The 6th International Conference on
Structural Health Monitoring of Intelligent Infrastructure
Hong Kong | 9-11 December 2013

OPTIMIZED CALIBRATION OF FINITE ELEMENT MODEL FOR VERRAZANO NARROWS BRIDGE SPAN

Z. Y. Wu [1], J. Zhao [1], R. Love [2], S. Bhide [3], A. Jeary [4] and T. Winant [4]
[1]Bentley Systems, Incorporated, Watertown, CT, USA. Email: zheng.wu@bentley.com
[2]Bentley Systems, Chicago, IL, USA.
[3]Bentley Systems, Incorporated, Sunrise, FL, USA.
[4]STRAAM LLC, New York, NY, USA.

ABSTRACT

The finite element (FE) model that can adequately simulate the existing conditions of a bridge is a desirable and useful for bridge performance assessment. However, such a FE model cannot be obtained unless the model is constructed and calibrated to capture the important dynamic characteristics of the structural system. Therefore, the FE model calibration, a process of deriving an adequately accurate FE model that represents the bridge dynamic integrity, has been and will continue to be an essential component of bridge management. Based on the authors' previous research work in model calibration and damage detection, a pilot study on bridge dynamic measurement and modeling has been undertaken for the approach span of the Verrazano Bridge in New York City. The bridge span selected for the study is not particularly sophisticated but represents the majority of highway bridges in the USA. The previously developed method for FE model calibration, using the Darwin parallel optimization framework, was improved with the weighted goodness-of-fit criteria. The calibration was conducted to minimize the difference between the measured and modeled dynamic responses. In particular, the model calibration proceeds in a progressive manner on a High Performance Computing cluster of many-core machines. Based on the sound engineering judgments together with the effective application of the FE model calibration optimization tool, the model was successfully identified and can serve as a valuable knowledge tool for bridge engineers. Such a model can be used to assess the bridge span condition and evaluate the condition-based maintenance solutions that are not only preventive or corrective but also diagnostic and prognostic for bridge management.

KEYWORD

Finite element model, model calibration, model updating, optimization, parallel computing.

INTRODUCTION

Bridges are the critical infrastructure for a society. According to the USA Federal Highway Administration, most bridges in the USA were built between 1951 and 1980. The designed service life for the average bridge is approximately 50 years. Simultaneously the approaching end of their service life is attracting attention to concerns about the safety and security of these infrastructure systems. As the ASCE reported, about one-quarter of the bridges across the country are either functionally obsolete or structurally deficient. With such a large number of bridges (for instance, there are more than 25,000 state-owned bridges in Pennsylvania alone) decisions must be made based on the sound engineering assessments as to which bridges should be upgraded and what the maintenance plan is for those that have been assessed. A structural analysis model such as a finite element model (FEM), which can be applied to adequately analyze the existing conditions of the bridges, is an essential and useful tool that enables such a technically-sound engineering practice.

In infrastructural engineering, a well-constructed and calibrated FEM yields a reliable prediction of dynamic responses under the prescribed loading, and also assesses the condition of the existing structure subject to the deterioration over time and ambient effects. It is, however, often discovered that when the predictions of a nominal FEM are compared with the observed responses, the degree of correlation is not sufficient to allow the application of the model with confidence. This phenomenon is not hard to understand. A nominal FEM is generally built upon the

architectural and structural drawing, and thus it does not reflect the practical situations. Unavoidably, the discrepancy between the nominal FEM and the real conditions of the structure results in consequential errors in the analyzed responses. In order to tackle this issue, significant research has been undertaken in updating the nominal FEM by using the dynamic measurements. Such research effort is categorized in the area of FE model updating or calibration.

FEM calibration is an iterative procedure of adjusting a set of the FEM parameters so that the discrepancy between the observed and the simulated responses is minimized (Mottershead and Friswell 1993). First of all, a nominal FE model must be constructed for the existing structural system. The task of model calibration is then to estimate the model parameters including the elastic modulus of individual element, material density, boundary conditions and several other contributing factors. In order to achieve a good FE model calibration such that the updated model can adequately emulate the dynamic responses of the subject structure, the FEM parameter estimation can be generally formulated as an optimization problem of searching for the near-optimal values of these parameters. Regardless of any specific formulation of a FEM calibration problem, the calibration criteria is the function of stiffness and mass matrices, which leads to nonlinearity (Sanayei et al. 2006). Soh and Dong (2001) proposed an inverse problem and solved for material uncertainty of Young's modulus. Hao and Xia (2002) applied a genetic algorithm (GA) (Goldberg 2002) to identify a set of Stiffness Reduction Factor (SRF) as the indicator of structure damages. SRF is defined as the ratio of stiffness reduction to the initial stiffness. One finite element is assigned with a SRF to be identified. Stiffness reduction seems to be more meaningful to indicate the structural property variation than Young's modulus variation, but both types of parameters are not directly related to the physical property of structure damage. More recently, Wu and Xu (2011) proposed a FE model identification formulation for damage detection. Structure damage is represented with a set of meaningful geometric parameters while a finite element model identification method is formulated by proposing a unified objective function that is based on the errors in both static and modal responses. A prototype of a software framework has been designed and implemented for FE model parameter estimation and structure damage detection. The framework draws advantages from the off-shelf structural analysis software. It offers the flexibility of parameter selection and the functionality of parameter grouping. To estimate the parameters efficiently, Darwin optimization framework (Wu et al. 2012), which is a generic parallel optimization tool, is applied to search for the near-optimal model parameters.

This paper reports a pilot study on the bridge heath modeling for the approach span of the Verrazano Narrows Bridge in New York City (NYC). According to the NYC Metropolitan Transportation Authority (MTA), the Verrazano-Narrows Bridge was the world's longest suspension bridge when it opened in 1964. Located at the mouth of upper New York Bay, the bridge not only connects Brooklyn with Staten Island but also serves as the major link in the interstate highway system, providing the shortest route between the middle Atlantic states and Long Island. The bridge span selected for this study is not particularly sophisticated but is typical of the majority of bridges in the USA. Therefore, this study provides a good opportunity to validate the effectiveness of the previously developed optimization framework, which has been integrated into the optimized sensor placement method and the model calibration technology. The complete study procedure consists of field dynamic measurements based upon the optimized sensor placement results, data analysis, initial FE modeling and model calibration. A companion paper elaborates on the sensor placement optimization (Wu et al. 2013).

VERRAZANO-NARROWS BRIDGE AND INITIAL MODELING

Before conducting the model calibration study, we first investigate the dynamic behaviors of the real structure and generate a nominal FEM for the bridge span.

Structural Layout and Field Data

In this project, one section of the approach span was selected for investigation, which is between pier EB14 and abutment EBU. The frame plan is shown in Figure 1.

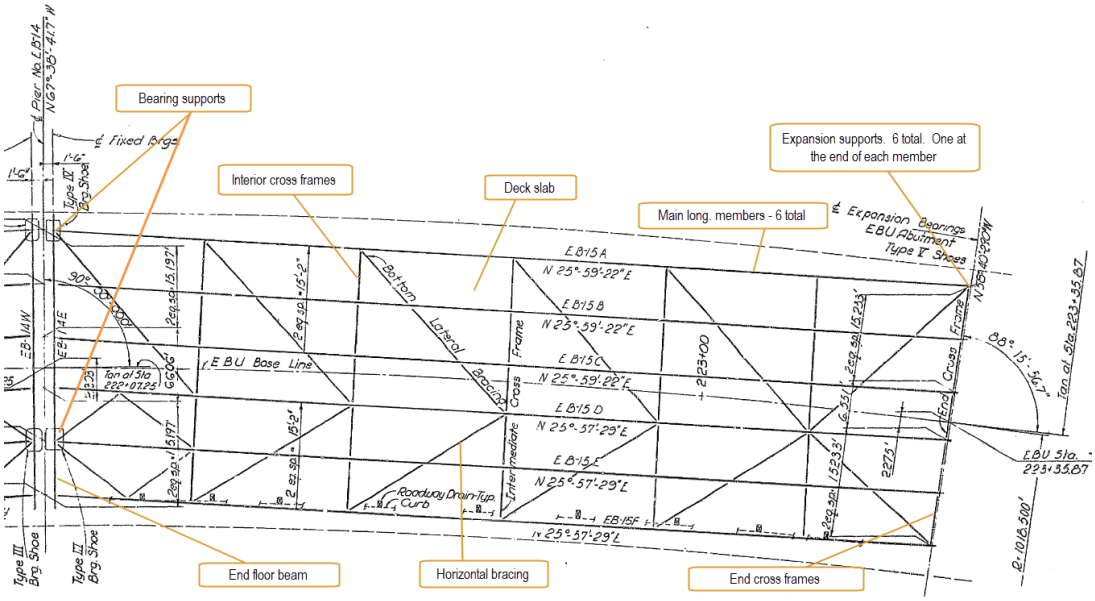

Figure 1

Figure 1. Layout of the bridge span

The necessary information to create the FEM model came from a series of original design drawing and consisted of the following:

- Geometric information that describes the bridge orientation and member layout
- Definition of the members that make up the model (deck slab, main longitudinal girders, intermediate cross frames, horizontal bracing, end cross frame at EBU abutment and floor beam at pier EB14)
- Member end support at bearings on pier EB14
- Member end support at abutment EBU
- Material properties

The dynamic measurements on the bridge span were conducted by placing five tri-axial accelerometers on the span and performing more than four-hours of continuous measurement of the bridge dynamic condition (Wu et al. 2013). With the collected data, the dynamic responses, such as frequencies and mode shape, were extracted using spectra and phase analysis. Six major vibration modes were identified with the frequencies as shown in Table 1. The extracted dynamic responses were used as the target values for the FEM calibration of the bridge span.

Table 1. Major modes and frequencies from field data

Modes	Descriptions	Frequency
1	1st Horizontal	3.00
2	1st Vertical	3.20
3	1st Anti-symmetric Torsion	3.70
4	Bird wing Flap Mode	6.20
5	1st Anti-symmetric Vertical	6.70
6	1st Anti-symmetric Torsion	8.90

Initial Finite Element Model

Using the information described above, the FE model, as shown in Figure 2, was constructed by using STAAD.Pro (Bentley 2012). The geometry information and the material properties of the nominal FEM are based on the design drawing. The main longitudinal girder, the cross frames, the horizontal bracing members and the floor beams at the boundary were all modeled with beam elements. The deck slab was modeled with plate elements that are 9 inches thick throughout the interior regions of the bridge deck.

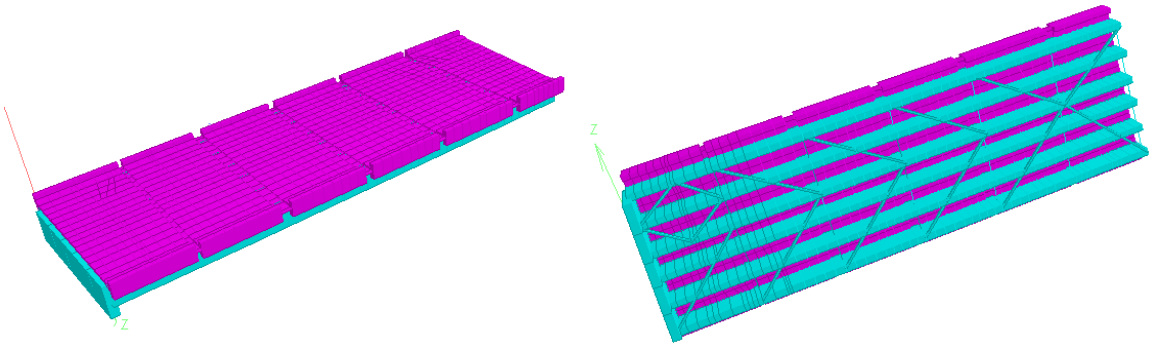

Figure 2. Finite element model constructed for the bridge span

One challenge in this pilot study is that the subject under investigation is just one approach span of the whole multi-span bridge structure. On one hand, the nominal FEM represents the boundary condition and the connection with the neighboring span. No interaction force or boundary information can be obtained from the design drawing. On the other hand, the boundary conditions (including the interactions with the adjacent span) play a significant role in the dynamic behavior of the structure under investigation. Especially in the low frequency range, constraints at the boundaries have a dominant effect on the structural response in comparison with the material attributes, and thus should be carefully characterized in any finite element analysis.

Traditionally, when setting up a FEM for an infrastructure system, fixed or free boundary conditions are applied to allow certainty about the degree of freedom of the boundary nodes. A free condition would represent a non-constraint-situation in the structure and typically represents a good approximation. However, a free boundary usually does not exercise the root support of the structure or the interaction between substructures. When a fixed boundary condition exercises the root support of the infrastructure system, it introduces uncertainty into this completely rigid assumption. The real world situation does not necessarily conform to such an approximation. The same thing happens when one isolates a bridge span from the whole structure and put it under investigation. Based upon these considerations, we propose that it is necessary to apply elastic support assumptions to certain boundary condition of the structural model, and thus to determine the elastic stiffness parameters in the model calibration procedure.

At pier EB14, since the six main longitudinal members of the bridge are constrained with the end floor beam, the six longitudinal members are constrained with the elastic support assumption in Y and Z directions, two spring constants k_{iy} and k_{iz} are specified. Here the subscript i denotes the boundary node index and the subscript y or z denotes the direction. Totally, there are 12 independent stiffness variables employed to specify the boundary condition at pier EB14, six k_{iy} variables and six k_{iz} variables (i = 36, …, 42).

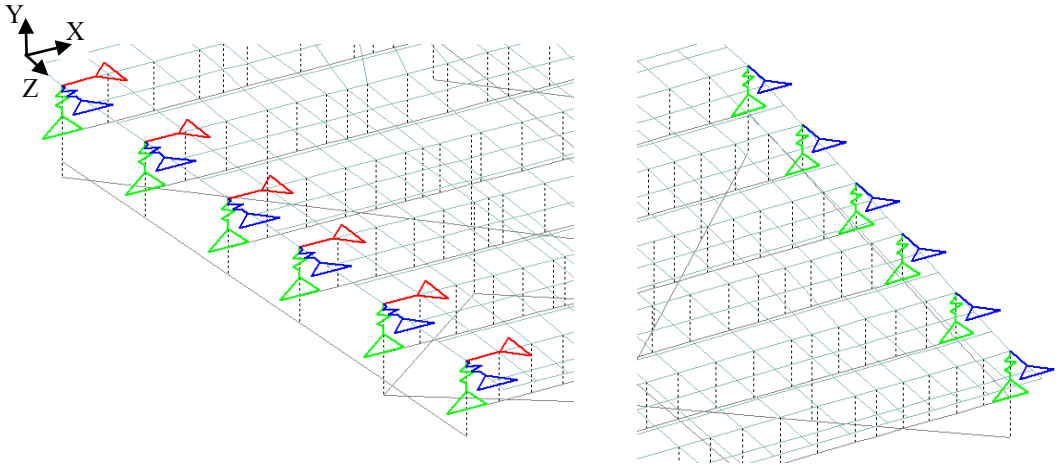

Figure 3. Modeling boundary supports of the bridge span

At abutment, the same elastic support treatment also applies to the vertical restraints. Totally, there are 6 independent k_{iy} stiffness variables appointed to specify the boundary condition at abutment (i = 43, …, 48). Totally 18 stiffness variables are employed in the nominal FEM. By several manually adjustment trials, we assign the horizontal stiffness variables $k_{iy} = 8000\,Kip/Ft$ and the vertical stiffness variables $k_{iz} = 500\,Kip/Ft$ in the nominal STAAD model. These merely are the initial values, which will be further optimized by applying the stiffness multipliers.

After the nominal FEM is generated, the next step is to investigate the dynamic behavior of the bridge span by using the analysis tool (Bentley 2012). The first six modes of the FEM are plotted in Figure 4. And the corresponding natural frequencies are also listed. Clearly, the discrepancy in dynamic behavior emerges when comparing the outcomes of the nominal FEM to the measured modes listed in Table 1. There are significant differences in the values of natural frequency. Undoubtedly the nominal FEM needs to be adjusted to minimize the difference, which is the target of model calibration.

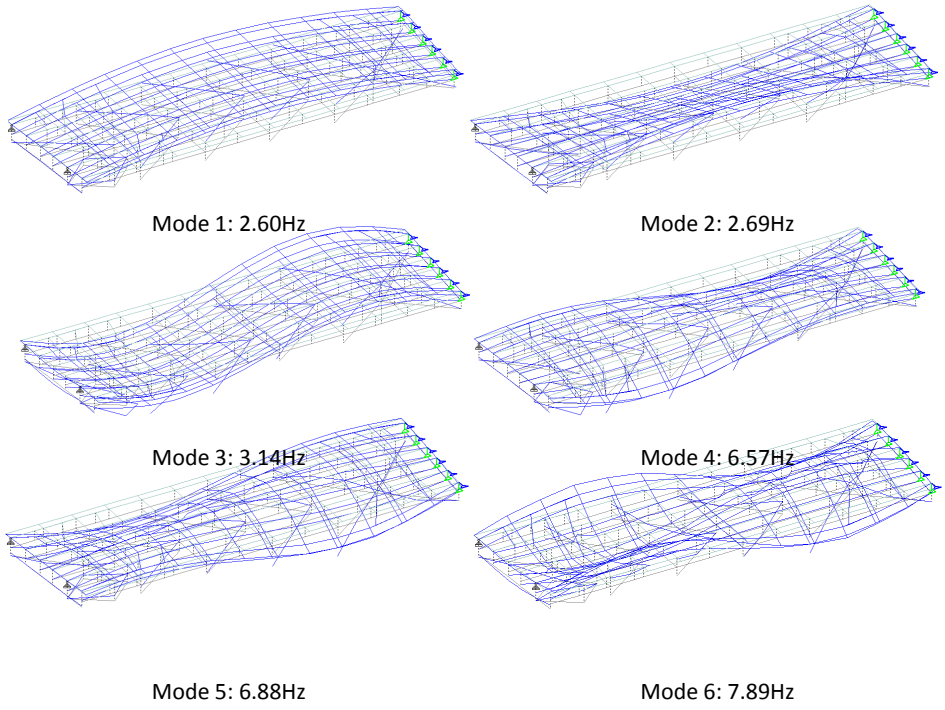

Figure 4. Simulated modes from initial FE model

MODEL CALIBRATION PROCEDURE

The discrepancy between the FEM-analyzed and the observed responses demonstrates that the nominal FEM represents the actual structure with a very limited accuracy. Thus the model must be calibrated and the calibration is formulated as nonlinear implicit optimization problem that is solved by using the Darwin parallel optimization tool.

Objective Function

The quantitative criterion, which measures the correlation between the FEM and the real structure, is calculated from the static response, the dynamic response or both of them. The unified the objective function is defined as:

$$E_T = E_{static} + E_{freq} + E_{mode} \tag{1}$$

where E_{static}, E_{freq} and E_{mode} are the static response-based, frequency-based and mode shape-based error function, respectively. In this study, the dynamic responses are used for the FEM calibration. Therefore, E_{static} is not included in the objective function. For most infrastructures, the lower modes consume the primary (up to 90%)

energy of the vibration, and thus it is far more important and desirable to have the mode match well with the observed ones for the primary modes. However, the calibration optimization process is relatively blind to the error of each individual mode, but is guided by the single value of the objective function value (the overall errors of all the modes). In order to better direct the optimization to search for a set of parameters such that the frequencies and mode shapes of the desired primary modes match up well with the observed values, the objective function is modified by allowing a user-specified weighting factor to be applied to each mode. The detailed definitions of the weighted frequency-based and mode shape-based error function are, respectively

$$E_{freq} = \sqrt{\sum_{i=1}^{n} \omega_i (f_i^A - f_i^O)^2} \quad (2)$$

$$E_{mode} = \sqrt{\sum_{i=1}^{n} \omega_i (\varphi_i^A - \varphi_i^O)(\varphi_i^A - \varphi_i^O)^T} \quad (3)$$

Here ω_i is the weighting factor applied to the i-th mode, f_i is the mode frequency and φ_i is the mode shape, the superscript A denotes the responses from the FE analysis and the superscript O denotes the observed response from the field data. Theoretically, for a structure with n degree-of-freedoms, all the mode frequencies and shapes are required to put into the calculation. In practice, however, only several lower primary modes can be measured due to the limit of techniques and complexity of structure. For FEM calibration, the primary modes hold sufficient information for a good finite element model of the real structure. In this study, only the first 6 modes are available, and thus $n = 6$ in Eq.(2).

Calibration Procedure

The target of optimized calibration is to search for a set of FEM parameters so that the discrepancy between the real structure and the nominal FEM is minimized. The FEM parameters to be optimized can be element geometry, elastic stiffness of the boundary condition and material attribute including element cross section, elastic modulus and mass density. In this study, the optimized parameters were limited to the elastic modulus of the elements and the elastic stiffness values of the boundary condition.

For most infrastructure systems, a FEM usually is constructed with a large number of elements. For the approach span in this study, there are more than 1000 elements used in the model. In order to reduce the dimension of the calibration optimization, similar elements were aggregated into one calibration group and assigned a common calibration factor or variable. By doing so, a total of 21 calibration groups were created to include all the beam elements. Similarly, a total of 14 calibration groups were used for the plate elements. For each calibration group, the same calibration parameter was applied to all the elements within the group. This treatment is for the elastic modulus adjustment. The boundary condition, as aforementioned, has more dominant effect on the dynamic behavior. Therefore, one stiffness variable is specified for one DOF of the selected boundary nodes. As a result, a total of 53 calibration variables, including 21 groups for beam elements, 14 groups for plate elements and 18 elastic stiffness variables, were optimized by the well-developed Darwin optimization framework (Wu et al. 2012).

To calibrate the FEM via optimization, the multipliers of the elastic modulus and the elastic stiffness were specified as the calibration parameter or decision variable for one group. The selected attribute of each element in the same calibration group was adjusted by multiplying the multiplier with the original value specified in the nominal model. Therefore, the model calibration problem is formulated to search for a number of model parameter multiplier as follows.

Search for: $\vec{X} = [x_1, x_2, \cdots, x_n]$ (4)

Minimize: E_T (5)

Subject to $x_i^{min} \leq x_i \leq x_i^{max}$ (6)

where x_i $(i = 1, \cdots, n)$ is the i-th calibration variable, n is the number of calibration variables , x^{min} and x_i^{max} are the minimum and maximum limits of variable x_i respectively. Here different ranges can be used for different calibration variables. For the element property multiplier, a parameter range of [0.5, 1.50] was used for this study. For the boundary stiffness multiplier, a wider optimization range is highly recommended, e.g. [0.01 10] in this research. Given the initial stiffness value, the possible fixed boundary condition scenario can be approximated when

the stiffness multiplier reach 10 while a free boundary condition is approximated by the stiffness multiplier close to 0.01.

Advanced Computing Techniques

As formulated in the previous section, FEM calibration is a typical implicit and nonlinear optimization problem. In this study, a genetic algorithm (GA) was employed as the search method due to its capability of escaping from local optima and exploring the solution space for a global solution. On the other hand, GA usually requires a large number of function evaluations or the FE analysis, which possibly leads to intensive computation. In order to speed up the computation, two essential techniques, the parallel optimization and hot start with a solution template, have been applied to ensure the scalable efficiency for the project.

A generic parallel GA tool, which is named the Darwin Optimization Framework, was developed for high performance analysis of infrastructure systems (Wu et al, 2011). The parallel optimization computer architecture is based upon a so-called manager-worker configuration as shown in Figure 5.

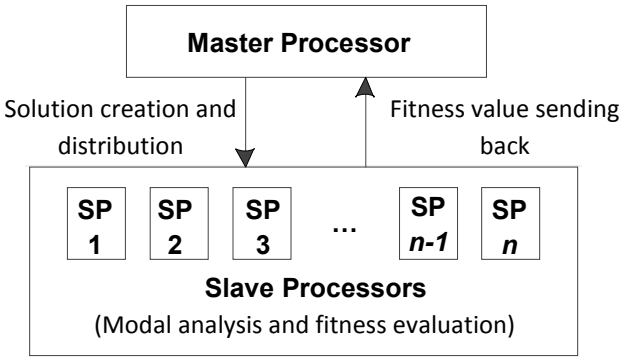

Figure 5. Manager-worker parallel optimization computing (Wu, Wang and Butala 2011)

The commutation and computation scope for manage-worker parallelism is as follows:
1) Manager processor creates and distributes solutions over worker processors;
2) Worker processors simultaneously implement model analysis and solution evaluation;
3) Worker processors send the fitness values back to manager processor;
4) Manager processor searches for an improved solution on the fitness values from worker processors.

Meanwhile, the Darwin Optimization Framework employs the 'hot start' technique. It restarts a new optimization run with the initial population of solutions generated around the near-optimal solution obtained from the previous run. Using Darwin, the top solution of each run can be saved and used as a template for the subsequent runs. The saved top solution is used as a template to generate the initial population of the solutions, which speed up the convergence of the new run to locate a better near-optimal solution. Usually a FEM calibration needs to take multiple optimization runs to gain a better understanding of the FEM characteristics to improve our knowledge on the problem, such as to identify the good ranges for calibration variables, to distribute the appropriate weighting factors among the modes etc. The hot start template effectively improves the efficiency of the overall calibration process.

CALIBRATION PERFORMANCE

With the method and the practice principles described above, this section demonstrates the performance of the model calibration within the Darwin optimization framework in a quantitative manner.

Computation Efficiency

An efficient evaluation of the possible solutions is essential for calibrating the FEM via optimization, because of tens or even hundreds of thousands of alternative solutions that must be automatically created and evaluated by

calling the FEM analysis solver. In this study, it was observed that one FEM run takes about 30 seconds. If the solutions were evaluated one after another in a sequential computing fashion, an optimization run of 100,000 evaluations, it would take more than 30 days to complete. In order to speed up the computation efficiency, the parallel optimization and hot start with a solution template were employed for the calibration task.

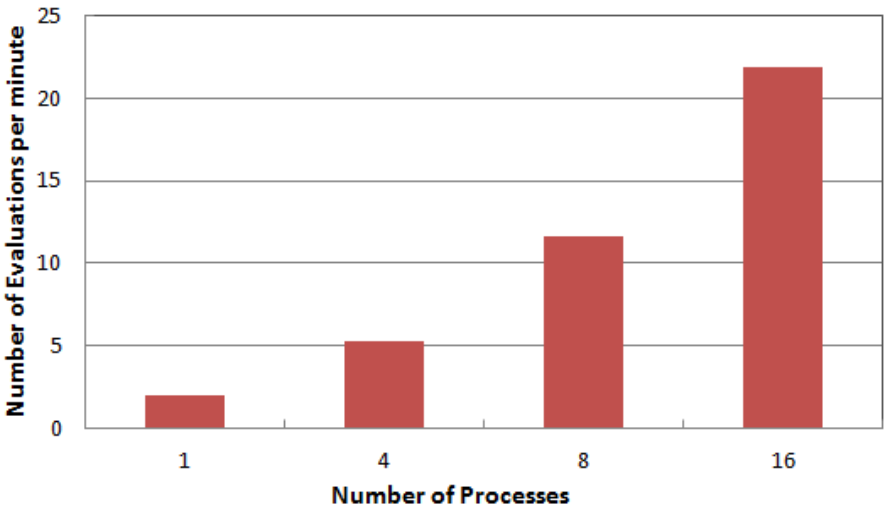

Figure 6. Computation efficiency speed up with parallel optimization

Figure 6 illustrates the computation efficiency with different parallel processes. Using one process (sequential computing), two FEM evaluations (runs) can be completed per minute, as the number of processes increase from 1 to 16, the number of FEM evaluations per minutes also increases to about 22. One parallel optimization run from scratch with 16 processes takes about 35 hours for 50,000 trials (evaluations). The solution is saved and exported as a template for the subsequent runs, which can be significantly speeded up with a hot start using the solution template. As shown in Figure 7, the optimization run with the solution template effectively improves the convergence of the subsequent run, which can be finished within 3 hours, and also enhances the final optimized solution by further minimizing the fitness with the adjusted parameter settings, e.g. modified decision variable ranges and/or weighting factors.

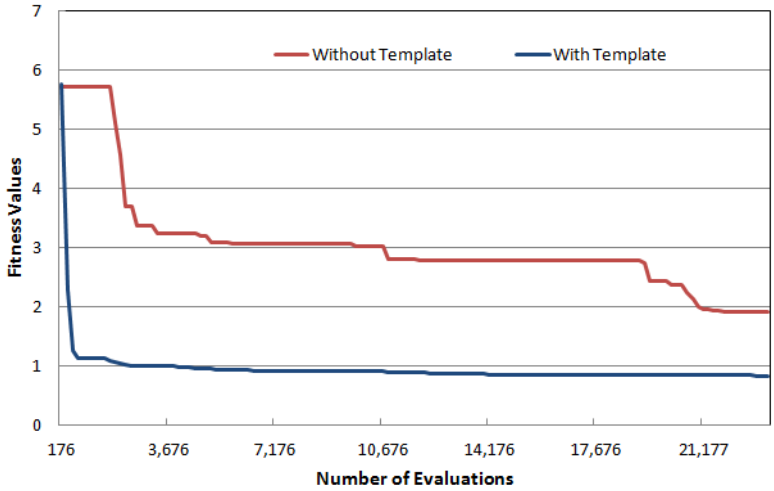

Figure 7. Comparison of convergence rate with and without hotstart

Calibrated Model

The FEM has been finally calibrated using the Darwin Optimization Framework with the techniques elaborated above, including the element aggregation (35 elemental attribute calibration groups), boundary condition

optimization and larger weighting factors for mode 1, 2 and 3. Meanwhile we employ an advanced computing technique such as multi-core parallel computing and hot-start with a solution template.

Before testifying the dynamic behavior of the calibrated FEM, Table 2 lists the updated stiffness variables in the calibrated FEM. As we can see, the stiffness coefficients in Y direction were identified with relative large values, some of which already reached the upper bound of the parameter range. The result implies that the vertical boundary condition of the longitudinal members can be represented as fixed support. The stiffness coefficients in the Z direction are identified with small values, which imply a more flexible constraint in the horizontal direction than in other directions. The identified stiffness parameters are physically significant and reflect the real boundary conditions with adequate accuracy.

Table 2. Calibrated boundary condition stiffness factors (Kip/ft)

Boundary Condition	Support Nodes	Allow Displacement			Spring Coefficient (KIP/FT)		
		X	Y	Z	X	Y	Z
West Bound	37 to 41	N	S	S	w/o	8000	500
East Bound	43 to 48	Y	S	N	w/o	3500	w/o

Table 3. Comparison of modal frequency

Modes	Measured Frequency	Calibrated Frequency	Absolute Error	Relative Error (%)
1	3.00	2.93	0.07	2.27
2	3.20	3.15	0.05	1.69
3	3.70	3.66	0.04	1.19
4	6.20	6.24	0.04	0.65
5	6.70	7.10	0.40	5.97
6	8.90	8.53	0.37	4.16

Table 4. Mass participation factors in percentage for the calibrated modes

Modes	X	Y	Z	SUMM-X	SUMM-Y	SUMM-Z
1	0.20	37.28	16.28	0.20	37.28	16.28
2	0.24	46.16	17.44	0.44	83.44	33.72
3	0.00	0.30	46.66	0.44	83.74	80.38
4	0.11	0.21	0.01	0.56	83.94	80.39
5	0.04	0.00	0.42	0.59	83.95	80.81
6	0.84	0.00	0.03	1.43	83.95	80.84

Table 3 lists the simulated frequencies of the calibrated FEM. For comparison purpose, we also include the measured data and the simulated frequencies of the initial FEM. It can be observed in the corresponding Figure 8 that the calibrated FEM results in less than 6% relative errors for all 6 principal modes. Meanwhile, the mass participation factor in percent is provided in Table 4. For y and z direction motion, the cumulative values of the first 3 modes already over 80%.

CONCLUSIONS

In this study, we have presented a finite element model calibration study for a span of the Verrazano Narrows Bridge in New York City. The nominal FE model is constructed for the real structure but represents it with a very limited accuracy. In order to minimize the discrepancy between the measured and modeled dynamic responses, the FEM calibration was undertaken to optimize the element elastic modulus of material attribute and the elastic stiffness of the boundary supports. The results obtained illustrate that the calibrated FE model of the bridge span was successfully achieved with the small relative errors in mode frequencies and was well-correlated with the vibration pattern in the mode shapes.

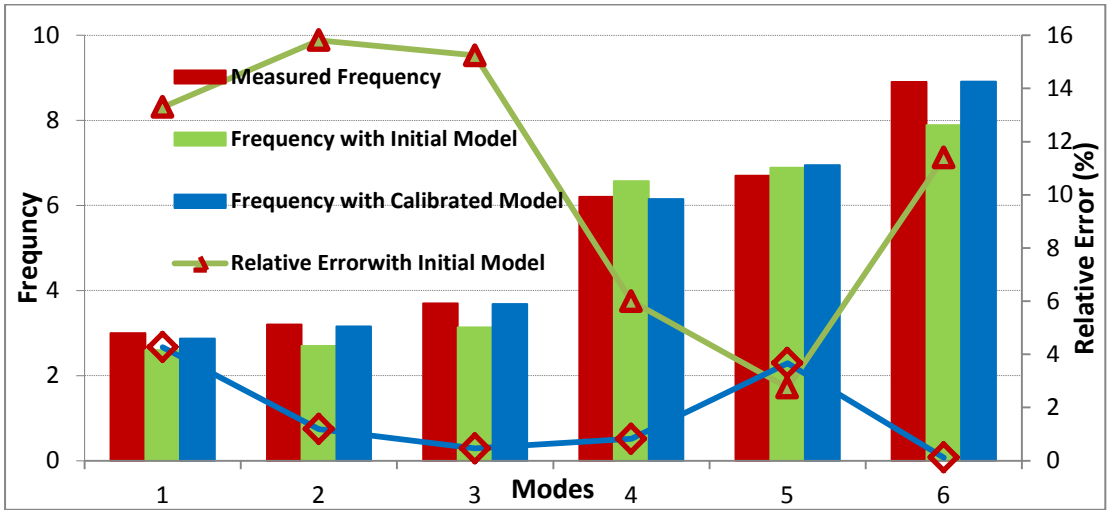

Figure 8. Comparison of modal frequency between the measured, the nominal FEM and the calibrated FEM

REFERENCE

Goldberg, D. (2002). *The Design of Innovation: Lessons from and for Competent Genetic Algorithms,* Kluwer Academic Publishers: Dordrecht.

Bentley Systems, (2012). *STAAD.Pro v8i User's Manual*, Yorba Linda, CA, USA

Hao, H., and Xia, Y. (2002). "Vibration-based Damage Detection of Structures by Genetic Algorithm", *Journal of Computing in Civil Engineering*, 16(3), 222-229.

Jeary, A. and Winant, T. (2012). *Structural Identification Report - Dynamic Measurements Conducted for Verrazano Narrows Bridge Span*, STRAAM Incorporation, New York City, NY, USA.

Mottershead, J.E. and Friswell, M. I. (1993). "Model updating in structural dynamics: a survey", *Journal of Sound and Vibration* 167 (2), 347-375.

Sanayei, M., Bell, E. S., Javdekar, C. N., Edelmann, J. L., and Slavsky, E. (2006). "Damage Localization and Finite-Element Model Updating Using Multiresponse NDT Data", *Journal of Bridge Engineering,* 11(6), 688-698.

Soh, C. K., and Dong, Y. X. (2001). "Evolutionary Programming for Inverse Problems in Civil Engineering", *Journal of Computing in Civil Engineering*, 15(2), 144-150.

Wu, Z. Y. and Xu, G. (2011). "Integrated Evolutionary Optimization Framework for Finite Element Mode Identification", *The First Middle East Conference on Smart Monitoring, Assessment and Rehabilitation of Civil Structures*, Feb. 8-10, 2011, Dubai, UAE

Wu, Z. Y., Wang, Q, Butala, S., Mi T. and Song Y. (2012). *Darwin Optimization User Manual*, Bentley Systems, Incorporated, Watertown, CT 06795, USA.

Wu, Z. Y., Yi, X., Zhao, J., Jeary, A, Winant, T., Love, R. and Bhide S. (2013). "Optimizing sensor placement and dynamic measurements of Verrazano Narrows bridge span", *The 6th International Conference on Structural Health Monitoring of Intelligent Infrastructure*, Dec. 9 - 11, 2013, Hong Kong, China.

Wu, Z. Y., Wang, Q. and Butala S. (2011). "Generalized Framework for high Performance Infrastructure System Optimization", *International Conference on Computing and Control for the Water Industry*, Sept. 5-7, 2011, Exeter, UK.