June, 2023 Vol.10 No.26 44-48

Experimental and numerical modal analysis of CFM56-3 Oil tank

Análisis modal experimental y numérico del tanque de aceite CFM56-3

ARREDONDO-VIZCAYA, Marlen Gricel†* & ROJAS-RAMIREZ, Sergio Raúl

Universidad Aeronáutica en Querétaro. Carretera Estatal 200 Querétaro – Tequisquiapan No. 22154 Col Parque Aeroespacial de Querétaro. Colon, Querétaro. México C.P 76278.

ID 1st Author: *Marlen Gricel, Arredondo-Vizcaya* / **ORC ID:** 0009-0006-8025-4351

ID 1st Co-author: *Sergio Raúl, Rojas-Ramirez* / **ORC ID:** 0000-0003-1728-4620

DOI: 10.35429/JQSA.2023.26.10.44.48 Received March 05, 2023; Accepted June 30, 2023

Abstract

To determine the dynamic behavior of components which is performed through the mechanical vibration analysis. The Experimental Modal Analysis (EMA) is the process to determine the modal parameters i.e. of natural frequencies, mode shapes, and damping. The Experimental Modal Analysis (EMA) of the CFM56 oil tank was performed using an impact hammer and a laser vibrometer. Finite Element Modeling (FEM) software, ANSYS APDL 19.2, was used to performed numerical modal analysis of the same oil tank. Thus, the results obtained for natural frequencies and mode shapes is presented in this document. It will contribute to the demonstration of the component functionality and to the validation of the proposed analysis methodology.

Modal Analysis, Oil Tank, CFM56-3

Resumen

Para determinar el comportamiento dinámico de componentes se realiza a través de análisis de vibración mecánica. El análisis modal experimental es el proceso de determinar los parámetros dinámicos por ejemplo las frecuencias naturales, formas modales y amortiguamiento. El análisis modal experimental del tanque de aceite montado en el motor CFM56-3 fue realizado usando un martillo de impacto y un vibrometro de laser. El programa de modelado en elemento finito es ANSYS APDL 19.2 usado para realizar el modelo numérico del mismo tanque de aceite. Los resultados obtenidos de frecuencias naturales y formas modales son presentados en este documento. Esto contribuye a la demostración de la funcionalidad del componente y a la validación de la metodología de análisis propuesta.

Análisis modal, Tanque de aceite, CFM56-3

Citation: ARREDONDO-VIZCAYA, Marlen Gricel & ROJAS-RAMIREZ, Sergio Raúl. Experimental and numerical modal analysis of CFM56-3 Oil tank. Journal of Quantitative and Statistical Analysis. 2023. 10-26: 44-48

* Correspondence to Author (e-mail: Marlen.arredondo@gmail.com)
† Investigador contribuyendo como primer autor.

June, 2023 Vol.10 No.26 44-48

1. Introduction

The CFM56-3 oil tank stores oil used to lubricate and cool the engine rotating components. It consists of a basic envelope and a cover.

The dynamic loads cause responses that may not be acceptable for the intended operation of the structure. When this is the case, the engineer must determine what if, anything, can be done to minimize or eliminate the undesirable response in the structure. Sometimes this can be very difficult if the cause of the unwanted response is unknown (Ewins).

Modal analysis is used to extract the modal parameters such as natural frequency, damping ratio and mode shapes (Hassan). Experimental Modal Analysis (EMA) uses experimental techniques to determine the modal parameters through the measurement of frequency response function (FRFs) on many points of the structure (P. Castellini *, 2003). Finite Element Modeling (FEM) is used for determination of modal parameters based on the material properties, structure dimensions and accurate boundary conditions; numerical results must be validated through the experimental ones. The EMA for the oil tank modal analysis was implemented using a laser vibrometer PDV100 (AB Stanbridge & DJ Ewins, 2005) and an impact hammer PCB086C03.The bandwidth was defined from DC to 800 Hz.

FEM was implemented with the information of Table 1, the material of the oil tank is aluminum alloy and the three-point soft support stiffness is shown. The support stiffness was estimated by a progressive static load test. The comparison of the first 3 frequencies and modal shapes obtained were compared. The results shown that are well within the reasonable error margin and this methodology is valid to characterize aeronautical components.

This paper is organized as follows:

Section 2 Component details which describe the component or object of study. Section 3 Experimental Modal Analysis describes the experimental methodology. Section 4 Finite Element Modeling describes the numerical procedure to characterize the oil tank, the results are provided in Section 5 EMA and FEA comparative results and Section 6 concludes the paper.

2. Component Details

The function of the oil tank is to store oil used for cooling and lubrication of engine rotating components; it has three-point soft mounting.

Oil Tank	
Material	Aluminum Alloy
Properties:	E=72 GPa
	ρ =2650 kg/m^3
	ν =0.3
Three-point soft support	
	Stiffness
Stiffness	33.89 Nm
33.89 Nm	56.49 Nm

Table 1 Oil tank and three-point support mounting details

Oil tank is made of light alloy weldment envelope with a sealed cast light alloy cover. It has an external flame-resistant coating and 5 inner bulkheads to reduce sloshing and strengthen the tank.

Total volume (usable oil plus residual oil plus air) of the oil tank is 5.3 gal (20 L). It has an oil capacity (usable oil plus residual) of 4.8 gal (18 L) (Manual, 1995).

3. Experimental Modal Analysis

The experimental setup was prepared on CFM56 engine, the oil tank is located on the right-hand side of the engine. It is secured to the fan Inlet case, at approx. 4.00 o'clock (Fig. 1 and 2).

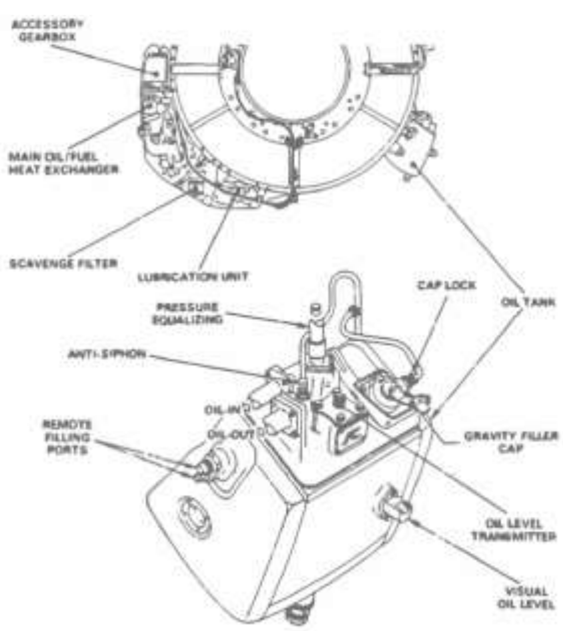


Figure 1 Oil tank location

ISSN 2410-3438

June, 2023 Vol.10 No.26 44-48

Figure 2 Experimental Setup

Modal testing was done using an impact hammer PCB086C03, Sensitivity: (±15%) 2.25 mV/N, with a rubber tip, The mass of the hammer is 0.34 lb.

For the impact and measured points the oil tank front face was divided into 25 grid points in order to achieve adequate spatial resolution of global structural mode shapes (Fig. 3).

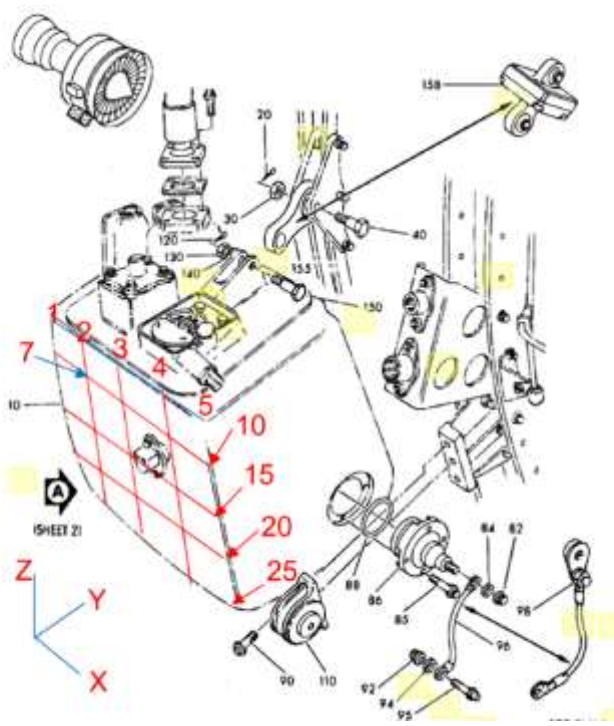


Figure 3 Impact Hammer Locations

The oil tank was excited at different points by using the impact hammer and the ensuing vibration is measured with the laser vibrometer (Polytec laser vibrometer PDV-100).

The laser beam hits on the oil tank (grid point #7). To provide proper reflection, a small, reflecting sticker is attached to the oil tank at the grid point #7. The experimental modal shapes and corresponding natural frequencies of the oil tank are presented in the Table 2.

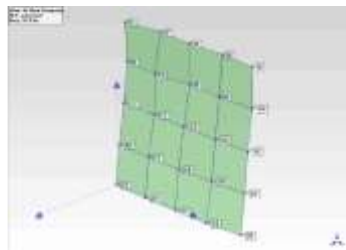
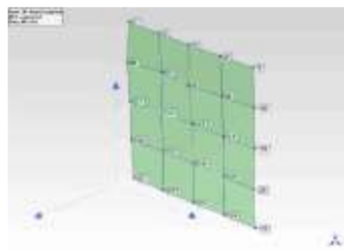
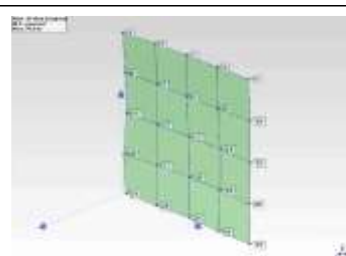
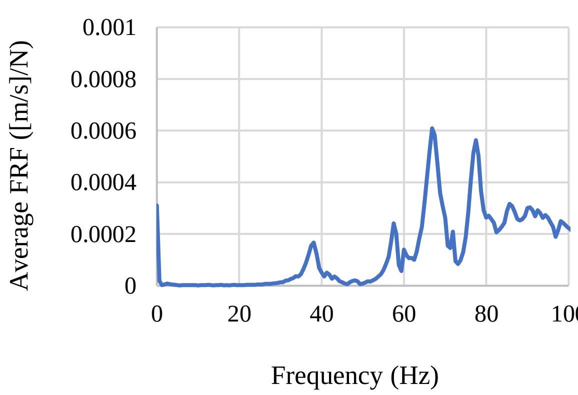
Mode	EMA Frequency (Hz)	EMA Modal Shapes
1	37.5	
2	66.3	
3	77	

Table 2 Experimentally obtained natural frequencies and modes shapes

The computed FRF based for #1-5 grid points measurements points is shown in Graphic 1.



Graphic 1 Computed FRFs

ISSN 2410-3438

The outcome curve is a set of modal parameters, which consist of natural frequencies for each identified modes within the range of frequency of interest, except the mode at 60 Hz which was an electric noise.

4. Finite Element Modeling

The oil tank, which is considered an important part of the CFM56-3 engine, can be considered as a complex structure, modelling the structure accordingly to the actual structure might be difficult, the finite element model can have very different levels of complexity.

A simplified geometry was modeled as a representation of the oil tank. The main characteristics to keep in the model are the total mass and stiffness in order to get the rigid body mode shapes related to the three-point mounting supports.

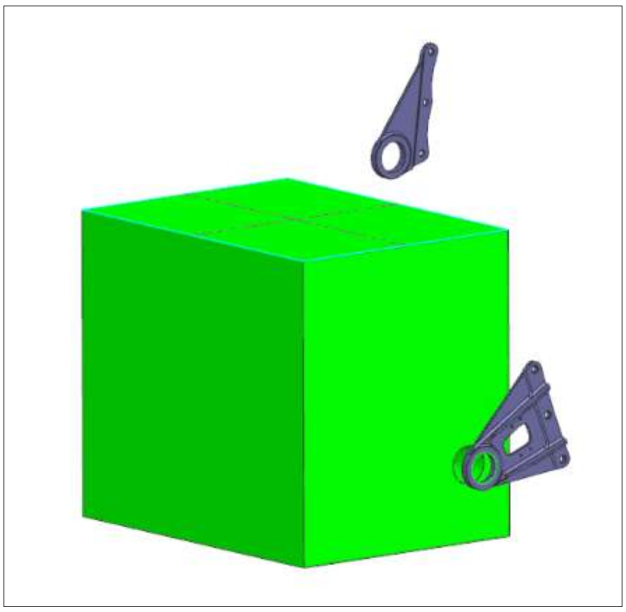


Figure 4 CAD Model

The finite element model of the oil tank was created by using 154660 elements of tetrahedral-8 shaped elements. The oil tank was assigned with material properties of aluminum and geometric parameters, which details are shown in Table 1.

For the three-point soft mounting, three torsional and rotational elements COMBIN14 were created, 2 of them with the same properties and the one at the top has an elastomer which the stiffness was obtained by a progressive static load test which details are shown in Table 1.

The corresponding numerically obtained modes shapes for different conditions of the oil tank are shown in Table 3.

In total 13 natural frequencies and corresponding mode shapes were calculated from 0-1000 Hz. The first 3 are shown in Table 3.

5. EMA and FEM Comparative Results

Correlation of data that was obtained through finite element analysis and modal testing to analyze the discrepancies existed between those sets of data.

The results shown in Table 3 compares the natural frequencies of EMA and numerical analysis.

The preliminary results obtained shows that the experimental and numerical results for the oil tank have a good correlation.

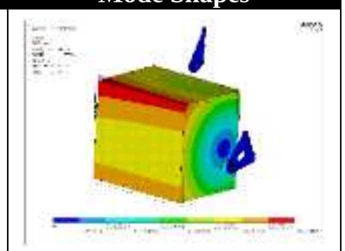
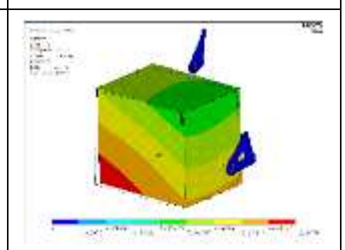
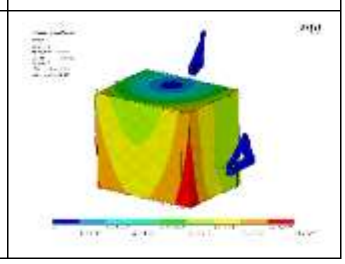
Mode Shape	EMA (Hz)	FEM (Hz)	FEM Mode Shapes
1	37.5	35.2	
2	66.3	58.4	
3	77	69.6	

Table 3 Comparison between natural frequencies and mode shapes obtained from FE models and EMA

The average percentage of error for the oil tank is about 9.8%.

It appears that it is very difficult to discard the errors and inaccuracies in finite element model when the structure in study is a complex structure.

There are few errors resulting from the assumptions made to characterize the mechanical behavior of the physical structure typically from geometrical simplifications, inaccurate assignments of mass, boundary conditions, incorrect joint stiffness (Sharma, 2018).

However, to reduce the existing discrepancies, model updating procedure can be applied on the finite element model of the oil tank, thus improving the model to have better correlation with the experimental results.

Several updating parameters (Young's modulus and all the thicknesses used in finite element analysis) maybe be considered to be included in model updating procedure. Besides that, during performing modal analysis using impact hammer test and run analysis in ME'Scope, the electrical noise may affect the results.

6. Conclusions

In this work, experimental and numerical modal analysis of the oil tank has been performed.

A finite element model of the oil tank was produced and the percentage of errors between those two sets of data was obtained. Defined that the outcome obtained from the numerical analysis is close to the modal testing and because of some existing error, will affect the results.

The average differences between finite element analysis and experimental analysis are within 10%.

Discrepancies of percentage error of obtained natural frequency for FEM and EMA makes both of the methods can be applied to determine the dynamic characteristic of the Oil tank structure.

It is recommended that further research be undertaken by performing modal updating in order to diminish the percentage of error.

Acknowledgement

We thank the Universidad Aeronautica en Queretaro UNAQ to provide the equipment and CFM56-3 engine.

Funding

This work has been funded by UNAQ.

References

AB Stanbridge & DJ Ewins. (2005). Modal testing using a scanning laser doppler vibrometer. *Journal of sound and vibration 281*, 439-451. doi:https://doi.org/10.1006/mssp.1998.1209

Ewins, D. (n.d.). *Modal Testing Theory, Practice and Application, @nd edn. Wiley.* . ISBN 978-0-86380-218-8.

Hassan, M. R. (n.d.). A revew of experimental modal analysis and finite element analysis. doi:https://doi.org/10.1177/1687814016649110

Manual, C.-3. T. (1995). *Academia.edu*. Retrieved from https://www.academia.edu/8851546/cfm_international_CFM56_3_TRAINING_MANUAL_EFFECTIVITY_CFMI_PROPRIETARY_INFORMATION

P. Castellini *, G. R. (2003). Measurement of Vibrational modal parameters using laser pulse excitation techniques. doi:10.1016/j.measurement.2003.07.004

Sharma, J. K. (2018). Theoretical and Experimental Modal Analysis of Beam. doi:http://dx.doi.org/10.1007/978-981-13-1642-5_16