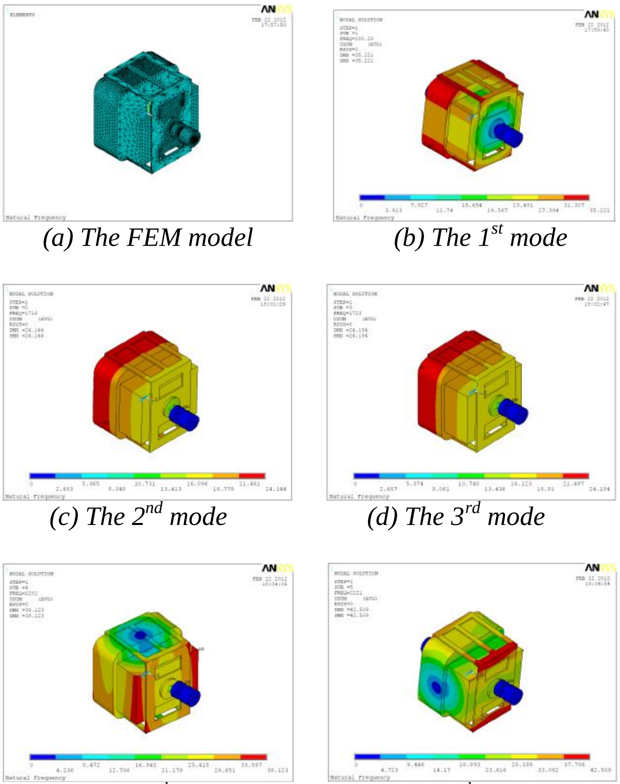
Mode Number	Natural Frequency				Change %
	Original		Modified		
1	360.68	1st Torsion mode about x axis	440.73	1st Torsion mode about x axis	+22.2
2	2066	1st longitudinal bending in ±x direction	1112.1	1st longitudinal bending in ±x direction	-46.17
3	3133.2	2nd Torsion mode about x axis	1406	2nd longitudinal bending in x direction	-55.13
4	3289.9	Combined longitudinal and lateral bending in x and -z directions	1593.9	Combined lateral bending and rotation in ±y directions	-51.55
5	3750	1st Vertical bending in y direction	2435.4	3rd longitudinal bending in ±x direction	-35.06

It is clearly shown that; decreasing the mass by *35%* and softening the structure by digging the *8* rectangular holes has obviously changed the modes of vibration's shape and amplitude. As expected, the mass lightning has improved the dynamic performance by increasing the *1st* rotational vibration mode to *440.73 Hz*

(22.2% higher than the original design). On the other hand, the next *4* modes have dramatically decreased by about *35%* to *55%* as shown in *Table 1*. But these results are still very satisfactory for the design.

3.2 The payload carrying frame with the Electro-Optical payload FEM Modal Analysis

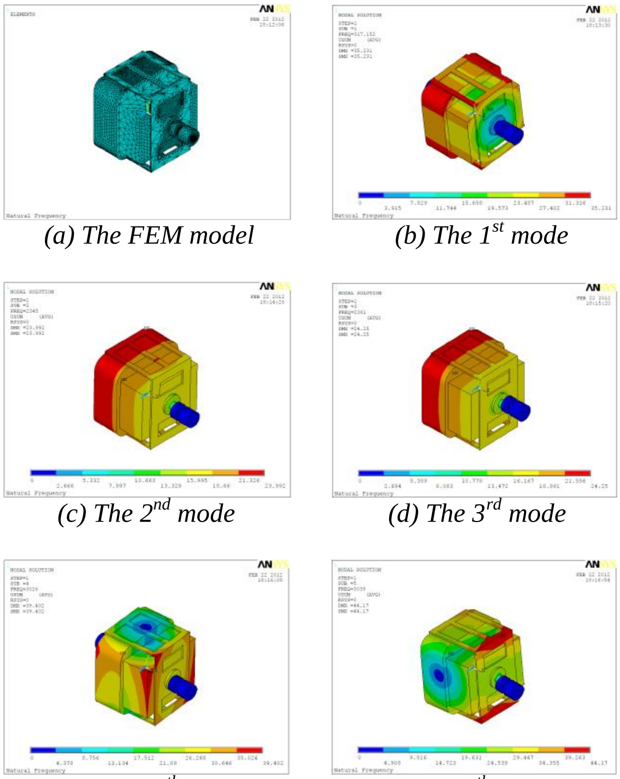
In this section the dynamic performance of the payload carrying frame with the Electro-Optical payload will be analyzed. The analysis will be created using the same condition as the discussed above for the payload frame Modal Analysis. The *3D* solid model is shown in *Fig. 1.b*, and the *FEM* Modal Analysis results for the first *5* natural frequencies are shown in *Fig. 5* and *Table 2* respectively. Herein, the overall mass of the structure has increased from *0.318 Kg* for only the payload carrying frame to *2.29 Kg* for both the frame and the payload. Moreover, the overall structure's stiffness is increased due to the tightening of the carrying frame on the payload.



(a) The FEM model *(b) The 1^{st} mode*
(c) The 2^{nd} mode *(d) The 3^{rd} mode*
(e) The 4^{th} mode *(f) The 5^{th} mode*

Fig. 5: *The FEM model and the first 5 modes of the Natural Frequency for the payload and the payload frame.*

Of course, the results are completely different from those shown in the previous section due to the great change in the overall structural stiffness, damping and mass distribution. The amplitude of the *1^{st}* torsional mode of vibration about the rotation axis *(x-axis)* has dropped to *180.29 Hz*, while the next four natural frequencies have also dropped, but still having high amplitudes. So, to improve the dynamic performance for this structure, we are going to stiffen it against twisting about the x-axis, by increasing the shaft shoulder diameter in the region connecting the rotating driving shaft with the payload carrying frame body from *12 mm* to *16.2 mm* for a length of *9 mm*. Although this amendment is not necessary from the point of view of the static and fatigue failure analysis, but it's so beneficial from the point of view of the dynamic performance. The results of the modified structure Modal Analysis are shown in *Fig. 6* and *Table 2* for the first *5* modes of Natural Frequencies.



(a) The FEM model *(b) The 1^{st} mode*
(c) The 2^{nd} mode *(d) The 3^{rd} mode*
(e) The 4^{th} mode *(f) The 5^{th} mode*

Fig. 6: *The FEM model and the first 5 modes of the Natural Frequency for the modified design.*

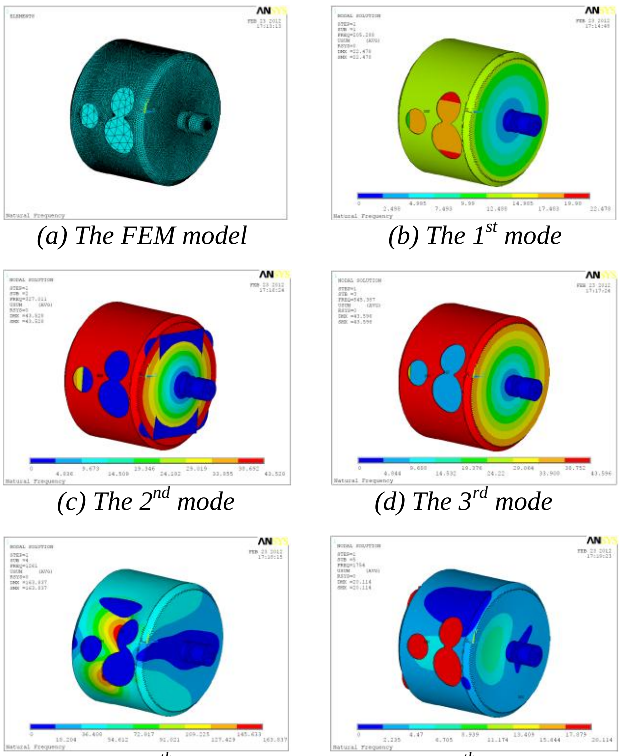
Table 2: *The original and modified first five modes of vibration for the payload and the payload frame*

Mode number	Natural Frequency			Description
	Origin al	Modified	Change %	
1	180.29	317.15	75.9	1^{st} Torsion mode about x axis
2	1716.3	2345	36.63	Combined vertical bending in y direction and longitudinal bending in x direction
3	1722.5	2360.8	37.06	Combined lateral bending in z and y directions
4	2201.6	3025.8	37.44	lateral Bending in ±Z direction
5	2220.5	3039	36.86	vertical bending in ± y direction

After applying the modifications to the structure, the *1^{st}* torsional mode of vibration about the rotation axis has been improved to *317.15 Hz*, increased about *75.9%*, and the following *4* modes of vibration have kept their shapes, but with an increasing in the Natural Frequency of about *37%* of the original value. The purpose for optimizing the design to keep its Natural Frequency as high as possible is to withstand the drop in it when augmented working components are added to the system. As mentioned before, the Natural Frequency of the system is a function of the overall structural stiffness, damping and mass distribution. So, in a working dual axis *ISP*, the two orthogonal rotating gimbals mechanisms are expected to gain a reduced structure's stiffness especially with the all rotating fine structures mounted on it. This will be declared when adding the front and rear hoods to the inner Elevation gimbal structure in the next section.

3.3 The whole inner gimbal with the front and back hoods FEM Modal Analysis

To check the convenience of the design optimization process for the inner gimbal, a final analysis for the whole inner gimbal with the front and back hoods has been made. The *3D* solid model is shown in *Fig. 2.c*. As a result for increasing the *3D* solid model size and details, the *FEM* model size has increased too. Despite the many assumptions and details' simplifications that have been carried out to the design to be handled with the *FEM* ANSYS software, the size of the *FEM* model is still too large. It was consisting of *66925* nodes and *242411* elements. The free mesh option couldn't mesh the model because of the fine details, so, the mesh tool with the smart size option was used to refine the mesh. The dynamic behavior of the model is shown in *Fig.7* and *Table 3*. Where, the first *5* Natural Frequency's modes are demonstrated and compared with those without the front and back hoods.



(a) The FEM model *(b) The 1st mode*
(c) The 2nd mode *(d) The 3rd mode*
(e) The 4th mode *(f) The 5th mode*

Fig. 7: *The FEM model and the first 5 modes of the Natural Frequency for the whole inner gimbal.*

Table 3: *The first five modes of Natural Frequency for the whole inner gimbal*

Mode number	Natural Frequency	Change %	Description
1	205.29	-0.35.27	Torsion about x axis for the inner components
2	327.01	-80.95	longitudinal bending in - x direction for the front and back hoods
3	545.39	-68.34	Front and back hoods contoured deformation
4	1261.4	-42.71	Optical sights aperture deformation
5	1753.9	-21.01	longitudinal bending in x direction for the inner components

Adding the front and rear hoods has increased the mass with *0.652 Kg* to achieve a system's new mass of *2.942 Kg* with an increase of about *28.47%*. Also, the cylindrical shape of the front and back hoods with the shell structure and Electro-Optical sighting devices' apertures have decrease the assembly's overall stiffness. Leading to new mode shapes depending on the accumulated subassemblies. An expected decrease in the *1st* torsional mode shape has happened with consequential decreasing in the amplitude of the following few orders of the natural frequencies. The new obtained Natural Frequency of the *1st* order mode of vibration is *205.29 Hz*, with a drop of *35.27%* from the previously obtained one without adding the front and back hoods. From the conservative design point of view, keeping the *1st* Natural Frequency of the inner gimbal higher than *150 Hz* is fairly enough to assure the convenient dynamic performance of the inner gimbal subassembly under the probability of some imperfections in the manufacturing process. Hence, the design optimization process of the *ISP's* inner gimbal using the *FEM* Modal Analysis has achieved successfully a desired dynamic performance.

4. Conclusion

The system's overall Natural Frequency is depending on the Natural Frequencies of its individual components and subassemblies. The shape and frequency of a natural vibration's mode are a function of the structural stiffness, damping and mass distribution. Due to the accumulation effect of the Natural Frequencies of the different *ISP's* component, the overall Natural Frequency was decreased from that of the individual ones. To withstand this reduction, the Natural Frequency of the individual components was optimized using *FEM* Modal Analysis. The payload carrying frame mass decreased by *35%*, and its Natural Frequency increased from *360.68 Hz* to *440.73 Hz*. For the subsystem of the payload and the carrying frame, the *1st* Natural Frequency increased from *180.29 Hz* to *317.15 Hz*. This led to a Natural Frequency of *205.29 Hz* for the overall inner gimbal assembly, which is above the desired value of *150 Hz* that assures the convenient dynamic performance of the inner gimbal. The *FEM* Modal Analysis of the *ISP's* Inner Gimbal has proved to be an effective method for optimizing the design to achieve the least mass and appropriate dynamic performance.

References

[1] M. K. Masten, Inertially Stabilized Platforms for Optical Imaging Systems, Tracking Dynamic Targets With Mobile Sensors, *IEEE Control Systems Magazine, Vol. 28*, 2008, pp. 47 –64.

[2] J.M. Hilkert, Inertially Stabilized Platform Technology, Concepts And Principles, *IEEE Control Systems Magazine, Vol. 28*, 2008, pp. 26 – 46.

[3] P. J. Kennedy, and R. L. Kennedy, Direct Versus Indirect Line of Sight (LOS) Stabilization: *Proc. IEEE Transactions on Control Systems Technology, vol. 11, no. 1*, 2003, pp. 3-15.

[4] Y. Man, J. Gong, H. Chen, Y. Hu, and Y. Jiang, Design and Research of Intelligent Vehicle Strapdown Line-of-Sight Stabilization Platform System: *Proc. IEEE Intelligent Vehicles Symposium*, 2009, pp. 1251-1254.

[5] Q. Zou, H. L. Ma, Y. H. Hu, and J. W. Ding, The lightweight design of the inner framework of the opto-electronic stabilization platform based on finite element analysis: *IEEE 18Th International Conference on Industrial Engineering and Engineering Management (IE&EM), Vol. 1*, 2011, pp. 521 –525.

[6] David V. Hutton, *Fundamentals of Finite Elements Analysis* (McGraw-Hill, NY 10020, 2006).

[7] Z. Bai, Y. Zhao, W. Ma, and H. Tian, Modal analysis for small satellite system with finite element method: *Proc. 2nd International Symposium on Systems and Control in Aerospace and Astronautics (ISSCAA),* 2008, pp.1-5.

[8] H. V. d. Auweraer, Structural Dynamics Modeling using Modal Analysis, Applications, Trends and Challenges: *Proc. of the 18th IEEE Instrumentation and Measurement Technology Conference, Vol. 3*, 2001, pp. 1502 –1509.

[9] Z. X. yang, Y. Ruixia, et al, Structure Optimal Design of Roll Gimbal for an Aerial Three-Axis ISP Based on FEM Modal Analysis: *3rd International Conference on Measuring Technology and Mechatronics Automation, Vol. 3*, 2011, pp. 373 – 376.

[10] Z. Zhu, et al, Integrated ADAMS+MATLAB Environment for Design of an Autonomous Single Wheel Robot: *IEEE 35th Annual Conference of Industrial Electronics*, 2009, pp.2253-2258.