

Hybrid Finite Element Analysis of a Rotorcraft

Nickolas Vlahopoulos
University of Michigan

Sungmin Lee
Michigan Engineering Services

Paul Braunwart and Jeff Mendoza
United Technologies Research Center

Donald Butts
Sikorsky Aircraft Corporation

ABSTRACT

The Hybrid FEA method is based on combining conventional Finite Element Analysis (FEA) with Energy Finite Element Analysis (EFEA) for mid-frequency computations. The difficulty in using conventional FEA at higher frequencies originates from requiring a very large number of elements in order to capture the flexible wavelength of the panel members which are present in a structure. In the Hybrid FEA the conventional FEA model is modified by de-activating the bending behavior of the flexible panels in the FEA computations and introducing instead a large number of dynamic impedance elements for representing the omitted bending behavior. The excitation is considered to be applied on the conventional FEA model and the vibration analysis is conducted. The power flow through the dynamic impedance elements is computed and applied as excitation to the EFEA model of the flexible panels. The EFEA analysis computes the vibration of the flexible panels. Gearbox excitation is an important contributor to the vibration induced in the fuselage of a helicopter. The frequency of excitation is typically in the 500Hz - 3,000Hz range. The excitation is applied at the highly rigid foundation structure and transmitted to the flexible panels of the fuselage through the frame structure. In order to demonstrate the validity of the Hybrid FEA and its utility for analyzing rotorcraft structures, a production rotorcraft was analyzed using the Hybrid FEA method. Excitation was applied separately through a shaker at two gearbox mount locations in the 250Hz - 3,000Hz frequency range. For each excitation the vibration at approximately thirty frame locations and at approximately ten panels was measured and compared successfully to the simulation results.

CITATION: Vlahopoulos, N., Lee, S., Braunwart, P., Mendoza, J. et al., "Hybrid Finite Element Analysis of a Rotorcraft," *SAE Int. J. Aerosp.* 6(1):2013, doi:10.4271/2013-01-1995.

INTRODUCTION

In rotorcraft applications computing the vibration and the noise during the design process is important for configuring the frame/panels/acoustic treatment arrangement for reducing the interior noise level with as low as possible increase in the weight. One of the most critical excitations for interior noise in helicopters is the one from the gearbox. Excitation from the gearbox originates from the meshing action of the gear teeth.

The gear tooth deflections caused by the transmitted loads along with variations in tooth profile, excites vibrations in the helicopter gearbox. Each gear mesh produces noise

components at frequencies corresponding to the tooth mesh rates and their higher harmonics. This gear induced vibration is transmitted to the gearbox casing and to the aircraft structure [1,2,3]. Computations from gearbox excitation are difficult to conduct due to the frequency range of the excitation (500Hz - 3,000Hz) and the nature of the power transfer mechanism to the helicopter cabin. The excitation from the gearbox is transmitted through the gearbox feet into the rigid fuselage frame and flexible skin. The vibration of the skin panels creates a substantial amount of noise which is transmitted through the acoustic treatment into the acoustic space inside the cabin. Due to the rigidity of the gearbox foundation and airframe, the global dynamic behavior of the

structure can be characterized by the transfer path of the gearbox excitation to the flexible skin panels. The global dynamic behavior of the helicopter's structural frame cannot be captured by an energy based method (i.e. EFEA). The EFEA is a finite element based method that utilizes a finite element model similar to those used for preliminary static structural design for conducting structural acoustic simulations at high frequencies. At the same time, the panels of the fuselage structure exhibit low enough flexibility that the conventional FEA requires a prohibiting large number of elements for capturing accurately the flexural wavelength within the panels above ~ 300 Hz. In rotorcraft applications the Hybrid FEA method [4,5,6,7,8,9,10,11] uses conventional FEA for modeling the load bearing parts of the rotorcraft system (i.e. gearbox foundation, frames, etc.) and the EFEA for modeling the flexible behavior of the panels. The entire Hybrid FEA analysis utilizes finite element models.

In this paper the technical background of the Hybrid FEA is reviewed. The functionality of the computational process which is used for conducting the rotorcraft analysis is presented. The development of the Hybrid FEA model for the production helicopter and the process used for the physical testing are discussed. Finally, the comparison between the test results and the numerical simulations is presented. The new elements presented in this paper are associated with conducting a finite element based vibration analysis for a complete production helicopter in the frequency range of 250Hz - 3,000Hz and comparing successfully simulation results to test data.

TECHNICAL BACKGROUND ON THE HYBRID FEA

The difficulty in using conventional FEA at higher frequencies originates from requiring a very large number of elements in order to capture the flexible wavelength of the skin panels which are present in a structure. In the Hybrid FEA the conventional FEA model is modified by deactivating the bending behavior of the panels in the FEA computations and introducing instead a large number of dynamic impedance elements for representing the omitted bending behavior. The excitation is considered to be applied on the conventional FEA model (i.e. on the load bearing parts of the structure) and the vibration analysis is conducted. The power flow through the dynamic impedance elements is computed and applied as excitation to the EFEA model of the flexible panels. The EFEA analysis computes the vibration of the flexible panels.

The values which are assigned to the dynamic impedances that represent the flexible behavior of the panels in the FEA part of the Hybrid FEA analysis are critical for the accuracy of the results [10]. In the Hybrid FEA method a Component Mode Synthesis (CMS) approach is applied to an analytical modal basis of each panel for deriving the appropriate conductance values. Technical details about the derivation of the dynamic impedances can be found in [7, 11], and an

overview is presented here. In order to compute the driving point impedance (and therefore the conductance) for the flexible members either an analytical expression for an infinite or semi-infinite member can be employed or the flexible panel can be approximated as a rectangular plate and then the driving point impedance at the boundaries of the flexible panel can be derived. The latter approach is employed in the Hybrid FEA. According to the CMS method the bending Degrees Of Freedom (DOF) of the rectangular plate are partitioned into the boundary and the interior DOF. Therefore, the displacement of any point P of the plate is considered as a superposition of two components, one induced by the movement of the boundary DOF and the other from the relative motion of the interior DOF with respect to the boundary DOF. The constraint normal modes are employed for defining the former and the internal normal modes are employed for determining the latter. The displacement at point P can be written as:

$$w(x, y, t) = \sum_{m,n} Y_{mn}(t) \psi_{mn}(x, y) + Y_s(t) \psi_s(x, y) \quad (1)$$

where $\psi_{mn}(x, y)$ is the internal normal modes, $Y_{mn}(t)$ is the corresponding modal participation factors, $\psi_s(x, y)$ is the constraint normal mode derived by considering a unit displacement at the retained DOF where the conductance is computed while all the remaining retained DOF are fixed, and $Y_s(t)$ is the corresponding modal participation factor. Introducing the displacement expression from equation (1) into the governing differential equation for the bending vibration of a plate results in:

$$D \left(\sum_{m,n} Y_{mn}(t) \nabla^4 \psi_{mn}(x, y) + Y_s(t) \nabla^4 \psi_s(x, y) \right) + \rho_s \left(\sum_{m,n} \ddot{Y}_{mn}(t) \psi_{mn}(x, y) + \ddot{Y}_s(t) \psi_s(x, y) \right) = f(x, y, t) \quad (2)$$

where D = the flexural rigidity, ρ_s = the mass density per area, $f(x, y, t)$ = the external force, and $\nabla^4 = \nabla^2 \nabla^2$, where ∇^2 is the Laplacian operator. The modulus of elasticity in the flexural rigidity is defined as a complex number in order to reflect the effect of the structural damping of the flexible panels in the computations. Introducing the analytical expressions for the normal modes and considering harmonic time dependency in equation (2) results in:

$$D \sum_{m,n} \left[\left(\frac{m\pi}{L_x} \right)^2 + \left(\frac{n\pi}{L_y} \right)^2 \right]^2 Y_{mn} \psi_{mn} - \rho_s \omega^2 \sum_{m,n} Y_{mn} \psi_{mn} + D Y_s \nabla^4 \psi_s - \rho_s \omega^2 Y_s \psi_s = f \quad (3)$$

where L_x and L_y are the lengths of the rectangular plate in x and y directions, respectively. Multiplying equation (3) with

an internal normal mode, ψ_{kl} , and integrating over the domain A , results in:

$$-\omega^2 \left(Y_s \int_A \psi_{kl} \psi_s dA + \sum_{m,n} Y_{mn} \int_A \psi_{mn} \psi_{kl} dA \right) + \frac{D}{\rho_s} Y_s \int_A \psi_{kl} \nabla^4 \psi_s dA + \sum_{m,n} \omega_{mn}^2 Y_{mn} \int_A \psi_{mn} \psi_{kl} dA = \frac{1}{\rho_s} \int_A f \cdot \psi_{kl} dA \quad (4)$$

The second and fourth terms of [equation \(4\)](#) are nonzero only when $m = k$ and $n = l$ because of the orthogonality of the internal normal modes. The third term in [Equation \(4\)](#) is set equal to zero according to the CMS method [12]. Since the objective of this derivation is to evaluate the conductance at a single point on the boundary of the plate, the external force $f(x, y, t)$ is considered to be applied on one of the retained DOF. Thus, the right hand side of [Equation \(4\)](#) is equal to zero since the external force is always applied on a retained DOF where the displacement of any internal normal mode, ψ_{kl} , is always equal to zero. [Equation \(4\)](#) is solved for the modal participation factors of the internal and of the constraint modes. The equation for the admittance is [11]:

$$Y = i\omega \left[-\frac{4\rho_s \omega^4}{L_x L_y} \sum_{m,n} \left(\frac{\int_A \psi_s \psi_{mn} dA}{\omega_{mn}^2 - \omega^2} \right) \int_A \psi_{mn} \psi_s dA + D \int_A \psi_s \nabla^4 \psi_s dA - \rho_s \omega^2 \int_A \psi_s \psi_s dA \right]^{-1} \cdot \int_A f \cdot \psi_s dA \quad (5)$$

The conductance (real part of the admittance) at the location where the force is applied is evaluated and it corresponds to the inverse of the damping coefficient which identifies the amount of power absorbed from the external excitation due to the vibration of the plate. The concentrated damping elements are applied in the conventional FEA model of all locations where the FEA model is connected to panels represented by the EFEA. The properties of the panel (including damping), its dimensions, and the distance between the connections influence the dynamic impedance values.

FUNCTIONALITY OF COMPUTATIONAL HYBRID FEA PROCESS

[Figure 1](#) presents a summary for the Hybrid FEA process. There are two main codes (FEA2HYB and HYB2EFEA) that enable the flow of information between the FEA and the EFEA solvers.

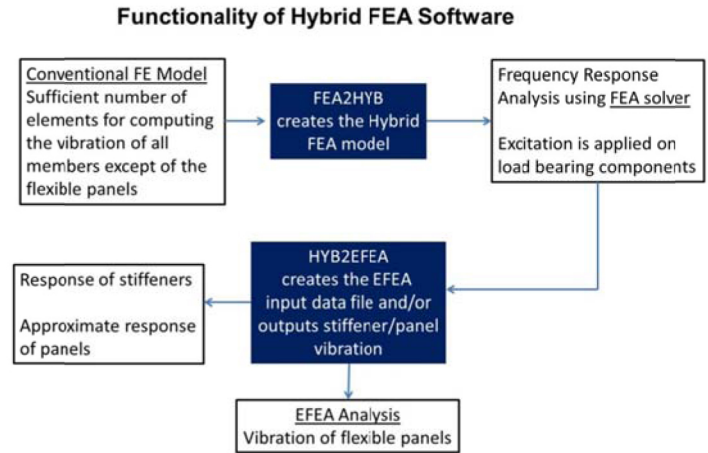


Figure 1. Hybrid FEA computational process

The FEA2HYB program generates an input file for the FEA part of the Hybrid FEA analysis from an FEA model, by representing flexible panels as dynamic impedances. Since damping elements (dynamic impedances) replace the bending behavior of the flexible panels, the corresponding bending stiffness is removed as a result. It should be noted that the finite elements representing the remaining behavior of the structure are not modified. Hence, it is required that the finite element model has a sufficient number of elements per wavelength for capturing the remaining behavior of the structure at the highest frequency analysis of interest. Based on the FEA model and the definition of the Property IDs (PIDs) for the panels that will have their bending behavior modeled by the EFEA, the locations for the joints between the FEA elements and the EFEA elements is automatically identified, the appropriate values of the dynamic impedances evaluated (based on the equations discussed in the previous Section), and the FEA model modified and prepared for the FEA part of the Hybrid FEA analysis.

Once the new modified finite element data file is created by the FEA2HYB code a frequency response analysis using a FEA software is conducted. The HYB2EFEA program is used to: either output vibration response at stiffeners and to provide an estimate for the vibration of the panels; or to generate an EFEA input data file for computing the vibration of flexible panels using the EFEA method. If stiffener and panel vibration responses are requested, the vibration of stiffeners is computed directly from the FEA analysis results and the vibration of the panels is estimated based on the power dissipated at the damping elements representing each panel in the FEA part of the Hybrid FEA analysis. If an EFEA input data file is requested, then the power dissipated by the group of dynamic impedances that represent each panel in the FEA part of the Hybrid FEA, comprises the excitation input power into each panel of the EFEA model. For the latter mode of operation the PID numbers representing each panel must be the same between the FEA and the EFEA models. The HYB2EFEA code utilizes the