An Iterative Eigenvalue Solver for Systems with Frequency Dependent Material Properties

Martin Gröhlich, Marc Böswald, René Winter
German Aerospace Center (DLR), Institute of Aeroelasticity, Department for Structural Dynamics and System Identification
37073 Göttingen, Germany, Email: martin.groehlich@dlr.de

Introduction

An essential requirement of aeronautic structures is their lightweight design. Such characteristic yields a considerable susceptibility to vibrations which contribute to an increased cabin-interior noise. In order to reduce structure-borne noise, viscoelastic materials can be integrated as damping layers in the fuselage structure. A special property of viscoelastic materials is the frequency dependence of their material parameters. At different excitation frequencies, both the stiffness and the damping capability of the material can vary significantly. Therefore, to evaluate the influence of local viscoelastic elements on a global structure regarding modal parameters such as eigenfrequencies and damping ratios, it is necessary to solve an eigenvalue problem with frequency dependent stiffness and damping matrices. However, a solver for such an eigenvalue problem is not available in current finite element software.

At first, an overview of the theoretical background is given. The principle of an iterative eigenvalue solver is introduced and demonstrated on a simple spring-mass system. In addition, an approach is provided to use the iterative eigenvalue solver in conjunction with MSC Nastran. Using a finite element model, the modal parameters of the iterative eigenvalue solver are finally compared with parameters identified from a frequency response analysis.

Theoretical background

In case of forced harmonic excitation, any material exhibiting both elastic and viscous properties can be described by a complex Young's modulus E^* in the form

$$E^* = E' + iE'' = E'(1 + i\eta) \tag{1}$$

where i is the imaginary unit, E' the storage modulus, E'' the loss modulus and η the material loss factor [1]. Under the assumption of isotropic behavior, the one-dimensional material law can be incorporated into a spatial model by means of a complex elasticity matrix $\mathbf{E}^*$

$$\mathbf{E}^* = E^*\,\mathbf{A} = E'\,\mathbf{A} + iE''\,\mathbf{A} = \mathbf{E}' + i\mathbf{E}''\,. \tag{2}$$

In this case, $\mathbf{A}$ corresponds to the classical elasticity matrix of isotropic solid material with the complex Young's modulus E^* factored out [2]. From eq. 2 it is evident that the real and imaginary part of the elasticity matrix are proportionally dependent on the corresponding part of the complex Young's modulus. Using the components of the complex elasticity matrix, the element stiffness $\mathbf{K}_E$ and element hysteretic damping matrices $\mathbf{D}_E$ can be assembled by solving the integral function over an element volume V_E

$$\mathbf{K}_E = \int\limits_{V_E} \left(\boldsymbol{\Theta}\boldsymbol{\varphi}^T\right)^T \mathbf{E}'\,\boldsymbol{\Theta}\boldsymbol{\varphi}^T dV_E\,, \tag{3}$$

$$i\,\mathbf{D}_E = i\int\limits_{V_E} \left(\boldsymbol{\Theta}\boldsymbol{\varphi}^T\right)^T \mathbf{E}''\,\boldsymbol{\Theta}\boldsymbol{\varphi}^T dV_E\,. \tag{4}$$

In those equations, $\boldsymbol{\varphi}$ indicates the shape function matrix and $\boldsymbol{\Theta}$ the differential operator matrix. The element matrices are sorted into the corresponding global stiffness $\mathbf{K}$ and hysteretic damping matrix $\mathbf{D}$, by multiplication with Boolean matrices $\mathbf{B}_E$ [2]

$$\mathbf{K} = \sum_E \mathbf{B}_E^T\,\mathbf{K}_E\,\mathbf{B}_E; \quad i\,\mathbf{D} = i\sum_E \mathbf{B}_E^T\,\mathbf{D}_E\,\mathbf{B}_E\,. \tag{5}$$

In general, a freely vibrating multiple degree-of-freedom (MDOF) system with hysteretic damping can be described by a complex equation of motion which consists of a mass matrix $\mathbf{M}$, a stiffness matrix $\mathbf{K}$ and a hysteretic damping matrix $\mathbf{D}$ [3]

$$\mathbf{M}\ddot{\mathbf{x}} + [\mathbf{K} + i\mathbf{D}]\,\mathbf{x} = \mathbf{0}\,. \tag{6}$$

The displacement is indicated by the vector $\mathbf{x}$. Assuming a solution of the form

$$\mathbf{x} = \hat{\mathbf{x}}e^{i\lambda t} \tag{7}$$

with t as the time, the r$^{\text{th}}$ eigenvalue λ_r contains the natural frequency ω_r in the real part and the modal loss factor η_r as the ratio of imaginary and real part [3]

$$\lambda_r^2 = \omega_r^2(1 + i\eta_r)\,. \tag{8}$$

For such systems the eigenvalue can be calculated directly from the constant system matrices. However, if a viscoelastic material is applied to the system, the stiffness matrix as well as the hysteretic damping matrix is not constant anymore. Instead, both matrices are dependent on the vibration frequency ω: $\mathbf{K} = \mathbf{K}(\omega), \mathbf{D} = \mathbf{D}(\omega)$. This yields the following equation of motion

$$\mathbf{M}\ddot{\mathbf{x}} + [\mathbf{K}(\omega) + i\mathbf{D}(\omega)]\,\mathbf{x} = \mathbf{0}\,. \tag{9}$$

As a consequence, the eigenvalues cannot be calculated directly anymore, since they also depend on the frequency. However, the corresponding eigenvalue problem can be defined using eq. 7 and 8

$$det\left[-\omega_r^2(1 + i\eta_r)\mathbf{M} + \mathbf{K}(\omega) + i\mathbf{D}(\omega)\right] = \mathbf{0}\,. \tag{10}$$

It is assumed that the vibration frequency on which the stiffness and damping matrix depend has to be a natural frequency of the system

$$\delta = \omega - \omega_r = 0\,, \tag{11}$$

where δ is the absolute error. It serves as a control parameter to check for convergence of the trial frequency ω towards the eigenfrequencies ω_r within the iterative loops of an iterative eigenvalue solver (IES) to be explained in the next section.

Structure of the iterative eigenvalue solver

Different approaches for determining modal parameters of structures containing viscoelastic material have been discussed in [4]. In contrast to the proposed algorithms, the IES presented in this paper includes an optimization which is based on eq 11. In order to find an eigenvalue, the difference between the „input“ and „output“ frequency has to be zero. It can be understood as an optimization in which the objective is to find the smallest error. At this point it should be mentioned that the IES cannot find all eigenvalues of a system at once, but one by one. To illustrate the function of the IES, the basic algorithm is described in Figure 1.

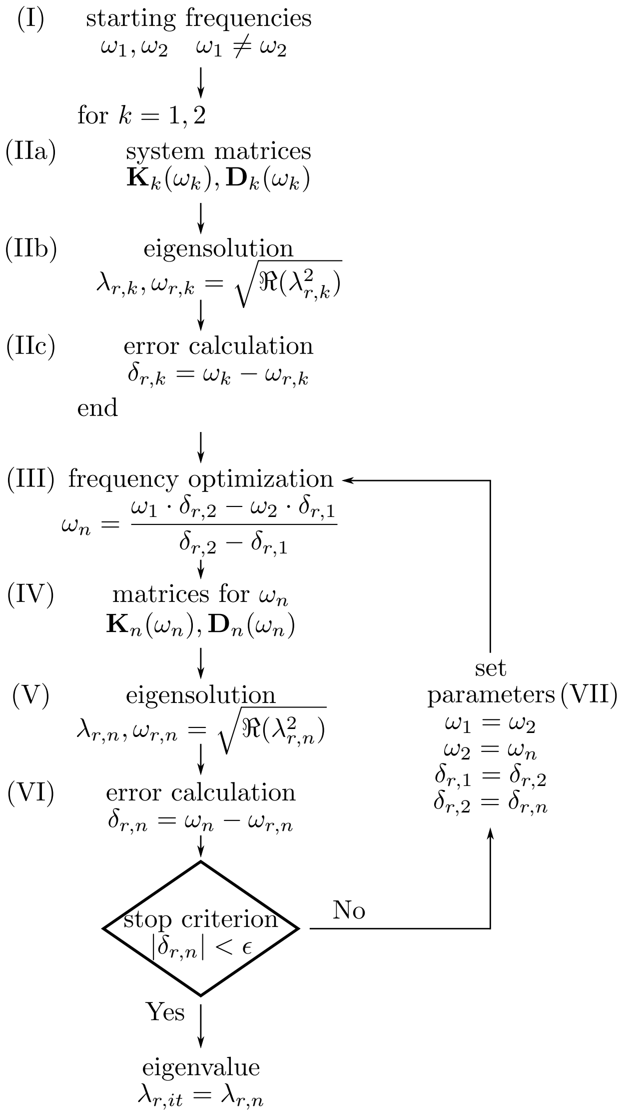

Figure 1: Algorithm of the iterative eigenvalue solver

At first, two different frequencies ω_1 and ω_2 have to be chosen as an input (I) and are referred to a loop. Inside the loop, the stiffness and damping matrices are generated (IIa) and the corresponding eigenvalue problem is solved for each of the chosen frequencies (IIb). The natural frequency ω_r is calculated from the resulting eigenvalue λ_r. After this, the absolute deviation δ_k between the „input“ frequency ω_k and the calculated natural frequency ω_r is determined. The secant method is used to compute a new improved frequency ω_n (III) which is closer to the sought natural frequency [5]. By using the improved frequency ω_n, the corresponding system matrices are generated (IV) and the eigenvalue problem is solved again (V). After performing the error calculation (VI), the algorithm checks, whether the calculated error $\delta_{r,n}$ is lower than a predefined threshold value ϵ. If not, the parameters have to be set according to (VII) and the algorithm restarts with the optimization step (III). If the stop criterion is fulfilled, the eigenvalue is finally found.

Example for a SDOF system

The capability of the IES is demonstrated by means of a single degree-of-freedom (SDOF) system. The system is shown in Figure 2 and consists of a constant mass m and a complex frequency dependent spring k^*

$$k^*(\omega) = k(\omega)\,(1 + i\eta(\omega))\;. \tag{12}$$

Both the stiffness k and the material loss factor η shall be linearly dependent on the frequency

$$k(\omega) = k_1 + k_2\omega\,, \tag{13}$$

$$\eta(\omega) = \eta_1 + \eta_2\omega\,. \tag{14}$$

In eq. 13 and 14, k_1 and k_2 are stiffness parameters and η_1 and η_2 are damping parameters. Although the system contains a frequency dependence, the eigenvalue problem can be solved analytically. By using the approaches of

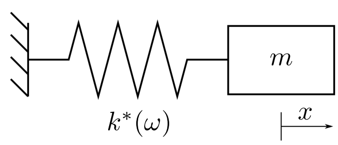

Figure 2: SDOF system with complex stiffness

eq. 7 and eq. 8, the eigenvalue problem is defined as

$$-\omega_r^2(1 + i\eta_r)m + k(\omega)\,(1 + i\eta(\omega)) = 0\,. \tag{15}$$

Applying eq. 13 and eq. 14 into eq. 15 yields

$$\begin{aligned} &-\omega_r^2(1 + i\eta_r)m + k_1 + k_2\omega \\ &+ i\left[k_1\eta_1 + k_1\eta_2\omega + k_2\eta_1\omega + k_2\eta_2\omega^2\right] = 0\,. \end{aligned} \tag{16}$$

In order to solve eq. 16, both the real and the imaginary part of the equation needs to become zero. As a consequence, eq. 16 can be divided up into two equations.

The real part results in

$$-\omega_r^2 m + k_1 + k_2\omega = 0\,, \tag{17}$$

while the imaginary part is

$$-\omega_r^2 \eta_r m + k_1\eta_1 + (k_1\eta_2 + k_2\eta_1)\omega + k_2\eta_2\omega^2 = 0\,. \tag{18}$$

From eq. 17, the natural frequency can be determined, if $\omega = \omega_r$

$$\omega_r = \frac{k_2 \pm \sqrt{k_2^2 + 4k_1 m}}{2m}\,. \tag{19}$$

Eq. 19 has two solutions for the natural frequency. Since all parameters can be assumed to be positive rationale numbers, eq. 19 always results in a positive and negative frequency. If the negative frequency is neglected, eq. 19 can be inserted into eq. 18 and solved for the corresponding modal loss factor

$$\eta_r = \frac{k_1\eta_1 + (k_1\eta_2 + k_2\eta_1)\omega_r + k_2\eta_2\omega_r^2}{m\omega_r^2}\,. \tag{20}$$

In the following, the system is evaluated for discrete virtual values, where the mass is $m = 1\,kg$, $k_1 = 1000\,Pa$, $k_2 = 10\,Pa\,s$, $\eta_1 = 0,02$ and $\eta_2 = 0,0005\,s$. As starting frequencies $\omega_1 = 40\pi\,s^{-1}$ and $\omega_2 = 80\pi\,s^{-1}$ are chosen for the IES algorithm. The threshold value is set to $\epsilon = 10^{-5}$. The results are illustrated in Tab. 1.

Table 1: Results of the SDOF system

Iteration step n	$\lvert\delta_{r,n}\rvert$ $[s^{-1}]$	ω_r $[s^{-1}]$	η_r $[-]$
1	0,3323	37,3400	0,0391
2	0,0115	37,0269	0,0385
3	$7,81 \cdot 10^{-6}$	37,0156	0,0385
analytical	-	37,0156	0,0385

It is evident that the iterated solution is already very close to the converged one after one iteration step, although the initial frequencies differ significantly from the resulting frequency. Since the stop criterion is reached, the algorithms stops after three iterations. The natural frequencies and the modal loss factors of the IES and the analytical solution coincide exactly up to the fourth digit after the decimal point.

Application with FEM software

The following interaction can be carried out with all finite element method (FEM) software which allow for the extraction of the system matrices. The principle is based on the fundamental equations of FEM, introduced in the first section. The Boolean operation from eq. 5 avoids retracing the contribution of each element matrix to the global system matrices. However, the factorized impact of a frequency dependent material (subscript $_f$) can be retrieved in the system matrices. This requires a manipulation of the corresponding material properties. Instead of using the real material values for the storage and loss modulus, both properties have to be set to a template value $E'_{f,temp} = E''_{f,temp} = 1$. At the same time, the damping contribution of other materials j of the studied system has to be neglected ($E''_j = 0$). This procedure ensures that the entries of the generated „template“ hysteretic damping matrix $\mathbf{D}_{temp}$ are factorized and can be referred to the frequency dependent material. Additionally, the contributions of the frequency dependent material are also known for the „template“ stiffness matrix $\mathbf{K}_{temp}$. By taking advantage of the proportional dependence of the complex Young's modulus, the real stiffness and hysteretic damping matrices can be determined at every frequency

$$\mathbf{D}(\omega) = E''_f(\omega)\,\mathbf{D}_{temp}\,, \tag{21}$$

$$\mathbf{K}(\omega) = \mathbf{K}_{temp} + \big(E'_f(\omega) - 1\big)\,\mathbf{D}_{temp}\,. \tag{22}$$

This means that the calculation of a FEM software has to be performed only once in order to get access to the system matrices. At the same time it does not matter by which type of elements the system was built. Afterwards, the system matrices can be manipulated in the proposed manner.

Numerical example using MSC Nastran

For demonstration purposes, a sandwich structure illustrated in Fig. 3 is considered. The cantilever beam has a length of $1000\,mm$ and consists of a viscoelastic core layer (B) that is constrained between two aluminum layers (A and C). The corresponding material and geometrical properties are listed in Tab. 2, the properties for the viscoelastic material are presented in Fig. 4.

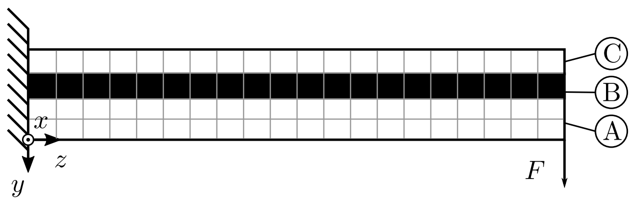

Figure 3: Schematic setup of the sandwich cantilever

Table 2: Material properties of the sandwich layers

layer	$\rho\,[kg/m^3]$	$E\,[Pa]$	$\nu\,[-]$	$h\,[mm]$
A	2700	$7,1 \cdot 10^{10}$	0,33	10
B	1250	-	0,499	5
C	2700	$7,1 \cdot 10^{10}$	0,33	5

In Tab. 2, ν is the Poisson's ratio, h the height of the layer and ρ the density. MSC Nastran 2018.2 CQUAD4 shell elements are chosen for the discretization. The membrane properties of the elements are used for representing the plain stress in the y-z plane. In the first instance, the material values for the storage and loss modulus are set as described in the previous section and the system matrices are extracted to Matlab R2018b. The resulting distributions of the matrix entries are presented in Fig. 5. While the stiffness matrix exhibits entries on the main diagonal and two off-diagonal bands, the hysteretic

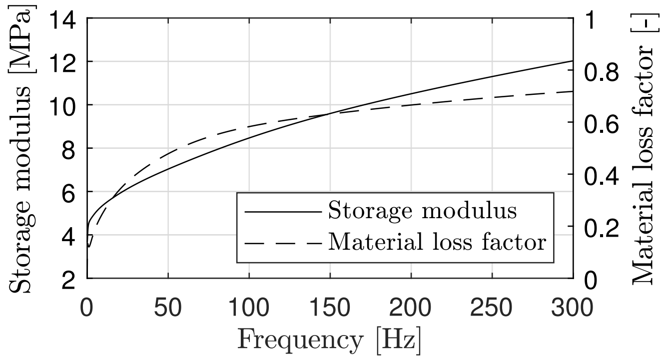

Figure 4: Storage modulus and material loss factor of the viscoelastic material

damping matrix is rather sparse. Only the viscoelastic material contributes to the content of the matrix. At the same time, the entries in the damping matrix indicate the areas of the stiffness matrix, where contributions from the aluminum have to be superimposed by the contribution of the viscoelastic material. The matrices with the real material properties from Fig. 4 can be obtained for a desired frequency by using eq. 21 and 22. In the first step, the IES is applied to the resulting system matrices in order to determine the modal parameters of the first four bending around the x-axis.

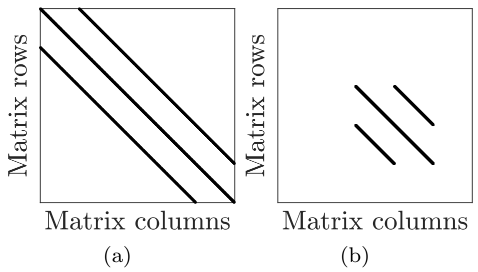

Figure 5: (a) Sparsity pattern of stiffness and (b) hysteretic damping matrix built up with CQUAD4 elements

In a second step, a frequency response analysis is performed. For this, the beam is subjected to a harmonic force F in y-direction (force amplitude $= 1\,N$), as visualized in Fig. 3. In order to validate the IES, the modal parameters are identified from the simulated frequency response functions by means of experimental modal analysis. The Least-Squares Complex Frequency domain method (LSCF) is used for this purpose [6]. Tab. 3 shows the deviation Δ of the identified eigenfrequencies f and damping ratios with those obtained from the IES. It should be noted that the following relation exists among the damping ratio and the loss factor: $2D_r = \eta_r$.

Considering the values of the eigenfrequencies, the IES results are almost exactly in accordance with the results of the LSCF method. Larger deviations can be observed however for the damping ratios. Furthermore it stands out that all values which are identified with the LSCF method are larger than the corresponding values of the IES. A reason for the deviation might be found in the linear system approach of the LSCF method that cannot represent frequency dependence properly.

Table 3: Comparison of IES and LSCF results

Mode	f_{IES} $[Hz]$	f_{LSCF} $[Hz]$	Δ_f [%]	D_{IES} $[-]$	D_{LSCF} $[-]$	Δ_D [%]
1	11,27	11,29	0,18	0,0507	0,0520	2,50
2	52,25	52,38	0,25	0,0641	0,0664	3,49
3	131,98	132,26	0,21	0,0581	0,0599	3,08
4	247,04	247,32	0,11	0,0450	0,0460	2,20

Conclusion

In this paper an alternative approach for an IES algorithm for frequency dependent systems was introduced. The results of the IES were compared to an analytically solved SDOF system and showed excellent accuracy regarding the modal parameters. Based on FEM theory it was presented, how system matrices from FEM software can be easily manipulated outside the software environment subsequently. Additionally, a sandwich cantilever beam with frequency dependent viscoelastic material properties was discretized by shell elements in MSC Nastran. The resulting system matrices were extracted and the system has been analyzed regarding its modal parameters by means of the IES. Furthermore, a frequency response analysis has been performed and the corresponding modal parameters have been identified using the LSCF method. The results of both methods showed good agreement, even if a small deviation exists for the damping ratios. Therefore, it has been proven that the IES is a viable tool for analyzing systems with frequency dependent material properties.

Acknowledgment

We would like to acknowledge the funding by the Deutsche Forschungsgemeinschaft (DFG, German Research Foundation) under Germany's Excellence Strategy – EXC 2163/1- Sustainable and Energy Efficient Aviation – Project-ID 390881007.

References

[1] A. D. Nashif, D. I. Jones, and J. P. Henderson. *Vibration damping.* John Wiley & Sons, 1985.

[2] K. J. Bathe. *Finite Element Procedures.* Prentice Hall, 1996.

[3] D. J. Ewins. *Modal Testing: theory, practice and application.* Research Studies Press Ltd., 2000.

[4] C. M. A. Vasques, R. A. S. Moreira, and J. Dias Rodrigues. Viscoelastic damping technologies - Part I: Modeling and finite element implementation. *Journal of Advanced Research in Mechanical Engineering*, 1(2):76–95, 2010.

[5] M. B. Allen and E. L. Isaacson. *Numerical Analysis for Applied Science.* John Wiley & Sons, 2011.

[6] P. Guillaume, P. Verboven, S. Vanlanduit, H. Van Der Auweraer, and B. Peeters. A poly-reference implementation of the least-squares complex frequency-domain estimator. In *Proceedings of IMAC 21*, Kissimmee, USA, 2003.