We are IntechOpen,
the world's leading publisher of
Open Access books
Built by scientists, for scientists

6,900
Open access books available

186,000
International authors and editors

200M
Downloads

Our authors are among the

154
Countries delivered to

TOP 1%
most cited scientists

12.2%
Contributors from top 500 universities

CLARIVATE ANALYTICS
BOOK CITATION INDEX
INDEXED

WEB OF SCIENCE™

Selection of our books indexed in the Book Citation Index
in Web of Science™ Core Collection (BKCI)

Interested in publishing with us?
Contact book.department@intechopen.com

Numbers displayed above are based on latest data collected.
For more information visit www.intechopen.com

MATLAB for All Steps of Dynamic Vibration Test of Structures

Abdurrahman Sahin and Alemdar Bayraktar

Additional information is available at the end of the chapter

Abstract

With the recent advances in computer technology and digital simulation software, it is now possible to rapidly and accurately build computer models for complex linear and nonlinear dynamic systems. MATLAB is a unique system that can be used for structural and earthquake engineering problems. This study presents MATLAB tools developed for numerical process of all steps of dynamic vibration test of structures. The functions of the tools are processing the signals obtained from forced and ambient vibration tests of structures, determining the dynamic characteristics of structural systems, and automatically updating the analytical finite element (FE) models. The software group is composed of three programs named as SignalCAD, ModalCAD, and FemUP. The SignalCAD program is developed for processing raw measured data obtained from forced and ambient vibration tests of engineering structures. The ModalCAD program is developed for dynamic characteristic identification and validation procedure. The peak picking method, complex exponential method, and polyreference time domain method are used for modal identification process. The FemUP program is developed for automatically updating the numerical models of structures compared to modal testing results. Each program has a unique graphical user interface and is designed as user friendly. The possibilities of the programs are demonstrated with the model vibration test of a steel cantilever beam. The obtained results are compared to the analytical model, and the FE model is automatically updated, whereas the experimental model is considered as the reference model. Finally, it is seen that MATLAB can be used as a scientific programming platform in all vibration test and modal analysis applications.

Keywords: MATLAB, SignalCAD, ModalCAD, FemUP, vibration test of structures, experimental modal analysis, operational modal analysis, FE model updating

1. Introduction

The recent developments in computer technology give us the opportunity to construct full-scale models of all kinds of structural systems. Advanced analysis methods can be used to determine the dynamic characteristics of the structures and to simulate the structural behaviors. Although the numerical methods have reached to an advanced level, experimental validations are still necessary to obtain realistic models. Full-scale vibration tests are one of the most reliable and widely used validation techniques. The dynamic vibration tests can be used for dynamic characteristic identification of all structural systems [1–5]. The determined dynamic characteristics are natural vibration frequencies, damping ratios, and mode shapes. These parameters may be used to validate and update the numerical models [6–9].

A complete structural evaluation process, including numerical and experimental studies, consists of some procedures. First, the numerical model of the structural system is constructed. The critical points for vibration test are determined and the structure system is equipped with the accelerometers. Then, the vibration test is performed via artificial or natural excitation sources. Generally, ambient vibration tests are carried out in large civil engineering structures, as it is not easy to record natural exciting sources in real time. After the vibration tests are completed, the raw measured acceleration records are first filtered to clean noise from the signal. Then, the data are processed and spectral functions are produced. These functions are frequency response function (FRF), cross-power spectrum (CPS), power spectral density (PSD), auto power spectrum (APS), and spectrogram. The system identification methods are applied to the produced spectral functions, and the dynamic characteristics of the structure are extracted. The obtained dynamic parameters are natural vibration frequency, damping ratio, and modal vectors of the structure of interest. These parameters are experimental dynamic characteristics of the structures and they are used for validating or updating the numerical models. In the model updating process, all parameters, including material properties, boundary conditions, stiffness distribution, connection details, and damaged parts over the structure, may be considered and the optimal numerical model is obtained, showing the closest behavior to the experimental model.

2. Forced and ambient vibration tests of structures

The dynamic vibration tests on structures are generally subdivided into two groups: (a) forced vibration test and (b) ambient vibration test. In the forced vibration test, the structure is usually excited by artificial means. The excitation force and the response of the structure are recorded at the same time and the spectral functions are developed using these data. In the ambient vibration test, the structure is excited by natural effects, such as wind load and traffic load. The response of the structure is recorded under operational conditions and the spectral functions are developed using these data. The dynamic parameters of the structures are determined from the produced spectral functions.

The modal parameter estimation stage of the forced vibration test is called the experimental modal analysis or input-output modal analysis. On the contrary, the modal parameter

estimation stage is called operational modal analysis or output-only modal analysis if the vibration test is carried out under operational conditions (ambient vibration test).

The experimental and operational modal analyses of structures are carried out in four distinct steps. In the first step, data are collected from the test structure. In the second step, digital signal processing is applied to collected raw measured data and the spectral functions are produced. In the third step, modal parameters (natural frequencies, damping ratios, and mode shapes) are extracted and these parameters are visualized and validated. In the last step, the numerical finite element (FE) model is updated by comparing to the experimental model. In the first part, the forced or ambient vibration test is carried out and data are collected from the structural system. This stage is the experimental procedure. The remaining stages are computational procedures and require the development of mathematical algorithms and programming tools. In this study, the developed tools in MATLAB platform for these numerical procedures are presented.

3. Digital signal processing for vibration test of structures

The first step of computational procedure is digital signal processing. In this stage, signals are converted from the time domain to the frequency domain usually through the Fourier transform. In the forced vibration test, FRFs are used to estimate the dynamic properties of a structure. Excitation force and response accelerations are used to obtain these functions. In large-scale civil engineering structures (such as bridges and towers), the structure is under excitation of natural sources such as traffic load and wind load. It is difficult to measure the input to the structure under the operational conditions. The CPSs may be used for output-only modal analysis. The response signals are used to obtain these functions.

3.1. Development of digital signal processing tool

The SignalCAD [10] program is developed for digital signal processing and may be used for forced and ambient vibration tests of structures. In the forced vibration test analysis process, first excitation forces and response signals are collected from the structure as shown in **Figure 1a**. The SignalCAD program reads these records and apply fast Fourier transform (FFT) to these signals using MATLAB Signal Processing Toolbox [11]. Then, coherence functions between input and output signals are generated.

In the ambient vibration spectral analysis process, the acceleration records are collected from the structure as shown in **Figure 1b**. These records may be single signals or signal groups. The ambient vibration test requires more time; therefore, the record time may be much longer. The collected signal may be divided into small signals and the signal series are processed in the system. The SignalCAD program reads these signal groups and applies FFT to these signals using MATLAB Signal Processing Toolbox [11]. The spectrum series, such as CPSs, PSDs, APSs, and spectrograms, are produced. The singular value decomposition (SVD) or averaging methods are applied to the produced spectrum series and the single spectra for each channel are produced. This process is repeated for every channel that collects signal from the structure.
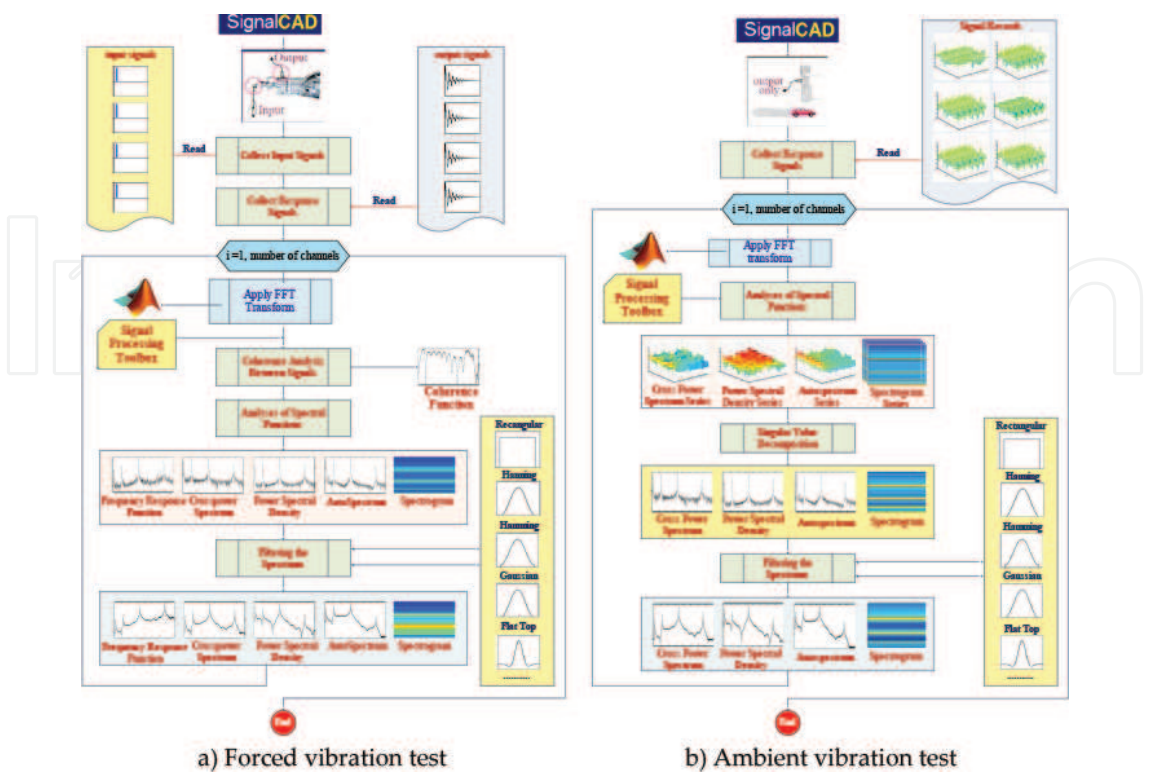
Figure 1. Flow chart of digital signal processing procedure with SignalCAD [10]: (a) forced vibration test and (b) ambient vibration test.

As can be seen in **Figure 1**, the leakage errors occur in spectral functions because of the FFT. Windowing functions need be applied to the spectra to eliminate these errors. In SignalCAD, some windowing alternatives may be applied. The main window of SignalCAD program is presented in **Figure 2**.
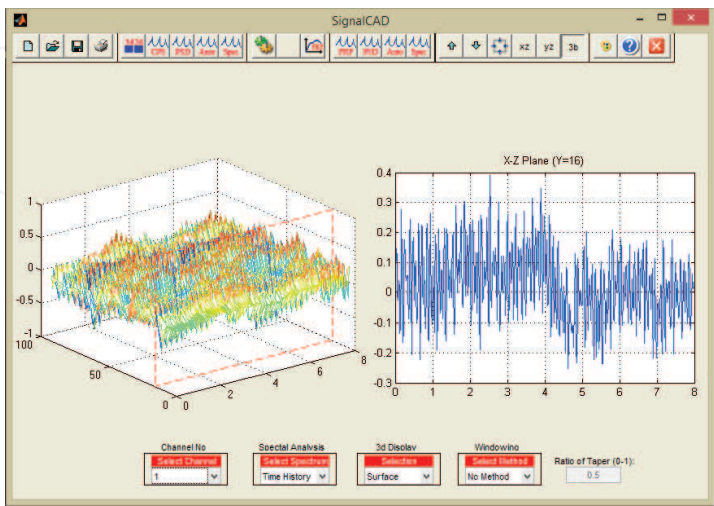

Figure 2. SignalCAD program main window [10].

4. Dynamic characteristic identification

After the collected raw measured acceleration records are processed and spectral functions are produced, dynamic characteristics are extracted from these spectra. MATLAB System Identification Toolbox [12] offers mathematical functions for system identification studies. It identifies time domain models from the data and can be used in modal identification studies. However, it is clear that some postprocessing is needed for the purpose of dynamic characteristic identification of structure. New functions need to be prepared to extract modal parameters from the spectral functions, to produce stabilization diagrams, to visualize
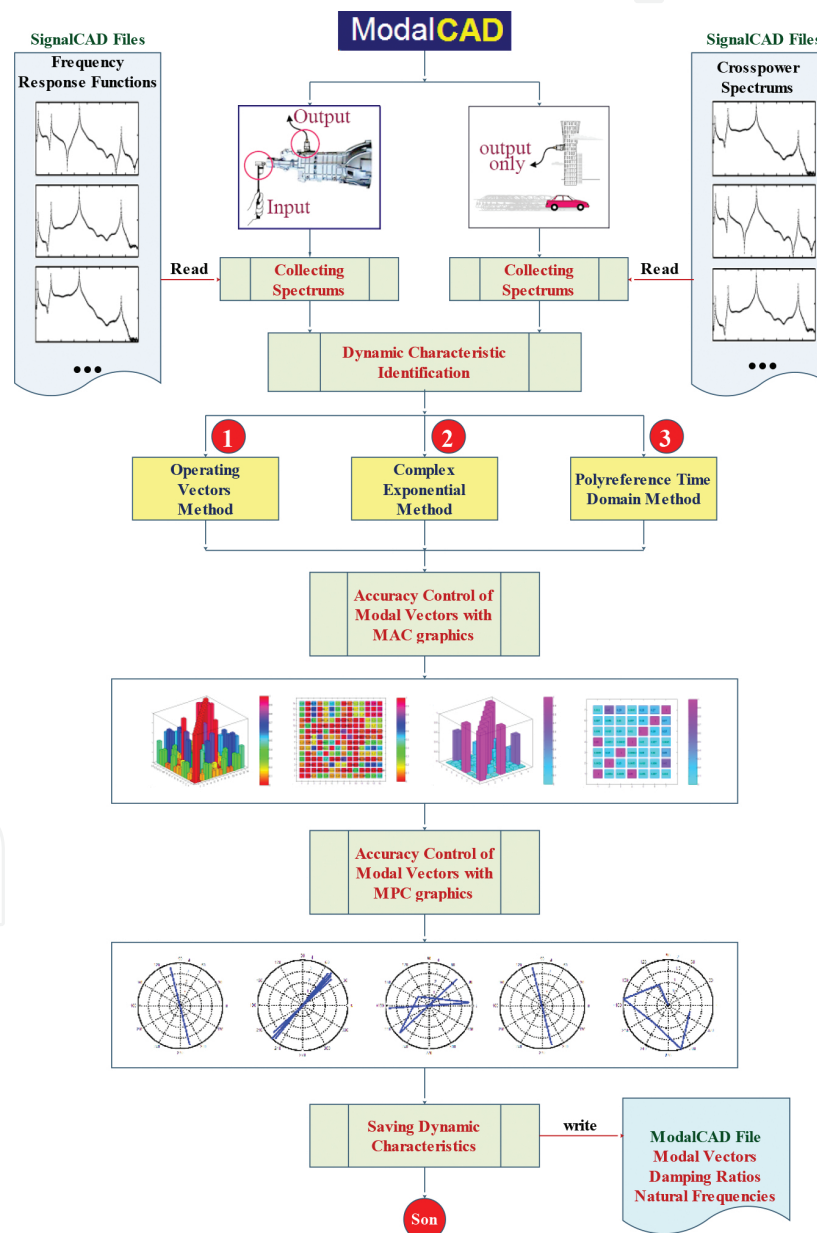

Figure 3. Flow chart of the ModalCAD program [13].

structure's geometry and mode shapes, and to evaluate modal validation tools. From this overview, about 250 new functions have been developed depending on the solution methods and are used together with general System Identification Toolbox functions.

4.1. Development of system identification tool

The ModalCAD [13] program is developed for system identification and may be used for the experimental and operational modal analyses of structures. In ModalCAD, three modal identification methods are used. First is the peak picking method or half-power band method in frequency domain. It may be called the operating vectors (OV) method. The second one is the complex exponential (CE) method in time domain and the polyreference time domain (PTD) method. The detailed explanations of these methods can be found in [13]. The flow chart of the program is given in **Figure 3** and the main window of ModalCAD is presented in **Figure 4**.
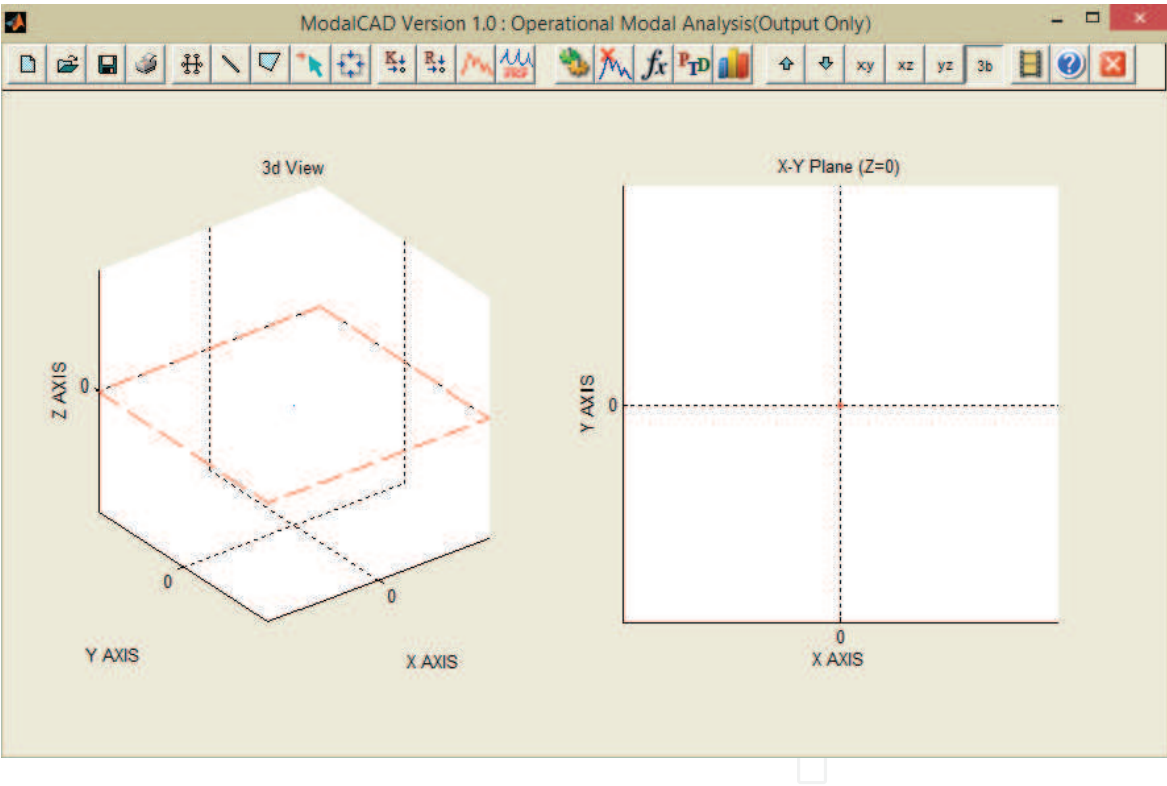

Figure 4. ModalCAD program main window [13].

5. FE model updating procedure

First, the FE model is developed using the initially estimated values for the unknown model parameters. FE modal analysis is then carried out to obtain the FE modal data. ANSYS [14] or

any other advanced FE codes may be used for analytical FE modal analysis. For the forced or ambient vibration test of the structure, the optimum points for the placement of sensors are chosen and test data are recorded. The measured raw data are processed with the SignalCAD software. In this process, FRFs are produced for experimental modal analysis and CPSs are produced for operational modal analysis. The experimental and operational modal analyses are then carried out using the OV, CE, and PTD methods to get the modal parameters via the ModalCAD software. For model updating, the modal frequencies and modal vectors are exported from ModalCAD to FemUP. The most common way to compare the analytical and experimental mode shapes is the use of modal assurance criterion (MAC), and it is obtained as follows:

$$MAC_j = \frac{\left|\Phi_{aj}^T \Phi_{ej}\right|^2}{\left(\Phi_{aj}^T \Phi_{ej}\right)\left(\Phi_{ej}^T \Phi_{aj}\right)} \tag{1}$$

where Φ_{aj} is the analytical modal vector that has been paired with the jth experimental modal vector Φ_{ej}. The value of the MAC is bound between 0 and 1. Higher value indicates better correlation between modal vectors. If the MAC value is zero, it is understood that there is not any correlation between the modal vectors. If the MAC value is 1, the highest correlation is obtained.

5.1. Development of computational FE model updating tool

The FemUP [15] program is developed for computational FE model updating automatically. It uses MATLAB Optimization Toolbox [16] for the optimal FE model determination process. A constrained optimization is performed using a sequential quadratic programming (SQP) algorithm. The optimization algorithm is supplied with start values, bounds, constraints, and optimization criterion. The optimization criterion chosen, which is to be minimized, is the sum of the differences in natural frequency within each correlated mode pair. Constraints are used on the correlation between analytical and experimental mode shapes using the diagonal values of the MAC matrix.

The FemUP program can read ANSYS FE models and run this code in batch mode using ANSYS Parametric Design Language. Because natural frequencies and mode shapes must be calculated many times during the updating procedure, ANSYS and MATLAB interact with each other. The objective and constraint functions, taking advantage of MATLAB's ability of reading and writing ASCII files, are used to transfer data between the two different software packages. The main window of FemUP is presented in **Figure 5**. The objective function in FemUP is defined as a sum of experimental and theoretical frequency differences. The constraint function, which includes nonlinear inequality constraints in FemUP, exports a vector that consists of the differences between MAC limit selected by the user and calculated MAC values.
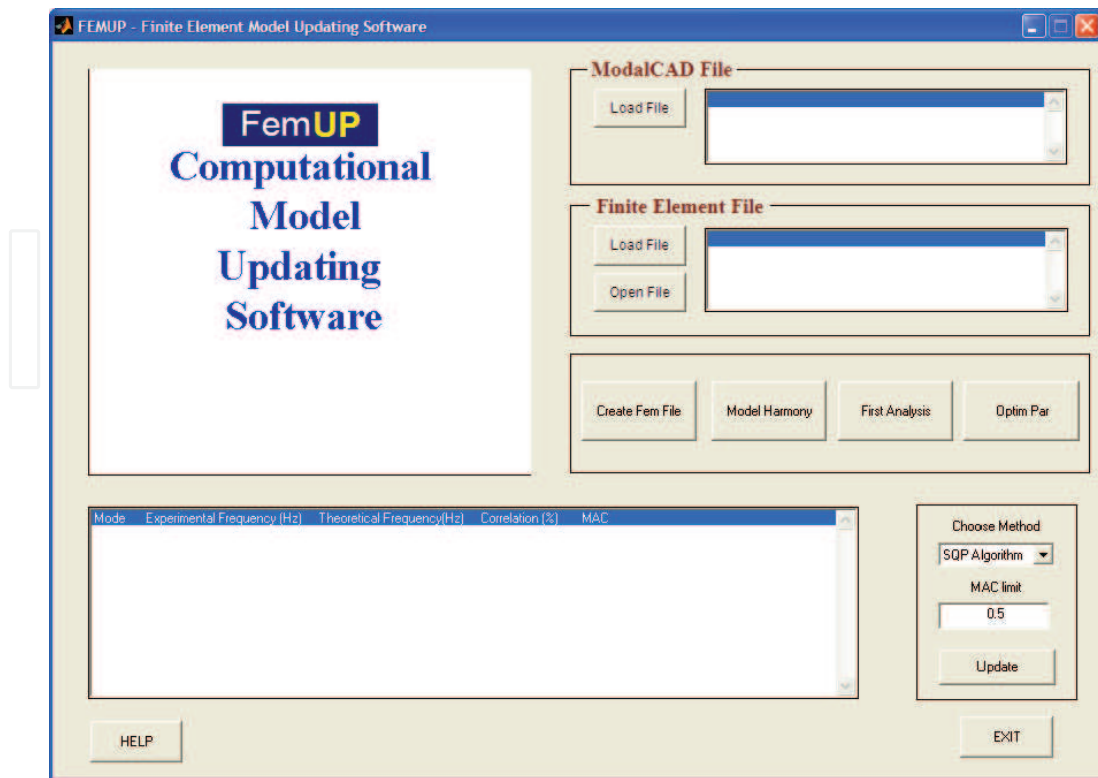
Figure 5. FemUP program main window [15].

6. Interaction between developed tools

As indicated previously, all tools have an interaction with each other and the data developed by a tool are used by others. The raw measured acceleration records are processed with SignalCAD and spectra are produced. The FRFs are produced for input-output modal analysis and the CPSs are produced for output-only modal analysis. These spectra are introduced to ModalCAD and dynamics characteristics are extracted from these spectral functions. The produced dynamic characteristics are input data of the FemUP program. The numerical model produced with ANSYS is also introduced to FemUP and numerical and experimental models are compared to each other. In the comparison process, the summation of natural frequency differences forms the objective function and coherence between modal vectors forms the constraint function. The MAC matrix is used to understand the agreement between experimental and numerical mode shapes. If there is a nontrivial difference between natural frequencies, the system parameters are automatically updated under the defined limits. After the automatic model updating process, the difference is checked again. The model updating process is repeated until the minimal difference is obtained, while the MAC values are about one. Finally, the updated model is presented. The interaction details between developed tools are presented in **Figure 6**.
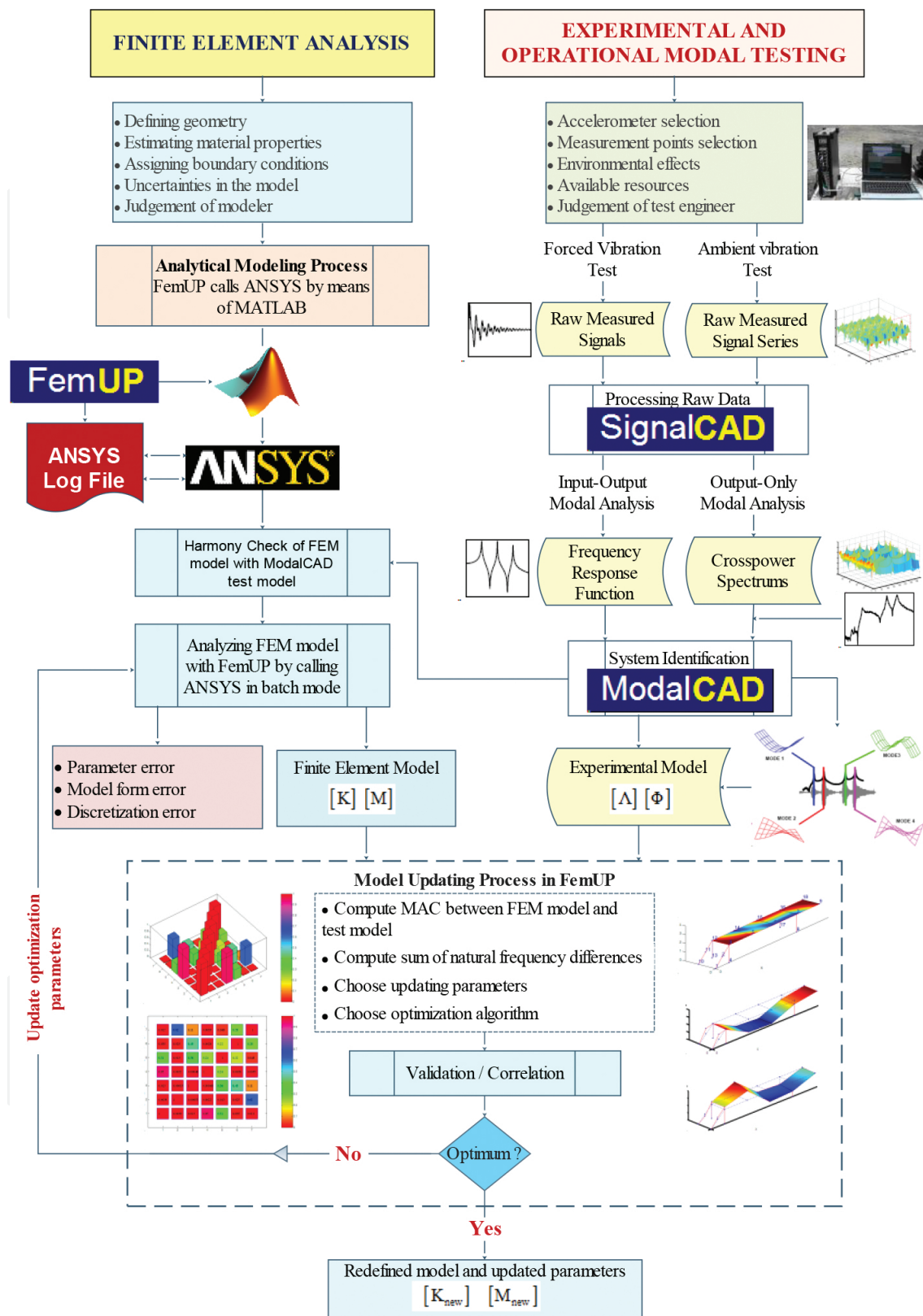
Figure 6. Interaction flow chart between MATLAB programs developed for signal processing, system identification, and FE model updating of structures [15].

7. Numerical application

A simple vibration test is carried out and the capabilities of developed tools are evaluated for a complete vibration test process. The example contains forced and ambient vibration tests, digital signal processing, modal parameter estimation, and automatic FE model updating of a steel cantilever beam model.

7.1. Vibration test and modal analysis

The vibration tests of the steel cantilever beam model are carried out. The beam model and test process are shown in **Figure 7**. The accelerometers are located on the surface of the model. The channel numbers and directions are given in **Figure 7**.


Figure 7. Cantilever beam model, acceleration set-up, and excitation with a hummer.
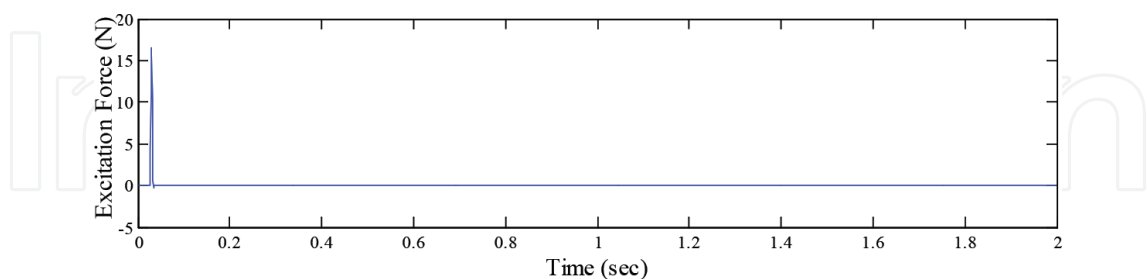
Figure 8. Input force signal collected during the forced vibration test.

First, the forced vibration test is carried out. The excitation force (**Figure 8**) and response of the model (**Figure 9**) are recorded simultaneously. These signals are processed with Signal-CAD and the FRFs are developed. These functions are introduced to ModalCAD, and the complex mode identification function (CMIF) is obtained from the developed FRFs as shown in **Figure 10**.
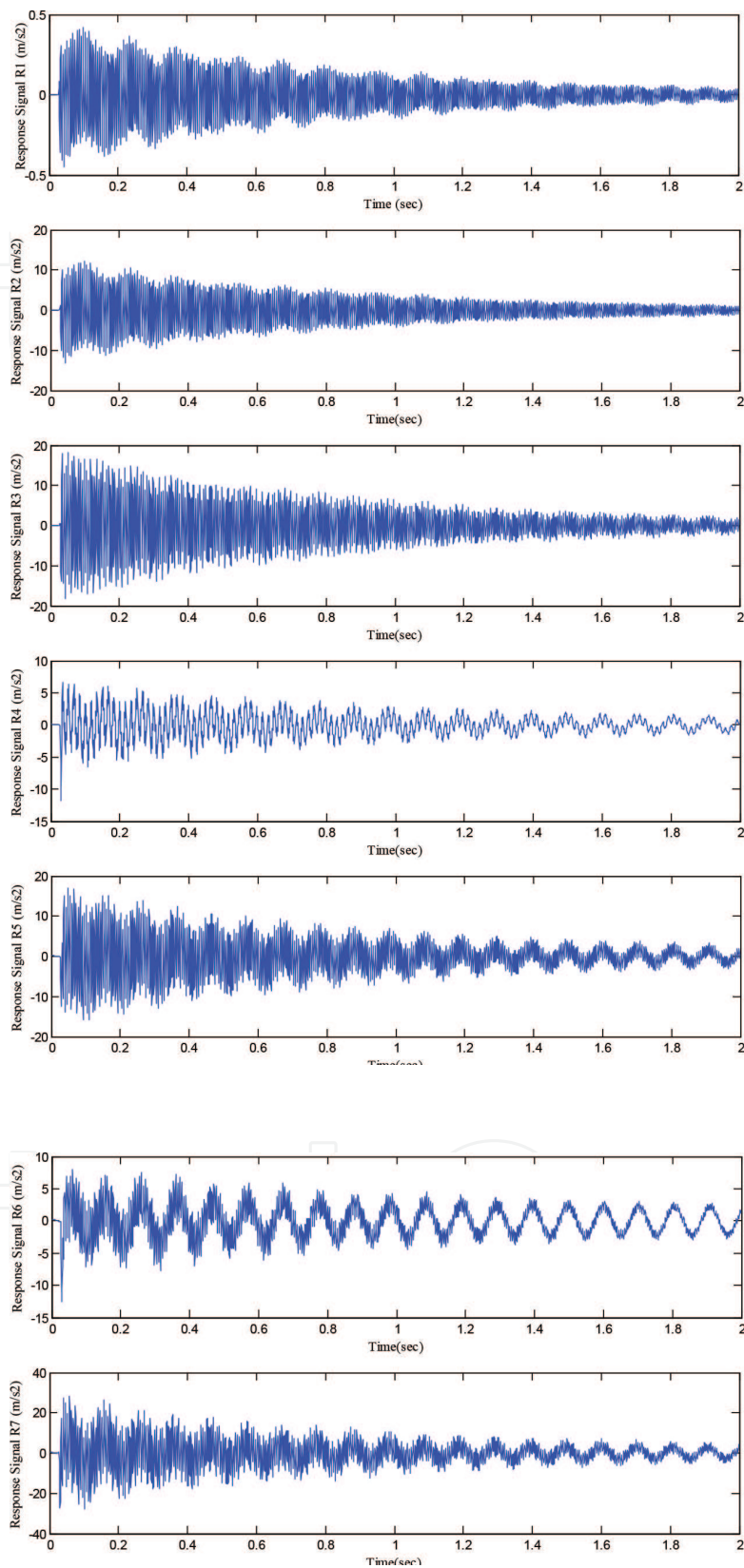
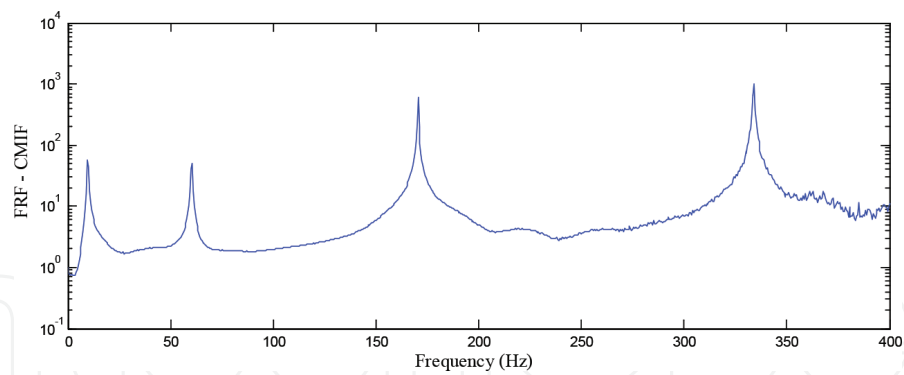**Figure 9.** Response accelerations obtained via the forced vibration test.
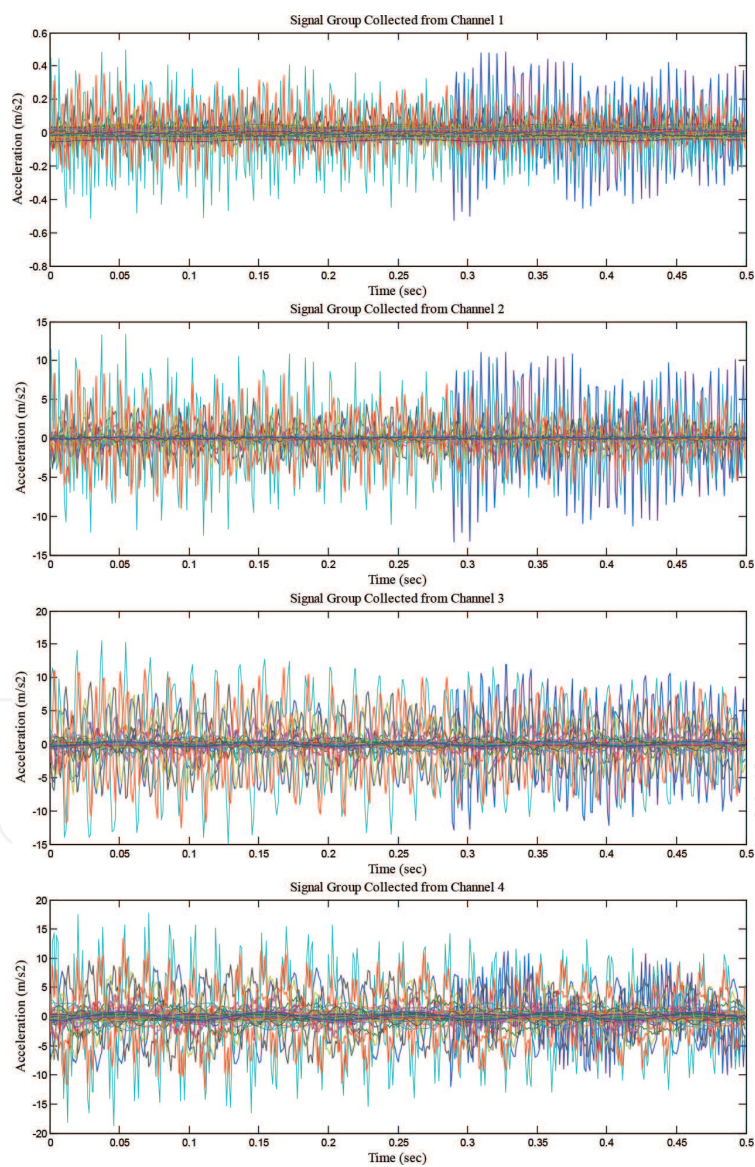
Figure 10. CMIF of calculated FRFs.

Figure 11. (a) Signal response set from the ambient vibration test (R1–R4).
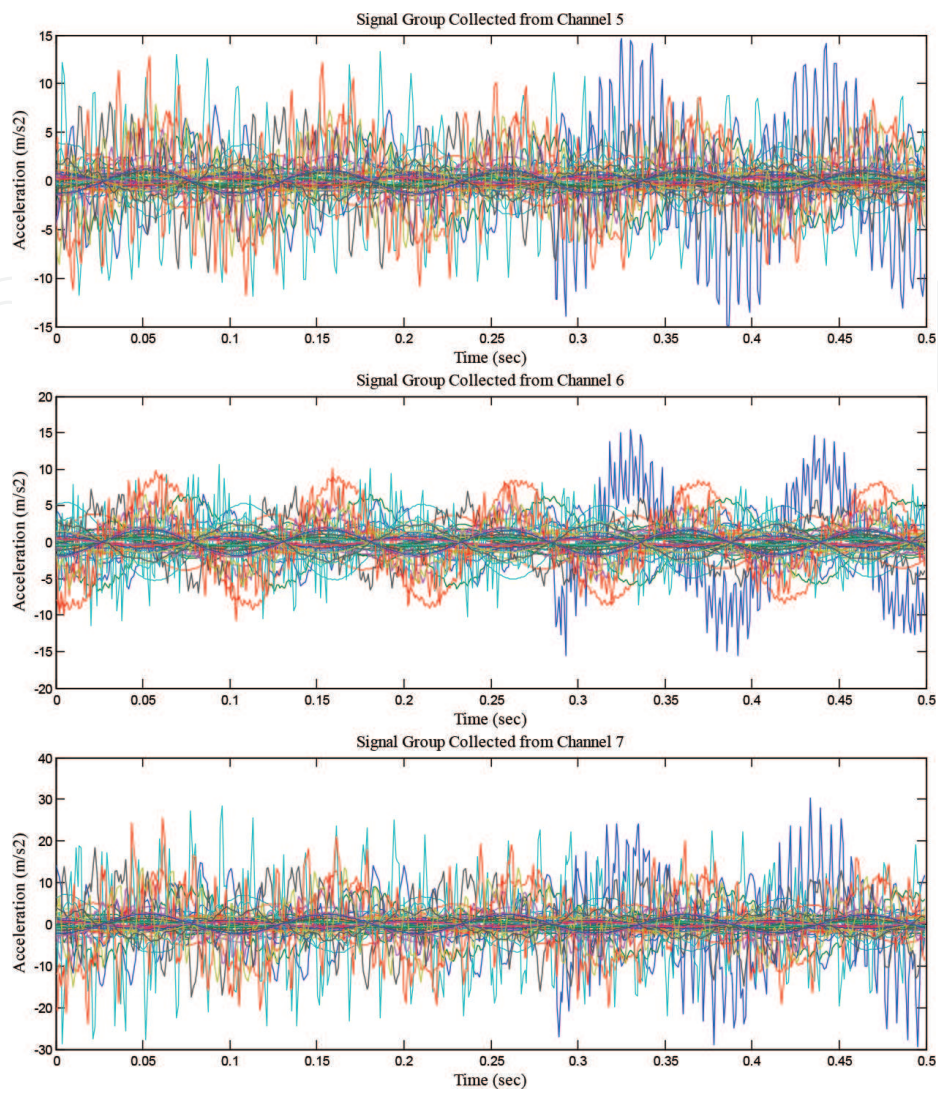
Figure 11. (b) signal response set from the ambient vibration test (R5–R7).
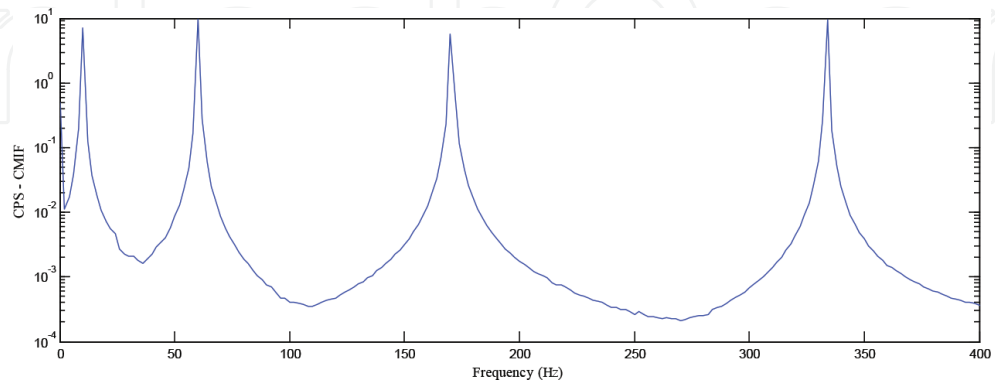

Figure 12. CMIF of calculated CPSs.

Second, the ambient vibration test is carried out and the response of the model is recorded. The collected signals are acceleration series for each channel as shown in **Figure 11**. These signals are processed with SignalCAD and the CPS series are developed. The CPS functions for each channel are produced by applying SVD to the spectrum series. The produced single functions are introduced to ModalCAD and the CMIF is obtained from the developed CPSs as shown in **Figure 12**.

The modal characteristics are then extracted with ModalCAD for input-output and output-only modal analyses. Same mode shapes are obtained from experimental and operational modal analyses with all methods. The obtained modal vectors are given in **Figure 13**.
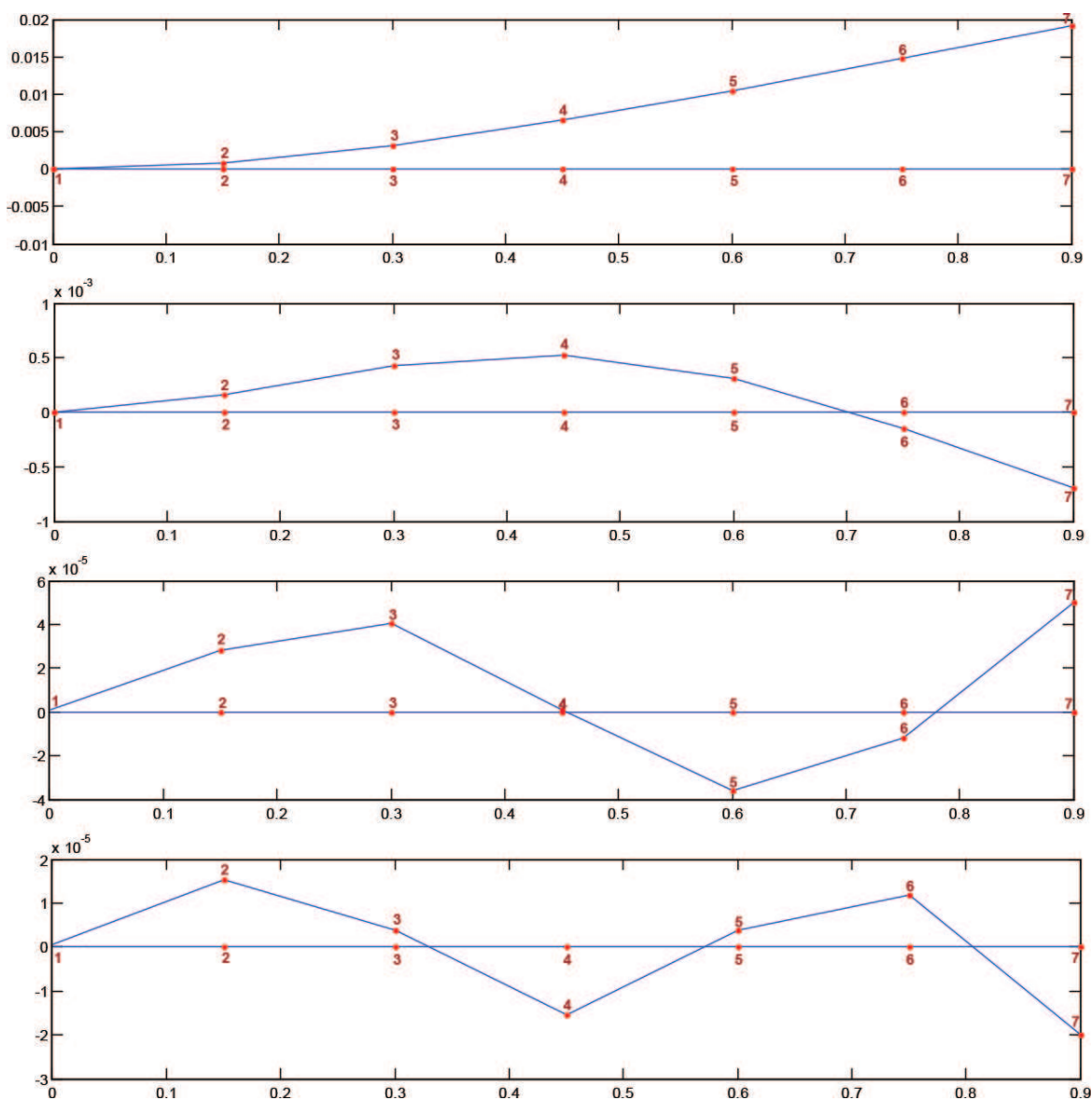
Figure 13. Experimental mode shapes of the cantilever beam model.

The natural vibration frequencies and damping ratios obtained using OV, CE, and PTD methods for input-output and output-only modal analyses are presented in **Tables 1** and **2**, respectively.

Mode no.	Input-output modal analysis (Hz)			Output-only modal analysis (Hz)		
	OV	CE	PTD	OV	CE	PTD
1	9.5	9.71	9.71	10	10	10.19
2	60.5	60.29	60.29	60	60.25	60.01
3	170.5	170.48	170.46	170	170.27	170.23
4	334	333.83	333.85	334	333.83	333.92

Table 1. Modal frequency values obtained from ModalCAD

Mode no.	Input-output modal analysis (%)			Output-only modal analysis (%)		
	OV	CE	PTD	OV	CE	PTD
1	16.3748	0.38409	0.37919	23.9592	0.35837	0.88837
2	2.8235	0.25131	0.2512	3.9965	0.21716	0.16947
3	0.53196	0.10881	0.11048	1.4897	0.12524	0.21411
4	0.38155	0.06087	0.05923	0.71889	0.06135	0.028594

**Table 2.** Modal damping values obtained from ModalCAD.

7.2. Analytical modal analysis and FE model updating

The analytical model of the cantilever beam is built in ANSYS. The natural frequencies and mode shapes are solved by the Lanczos method. The obtained mode shapes are presented in **Figure 14**. The analytical model is compared to the experimental model. As a reference data, the experimental model results obtained using the PTD method for input-output modal analysis are used because the experimental results are close to each other. The comparison of dynamic characteristics between the initial analytical model and the experimental model shows that the analytical natural frequencies are higher than the corresponding natural frequencies obtained experimentally. The differences in modal frequencies are higher than 20% for all modes as shown in **Table 3**. These differences are based on physical parameters. To achieve an analytical model that correlates better with the experimental results, the material properties are updated. There is no need to add mass and update boundary conditions in this model. The model is automatically updated by running the ANSYS model many times and the iterations are terminated until the aim function reaches the minimum value. The material properties of the beam model before and after the updating process are shown in **Table 4**. As shown in **Table 4**, the modulus of elasticity has been changed by FemUP. This change primarily affects the frequency values and modal vectors. The other parameters such as density and poison ratio have not been changed. In the updating step, three parameters are included in the

automated updating procedure. The correlation between mode shapes of the analytical and experimental models is evaluated using the MAC matrix. After the model updating process is completed, it can be said that the correlation is good. All differences in natural frequencies are below 1%. The experimental modal vectors are just same with the analytical modal vectors. This good harmony after the updating process may be observed from the MAC graphics given in **Figure 15**. As a result of the optimization study, it can be said that the most effective physical parameter of the model for model updating is the modulus of elasticity.
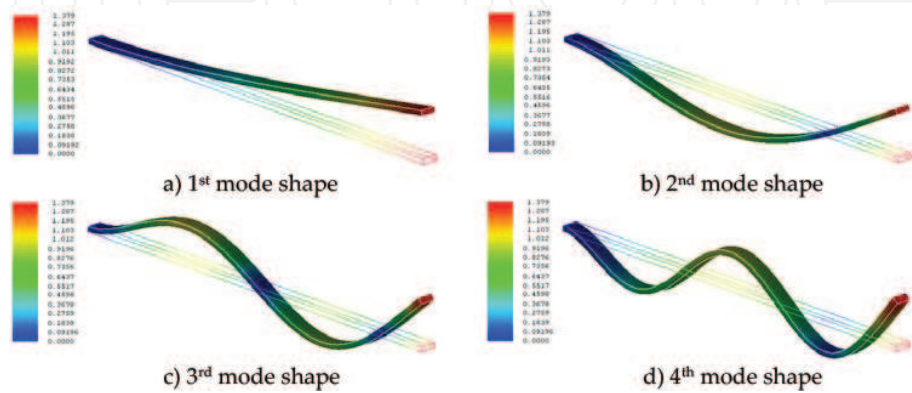
Figure 14. Numerical mode shapes of the cantilever beam model: (a) first mode shape, (b) second mode shape, (c) third mode shape, and (d) fourth mode shape.
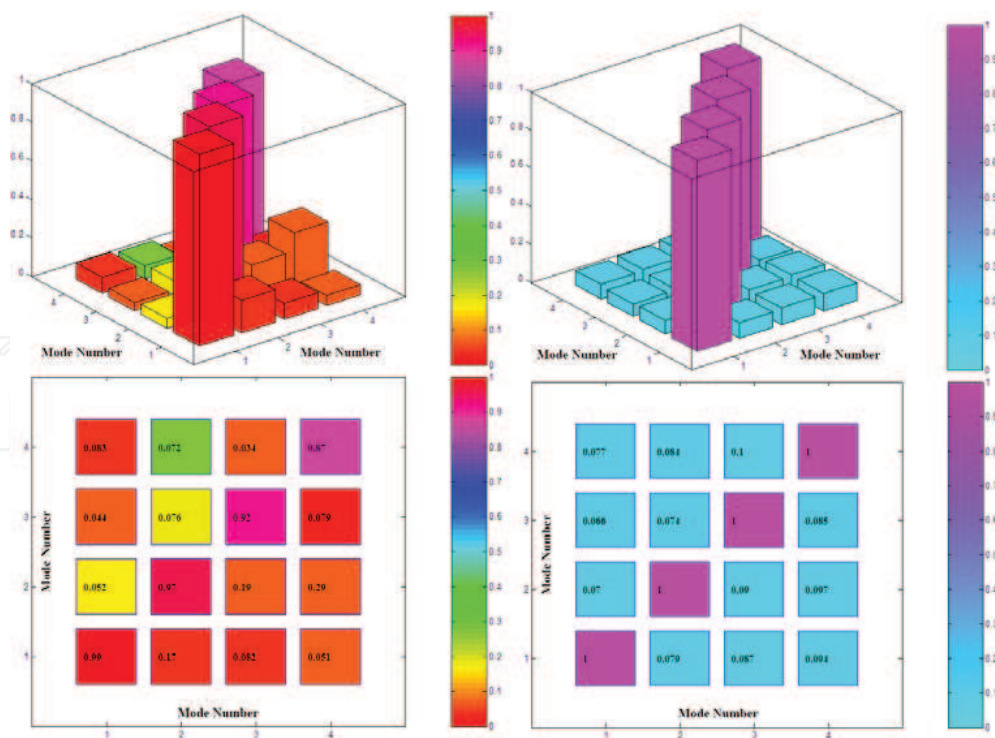
Figure 15. MAC matrices between the experimental model and the numerical model before and after the update process.

Mode no.	Test frequency (Hz)	Analytic frequency before the update (Hz)	Error before the update (%)	MAC before the update	Analytic frequency after the update (Hz)	Error after the update (%)	MAC after the update
1	9.7060	11.8415	−22.00	0.9950	9.7162	−0.11	1.0000
2	60.2896	74.2084	−23.09	0.9746	60.8814	−0.98	0.9995
3	170.4636	207.9209	−21.97	0.9217	170.4305	0.02	0.9963
4	333.8520	408.3915	−22.33	0.8741	333.8636	−0.00	0.9974

Table 3. Initial correlation analysis results between the experimental and analytical models.

Parameter	Before the update	After the update
Modulus of elasticity	$2.75 \times 10^{11} N / m^2$	$1.85147 \times 10^{11} N / m^2$
Density	7800 kg/m³	7800 kg/m³
Poisson ratio	0.3	0.3

Table 4. Model parameters before and after the update.

8. Conclusion

In this study, the importance of the computational part of vibration tests is highlighted and the capabilities of MATLAB for the possible use of all steps of dynamic vibration test of structures are explained. Three computer programs for these steps have been developed in MATLAB platform. The general properties of these tools are introduced and some flow charts for the general algorithms are also presented. The first tool is an interactive and comparative digital signal processing software developed for vibration test of structures and named as SignalCAD. The Signal Processing Toolbox functions are used for some of the operations. The software provides the capability to simulate vibration tests and perform spectral analysis including FRFs, CPSs, PSDs, coherence functions, transfer functions, and spectrograms. The second tool is the system identification software named as ModalCAD. The System Identification Toolbox functions and new developed functions are used for modal identification process. The software provides the capability to simulate vibration tests, perform experimental and operational modal analyses including structural identification with OV, CE, and PTD methods, and validate/visualize modal analysis results. The last tool is a computational FE model updating software called as FemUP. The SQP algorithm in MATLAB Optimization Toolbox is used to minimize the difference between analytical and experimental natural frequencies. Constraints are used on the correlation between the analytical and experimental mode shapes using the MAC matrix. The natural frequencies and mode shapes are solved by ANSYS. A simple vibration test is carried out and the capabilities of developed tools are evaluated for a complete vibration test process. The example contains forced and ambient vibration tests, signal analyses, system identification, and automatic FE model updating

process of a steel cantilever beam model. Beside this simple example, the applications of the developed tools for more detailed civil engineering structures can be seen in [17, 18]. The obtained results show that developed tools work well and can be used for the vibration test of structures.

Author details

Abdurrahman Sahin[1*] and Alemdar Bayraktar[2]

*Address all correspondence to: abdsahin@yildiz.edu.tr

1 Department of Civil Engineering, Yıldız Technical University, Istanbul, Turkey

2 Department of Civil Engineering, Karadeniz Technical University, Trabzon, Turkey

References

[1] Farrar C. R., James G. H. III. System identification from ambient vibration measurements on a bridge. J. Sound Vib. 1997; 205(1): 1–18.

[2] Brownjohn J. M. W., Dumanoglu A. A., Severn R. T. Ambient vibration survey of the Fatih Sultan Mehmet (Second Bosporus) Suspension Bridge. Earthquake Eng. Struct. Dyn. 2004; 21(10): 907–924.

[3] Atamturktur S., Fanning P., Boothby T. Traditional and operational modal testing of monumental masonry structures. International Operational Modal Analysis Conference, Copenhagen, Denmark. 2007.

[4] Gentile C., Saisi A. Ambient vibration testing of historic masonry towers for structural identification and damage assessment. Constr. Build. Mater. 2007; 21(6): 1311–1321.

[5] Ren W.-X., Peng X.-L., Lin Y.-Q. Experimental and analytical studies on dynamic characteristics of a large span cable-stayed bridge. Eng. Struct. 2005; 27(4): 535–548.

[6] Hartley M. J., Pavic A., Waldron P. Investigation of pedestrian walking loads on a cable stayed footbridge using modal testing and FE model updating. 17th International Modal Analysis Conference (IMAC XVII), Kissimmee, FL. 1999; 3727(2): 1076–1082.

[7] Jaishi B., Ren W. X. Structural finite element model updating using ambient vibration test results. J. Struct. Eng. 2005; 131: 617–628.

[8] Foti D., Diaferio M., Giannoccaro N. I., Mongelli M. Ambient vibration testing, dynamic identification and model updating of a historic tower. NDT E. Int., 2012; 47: 88–95.

[9] El-Borgi S., Choura S., Ventura C., Baccouch M., Cherif F. Modal identification and model updating of a reinforced concrete bridge. Smart Struct. Syst. 2005; 1(1): 83–101.

[10] Sahin A., Bayraktar A. SignalCAD—A digital signal processing software for forced and ambient vibration testing of engineering structures. J. Test. Eval. 2010; 38(1): 95–110.

[11] MATLAB Signal Processing Toolbox User's Guide. MathWorks, Natick, MA; 2009.

[12] MATLAB System Identification Toolbox User's Guide. The MathWorks, Natick, MA; 2009.

[13] Sahin A., Bayraktar A. ModalCAD—Interactive dynamic characteristic identification software for experimental and operational modal analysis of engineering structures. J. Test. Eval. 2010; 38(6): 738–758.

[14] ANSYS Finite Element Analysis System. (2007). ANSYS, Inc. 2600 Ansys Drive Canonsburg, Pennsylvania, USA

[15] Sahin A., Bayraktar A. Computational finite element model updating tool for modal testing of structures. Struct. Eng. Mech. 2014; 51(2): 229–248.

[16] MATLAB Optimization Toolbox User's Guide. MathWorks, Natick, MA; 2009.

[17] Sahin A., Bayraktar A. Forced vibration testing and experimental modal analysis of steel footbridge for structural identification. J. Test. Eval. 2014; 42(3): 695–712.

[18] Sahin, A., Bayraktar, A., Ozcan, D. M., Sevim, B., Altunisik, A. C., Turker, T. Dynamic field test, system identification, and modal validation of an RC Minaret: Preprocessing and postprocessing the wind-induced ambient vibration data. J. Perform. Constr. Facil. 2011; 25(4): 336–356.