Uludağ University Journal of The Faculty of Engineering, Vol. 25, No. 1, 2020 RESEARCH

DOI: 10.17482/uumfd.643339

IDENTIFICATION MATERIAL PROPERTIES BY MODAL CALIBRATION METHOD BASED ON AMBIENT VIBRATION TESTS

Yusuf YANIK [*]
Temel TÜRKER [*]
Ömer YILDIRIM [**]
Tayfun DEDE [*]

Received: 05.11.2019; revised: 20.02.2020; accepted: 25.02.2020

Abstract: Nowadays, different structural members can be used in structures. The mechanical properties of the materials are variable. For this reason, it is very important to identify the material properties correctly. The main parameter used to define the material properties is the elasticity modulus. In this study, the elasticity modulus of structural members was determined by Model Calibration Method using dynamic characteristics obtained from Ambient Vibration Test data. The application of the proposed method was presented on a steel structure member. First of all, experimental natural frequencies were obtained by conducting Environmental Vibration Test on the selected model. A measurement system consisting of accelerometers for experimental measurements and an electronic circuit used as signal collection unit are designed. The data obtained from this measurement system were transferred to the computer environment and analyzed in the Matlab program. Using the SAP2000 package program, the theoretical natural frequencies were found by Finite Element Method. The difference between the experimental and theoretical results was reduced to the lowest as a result of repeated analysis by modifying the elasticity modulus in the Finite Element model created in SAP2000 program via software generated in Matlab. The calibrated finite element model and the elasticity modulus of the steel structural member are determined by the process.

Keywords: Elasticity modulus, Natural frequencies, Material property, Modal analysis, Modal calibration, Vibration test

Malzeme Özelliklerinin Titreşim Testi Verilerine Dayalı Model Kalibrasyon Yöntemiyle Belirlenmesi

Öz: Günümüzde yapılarda çok farklı türde yapı malzemeleri kullanılabilmektedir. Malzemelerin mekanik özellikleri değişkenlik göstermektedir. Bu nedenle malzeme özelliklerinin doğru olarak belirlenmesi oldukça önemlidir. Malzeme özelliklerinin tanımlanmasında kullanılan başlıca parametre elastisite modülüdür. Bu çalışmada, yapı elemanlarının elastisite modülü Çevresel Titreşim Testi verilerinden elde edilen dinamik karakteristikler kullanılarak Model Kalibrasyon Yöntemiyle belirlenmiştir. Önerilen yöntemin uygulaması bir çelik yapı elemanı üzerinde sunulmuştur. İlk olarak seçilen çelik yapı elemanı üzerinde Çevresel Titreşim testi yapılarak deneysel olarak doğal frekanslar elde edilmiştir. Deneysel ölçümler için ivmeölçerlerden ve sinyal toplama ünitesi olarak kullanılan elektronik devreden oluşan bir ölçüm sistemi tasarlanmıştır. Bu ölçüm

---

[*] Karadeniz Teknik Üniversitesi, Mühendislik Fakültesi, İnşaat Mühendisliği Bölümü, TRABZON

[**] Gümüşhane Üniversitesi, Bilgisayar Teknolojisi Bölümü, GÜMÜŞHANE

Correspondence Author: Temel TÜRKER (temelturker@ktu.edu.tr)

sistemiyle alınan veriler bilgisayar ortamına aktarılmış ve Matlab programında analiz edilmiştir. SAP2000 paket programı kullanılarak Sonlu Eleman Yöntemiyle de teorik doğal frekanslar bulunmuştur. Deneysel ve teorik sonuçlar arasındaki fark, Matlab'da oluşturulan yazılım aracılığıyla SAP2000 programında oluşturulan Sonlu Eleman modelinde elastisite modülü değiştirilerek yapılan tekrarlı analizler sonucunda en aza indirilmiştir. Bu işlemler sonucunda çelik elemanın kalibre edilmiş Sonlu Eleman modeli ve elastisite modülü elde edilmiştir.

Anahtar Kelimeler: Elastisite modülü, Doğal frekanslar, Malzeme özelliği, Modal analiz, Model kalibrasyon, Titreşim testi

1. INTRODUCTION

The behavior of the structures under the effect of static and dynamic loads varies depending on the material properties. Accurate determination of structural behavior is possible by correctly determining the elasticity modulus, which is the main mechanical property of the structural elements. There are two main methods for determining the elasticity modulus. In the first method, when the samples are examined in the laboratory environment, the second method is examined on the structure element with undamaged procedure. Today, nondestructive evaluation in many structures, especially historical buildings are preferred. There are many methods to define the properties of materials without damaging by developing technology. At the beginning of these methods, the evaluations based on the vibration test data come first.

There are a lot of studies by using vibration test intended for material properties of structural structures, structural behavior and damage detection. In these studies, it was determined that Model Calibration-Update Methods were mostly used. Brownjohn and Xia (2000) tried to determine the dynamic behavior of the Safti Link Bridge, the suspension bridge in Singapore. The changes in the elasticity modulus of concrete were determined by using Finite Element Model Analysis. In this study, Finite Element Analysis was performed by using ANSYS together with FEMtools. In the study, based on the modal data obtained from the first measurement experiments, it has been tried to improve the finite element model by correcting certain non-specific structural parameters such as structural geometry and elasticity modulus of the concrete. Gibson (2000) has identified the elasticity modulus and the damping factors of the composites and components of the modal test under various environmental conditions in single-mode or multiple-vibration mode. The modal test method has been demonstrated to be a fast and accurate approach for both internal properties and quality control. Measurements, the distribution of reinforcing fibers within the composites with general elastic constants of composites, the time-domain creep response of composites, the elevated temperature behavior of composites and their components are applied to characterize the fracture toughness of the composites in the intermediate layers. Kaewunruen and Remennikov (2005) used the experimental modal test method to solve the problems related to the identification of the dynamic properties of railway track rails in laboratory and field conditions. The modal test was used to determine the dynamic stiffness and damping parameters of different age track pads. The frequency ranges used in the tests were taken as variable for the range of resonance frequencies and mode shapes that are important for each specific element. Shi et al., (2006) have developed an inverse method for material identification according to measured flexural resonance frequencies of sandwich beams. The method was applied to numerically generated test data and experimentally measured data in the literature. An error estimation system was designed to explore and discuss the main sources of error. Bayraktar et al., (2007) in order to investigate the effect of the Finite Element Model improvement on the earthquake behavior in the Akçaabat district of Trabzon Province, they conducted a three dimensional modal analysis with ANSYS. They determined the natural frequency and mode patterns in the analysis. Then, the dynamic characteristics of the bridge were determined by Operational Modal Analysis Method. Experimental and theoretical characteristics were compared with each other and the Finite Element model reflecting the current state of the bridge was prepared. Abeele et al., (2010) studied a material identification technique following a reverse engineering scheme. The planar elastic properties were determined by a dynamic module definition using resonance frequencies. Türker (2011) has worked on an approach that enables the detection and evaluation of the damage conditions of structures by considering the experimentally measured dynamic characteristics under environmental vibration. From the investigations carried out on laboratory models representing the basic engineering structures, the approach was found to be highly effective in determining and assessing the damage status of the structures. Dos Santos et al., (2013) evaluated the results of the elasticity modulus obtained for three different methods. The elasticity modulus was determined by using different methods static and dynamic by using the samples having dimensions of 150*30*20mm^3. It has been observed that the results obtained using different physical tests such as tensile, pressure, fracture, RFDA and ultrasonic velocity measurements of the modulus of modulus are consistent. Türker et al., (2015) obtained the dynamic characteristics of the 1/10-scale stone arch bridge model they have created in the course of their studies by the Environmental Vibration Test Method. Then, they used modal analysis of the model by using SAP2000 program. They evaluated the differences by comparing the experimental results and the results of the modal analysis. Kömür and Deneme (2015) have determined the dynamic characteristics of two symmetric and non-symmetric steel structures constructed by the laboratory by the Operational Modal Analysis Method. For this purpose, experimental measurements were taken using six uniaxial accelerometers in both steel structures. In the evaluation of these data, they tried to obtain the desired findings by using decomposition and stochastic subfield determination methods in the frequency domain of the ARTEMIS program. There were some differences between the results of both methods. Afterwards, these two steel structures were modeled numerically to obtain mode shapes and natural frequencies. It has been seen that there is quite a difference between the frequency values obtained from Finite Element modeling and the values obtained from experimental measurement. This difference is more evident in the non-symmetrical structure. Considering the finite element modeling of the accelerometer weights, the results were found to be more consistent. Prashant et al., (2015) used the Experimental Modal Analysis Method to obtain the natural frequencies, damping ratios and mode shapes of the rectangular cantilever beam. Frequency Response Functions and then natural frequency, damping ratios, and mode shapes were performed using the LabVIEW program using pulse hammer as the exciter source. The obtained modal parameters were then controlled using theoretical modal analysis. Experimental results and theoretical results were similar. Mansour et al., (2016) conducted the modal test of viscoelastic composite materials using a genetic algorithm by minimizing the difference between analytical experimental tests and calculated response, a function of modal parameters. The analytical transfer functions provided a sub-configuration to define the modes as a function of the damped natural frequencies and the lost factors of a complex structure and moved away from the experimental noise and the modal bonding effect. A cantilever of FEM was confirmed by calculating the elasticity modulus of the steel beam and compared with the results of the proposed algorithm. The effectiveness of the proposed method has been demonstrated on the static and dynamic behaviors of epoxy cantilever beam samples reinforced with silicon nanoparticles. Türker and Bayraktar (2017) have studied the effects of construction phases on the modal parameters of reinforced concrete buildings. For this purpose, a 1/3-scale three-storey reinforced concrete building model was constructed and modal test measurements were made by using the Operational Modal Analysis Method in the framework of building model framed, brick walls and plastered. The natural frequencies of the building model were quite different from the construction stages. Frequencies decreased with increasing number of floors. However, brick walls caused a significant increase in natural frequencies and the longitudinal response was determined to occur in the first mode. Slim et al., (2017) measured the elasticity modulus of coatings and substrates by the Pulse Excitation Method. A numerical Finite Element model was developed and used to find systematic errors from four analytical models. They then carried out an uncertainty study to assess errors and sources of uncertainty and to assess their contribution and evaluated the findings.

In this study, a method has been tried to be developed by using Ambient Vibration Test data to obtain the desired parameters without damaging the tested material beyond the traditional methods used to determine the material property. The developed system consists of hardware and software. In the hardware part, vibration measurement system was established to provide measurement from the structure. In the software phase, a software part was created to determine the structural parameters from the measured signals. After these parts were formed, measurement was carried out on the selected model, and theoretical analyzes of the model were made with FEM in order to compare the experimental and theoretical obtained values. The difference between the experimental and theoretical results was reduced to the lowest as a result of repeated analysis by modifying the elasticity modulus via generated software. The calibrated finite element model and the elasticity modulus of the steel structural member are determined.

2. METHOD

In this study, Model Calibration Method was used to determine the elasticity modulus. Model Calibration was the process of minimizing the difference between experimental and analytical natural frequencies with changes in the Finite Element model based on the measurement data obtained from the vibration test for the undamaged condition of the structure. Model Calibration process on the Finite Element model from material properties to the boundary conditions could be taken into account the change in many parameters. In order to perform the Model Calibration process, the characteristic values which are obtained experimentally and theoretically, were needed. It had been determined that natural frequencies are the most appropriate values for Model Calibration.

In this study, the elasticity modulus was taken into account as a variable and the difference between the Model Calibration process and the experimentally and analytically determined frequency values was minimized. For this, the calibration interval of the parameter was entered by entering the upper and lower limit values and the error amount in the program. When the difference between the frequency values reaches the desired level, the calibration process stoped automatically. Processes were presented systematically in block diagram in Figure 1 below.

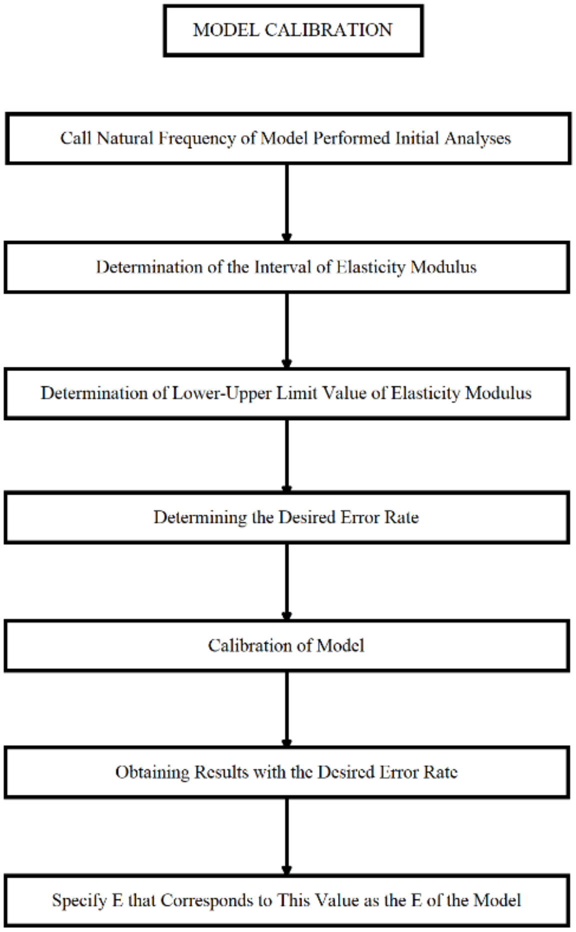


Figure 1:
Modal Calibration block diagram

3. APPLICATIONS

In this study, it was aimed to determine material properties by using Model Calibration Method. With this method, the characteristic values structure-specific which were determined experimentally and theoretically, were needed. For the theoretical characteristic values, natural frequencies calculated from the specially designed vibration measurement system were used as experimental characteristic values when using natural frequencies calculated on the Finite Element model. The developed measurement system consisted of two parts as hardware and software. The hardware part consisted of sensors and data collection unit. The software part consisted of an interface including signal processing, Finite Element model transfer and Model Calibration. Software user interface GUI feature was used in Matlab platform (1999). In order to determine the material properties of the selected steel model, the initial analytical model was created, and measurements were made under environmental vibrations on the accepted model representing the undamaged condition. Then the initial analytical model was calibrated according to undamaged experimental measurement data and material properties were determined.

3.1. Measurement Setup

Vibration measurement was primarily supported by the structure to be measured and the sensors that detect the vibrations of the structure were connected to the structure at regular intervals. The sensors were then connected to the developed data acquisition board. Vibration data from the sensors were arranged in the data acquisition card and transferred to the computer environment by using the aux cable to input the sound card. Thus, the physical connections of the measuring device were completed. Finally, the interface program developed for receiving vibration signals in the computer environment can be opened and measurements could be made after necessary adjustments.

3.1.1. Hardware Part

The hardware part was all of the equipment used to detect and transfer vibration data through structures and transfer them to the computer environment. Generally, the items forming the hardware part could be grouped under two subheadings as sensors and data acquisition card.

Sensors

Sensors can be considered as sensory organs in the human body. The devices used to detect environmental changes (vibration, pressure, temperature, etc.) that can be perceived by human senses in electronic environments were called sensors. In this study, three-axis analog accelerometers were used to detect the vibrations of the structure to be measured electronically (Figure 2). These accelerometers were able to detect vibrations in the X, Y and Z axes by means of the semiconductor technology within the axes and transmit them to the external environment as an analog and continuous signal. The general characteristics of the accelerometer were given below.

- Dimensions W*L*H: 19mm*19mm*3.14mm
- Weight without pins: 1.27gr
- Operating input voltage: 5V–5.5V
- Measuring range min: +-3G
- Measuring range typical: +-3.6G
- Measurement error: +-%0.30
- Error between flows: +-%1
- Sensitivity ratiometric (3V) min: 270mV/G
- Sensitivity ratiometric (3V) typical: 300mV/G
- Sensitivity ratiometric (3V) maks: 330mV/G
- Zero G output voltage (3V) min: 1.35V (Z=1.2V)
- Zero G output voltage (3V) typical: 1.5V
- Zero G output voltage (3V) maks: 1.65V (Z=1.8V)
- Operating current typical: 0.5mA
- Initial reaction time: 1mSecond

There were six pins on the accelerometer module. The Positive Supply Voltage (VCC) pin from these pins was the pin with a five-volt voltage connected to the module via USB. The Negative Supply Ground (GND) pin represented the reference end, that is, the ground end. X, Y and Z were the pins on which the analog output of the vibrations in the axes was made. The self-test (ST) pin was the pin in the accelerometer, which collects the vibrations from all axes without regard to any axis. These pins were transferred to the data acquisition card using a five-core and shielded cable.

Figure 2:
ADXL335 triaxial analog accelerometer module

Data Acquisition Card

The most important part of the developed system consisted of the data collection card, which enables the transfer of vibration data to the computer environment. Vibration data from three-axis analog accelerometers and converted into electrical signals are collected by means of a data acquisition card. This analogue data was converted back to the computer environment and converted to digital data that the computer could process.

The electrical signals from two different analog accelerometers were placed in two different locations in the collection circuit of the data acquisition card. Considering the frequency analysis in the developed system, the vibration data from the analog accelerometers placed on different places on the structure did not cause any loss or change in the frequency contents of these signals. As it was known from the Fourier transform, when the two signals with different frequencies were collected and the Fourier transform was applied to these signals, the frequency information of the signals could be obtained separately. Therefore, by using two analog accelerometers, data from these sensors were collected by means of the collecting circuit and made into a single signal to detect different frequency components that were dominant in different parts of the structure. In addition, low amplitude signals from analog accelerometers were subjected to a small pre-upgrade by means of the collection circuit. The design of the data acquisition card and the drawing of the printed circuit were performed with the Proteus 8 (2017) program (Figure 3).

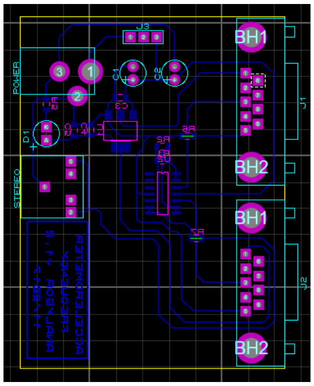


Figure 3:
Data collection card designed in Proteus 8

Top and bottom view of the data acquisition card designed in the Proteus 8 program and the final design of the data acquisition card produced on the copper plate as a result of the design is shown in Figure 4.

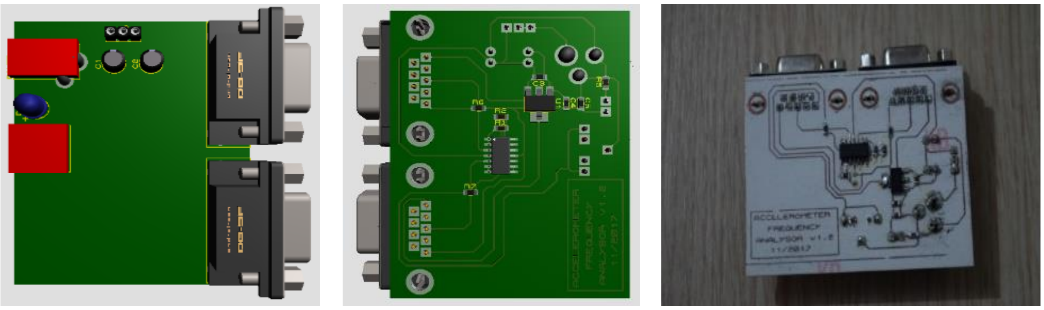


Figure 4:
The top and bottom view of the data collection board and the view on the copper planet

3.1.2. Software Part

There were several software packages for signal processing. The use of ready-made software packages had many advantages and sometimes disadvantages. Therefore, it was more advantageous to create the desired software by writing original and purpose oriented functions. In this study, the signals taken from the model to determine the material properties of the selected steel model were evaluated by the program prepared in Matlab environment developed by Yanık (2018). The interface program created in Matlab environment consists of three parts (Figure 5).

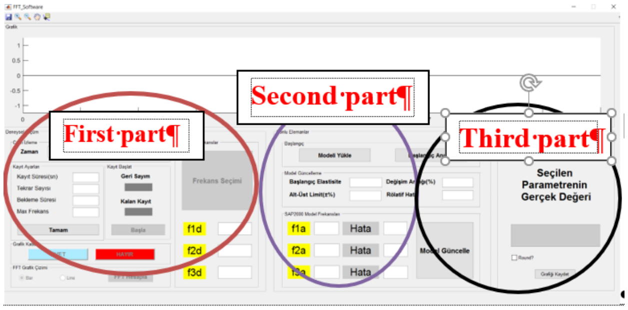
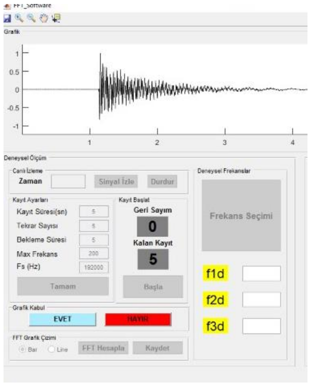


Figure 5:
Measurement and calibration interface formed in Matlab environment

Part One: Experimental Modal Analysis

The first part was the part of the FFT transformations of experimental measurements and vibration data (Figure 6). Thanks to this part, the digital vibration data received on the sound card was transferred graphically to the interface. After the sensor connections were made. Thanks to this part, there was 'Live Monitoring' command which allows the sensors to be checked in real time and to check whether the sensors provide the desired outputs. This part also contained 'Recording Settings', which consists of recording time, number of repetitions, waiting time, maximum frequency and sampling frequency (fs) required for measurement; 'Start Record', which consists of beginning the measurement, selecting and saving data; 'Graphic Acceptance and Save' commands. The graphic in which the FFT transformation of a typical measurement signal was plotted is shown in Figure 7.

In addition, thanks to ‘Save’ button, the data were recorded in ‘.txt’ format and analyzed in the Operational Modal Analysis (2006) program.



Kayıt Süresi(sn) 5
Tekrar Sayısı 5
Bekleme Süresi 5
Max Frekans 200
Fs (Hz) 192000
Geri Sayım 0
Kalan Kayıt 5

Figure 6:
Experimental Modal Analysis part

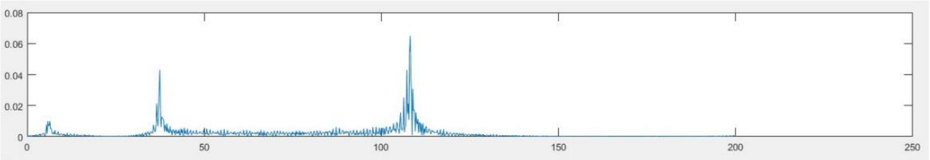


Figure 7:
Graphic of FFT conversion

Part Two: Finite Element Modeling Part

In the second part of the Finite Element section, the element which was examined in the SAP2000 (2008) program is called by the ‘Load the Model’ command in the interface and initial Finite Element analysis was performed and the natural frequency values of the model were defined. These frequency values are automatically transferred to the interface in Matlab environment (Figure 8).

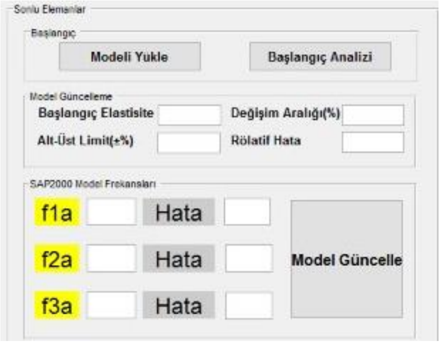
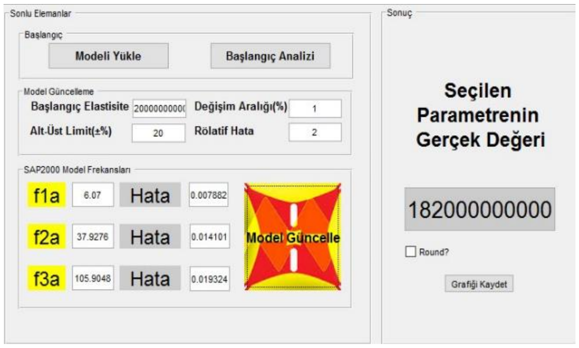


Figure 8:
Finite Element model transfer screen

Part Three: Model Calibration Part

The third part, Model Calibration, involved the reduction of the difference between experimental and analytical frequency values. In this study, the elasticity modulus was considered as variable and the difference between the Model Calibration process and the experimental and analytical determined frequency values was minimized. When calibrating the model, the initial value of the elasticity modulus selected as the change parameter was automatically transferred from the SAP2000 program to the interface in Matlab environment (Figure 9). Then, the change interval of the parameter, the upper and lower limit values and the amount of error were entered into the program and calibrated. When the difference between the experimental and analytical frequency values was reached to the desired level, the calibrating process stopped and the elasticity modulus of the selected material was automatically written to the result section.



Başlangıç Elastisite 2000000000
Değişim Aralığı(%) 1
Alt-Üst Limit(±%) 20
Rölatif Hata 2

f1a 6.07 Hata 0.007882
f2a 37.9276 Hata 0.014101
f3a 105.9048 Hata 0.019324

Seçilen Parametrenin Gerçek Değeri
182000000000
☐ Round?

Figure 9:
Model Calibration part

3.2. Steel Model Application

The selected steel model was supported with a cantilever beam. The cantilever beam consisted of a 3.0cm*0.5cm rectangular steel profile and the total length is 80cm. Figure 10 showed the geometrical properties of the steel cantilever beam model in three dimensions.

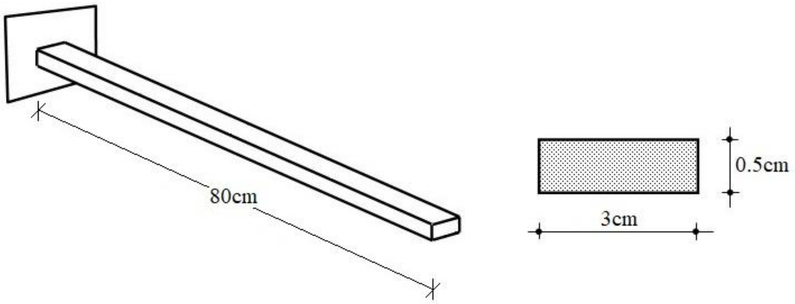


Figure 10:
Three-dimensional appearance and dimensions belong to steel console beam model

The steel cantilever beam model was modeled analytically by the Finite Element Method (FEM) using the SAP2000 program. The bearing conditions at one end of the model were accepted as fixed end and the others were released. The analytical model created to represent the cantilever beam was given in Figure 11.

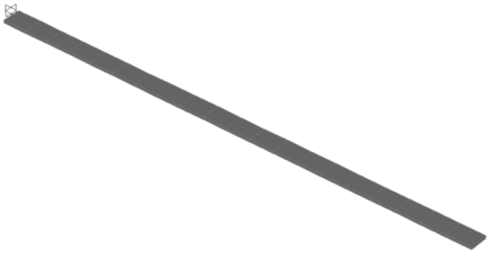


Figure 11:
Finite element model of steel cantilever beam

In order for the analytical model to represent the real behavior of the cantilever beam, the model must be divided into elements. The analytic model was divided into 20 equal parts by considering the Mesh Convergence and it was concluded that the results obtained were sufficient. The material properties used to form the analytical model of steel cantilever beam are presented in Table 1.

Table 1. Material properties and values of steel model

Material Properties	Value
Elasticity Modulus (N/m^2)	$2.0*10^{11}$
Unit Mass (kg/m^3)	7697.286
Section Area (m^2)	$1.5*10^{-4}$
Sections Moment of Inertia (m^4)	$0.3125*10^{-9}$
Poisson Ratio	0.25

Under the vibration and random vibrations generated by the environment, steel cantilever beam model was measured by Environmental Vibration Test Method. In the experimental measurement performed on the steel cantilever beam elements, measurements were made using two three-axis accelerometers. The accelerometers were placed on the model to measure the responses in the vertical direction. This placement had been carried out at regular intervals. In the vibration measurement, the support conditions of the steel model were assumed as completely fixed in one end and the weight of the accelerometers were ignored. Figure 12 showed the measurement device for the steel cantilever beam model.

Figure 12:
Measuring system for steel cantilever beam model

The generated measurement system had the capability to receive 192000 data in one second. The sampling frequency range was selected as 0-1000Hz in the measurement performed on the steel console model. The recording time was selected 180 seconds. During the experimental measurement, the signals from the accelerometers were collected in the data acquisition card and then simultaneously transferred to the computer with Fourier transforms in the interface program. An example of the received signal was given in Figure 13. After FFT transformations of the signals, frequencies are obtained graphically. Then, these frequencies on the graphic were obtained numerically using the Peaks' Selection Method. Experimental measurement frequencies of the steel cantilever beam element were obtained using this method. These frequencies obtained by using the environmental vibration data were transferred to the buttons on the program.

Kayıt Süresi(sn) 180
Tekrar Sayısı 1
Bekleme Süresi 1
Max Frekans 200
Fs (Hz) 1000

Figure 13:
Received signal example from steel model

The first three frequency values obtained from the processing of the signals obtained from the measurement in the steel console model were given in Table 2. The mode shapes were not obtained from the experimental measurement.

Table 2. Natural frequencies obtained from experimental measurements by Matlab

Mode Number	Frequency (Hz)
1	6.02
2	37.40
3	107.99

In addition, the signals obtained were analyzed in the Operational Modal Analysis program to find natural frequencies and damping rates as shown in Figure 14. The frequencies obtained were found to be compatible with each other. The transmitted signal sample was transferred to the Operational Modal Analysis program. The diagram showing the frequency values obtained by processing the signals obtained for the steel model with the help of SSI-PC command is given in Figure 15.

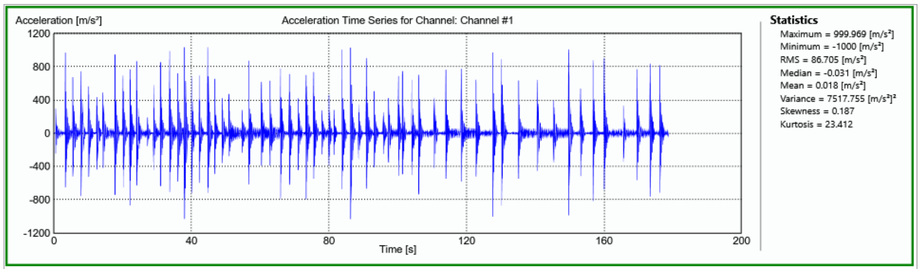


Figure 14:
Signal sample transferred to Operational Modal Analysis program

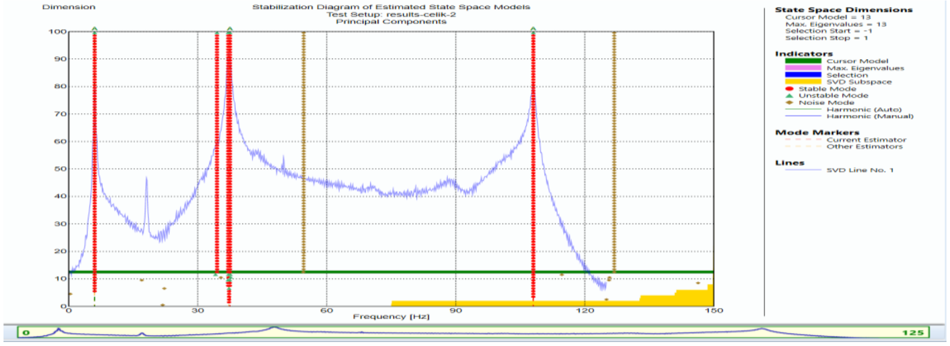


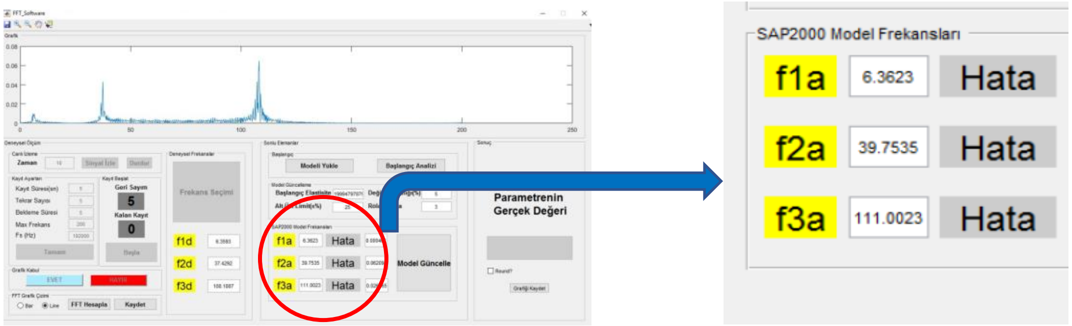
Figure 15:
Stability diagram obtained with OMA software

The frequency and damping values obtained from the OMA program were presented in Table 3.

Table 3. Natural frequencies and damping rates obtained in the OMA program

Mode Number	Frequency (Hz)	Damping Ratio [%]
1	6.02	1.20
2	37.49	1.06
3	108.03	0.45

In the interface (GUI) formed in the Matlab program, the model that was previously prepared in the SAP2000 program was called to the Matlab environment by pressing the ‘Finite Element’ button in the program to print the analytical frequency values of the steel cantilever beam model in the section prepared as the Finite Element section. After calling the model, the initial natural frequency values of the model were obtained by pressing the ‘Initial Analysis’ button. These values were automatically printed in the corresponding section (Figure 16).



SAP2000 Model Frekansları

f1a 6.3623 Hata

f2a 39.7535 Hata

f3a 111.0023 Hata

Figure 16:
The initial frequency values obtained after the finite element method

During the generated of the initial analytical model of the steel cantilever beam model, acceptance was made under material properties and boundary conditions. The acceptance elasticity value in the material properties was at the given standards. In the boundary conditions, it could be said that the acceptance of one blade of the steel cantilever girder was defined as fixed end.

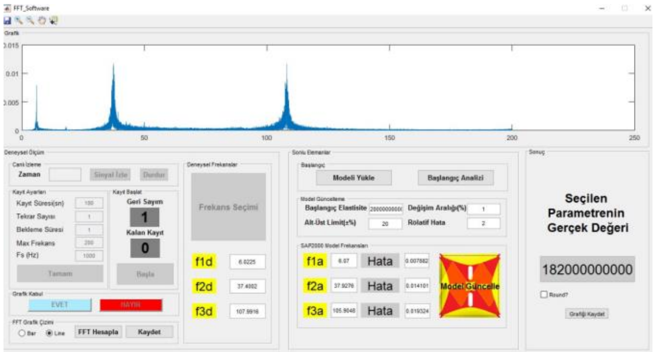
When the experimental and analytical study results were examined, it was seen that there was a difference between the frequency values obtained (Table 4). The reasons for the differences between experimental and analytical results could cause the assumptions in the finite element modeling phase including support condition, weight of accelerometer, initial material properties, etc. In this study, it was assumed that the reason of the differences occurred from the initial material properties. The differences need to be minimized. Experimental and analytical frequency values were tried to be converged by selecting the elasticity modulus as the change parameter. In this way, the material properties of the structural element examined would be determined.

Table 4. Experimental and initial analytical frequencies of the cantilever beam model

Mode Number	Natural Frequencies (Hz)		Difference*(%)
	Experimental	Analytical	
1	6.02	6.36	5.65
2	37.40	39.75	6.28
3	107.99	111.00	2.79

*The difference value is calculated by absolute value.

In the initial analytical model, the elasticity value change step was assumed to be 1%, the lower-upper limit of the elasticity module was 20% and the error rate between the frequencies was 2% (Figure 17).



Kayıt Süresi(sn) 100
Tekrar Sayısı 1
Bekleme Süresi 1
Max Frekans 200
Fs (Hz) 1000
Geri Sayım 1
Kalan Kayıt 0
f1d 6.0225
f2d 37.4002
f3d 107.9916
Başlangıç Elastisite 2000000000
Değişim Aralığı(%) 1
Alt-Üst Limit(±%) 20
Rölatif Hata 2
f1a 6.07 Hata 0.007582
f2a 37.9276 Hata 0.014101
f3a 105.9048 Hata 0.019324
182000000000

Figure 17:
Screenshot of calibrating the initial model of the steel model

As a result of the iterations, the allowed convergence criterion was obtained and the natural frequencies were gained for the attained elasticity value. The values obtained as a result of the calibration taking into account the change in the elasticity value of the initial model were given in Table 5.

Table 5. The experimental and calibrated initial analytical frequencies of the cantilever beam model and the elasticity modulus

Mode Number	Natural Frequencies (Hz)		Difference*(%)	Determined Elasticity Modulus (N/m^2)
	Experimental	Analytical		
1	6.02	6.07	0.83	$1.82*10^{11}$
2	37.40	37.93	1.42	
3	107.99	105.90	1.94	

* The difference value is calculated by absolute value.

As a result of calibrating the initial analytical model of the steel model, the average difference between frequencies could be reduced from 4.91% to 1.4%. In contrast to this reduction, the elasticity of the steel model was reduced from $2.0*10^{11}$N/m^2 to $1.82*10^{11}$N/m^2. When the results were evaluated, the correlation between frequency was obtained by changing the elasticity module by 9%.

4. CONCLUSION

In this study, according to the Ambient Vibration Test and Finite Element data, a method had been proposed to determine the material properties of the structural elements by using Model Calibration Method. The proposed method was applied on the steel cantilever beam model. In the study, an interface system for the Ambient Vibration Test and an interface program for Model Calibration was developed. The current value of the elasticity modulus was determined by taking into account the differences between experimental and analytical natural frequency values. The results of this study were listed below:

- A unique, open-coded and developed measurement system was generated for the measurements by the Experimental Modal Analysis Method.

- For the Model Calibration of the Finite element, an interface program consisting of the Environmental Vibration Test, Finite Element Analysis and Model Calibration is formed.
- Natural frequencies of the selected steel model's from Vibration Test were determined as 6.02Hz, 37.40Hz and 107.99Hz.
- Natural frequencies from the Finite Element Analysis of the steel model were calculated as 6.36Hz, 39.75Hz and 111.00Hz.
- Between the experimental and analytical frequencies obtained for the steel model, an average difference of 4.91% was determined. The average difference between the frequency values with the model calibration process performed on the variable elasticity modulus was reduced to 1.4%.
- The natural frequencies of the calibrated steel element were determined as 6.07Hz, 37.93Hz and 105.90Hz.
- While the elasticity modulus of the steel model was initially $2.0*10^{11}N/m^2$, it was determined as $1.82*10^{11}N/m^2$ for the current situation as a result of Model Calibration.

REFERENCES

1. Abeele, V. D. F., Oliveira J. J. R. and Huertos F. J. (2010) Identification of the Complex Moduli of Orthotropic Materials Using Modal Analysis, Excerpt from the Proceedings of the COMSOL Conference, Paris
2. Bayraktar, A., Altunışık, A. C., Türker, T. and Sevim, B. (2007) Effect of Finite Element Model Improvement to the Earthquake Behavior of Historical Bridges, Sixth National Earthquake Engineering Conference, October, Istanbul, Proceedings Book: 29-39
3. Brownjohn, J. M. and Xia, P. Q. (2000) Dynamic Assessment of Curved Cable-Stayed Bridge by Model Updating, Journal of Structural Engineering, 126, 2, 252-260. https://doi.org/10.1061/(ASCE)0733-9445(2000)126:2(252)
4. Dos Santos, J. P. L., Amaral, P. M., Diogo, A. C. and Rosa, L. G. (2013) Comparison of Young's Moduli of Engineered Stones Using Different Test Methods, In Key Engineering Materials, 548, 220-230, Trans Tech Publications. https://doi.org/10.4028/www.scientific.net/KEM.548.220
5. Gibson, R. F. (2000) Modal Vibration Response Measurements for Characterization of Composite Materials and Structures, Composites science and technology, 60, 15, 2769-2780. https://doi.org/10.1016/S0266-3538(00)00092-0
6. Kaewunruen, S. and Remennikov, A. (2005) Application of Experimental Modal Testing for Estimating Dynamic Properties of Structural Components, Australian Structural Engineering Conference, Newcastle, Australia. https://ro.uow.edu.au/engpapers/283/
7. Kömür, M. A. and Deneme, İ. O. (2015) Operational Modal Analysis of Symmetric and Non-Symmetrical Steel Structures, Ömer Halisdemir University Journal of Engineering Sciences, 5, 1, 64-72
8. Mansour, G., Tsongas, K. and Tzetzis, D. (2016) Modal Testing of Nanocomposite Materials Through an Optimization Algorithm, Measurement, 91, 31-38. https://doi.org/10.1016/j.measurement.2016.05.032
9. Matlab, (1999). Mathworks Inc, MATLAB User Guide, Natick, MA
10. OMA, (2006). Operational Modal Analysis, Release 4.0. Structural Vibration Solution A/S, Denmark.

11. Prashant, S. W., Chougule, V. N. and Mitra, A. C. (2015) Investigation on Modal Parameters of Rectangular Cantilever Beam Using Experimental Modal Analysis, Materials Today: Proceedings, 2 (4-5), 2121-2130. https://doi.org/10.1016/j.matpr.2015.07.214

12. Proteus, (2017). Beechcroft House 21 Hardy Grange Grassington North Yorkshire BD23 5AJ, https://www.labcenter.com/contact

13. SAP2000, (2008). Integrated Finite Element Analysis and Design of Structures, Computers and Structures Inc, Berkeley, California, USA

14. Shi, Y., Sol, H. and Hua, H. (2006) Material Parameter Identification of Sandwich Beams by an Inverse Method, Journal of Sound and Vibration, 290(3-5), 1234-1255. https://doi.org/10.1016/j.jsv.2005.05.026

15. Slim, M., Alhussein, A., Billard, A., Sanchette, F. and François, M. (2017) On the Determination of Young’s Modulus of Thin Films with Impulse Excitation Technique, Journal of Materials Research, 32, 3, 497-511. https://doi.org/10.1557/jmr.2016.442

16. Türker, T. (2011). Structural Damage Detection and Evaluation by Using Ambient Vibration Data, PhD. Thesis, Karadeniz Technical University, Institute of Science and Technology, Trabzon, Turkey

17. Türker, T., Bayraktar, A., Kocaman, İ. and Çoruhlu, B. (2015) Experimental and Analytical Investigation of the Dynamic Behavior of the Scale Masonry Belt Bridge Model, 5. Strengthening of Historical Artifacts and Safely Transferring to the Future Symposium, Erzurum, Proceedings: 113-126

18. Türker T. and Bayraktar A. (2017) Vibration Based Modal Testing of a Scaled Reinforced Concrete Building for Construction Stages, Bulletin of Earthquake Engineering, no.5, pp.1-18. https://doi.org/10.1007/s10518-015-9852-9

19. Yanik, Y., (2018). Determination By Modal Calibration Method Based Upon Vibration Test Data of Material Properties, Master Thesis, Karadeniz Technical University, Institute of Science and Technology, Trabzon