AER501: Assignment 2: Vibration Analysis

Emma Seabrook (1005834563)

November 18, 2023

1 Introduction

In this report, I will describe the analysis performed of various vibration-damping models. The frame structure that will be used for this analysis is depicted in Figure 1. Four different meshes will be used, each with one, two, three, and four elements per frame member respectively. The natural frequencies and mode shapes as well as the displacements will be determined and examined using the four meshes. The goal of this assignment is to determine the effects of mesh refinement on the different techniques as well as to compare the accuracy of the two forced-vibration techniques.

2 Matlab Code Structure

The following subsections describe the main analysis functions present in the code.

2.1 Free Vibration Analysis

The *freevibration.m* function takes the stiffness and mass matrices as inputs and outputs the natural frequencies and the mode shapes of the system. I used the MATLAB *eig* function to solve the following problem for ϕ and λ:

$$\mathbf{K}\phi = \lambda\mathbf{M}\phi \tag{1}$$

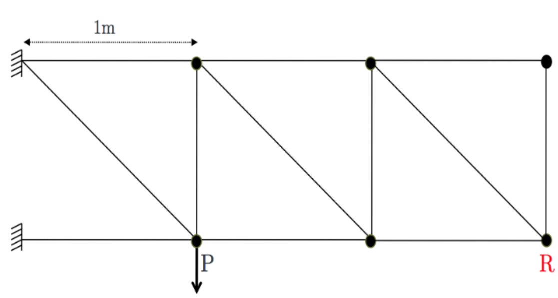


Figure 1: Two-dimensional frame under analysis.

2.2 Full Frequency Analysis

The *directfrfanalysis* function uses the stiffness (K), mass (M), and damping (C) matrices as well as the excitation frequency Ω and the excitation forces (f) to determine the displacement response (u0). The full-order dynamic stiffness matrix (D) is defined as:

$$\mathbf{D} = \mathbf{K} - (\Omega^2)\mathbf{M} + j\Omega\mathbf{C} \tag{2}$$

The relationship between the dynamic stiffness matrix and the displacement response is as follows:

$$\mathbf{f_0} = \mathbf{D}\mathbf{u_0} \tag{3}$$

2.3 Modal Frequency Analysis

The *modalfrfanalysis* function takes the excitation frequency (Ω), the excitation modes (Φ), the natural frequencies (Λ), the input forces (f0), and the number of modes to use (m) as inputs, and produces the displacement response (u0).

This is achieved using the following formulation:

$$\mathbf{A} = \Lambda_m - \Omega^2\mathbf{I_m} + j\Omega(\gamma_1\Lambda_m + \gamma_2\mathbf{I_m}) \tag{4}$$

$$\mathbf{b} = \Phi_m\mathbf{f_0} \tag{5}$$

$$\mathbf{Aq} = \mathbf{b} \tag{6}$$

$$\mathbf{u} = \Phi_m\mathbf{q} \tag{7}$$

Where $\mathbf{I}$ is the identity matrix, j = $\sqrt{-1}$, γ_1 is 0, and γ_2 is 10.

3 Free Vibration Analysis

For the free-vibration analysis, we look at the differences in the first 12 natural frequencies identified when the mesh is refined. The four meshes used for this study had one, two, three, and four elements per member so as the mesh number increases, the mesh becomes more refined. The results are shown in figure 2. Though all of the meshes produce similar frequencies for the first couple of nodes, they diverge as the mode number increases. However, as the number of elements per member increases (ie. as the mesh gets finer), the results converge.

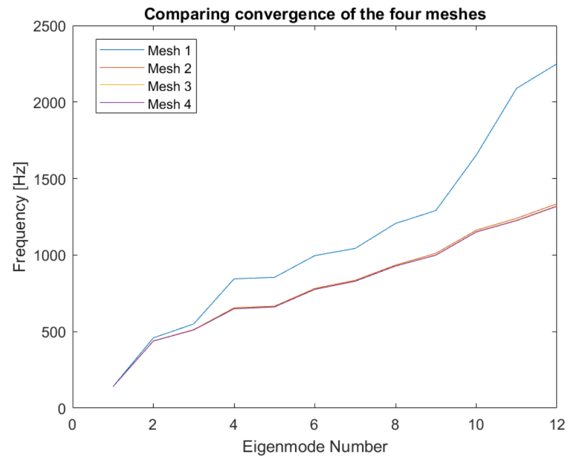


Figure 2: Free Vibration Analysis using four different levels of mode refinement.

4 Forced Vibration Analysis

We apply an excitation of the form $P(t) = e^{(j\Omega t)}$ at node P and analyse the displacement response of this excitation at node R. (See fig 1 for node placement)

4.1 Full-order Dynamic Response

The *directfrfanalysis.m* function uses the relationship described in Equations 2 and 3 to determine the displacement response.

4.1.1 Mesh Refinement Study

In figure 3, we see that the first mesh does not provide a fine enough model to adequately determine the displacement response at the higher frequencies.

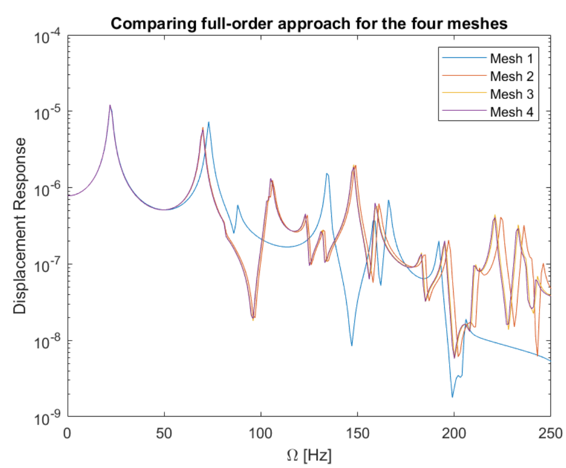


Figure 3: Mesh Refinement Study using Full Order Dynamic Stiffness Matrix

4.2 Modal Analysis Response

The *modalfrfanalysis.m* function uses the relationship described in Equations 4 to 7 determine the displacement response based on the Modal analysis technique.

4.2.1 Mesh Refinement Study

In figure 4, we see that the first mesh does not provide a fine enough model to adequately determine the displacement response at the higher frequencies.

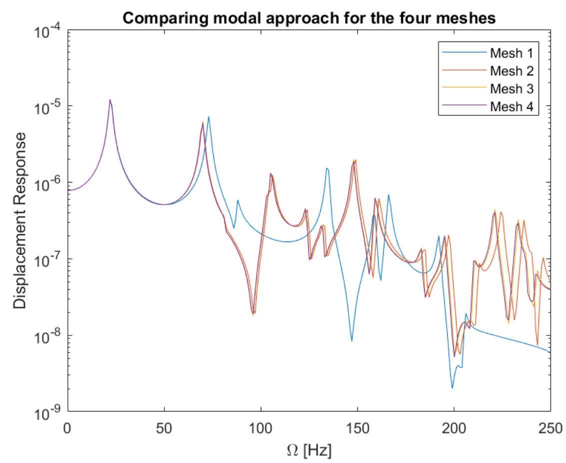


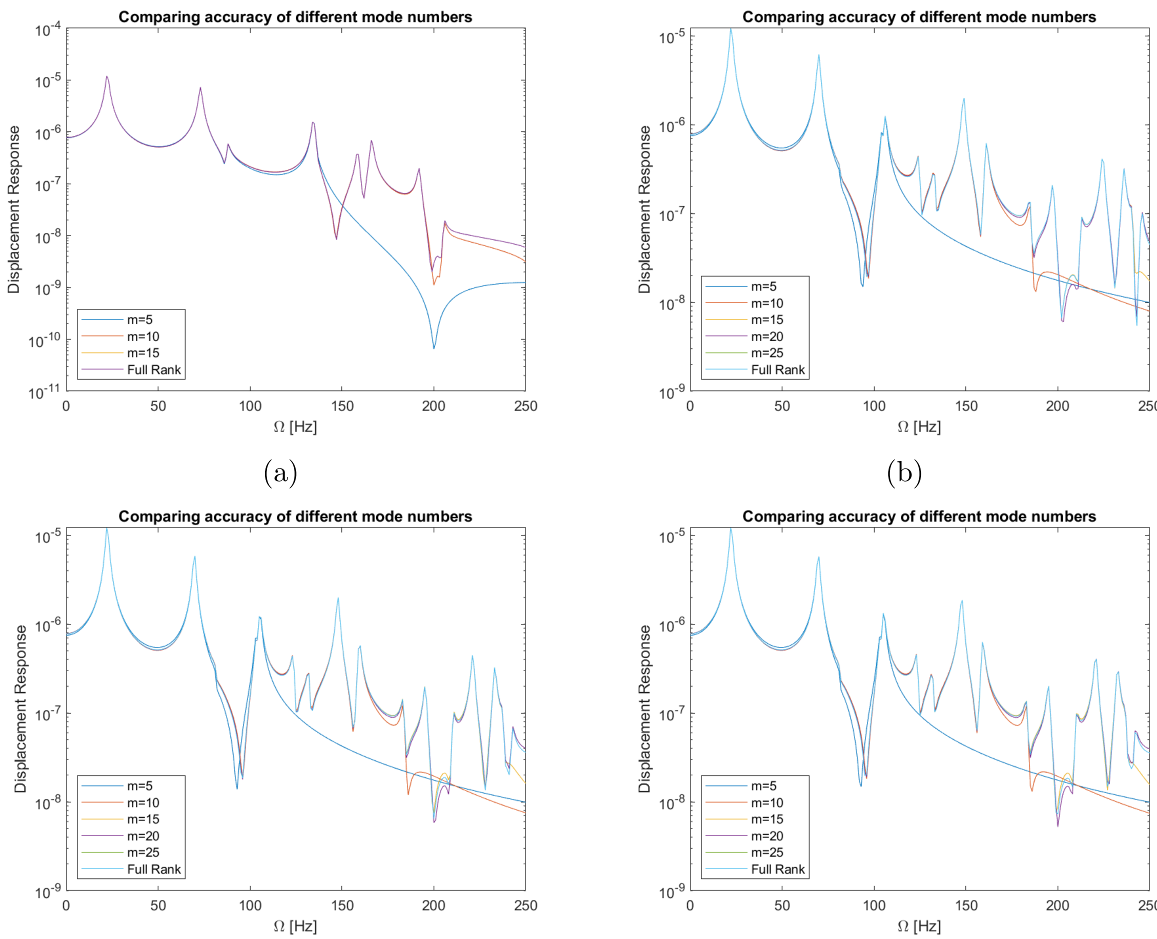
Figure 4: Mesh Refinement Study using Modal Analysis technique

4.2.2 Mode Number Study

When using a modal analysis, the number of modes can be limited to improve computation time. "Full-Rank" is when all of the available modes are used in the analysis and is the highest accuracy approximation available. Figure 5 compares the modal approximation of the four meshes using different mode numbers. From the figure, we can determine that for the first mesh, at least 15 modes are needed for an accurate approximation. The number of modes needed for an accurate approximation for the second, third, and fourth meshes is 20. Table 1 shows the convergence of the L2 norm of the error between the full-rank and the mode-limited approximations as the number of modes increases.

4.3 Comparison of Analysis Techniques

In figure 6 we compare the displacement response of the four meshes using both techniques. In all four cases, the two methods produce very similar results at the lower frequencies ($<$ 150 Hz) but begin to diverge at the higher frequencies.



(a) (b) (c) (d)

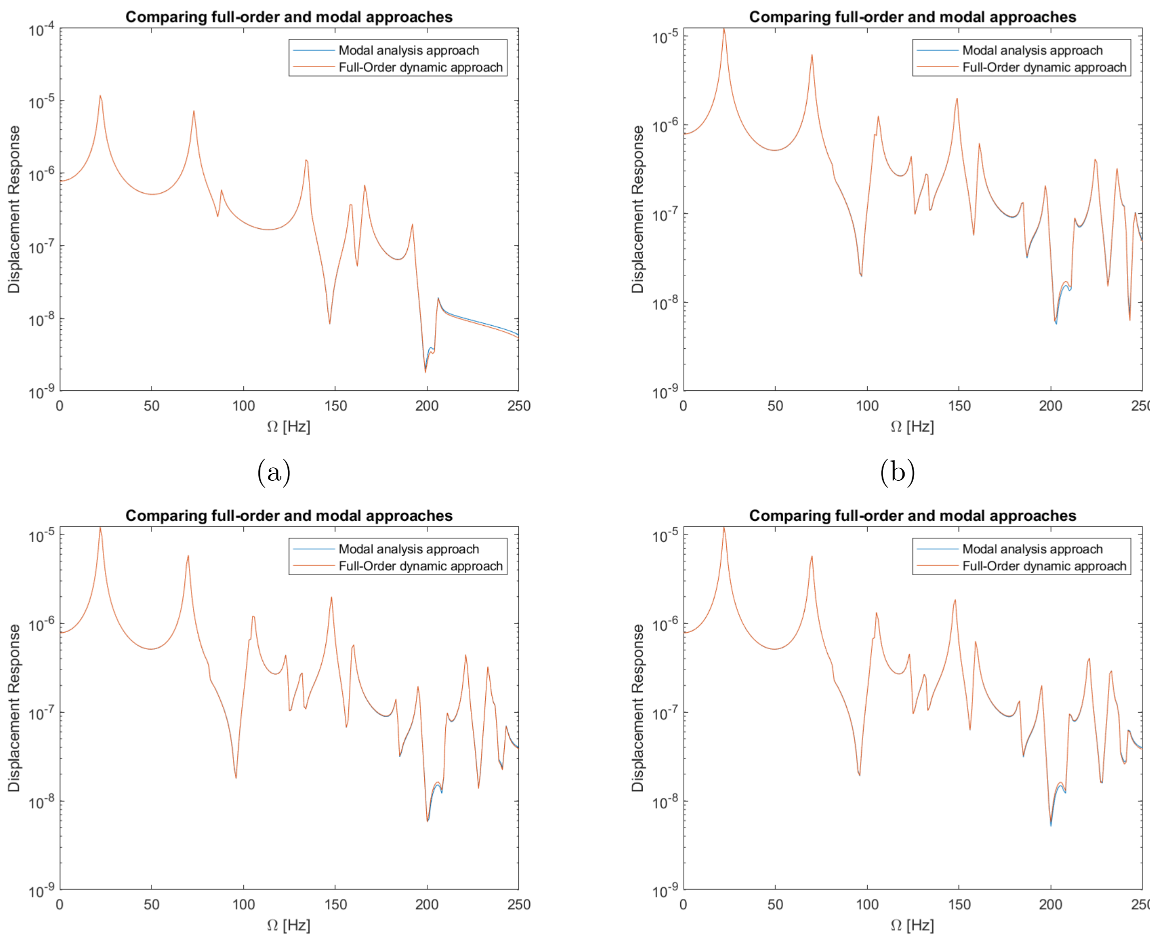
Figure 5: Comparison of mode number used for each mesh size.

5 Conclusions

From the analysis conducted on the different techniques and meshes, it appears that the accuracy of the displacement response is affected more by the degree of refinement of the mesh than the technique used for analysis. The modal approximation allows us to significantly reduce the computation cost of very large meshes by providing an accurate approximation using only the first m nodes. This is especially useful when working with very large structures that require very fine meshes. Unfortunately, this technique is only appropriate for low-frequency vibration analysis as these are the modes that would primarily be included in the approximation.

Table 1: Comparing L2-norm of the error between the full-rank and mode-limited approximations.

	Mesh 1	**Mesh 2**	**Mesh 3**	**Mesh 4**
m = 5	1.29e-06	3.47e-06	3.38e-06	3.35e-06
m = 10	2.39e-08	8.97e-07	8.95e-07	8.85e-07
m = 15	7.59e-09	1.32e-07	9.34e-08	9.18e-08
m = 20	0	1.41e-08	1.56e-08	1.56e-08
m = 25	0	4.11e-08	3.27e-08	3.02e-08



(a) (b) (c) (d)

Figure 6: Comparison of analysis techniques for each mesh size.

6 Bonus: Quasi-Static Correction Scheme

The Quasi-static correction scheme aims to reduce the error of the approximation by adding a correction factor to the modal approximation. It is used to reduce the normalized residual error norm to allow for the use of more accurate modal analysis approximations.

$$\mathbf{u} = \widehat{\mathbf{u}} + \Delta\mathbf{u} \tag{8}$$

In matrix-vector notation, the correction factor can be defined as follows:

$$\Delta u \approx K^{-1} f(t) - \Phi_m \Lambda_m^{-1} \Phi_m^T f(t) \tag{9}$$

The *quasi-static.m* function will calculate the quasi-static correction scheme. It takes the stiffness matrix (K), the input forces (f), the excitation modes (PHI), the modal frequencies (LAM), the modal approximation (u_{modal}), and the number of modes (m) as arguments and produces the corrected approximation.

6.1 Comparison to Modal Analysis

In Figure 7, we compare the results of the quasi-static approximation for the four mesh sizes. From this, we can see that in the regions where the modal analysis deviates from the full rank approximation, the quasi-statically corrected modal analysis produces a closer approximation.

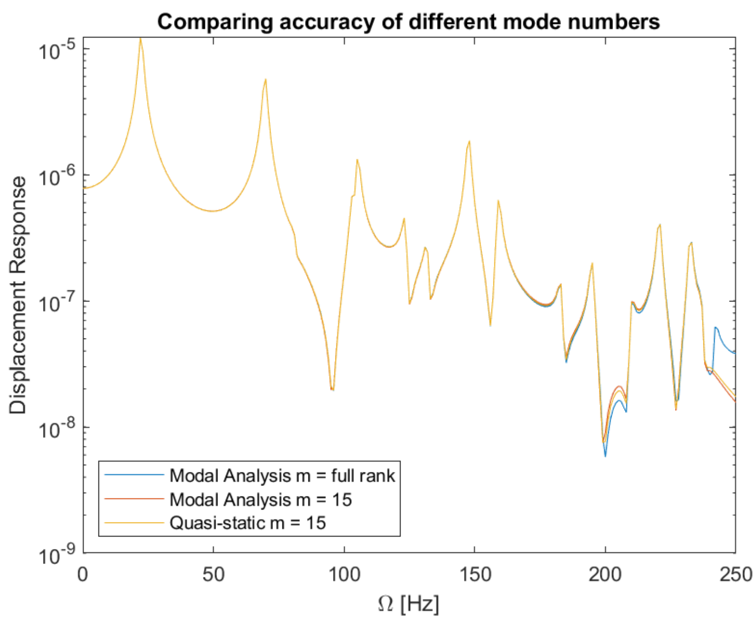


Figure 7: Comparison of quasi-static correction scheme for the four-element mesh size.