



**HAL**  
open science

## System identification and model calibration of a steel road bridge

Falk Hille, Ronald Schneider, Patrick Simon, Ralf Herrmann, Matthias Baessler

► **To cite this version:**

Falk Hille, Ronald Schneider, Patrick Simon, Ralf Herrmann, Matthias Baessler. System identification and model calibration of a steel road bridge. 11th International Operational Modal Analysis Conference, Institut National de Recherche en Informatique et en Automatique [Inria], May 2025, Rennes, France. <hal-05232756>

**HAL Id: hal-05232756**

**<https://hal.science/hal-05232756v1>**

Submitted on 1 Sep 2025

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire HAL, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.



Distributed under a Creative Commons CC BY 4.0 - Attribution - International License



# International Operational Modal Analysis Conference

20 - 23 May 2025 | Rennes, France

## System identification and model calibration of a steel road bridge

*Falk Hille<sup>1</sup>, Ronald Schneider<sup>2</sup>, Patrick Simon<sup>3</sup>, Ralf Herrmann<sup>4</sup>, and Matthias Baeßler<sup>5</sup>*

<sup>1</sup> Bundesanstalt für Materialforschung und -prüfung (BAM), Berlin, Germany, falk.hille@bam.de

<sup>2</sup> BAM, ronald.schneider@bam.de

<sup>3</sup> BAM, patrick.simon@bam.de

<sup>4</sup> BAM, ralf.herrmann@bam.de

<sup>5</sup> BAM, matthias.baessler@bam.de

### ABSTRACT

The Bundesanstalt für Materialforschung und -prüfung (BAM), in cooperation with the Netherlands Organization for Applied Scientific Research (TNO), is working on a framework for integrating frequently updated structural models into an asset management process for bridge structures. A multi-span steel road bridge was selected as a test case for the development of this framework. In order for the structural model to represent the real behavior of the bridge with sufficient accuracy, model calibration is required. In this case, we have planned to calibrate the model based on the dynamic response of the bridge. To determine its dynamic properties, a multi-setup operational modal analysis was performed on one of the bridge spans. In parallel, a structural model of the span was developed based on the available design and service life information. Both eigenfrequencies and mode shapes were used as reference parameters to calibrate the model. A sensitivity analysis was performed to identify the most influential design parameters. Subsequently, a genetic algorithm was applied for minimizing the difference between measured and simulated characteristic responses. In the proposed paper, we summarize the measurements as well as the determination of the modal response of the bridge and describe the process of calibration of the structural model using the identified dynamic response.

*Keywords: Bridge structure, Operational modal analysis, Model calibration*

### 1. INTRODUCTION

The analysis of the structural condition of bridge structures is mainly based on numerical models that represent the real behavior of the structure as accurately as possible. In the context of the digital twin paradigm, the model as a fundamental component should generate as accurate a representation as possible of the real structural response of a bridge to its loading. The development of numerical models of

a structure involves assumptions and simplifications that can lead to errors, usually associated with inaccuracies in the finite element discretization, variations in geometry and material properties, and uncertainties.

Therefore, to ensure the required accuracy of the numerical simulation, it is necessary to calibrate the previously developed model using measured structural responses due to defined loads or excitations [1]. There are several established methods for calibrating structural models. A very common method is to compare the deformation behavior of real structures under static or slowly moving defined loads, such as bridge load tests with loaded trucks or locomotives. Another well-established approach for model calibration is to use the dynamic properties, i.e. the vibrational behavior of a structure [1, 2]. In particular, natural frequencies and mode shapes characterize the structural behavior and can be determined in a modal analysis campaign. These procedures can be straightforward, depending on the number and location of measurement points as well as the form of excitation. In a subsequent analysis, the modal information is extracted from the measurement data and can be compared with the calculated values of the numerical model. The model update or calibration routine then adjusts the sensitive structural parameters so that the modal variables of both models (measurement vs. numerical) eventually match with sufficient accuracy.



**Figure 1.** Road bridge structure consisting of ten simply supported steel box girders.

A multi-span road bridge shown in Figure 1, was selected as a real-world application to test a possible update routine. As part of the project, a multi-setup operational modal analysis was performed on one of the bridge spans to determine its dynamic properties in terms of natural frequencies and mode shapes. This information was then used as the basis for calibrating a developed FE model of the span. The paper describes the modal analysis based on the measured dynamic response and the application of the calibration routine to adjust the dynamic behavior of the FE model of the structure. The model updating process is performed using commercial software that includes preconditioned algorithms for sensitivity analysis and the optimization process.

## 2. STEEL BRIDGE STRUCTURE

The road bridge is approximately 1 km long and consists of ten spans of approximately 100 m (see Figure 1). The road has five lanes in each direction, including an emergency lane and a cycle lane. The primary structure of each span is a simply supported steel box girder with a symmetrical cross-section, as shown in Figure 2. The maximum height of the girder is 3.5 m and its deck slab is 42.9 m wide. Along the longitudinal axis, each girder is composed of five regular segments of 16.67 m and two end segments of 7.29 m and 9.38 m. The deck plate, webs and bottom plate of the box girder are orthotropic steel plates with trapezoidal stiffeners. Along the box girder, transverse bulkheads are typically spaced 4.17 m apart and a longitudinal bulkhead is located at its centerline. In longitudinal direction, the distance between the supports is 100 m and in transverse direction 12.8 m. In 2005, a high-strength concrete overlay of approximately 7 cm was applied to the orthotropic deck slab to mitigate fatigue damage due to increased traffic loads.

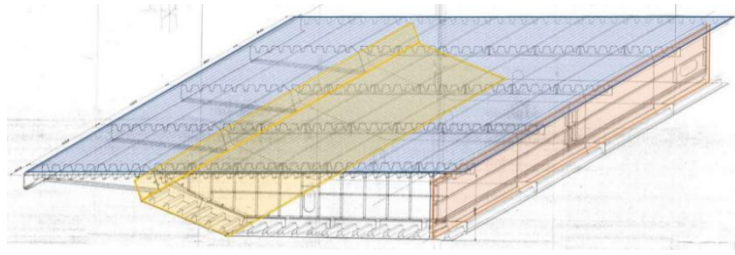


Figure 2. Half of a regular segment of the box girder [3].

### 3. VIBRAION-BASED SYSTEM IDENTIFICATION

#### 3.1. Measurement campaign

In order to conduct an Operational Modal Analysis (OMA) based system identification, an output-only vibration measurement under ambient conditions was performed on the south-easternmost bridge span [3]. The measurements were performed in three setups using a 30-channel system with 25 vertical geophones (vibration velocity sensors), four horizontal geophones, and one vertical force balance accelerometer. The outline of the three-setup campaign is shown in Figure 3, above. During the three setups seven vertical and four horizontal reference sensors were in fixed positions while the position of 18 vertical geophones was changed from setup to setup (see Figure 3, top).

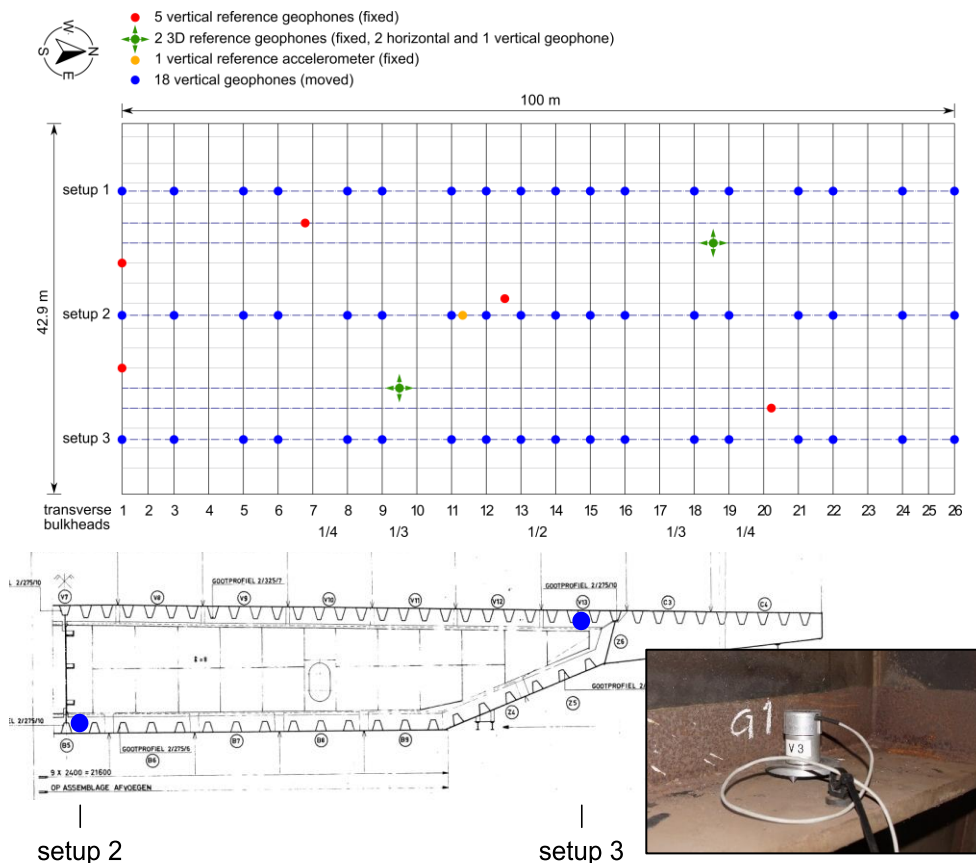


Figure 3. Top view (above) and cross-sectional view (below) of the multi-setup measurement, along with a typical vertical geophone sensor. [3].

Two reference sensors were placed on the south supports of the box girder. The remaining reference sensors were placed on the bottom plate of the box girder at the positions shown in Figure 3. In Setups 1 and 3 (west and east sides of the box girder), the 18 vertical geophones with changing positions were placed on the top horizontal stiffener of the corresponding transverse bulkheads (see Figure 3, bottom).

In Setup 2 (centerline of the box girder), these sensors were positioned on the bottom plate. All sensors were mounted on base plates with three conical pins (Figure 3, bottom). For each sensor setup, a 60-minute record of geophone data and accelerations was measured at a sampling rate of 1000 Hz.

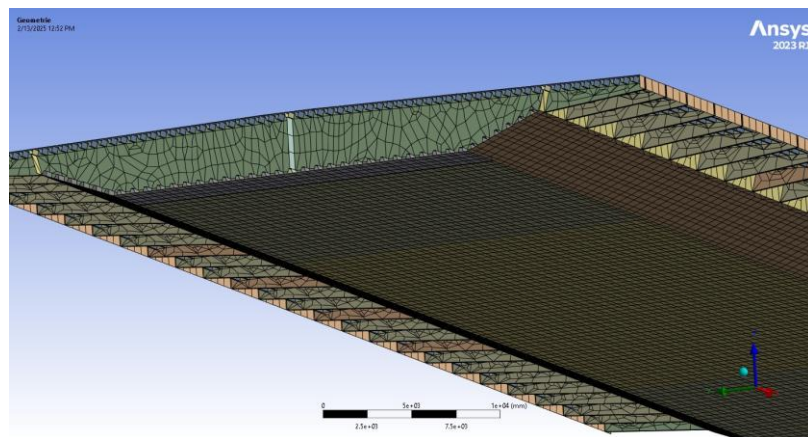
### 3.2. Operational Modal Analysis

The eigenfrequencies and mode shapes of the southernmost box girder were determined from the measured geophone signals. In a first step, each individual geophone signal was filtered using an 8th order Butterworth bandpass filter with cutoff frequencies of 0.75 Hz and 300 Hz, with the upper cutoff frequency chosen to avoid aliasing. In a second step, the individual geophone signals were transformed into the frequency domain and calibrated as described in [3]. In a third step, the calibrated geophone signals from the different measurement setups were merged and normalized. Finally, the natural frequencies and mode shapes were determined using the frequency domain decomposition (FDD) technique [4] in conjunction with peak picking. The third and fourth steps were performed using ARTeMIS Modal [5].

## 4. FINITE ELEMENT MODEL CALIBRATION

### 4.1. Modelling and numeric modal analysis

A linear-elastic finite element (FE) model of the box girder was developed by BAM in ANSYS Mechanical APDL [6] to determine the structural behavior under complex loading scenarios and to locate areas with potential for fatigue damage.



**Figure 4.** Numerical model of the steel bridge deck.

The deck slab, webs, floor slab, bulkheads, and trapezoidal stiffeners of all orthotropic slabs were modeled with shell elements (see Figure 4). The concrete overlay of the road, the asphalt and the epoxy resin layer were captured by modeling the deck slab with composite shell elements. All properties of the ANSYS model were defined based on information collected during the measurement campaign, design drawings, assessment reports and a highly detailed DIANA [7] model provided by TNO. The material property and the thickness of the overlay were described without spatial variation. Using the developed FE model, a numerical modal analysis was performed and the first nine global modes were extracted for comparison and the planned model update. The frequencies are shown in Table 1 and the mode shapes in Table 2.

### 4.2. Comparison of modal results

Tables 1 and 2 show the experimentally determined eigenfrequencies of the real bridge structure in comparison to the numerically determined eigenfrequencies of the preliminary model. The OMA was performed by Frequency Domain Decomposition (FDD) using the ARTeMIS software package. For

the frequency range from 0.75 Hz to 10.5 Hz, nine modes were identified using peak picking. The natural frequencies of the experimentally identified modes are shown in Table 1.

A qualitative comparison of the measured mode shapes with the modes obtained from the modal analysis of the ANSYS model is given in Table 2. The identified and matched numerical mode shapes appear to be in very good agreement. For a quantitative evaluation of the similarity of the mode shapes, the commonly used Modal Assurance Criterion (MAC) is applied [8]. The MAC value is a scalar that measures the correlation between two mode shapes, where a value of 1 indicates perfect similarity and a value of 0 indicates no correlation. The last column of Table 1 shows the MAC value for the identified first nine global modes of the bridge structure. The high MAC values reflect a strong agreement between the experimentally identified modes and those predicted by the initial ANSYS model, indicating an already satisfactory model accuracy.

**Table 1.** Comparison between experimental and matched numerical results.

Mode	Frequency			MAC
	Experimental [Hz]	Numerical [Hz]	Error [%]	
1	0.86	0.86	0.15	0.999
2	2.03	2.05	1.08	0.998
3	2.61	2.83	8.26	0.995
4	4.22	4.32	2.53	0.992
5	4.74	5.11	7.76	0.977
6	6.33	6.45	1.77	0.983
7	7.26	7.11	2.10	0.924
8	8.00	8.23	2.81	0.833
9	9.01	9.10	1.04	0.862

## 5. CALIBRATION OF THE FE-MODEL

One of the goals of this work is the application of a routine for the calibration of the FE model. For a significant good a priori agreement of the numerical with the experimental modal analysis results, it is of interest in which form and to which degree a further robust adjustment of the FE model is achievable.

In general, it is advantageous to start the calibration routine with a sensitivity analysis to identify those variable input parameters that have the most significant influence on the target (output) variables. This approach involves evaluating all parameters over the full range of possible values. A sensitivity analysis provides an understanding of the structural system and is therefore necessary to select the most important uncertain parameters. Prior to optimization, this sensitivity-based parameter selection is essential for the optimization routine to function effectively.

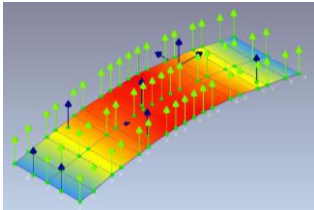
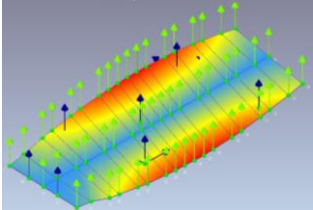
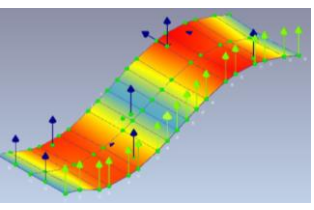
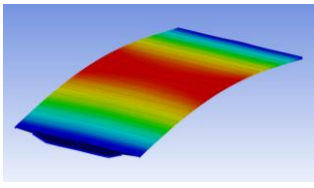
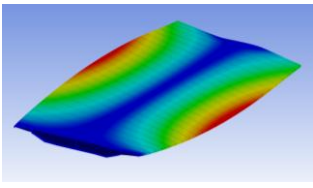
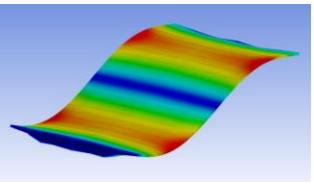
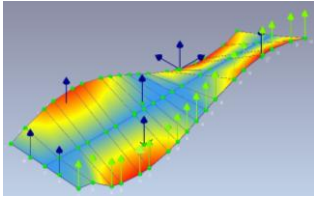
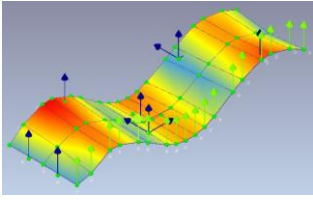
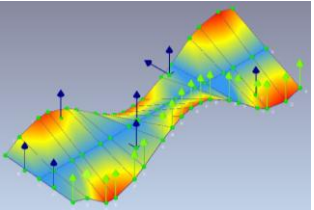
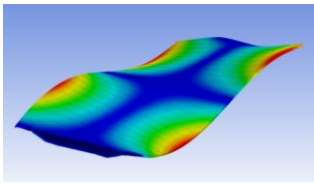
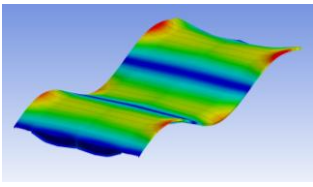
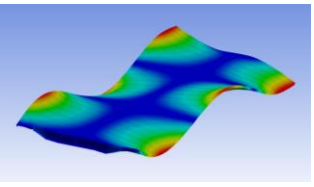
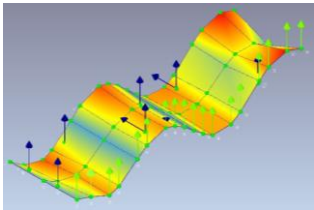
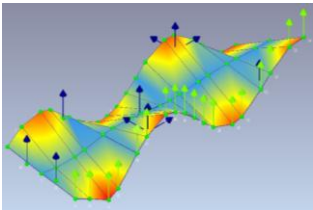
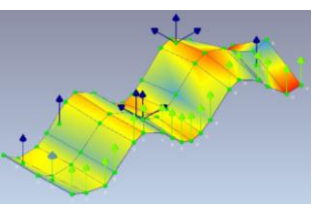
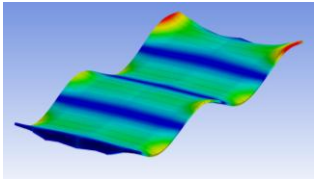
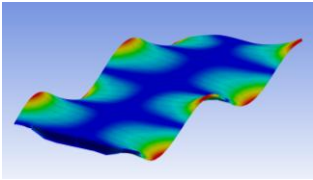
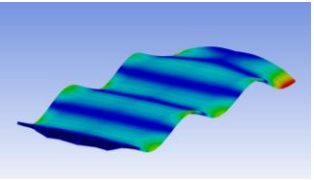
Within the ANSYS optiSLang software tool used here, the sampling method can be selected from several stochastic sampling methods. The data are then analyzed by calculating correlation coefficients or other methods such as principal component analysis or coefficient of determination [10].

### 5.1. Sensitivity Analysis

The sensitivity analysis is initiated with a preliminary collection of parameters considered relevant in influencing the global dynamic response of the bridge structure. This comprehensive collection includes 48 parameters, including material parameters, cross-sectional geometric measures of all modeled components, and stiffness at all structural supports. The sensitivity analysis was performed using ANSYS optiSLang [10]. The target variables were the corresponding measured and numerically determined eigenfrequencies and mode shapes of the first nine global modes. While the eigenfrequencies are

directly processed to compare the similarity of the numerical with the measured modes for mode shapes, the calculated Modal Assurance Criterion (MAC) is used.

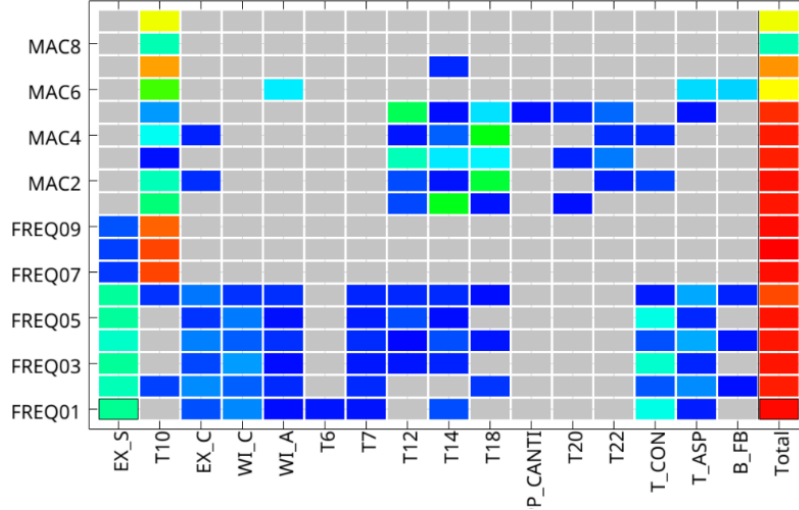
**Table 2.** Comparison between the experimental and the matched numerical mode shaped.

	1. Mode	2. Mode	3. Mode
Measurement			
Simulation			
	4. Mode	5. Mode	6. Mode
Experimental			
Simulation			
	7. Mode	8. Mode	9. Mode
Measurement			
Simulation			

A total of 100 samples were generated using the implemented Advanced Latin Hypercube Sampling method. The resulting data is shown in Figure 5 as a Coefficient of Prognosis (CoP) matrix, which here includes the 16 most relevant parameters influencing the vibrational behavior. The CoP is a model-independent measure of the model quality and accounts for prediction errors during sampling. Within the matrix red indicates a high influence of the parameters under consideration while blue indicates a weak influence. The CoP matrix in Figure 5 the rows contain the target values (modal measures) while

the columns contain the 16 considered structural parameter as input values. The prefix  $T$  indicates thicknesses of structural members,  $EX$  the elasticity modulus and  $WI$  the density of the models material.

The sensitivity analysis did not reveal any significant phenomena. The modulus of elasticity and the density of the structural material are the most relevant material parameters, while the thickness values of the steel plates, the concrete and the asphalt layer significantly influence the global dynamic behavior. In particular, the thickness of the outer plates of the box girder in the longitudinal direction, especially for T10, has an influence on the higher modes and, to a lesser extent, on all mode shapes. The parameters displayed in the CoP matrix revealed sensitivities and are used in the optimization process.



**Figure 5.** Result of sensitivity analysis: Coefficient of Prognosis (CoP) matrix [10], displaying the sensitivities of individual structural (input) parameter on individual modal (target) values.

## 5.2. Optimization

The optimization step allows to obtain the parameter set that minimizes the differences between the numerical and experimental modal responses. This is the objective function  $g$ , defined here for each output frequency as

$$g = W_f \sum_{i=1}^n \left( \frac{f_{num} - f_{exp}}{f_{exp}} \right)^2 + W_{MAC} \sum_{i=1}^n (1 - MAC_i)^2 \rightarrow \min \quad (1)$$

where  $f_{exp}$  and  $f_{num}$  specify the experimental and numerical obtained eigenfrequencies and  $MAC_i$  the MAC values of the function  $n$  modes respectively.  $W_f$  and  $W_{MAC}$  are weighting factors balancing the contribution of both terms. According to [6], weighting factors of 0.75 and 0.25 were used because the results of the preliminary FE model already gave good MAC ratios except for the two highest modes considered.

The optiSlang software uses a genetic algorithm to estimate a set of parameters focused on minimizing the residuals of the objective function. This process is repeated iteratively until the defined maximum number of generations is reached. The optimization of the bridge deck model involved the adjustment of 16 design variables, nine natural frequencies and nine mode shapes. As mentioned before, the preliminary FE model shows a good agreement with the real structure. The objective value in (1) is with  $g_{pre} = 0.066$  already significantly small. The applied optimization procedure resulted in several designs with a significant lower objective value ( $g_{b1} = 0.033$ ,  $g_{b2} = 0.036$ ), but it turned out that those designs are not similar in their parameter variations. That means, they map just local minima which leads to the conclusion that the applied optimization is not sufficiently robust. The main reasons for this is the use of only globally acting varying parameters. A reasonable adjustment could introduce an additional spatial variability of the design parameter. In this way higher modes will be better influenced. Though it has to be stated that this approach would increase the analysis effort significantly.

## 6. CONCLUSIONS

To obtain data from a real bridge structure, the vibration response of one of the simply supported spans of a road bridge was measured under ambient conditions. The recorded data were analyzed using operational modal analysis techniques to identify its modal properties. An initial structural model of the span was developed, which already showed a significant agreement with the real structure. To further minimize the difference between the identified and numerically determined modal properties, a model calibration procedure was applied. The results of the calibration procedure show a non-robust minimization of the objective. The reason for this is the integration of a large number of mostly global parameters. Since there is a large agreement between measured and calculated dynamic behavior, especially in the lower modes, it will be necessary to vary parameters spatially. However, since this leads to a significant increase in computational effort, it is outside the scope of this study.

The calibrated numerical model can potentially be used as a basis for predicting stresses at the critical fatigue hotspots in conjunction with site-specific traffic load models. Based on planned hotspot stress history simulations and available inspection results, hotspot fatigue models and fatigue life predictions can be updated. This information can then be used to support decisions on future maintenance actions. In addition, the calibrated model can be used to evaluate the performance of damage detection methods based on simulated damage processes and monitoring results.

## ACKNOWLEDGEMENTS

The authors would like to thank Caroline Den Besten, Bernard Oldescheper, Gerard van Melsen and Twan Wanrooij from Rijkswaterstaat for the opportunity to use the bridge as a test case, Rob Jansen from TNO unit High Tech Industry as well as Stefen Verdenius and Erik Slis from TNO unit Mobility & Building Environment for their support in preparing and conducting the vibration measurements and building the ANSYS model.

## REFERENCES

- [1] F. Hille, R. Herrmann, S. Said, M. Baessler and F. Wille. FE model updating of a drop test tower based on experimental modal parameters. In *Proc. 10th International Conference on Structural Health Monitoring of Intelligent Infrastructure*, Porto, Portugal, 2021.
- [2] L. Frýba, M. Pirner. Load tests and modal analysis of bridges. *Engineering Structures*, 23(1), 102-109, 2001
- [3] R Schneider, P Simon, F Hille, R Herrmann and M Baeßler. Vibration-based system identification of a large steel box girder bridge. In *Proc. 12th International Conference on Structural Dynamics – EURO DYN*, Deft, Netherlands, 2023.
- [4] R. Brinker, L. Zhang, and P. Andersen. Modal identification of output-only systems using frequency domain decomposition. *Smart Materials and Structures*, 10(3), 441-445, 2001.
- [5] Structural Vibration Solutions A/S. ARTEMIS Modal. <https://svibs.com/artemis-modal/> (accessed 2025).
- [6] B. Barros, B. Conde, M. Cabaleiro and B. Riveiro. Deterministic and probabilistic-based model updating of aging steel bridges. *Structures*, 54, 89-105, 2023.
- [7] Ansys Inc. Ansys Mechanical 2024 R1, Finite Element Analysis (FEA). <https://www.ansys.com/products/structures/ansys-mechanical/> (accessed 2025).
- [8] DIANA FEA BV. "DIANA FEA. <https://dianafea.com/index.php/> (accessed 2023).
- [9] M. Brehm, V. Zabel, and C. Bucher. An automatic mode pairing strategy using an enhanced modal assurance criterion based on modal strain energies. *Journal of Sound and Vibration*, 329(25), 5375-5392, 2010.
- [10] Ansys Inc. Ansys optiSLang 2024 R1, Process Integration & Design Optimization: <https://www.ansys.com/products/connect/ansys-optislang/> (accessed 2025).