IOMAC'19

8th International Operational Modal Analysis Conference

2019 May12-14 Copenhagen

A NUMERICAL BENCHMARK FOR SYSTEM IDENTIFICATION UNDER OPERATIONAL AND ENVIRONMENTAL VARIABILITY

Konstantinos Tatsis [1], *and Eleni Chatzi* [2]

[1] PhD, ETH Zürich, tatsis@ibk.baug.ethz.ch.

[2] Associate Professor, ETH Zürich, chatzi@ibk.baug.ehhz.ch.

ABSTRACT

Structural Health Monitoring (SHM) has been experiencing a very rapid growth during last decades, with a wide range of methods developed to support decisions for maintenance planning over a broad spectrum of operational and environmental conditions. Despite this progress, performance evaluation of various approaches is usually carried out on different case studies, rendering side-by-side comparison difficult. This contribution presents a numerical benchmark for system identification under changing environmental and operational conditions. The considered system is modelled with an open access Finite Element code, which is supplemented with a Graphical User Interface (GUI) and enables the extraction of modal properties and dynamic response data for a number of predefined damage patterns and operational as well as environmental scenarios. This benchmark, generated as part of COST Action TU1402 on Quantifying the Value of Information, is to serve as a reference case study for validation of decision-making tools relying on the Value of Information.

Keywords: Structural dynamics, Damage assessment, Data analysis, Value of information, Benchmark

1. INTRODUCTION

The very rapid growth of research in the field of SHM, renders benchmarking an important step for performance assessment. The existence of a baseline, which will enable the straightforward and quantitative evaluation of the large assortment of solution methods, will undoubtedly assist in consolidating the recent advances in the field. Among various identification methods, a number of contributions is focused on the identification of systems featuring environmental variability under different operational conditions. To this end, the present work aims at providing a simple numerical benchmark that can address such aspects.

Benchmarking is not a new idea in the context of SHM, since a number of standardized problems is already available for performance evaluation. The first study within that context is provided by Farrar et al. [1] on the dynamic testing of the I-40 bridge. Using the same structure as test-bed, Maeck and De Roeck [2] conducted a well-known benchmark by imposing a series of damage scenarios thereto, offering thus the material for testing and validation of detection, localization and quantification methods. Similarly, Molina et al. [3] established an experimental study using a composite steel-concrete two-story frame. A full-scale study on a bridge structure was set-up by the Center of Structural Monitoring and Control at Harbin Institute of Technology [4] in recent years, while Smith and Randall [5] established a benchmark based on the data from the Case Western Reserve University (CWRU) Bearing Data Centerand Joyce et al. [6] offered a test case for high-rate problems. Such contributions are not limited to experimental cases, since a number of noteworthy numerical studies is also available, with the IASC-ASCE SHM Task group offering one of the most influential benchmarks [7], which is essentially the numerical part of the companion experimental work by Dyke et al. [8]. Lastly, it should be underlined that benchmark studies are well extended to more complex and challenging systems featuring nonlinear behavior, like the work carried out by Tiso and Noël [9].

This paper introduces a simple numerical benchmark which is generated as part of COST Action TU1402 on Quantifying the Value of Information and is to serve as a reference case study for validation of decision-making tools relying on the Value of Information. The benchmark model is herein described along with the open source code for generation of simulated data. Although a default set-up of the benchmark is presented, the interested researchers are encouraged to extend and enhance the functionalities as desired. Within this context, the paper is organized as follows: the second section presents the numerical model used as benchmark along with the employed damaged patterns and the different operational loads. The third section describes the process followed for response simulation, while the fourth section serves as a manual for the Graphical User Interface (GUI), which is publicly available in GitHub.

2. BENCHMARK MODEL

The structure considered as benchmark model is a two-span beam as illustrated in Figure 1. The height of the beam is $h = 0.6$ units, the width is equal to $w = 0.1$ units and each span has a length of $L = 10$ units. The model is assigned a linear elastic material with Young's modulus $E = 2.1e11$, Poisson's ratio $\nu = 0.3$ and material density $\rho = 7800$ units. The structure is represented by a FE model which is constructed with isoparametric quadrilateral elements for plane-stress problems and a predefined discretization scheme with 200 and 6 elements in x and y directions, respectively, is used. To minimize the effect of locking, the implemented elements comprise 9 nodes and their stiffness and mass matrices are integrated with a Gauss scheme of 3×3 point rule.

Elastic boundaries are assumed for all three support points, in both x and y directions, with a support

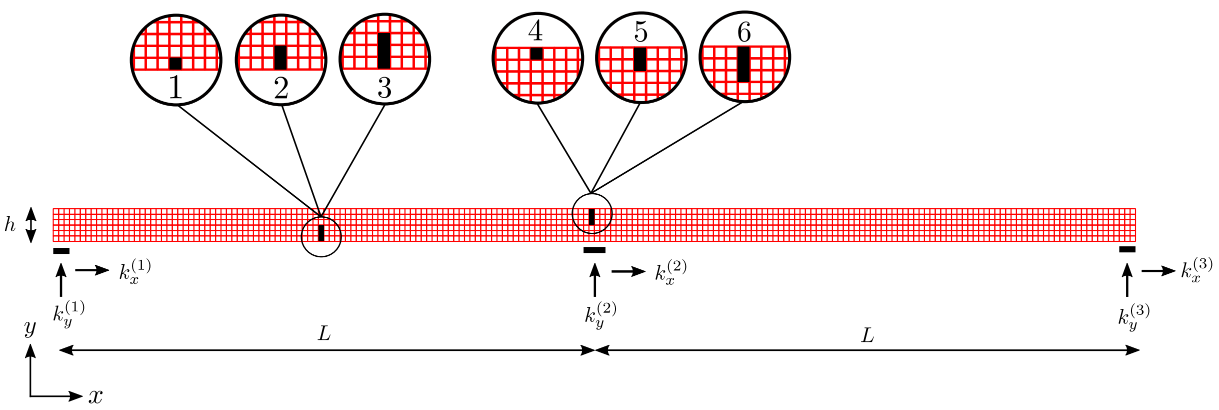


Figure 1: Finite element model of the benchmark structure.

width of 0.3 units for the two extremities and 0.4 units for the midpoint. It should be noted that all mentioned properties are merely proposed as default values for the reference state and the users are allowed to adjust them as deemed necessary. Except for the system dimensions, as will be presented in Section 4., a temperature-dependent model can be assigned to each variable, including boundary conditions, enabling thus the simulation under different environmental conditions.

2.1. Damage patterns

In addition to the reference structure described above, two groups of damaged cases, with a total of six scenarios as numbered in Figure 1, are studied, where each group is materialized with a stiffness reduction in the specified area. The first group assumes a damaged region in the mid of the first span, starting from the lower level of the cross section, while the second one comprises a damaged region above the mid-point support. Both groups advance from an area consisting of a single element up to a zone of three elements, as shown in Figure 1, with a width of 0.1 units and a height ranging from 0.1 to 0.3 units, with the latter being equal to half of the cross section height. It should be noted that these scenarios are not intended to represent the mechanisms and physics of actual damage but instead, provide the means for performance testing of various SHM methods in system changes. To this end, the amount of damage for each scenario is not *a priori* defined, but it can be specified through the percentage of stiffness reduction which ranges between 10 and 100%. To further enable the simulation of degradation effects, such as corrosion [11, 12], the thickness of the structure can be spatially reduced by specifying a thickness wastage pattern in the longitudinal direction.

2.2. Load cases

For each model case, three default sets of excitations can be used, as illustrated in Figure 2. These include (a) a mass of 100 units moving along the structure with a unit speed towards positive direction of x axis, and two fixed-point loads at the (b) left- and (c) right-hand side spans respectively. Although the excitation signal of the moving load is of deterministic nature, the signals of fixed-point loads are modeled as independent Gaussian white noise whose time series are stored in data files which can be modified accordingly.

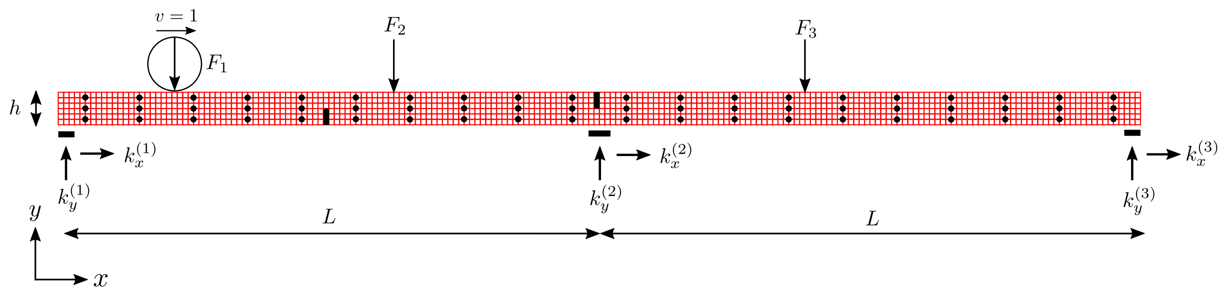


Figure 2: Load cases and measurement positions

3. RESPONSE SIMULATION

This section is focused on the numerical integration of the equations of motion, which is carried out to obtain the response induced by the different load cases. Given the initial order of the model, the equations are solved in the modal domain which is described by the generalized coordinates $\mathbf{q}$. These are related to the physical degrees of freedom by $\mathbf{u} = \mathbf{\Phi q}$, where $\mathbf{\Phi}$ is the mode shape matrix and the equations of motion in modal coordinates are then given by

$$\hat{\mathbf{M}}\ddot{\mathbf{q}}(t) + \hat{\mathbf{C}}\dot{\mathbf{q}}(t) + \hat{\mathbf{K}}\mathbf{q}(t) = \mathbf{\Phi}^{\mathrm{T}}\mathbf{F}(t) \tag{1}$$

where $\hat{\mathbf{M}}$, $\hat{\mathbf{C}}$ and $\hat{\mathbf{K}}$ are the modal mass, damping and stiffness matrices, respectively, while $\mathbf{F}$ is the vector of forces acting on the physical degrees of freedom. Although the mass and stiffness matrices are internally defined upon selection of a model case, the damping matrix is assumed to be proportional, obtained as follows

$$\mathbf{C} = \alpha \mathbf{M} + \beta \mathbf{K} \tag{2}$$

and the values of coefficients α and β are left for the user to specify, as will be described in the following section.

By default, the equations of motion are integrated using an implicit Newmark scheme [13], with parameters $\gamma = 1/2$ and $\beta = 1/6$. To render the analysis computationally affordable and enable the efficient execution of a series of jobs, the dynamics of the system are represented by the first ten vibration modes, which also constitutes a default setting and can be modified as deemed necessary. For each model scenario and load case, the acceleration and strain response from a number of structural points are exported as measured signals. The location of these points is depicted in Figure 2, for which the vertical component of acceleration and all three components, ϵ_{xx}, ϵ_{yy} and τ_{xy} of strains are measured.

4. DATA GENERATION

The modeling features described above, along with the response simulation scheme, are included into an open access Python code which is made available through GitHub. The code is wrapped with a Graphical User Interface (GUI) and offers the possibility of performing eigenvalue and time history analyses on the model under different operational and environmental conditions. Moreover, it provides the capability of extracting acceleration and strain measurements from a number of sensing points, which are arranged in a structured grid as shown in Figure 2. It should be noted here that both back-end and front-end of the benchmark model are publicly available in the form of an open source code and the readers are encouraged to contribute and freely extend the functionalities presented herein.

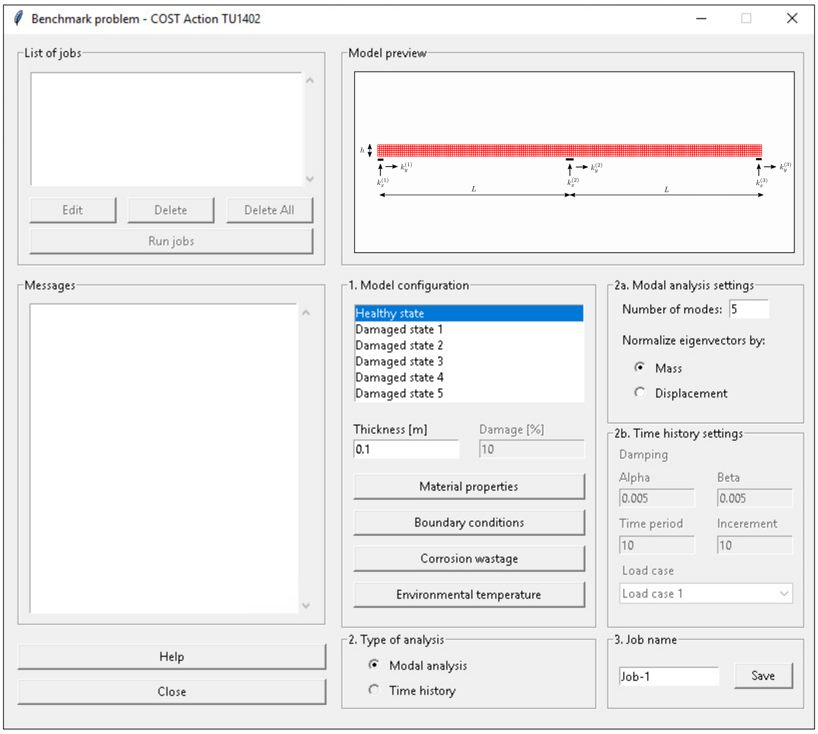
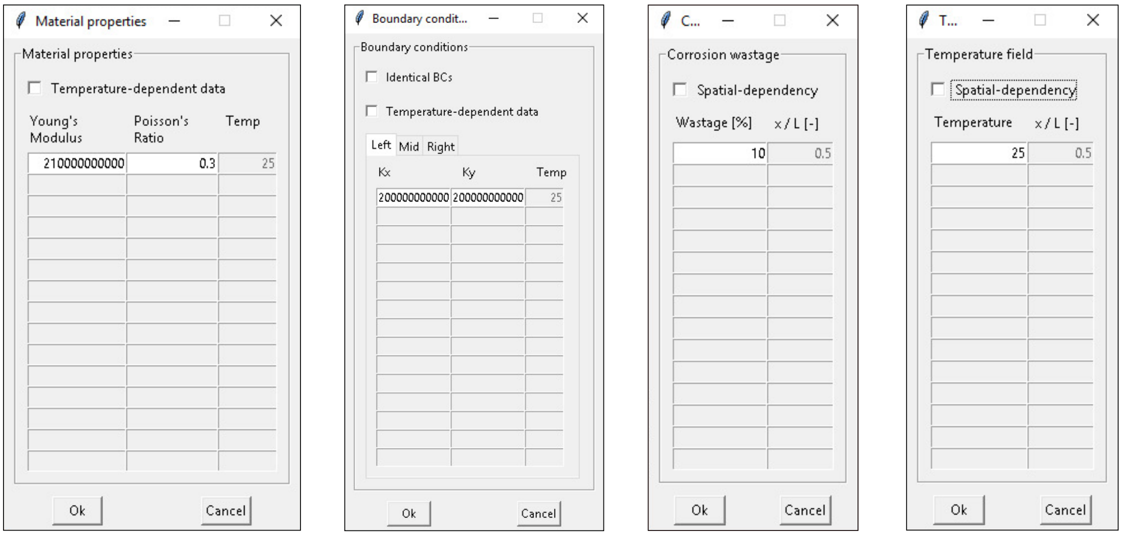


Figure 3: Graphical user interface for generation of simulated data.

A snapshot of the GUI for data generation is shown in Figure 3, where the user is given the capability of creating a list of different model scenarios, which are shown in the top-left frame named ”List of jobs”. Such a scenario may be created by following the numbered frames under model preview, starting



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

Figure 4: Graphical user interface for specifying (a) material properties, (b) boundary conditions, (c) corrosion wastage and (d) environmental temperature

from ”1. Model configuration”. This step includes the selection of a model state, choosing among the healthy and a number of damaged ones as described in Section 2., and the specification of a thickness and damage value. The latter is evidently relevant only for damaged models and defines the percentage of stiffness reduction applied on the damaged region of each model case. To finalize the model set-up, the user is called to specify the temperature-dependent material properties and elastic boundary conditions, as well as the spatially-distributed corrosion wastage and environmental temperature. A snapshot of the corresponding windows, which can be opened by the buttons under thickness and damage entries, is shown in Figure 4.

Upon definition of the model parameters, the user is asked to select the type of analysis, which can be either modal or time history. Depending on the choice, the corresponding settings should be then specified in either ”2a. Modal settings” or ”2b. Time history settings” frame, respectively. Once the scenario is fully defined, it can be added to the list of jobs by specifying the name and subsequently clicking on ”Save” button in frame ”5. Job name”. The name of the job will be displayed then in the list of jobs and can be retrieved for editing upon selection and clicking on the ”Edit” button. Similarly, any job, or all them, can be deleted by clicking on ”Delete” or ”Delete All” buttons, respectively. The available jobs can be submitted for analysis by hitting on ”Run jobs” button and the progress of execution will be displayed in the ”Messages” frame. The results of each job will be finally stored in a .dat file named in accordance with the job name.

5. CONCLUSIONS

This paper presented a benchmark study for system identification under environmental and operational variability. The considered structure is a simple two-span beam supported on elastic boundaries and modeled with plane stress elements. The system is supplemented with a number of predefined damage patterns and loading conditions, while it provide the possibility of defining temperature-dependent parameters. Along with the paper, an open source python code, which is wrapped with a GUI, is provided in order to enable the generation of dynamic response signals.

ACKNOWLEDGEMENTS

The authors would like to gratefully acknowledge the European Research Council via the ERC Starting Grant WINDMIL (ERC-2015-StG #679843) on the topic of Smart Monitoring, Inspection and Life-Cycle Assessment of Wind Turbines.

REFERENCES

[1] Farrar, C.R. and Baker, W.E. and Bell, T.M. and Cone, K.M. and Darling, T.W. and Duffey, T.A. and Eklund, A. and Migliori, A. Dynamic characterization and damage detection in the I-40 bridge over the Rio Grande. *Los Alamos National Lab*, NM(United States).

[2] Maeck, J. and De Roeck, G. (2003) Description of Z24 benchmark. *Mechanical Systems and Signal Processing*, 17(1), 127-131.

[3] Molina, F. and Pascual, R and Golinval, J. (2003) Description of the steelquake benchmark. *Mechanical Systems and Signal Processing*, 17(1), 77-82.

[4] Li, S. and Li, H. and Lan, C. and Zhou, W. and Ou, J. (2014) SMC structural health monitoring benchmark problem using monitored data from an actual cable-stayed bridge. *Structural Control and Health Monitoring*, 21, 156-172.

[5] Smith, W. and Randall, R. (2015) Rolling element bearing diagnostics using the Cave Western Reserve University data: A benchmark study. *Mechanical Systems and Signal Processing*, 64-65, 100-131.

[6] Joyce, B. and Dodson, J. and Laflamme, S. and Hong, J. (2018) An experimental test bed for developing high-rate structural health monitoring methods. *Shock and Vibration*, 2018.

[7] Johnson, E.A. and Lam,H.F. and Katafygiotis, L.S. and Beck, J.L. (2004) Phase I IASC-ASCE Structural Health Monitoring Benchmark Problem Using Simulated Data. *Journal of Engineering Mechanics*, 130(1), 3-15.

[8] Dyke, S. and Bernal, D. and Beck, J. and Ventura, C. (2003) Experimental phase of the structural health monitoring benchmark problem. *Proceedings of the 16th ASCE engineering mechanics conference*.

[9] Tiso, P. and Noël, J. (2017) A new, challenging benchmark for nonlinear system identification. *Mechanical Systems and Signal Processing*, 84, 185-193.

[10] Odgaard, P. and Stoustrup, M. and Kinnaert, M. (2009) Fault tolerant control of wind turbines - a benchmark model. *Proceedings of the 7th IFAC Symposium on Fault Detection, Supervision and Safety of Technical Processes*.

[11] Dunbar, T. and Pegg, N. and Taheri, F. and Jiang, L. (2004) A computational investigation of the effects of localized corrosion on plates and stiffened panels. *Marine Structures*, 17(4), 385-402.

[12] Sultana, S. and Wang, Y. and Sobey, A.J. and Wharton, J.A. and Shenoi, R.A. Influence of corrosion of the ultimate compressive strength of steel plates and stiffened panels. *Thin-Walled Structures*, 96, 95-104.

[13] Chopra, A. (2017) Dynamics of structures: theory and applications to earthquake engineering. *Prentice-Hall* Englewood Cliffs, NJ, USA.