Paper ID: 46420
interactive

IAF MATERIALS AND STRUCTURES SYMPOSIUM (C2)
Interactive Presentations - IAF MATERIALS AND STRUCTURES SYMPOSIUM (IP)

Author: Mr. Benjamín Pinazo-Herrero
University of Valencia, Spain

Mr. Pedro Rodríguez-Martínez
University of Valencia, Spain
Mr. José Luis Gasent-Blesa
University of Valencia, Spain

SPACE SYSTEMS STRUCTURAL ANALYSES FROM MODAL PARAMETERS USING A PYTHON DEVELOPED TOOLSET, AND ADDITIONAL PRE/POST-PROCESSING FEATURES

Abstract

Frequency Response and Random analyses are mandatory on space projects, but usually commercial FEM software is general-purpose and only provides raw results; hence these outputs have to be handled further with customized programs to obtain the final ad-hoc reports. Input profiles are frequently updated during a space project development (e.g. in a notching discussion), and it would be very useful to have an interactive tool available. Moreover, It is not easy to deal with FEM software, and both in-depth knowledge and high skills are required to get the desirable results.

This paper presents a Python-based toolset developed to post-process modal parameters of proportionally damped structures, coming from a MSC Nastran modal analysis, optimizing the use of a shared license. This toolset includes sine sweep and random analyses of hard-mounted structural components under its base excitation. It also provides further pre and post-processing features.

The code, based on standard free Python packages, is organised in functions bundled into a library. Each task is tailored and executed in a Jupyter Notebook App (i.e. an open-source web application). MSC Nastran solves and stores the whole set of intermediate results for all the analysed nodes or elements (items), in each internal calculation step. The developed toolset improves this calculation procedure; modal analysis parameters are taken from the MSC Nastran output file to compute the required dynamic analysis results, separately for each item.

This approach involves significant advantages compared to the Nastran standard full-run analysis, namely: (1) handling of sine-sweep and PSD input profiles under a user-friendly environment; (2) more efficient use of proprietary software license; (3) optimization of both data storage and RAM usage; (4) to take advantage of multiprocessing technique when needed; (5) interactive fast representation of graphical results; (6) customizable results (graphs and tables) with an automated process to obtain prompt reports; (7) customizable notebook to include new analyses/tools.

In addition, with this toolset not only FEA software experts, but also any qualified person would be able to make post-processing tasks for dynamic analyses and reports.

This work also highlights the benefits of the toolset by showing an application for the structural analysis of a scientific payload system of an ESA mission.