16th World Conference on Earthquake, 16WCEE 2017
Santiago Chile, January 9th to 13th 2017
Paper N° 3628
Registration Code: S-3628

MODEL UPDATING AND SEISMIC ANALYSIS OF THE PORT MANN BRIDGE

S. McDonald[(1)], C. Ventura[(2)]

[(1)] *Structural Engineer in Training, AECOM, steven.mcdonald@aecom.com*
[(2)] *Professor, University of British Columbia, ventura@civil.ubc.ca*

Abstract

Model updating serves the purpose of providing a more representative finite element model of a structure. Engineers and researchers need a tuned model for many applications such as damage detection, structural modification assessments, or seismic assessments. It is very common for experimental results to contradict the results taken from a computer model, which demonstrates a need for having a properly calibrated model. In this paper, a model of a three-span cable-stayed bridge is updated using an ongoing structural health monitoring network, and a linear time history analysis is carried out to compare results before and after updating. A finite element model is created based on design drawings, and both manual and automatic updating methods are applied. First, the loading on the deck is manually calibrated to reasonably agree with experimental results; and second, important parameters are identified based on a sensitivity analysis, and then the model is manually updated to closely match the frequencies and mode shapes of the experimental results. Special attention is also paid to the boundary conditions at the ends of the bridge and at each tower, as well as the stiffnesses at the foundations. In order to provide a comparison between the original and updated models, a suite of nine scaled ground motions are selected for a time history analysis. Three crustal, three subcrustal, and three subduction earthquake motions are selected for the analysis. Observations from the seismic analysis reveal the importance of having a more confident, calibrated finite element model.

Keywords: Model updating; Ambient vibration testing; Linear time history analysis; Cabled-stayed bridge

1. Introduction

Engineers and researchers strive to better understand the behavior of structures in order to improve the confidence in their designs and analyses. Computer models have developed over the years to provide more capabilities for larger structures and rigorous, complex analyses. Many structures are analyzed in finite element programs to determine capacity, changes in construction, retrofits, and even damage detection. Given these applications, it is imperative that there is a high level of confidence in the finite element model; however, models still tend to differ from their experimental counterparts. The art of model updating has been around for years, but its application has not been adopted very often.

In order to demonstrate the effectiveness of model updating, as well as document the process, this paper provides a case study of its application and a comparative study based on a linear time history analysis. While the application remains in the linear range, it still allows the results to be compared to see just how significant model updating can be. The current study is a small part of a potentially large and far-reaching project—there is still much to be explored, and many components of the study were not able to be included.

1.1 Sources of Model Error

When creating a finite element model, many assumptions and idealizations have to be made in order to represent the physical structure. These assumptions can lead to errors in the results. Examples of idealization errors include improper boundary conditions, joint connections, or external loads. The finite element method also involves discretization of the mathematical model into individual elements and approximating the structural response based on these elements. There are some errors associated with this discretization, but it is generally not possible to compensate for them. Another source of error is in the assumed physical properties, such as elastic modulus, mass density, or cross sectional properties. These are the main errors that can be compensated for in the model updating process. Many sources of error are described in the paper by Mottershead [1].

1.2 Ambient Vibration Testing

As the most important piece for model updating, ambient vibration testing serves to provide the results to which the finite element model should aspire to portray. There are sources of error in ambient vibration testing, but with advanced techniques in signal processing and modal identification, many of these errors have been reduced. It is therefore reasonable to assume the proportion of error with ambient vibration testing is much lower than that associated with finite element modelling, and for this reason the results are used as the main target for finite element model calibration.

1.3 Model Updating Approach

The art of model updating requires three main components: an experimental model which identifies the main mode shapes of the as-built structure; a finite element model which matches as closely as possible to what was designed; and a high level of engineering judgment. Essentially the process involves modifying various properties, referred to as parameters, in order to match the results obtained from the experimental model. The difficulty is in selecting appropriate parameters and using discretion as to how much to vary them and finding a combination of parameter changes that still reflect a physically realistic structure. After all, a physically meaningful model is required for most applications. The modification of parameters can be done manually or by various automated, iterative procedures [2].

2. Description of the Port Mann Bridge

The Port Mann Bridge was completed in 2012 to replace the original aging bridge. The older Port Mann Bridge had opened in 1964 and it became one of the most travelled bridges in North America. The new bridge spans a total of 2020 metres and boasts an impressive width of 65 m. It had claimed the Guinness World Record for the world's widest bridge until a year later, in 2013, when the new San Francisco Bay Bridge was opened.

The Port Mann Bridge has three main sections: the cable-stayed main span, a north approach, and a south approach. The cable-stayed main span will be the focus of the current research. The main span is 850 m long and is composed of steel girders and cross beams that support precast concrete deck panels. The deck is divided into two separate but similar sections, and it is connected together with hollow steel median struts. The north and south approaches are constructed using concrete box girder sections. A total of 288 steel strand-type cables are used for the bridge, and they connect into two large concrete piers. Fig. 1 shows a picture of the Port Mann Bridge looking from the south side. In it you can see the stay cables, the steel girders, and the main concrete piers. Fig. 2 displays a simplified elevation of the Port Mann Bridge that will be the main area of focus for the study. This study uses only the main span in its analysis, and the piers are labelled N1, S2, N2, and S2.

Fig. 1: Port Mann Bridge looking south

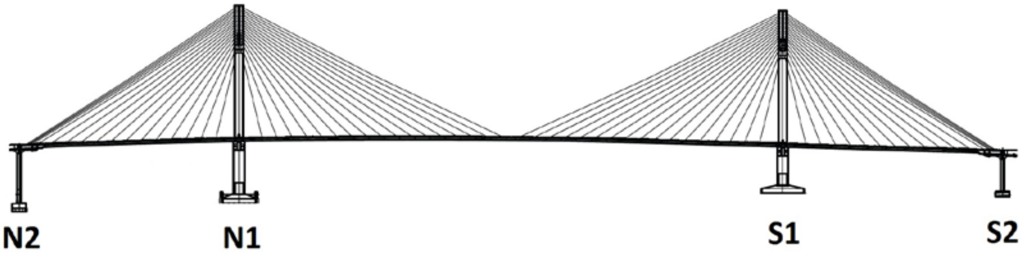

Fig. 2: Simplified elevation of Port Mann Bridge

3. Experimental Results

The first step to model updating involves gathering experimental data that captures the dynamic behaviour of the structure. The Port Mann Bridge is equipped with multiple accelerometers as part of its structural health monitoring program. These sensors are installed at important points along the structure to be able to gain insight into its operating conditions, as well as provide a means for monitoring the structure after large events such as earthquakes. The location of each sensor used in the analysis is shown in Fig. 1. The data from these sensors were downloaded and analyzed using the program ARTeMIS [3]. A modal analysis was carried out to identify the frequency characteristics of the Port Mann Bridge. The model created with the associated channel orientations is shown in Fig. 2.

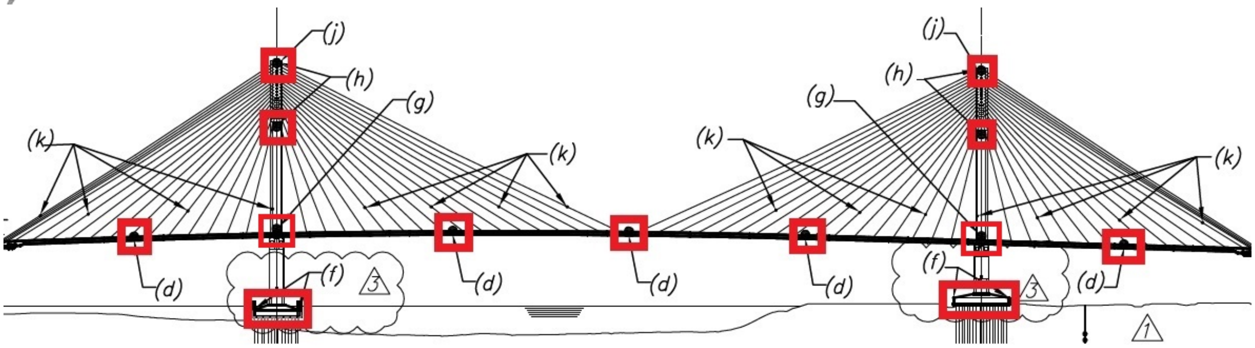

Fig. 3: Location of sensors used for experimental data

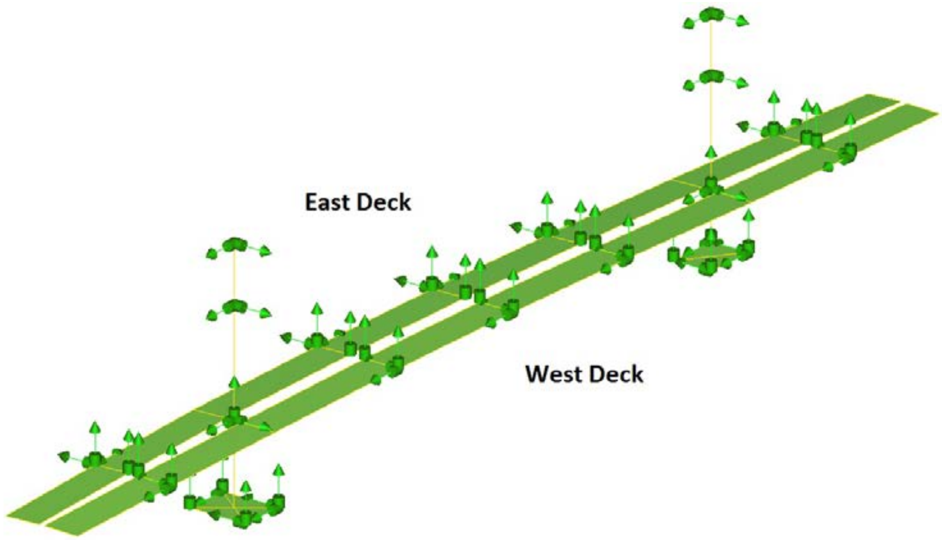

Fig. 4: Model created in ARTeMIS for modal analysis

As can be seen, there are sensors installed along the deck and the main towers. There are also accelerometers at the foundations and on the cables, but these were not able to be incorporated into the current study. It would be beneficial in the future to include an analysis of these areas. The data was analyzed using the frequency domain decomposition technique and further validated by using a stochastic subspace identification method. Table 1 lists the first five mode shapes that were identified. A larger study [4] included a total of 13 mode shapes, but they are not shown in this paper for brevity. The first five mode shapes include two verticals and three torsional-based modes. There were no clear transverse deck mode shapes that were identified with the data.

Table 1: Summary of first five mode shapes from experimental data

Mode Number	Frequency (Hz)	Damping Ratio (%)	West Description
1	0.233	0.68	1st Vertical
2	0.251	0.84	1st Torsional
3	0.272	0.64	2nd Torsional
4	0.302	0.66	2nd Vertical
5	0.432	1.14	Mid-span Torsion

4. Finite Element Model

The Port Mann Bridge was modelled by the original structural engineers using the program ADINA [5]. However, for the scope of the research, it was more beneficial to have the model in SAP2000 [6] because it is more accessible and easy to modify properties by editing tables. SAP2000 also works alongside the program FEMTools [7], where the two programs can be run together to aid in the model updating process. This was not done in the current study, but having the model in SAP2000 allows future research to take full advantage of the FEMTools capabilities.

Transferring the model into SAP2000 was not a straightforward process; many properties and elements had to be recreated entirely. The total model consists of 9707 frame elements which make up the main towers, approach piers, steel deck girders, and steel deck cross beams. The final model, which only includes the main span section of the bridge, is displayed in Fig. 3.

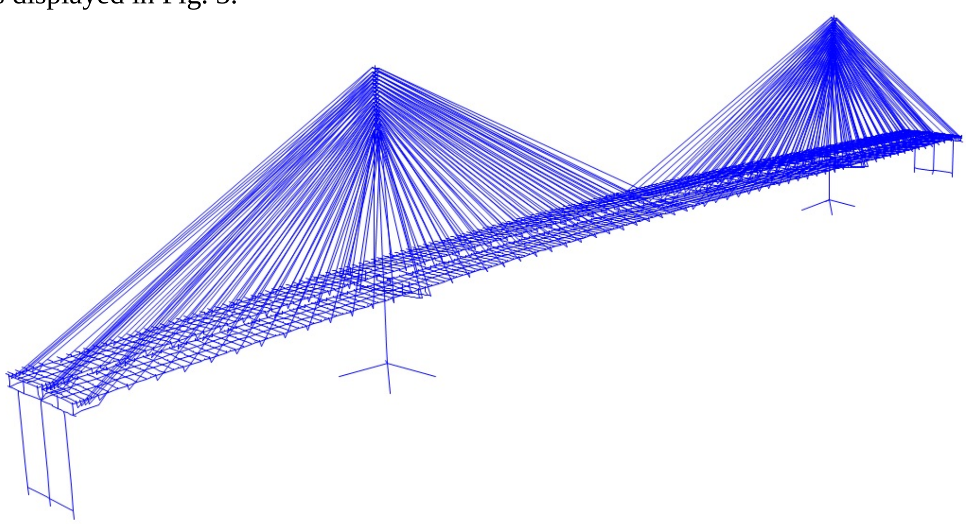

Fig. 5: Finite element model in SAP2000

Many important components must be included in the model in order to properly replicate the real structure. The Port Mann Bridge has the following main components: foundations, north and south approach piers, north and south main pylons, the deck, and the cables. Even with the most diligent and precise creation of the model, it is impossible to determine the exact properties and boundary conditions of the constructed bridge. On top of that, there are also limitations in the finite element method itself. It is for this reason that model updating can benefit—it allows a rigorous treatment of parameters that can converge on a suitable solution to the physical behaviour. The other important parts of the finite element model include:

- Modelling of the foundation stiffnesses

- Method for modelling cables (catenary behaviour, truss elements, etc.)
- Restraint conditions at deck-to-pier connections

The foundation was originally modelled as a rigid spring at each pier, and it was intended to investigate the effect of this stiffness with a sensitivity analysis. The stay cables are modelled as simple truss elements, but it might be useful to research in the future the effect of different modelling techniques. Lastly, the restraints for the pier connections are modelled as closely to that as was intended in the design, but a sensitivity analysis will be carried out to determine what effect changes in these restraints will have on the results.

After carrying out a modal analysis in SAP2000, the results are compared with what was originally obtained from the experimental data. Table 2 shows the comparison including the frequency differences and the modal assurance criterion (MAC) values. The MAC values are a correlation between two different mode shapes: a value of 100% means a perfect match. The table shows an average difference of 8% in frequencies, and an average MAC value of 85.6%. It is clear that there is room for improvement in the new finite element model created.

Table 2: Comparison of first five mode shapes from experimental and SAP2000 data

FEA Mode	SAP2000 Freq. (Hz)	Test Freq. (Hz)	Diff. (%)	MAC (%)
1	0.263	0.233	12.59	99.0
2	0.275	0.251	9.31	80.7
3	0.298	0.272	9.56	86.2
4	0.329	0.302	8.85	97.5
5	0.431	0.432	-0.28	64.7

5. Model Updating

Model updating uses the experimental and analytical models in order to refine the finite element model. The experimental model was used as a benchmark with which to compare the finite element model. The program FEMTools was used as an aid in correlating the two models and carrying out a generic sensitivity analysis.

As part of model updating, you must first choose what response values you want to use as a target. In this case, the frequencies and MAC values are selected for updating. Secondly, a set of parameters must be decided upon that will be modified in order to calibrate the model. The selection of parameters is the most important part of model updating; a significant amount of engineering judgment must be exercised in weighing the uncertainty in various properties and the sensitivity of changes in these properties to the dynamic behaviour. The engineer or researcher must establish a set of parameters that have enough sensitivity, are physically meaningful, and are diverse enough to be able to converge on a solution to the model calibration problem.

Both the experimental and analytical model were imported into FEMTools and overlapped, as shown in Fig. 4. The mode shapes were correlated together based on frequency and MAC value thresholds. These paired mode shapes are used as the targets for model updating.

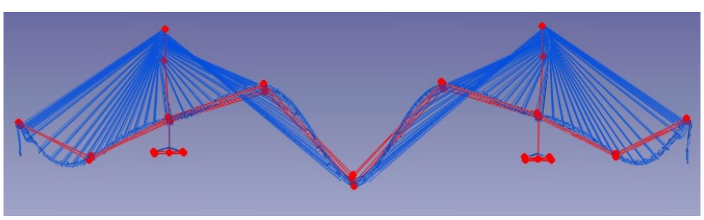

Fig. 6: Example of correlated mode shape

In order to determine suitable parameters for model updating, various sensitivity analyses were carried out. The program FEMTools was utilized to carry out a sensitivity analysis with various material properties. The sensitivity matrix shown in Fig. 5 is an example of how sensitive the mode shapes are to changes in the elastic modulus and moment of inertia of various members. This sensitivity analysis narrowed down the potential set of parameters to an amount that was more feasible.

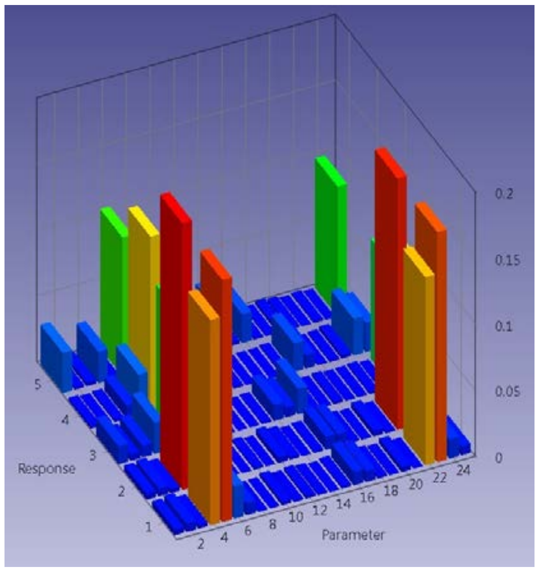

Fig. 7: Sensitivity matrix for E and I values

On top of the automated sensitivity analysis, sensitivities were explored in the SAP2000 model through trial-and-error. These included investigations into the significance of the foundation stiffnesses, the cable properties, and the boundary conditions. After considering the uncertainties and the sensitivities, a final set of parameters were decided upon:

- E for main piers N1, S1 and for the steel girders
- Moment of inertia for main piers N1,S1 and for steel girders
- Mass on top of the deck

While it was found that the boundary conditions and cable properties had a significant effect on the dynamic properties of the model, they were excluded as parameters. The cables were not considered in the updating procedure for two main reasons: the steel material properties are considered to have a higher confidence than other structural components, and each cable in the model is assigned its own slightly varying material properties. Thus, if the cables were included, it would be computationally cumbersome to modify all 288 different properties. The boundary conditions for the main piers were found to have little effect on the overall analysis results, but there was a notable difference in changes to the approach piers. If a translational release was provided at the approaches, the effect disproportionately changes the average difference in frequencies. A disproportionate change in frequencies indicates that a rigorous component-level model updating approach may be necessary; but this increases the difficulty in obtaining a model that converges to a solution and significantly increases computational demand, so it is was decided to exclude translational stiffness as a potential parameter.

It was originally intended to use FEMTools and its automatic updating tools, but due to the size of the model, there were problems encountered when trying to converge to a solution. Therefore, the model was manually updated and carried out in three steps. The first step involved adding mass to the deck, as there was a high level of uncertainty. The second step calibrated the parameters for the piers N1 and S1; and the third step calibrated the parameters of the steel girders. The final mode shape comparison after model updating is shown in Table 3, and the summary of parameter changes is shown in Table 4.

Table 3: Mode shapes after model updating

FEA Freq. (Hz)	Test Freq. (Hz)	Diff. (%)	MAC (%)
0.234	0.233	0.33	99.0
0.251	0.251	-0.11	85.7
0.274	0.272	0.85	88.3
0.296	0.302	-1.97	97.7
0.432	0.432	-0.04	65.4

Table 4: Parameters modified for model updating

Parameter	Parameter	Modifiction
Pier N1 and S1	E	-10%
	I_2	-10%
Deck Girders	E	-5%
	I_2	-10%
	I_3	+20%
Deck Weight Addition	W	12.25 kN/m

After model updating, the average frequency difference went from 8% to 0.6%, resulting in a very good correlation between mode shapes. The parameters were changed by 5-10% with the exception of the moment of inertia of the deck girders. Generally the parameter changes were limited to a maximum of 10% in order to maintain a physically realistic model.

5. Linear Time History Analysis

A dynamic analysis was carried out in SAP2000 to demonstrate the significance of model updating. This analysis is intended to provide a means of comparison between the original model and the updated model. It can show how significant some design values may change if the finite element model is updated to match the as-constructed experimental data.

A comprehensive set of ground motions were selected to capture a wide range of possible dynamic inputs that the structure may be subject to. These ground motions were selected and scaled based on the bridge's experimental periods. In total, three from each of crustal, subcrustal, and subduction ground motions were selected. A summary of the ground motions are presented in Table 5.

Table 5: Ground motions selected for linear time history analysis

	Name	Magnitude	Peak Accel. (g)	Scale Factor
Crustal:	Chi-Chi 1999	6.2	0.15	2.21
	Loma Prieta 1989	6.9	0.97	0.60
	Gazli 1976	6.8	1.26	0.74
Subcrustal:	Miyagi	7.2	0.15	2.60
	Olympia	7.1	0.27	2.26
	Geiyo	6.4	0.47	3.87
Subduction:	Tohoku1	9.0	0.09	2.34
	Hokkaido	8.0	0.10	2.51
	Tohoku2	9.0	0.09	2.67

The ground motions were imported into SAP2000 and a linear time history analysis was carried out. There were some key response values that were used for comparison: the shears and moments at the bases of the main pylons N1 and S1, and the approach piers N2 and S2, were reported, as well as the displacements at the piers and mid-span of the decks. Table 6 concisely summarizes the absolute maximum changes in responses due to model updating. For example, the original model results showed a maximum transverse displacement of 148 mm at the mid-span of the deck. After model updating, this value increased by 79% to become 265 mm. On average, these key responses changed by 37%. The effect of model updating is quite significant—an increase in elastic forces or displacements by 20-70% demonstrates this.

Table 6: Largest changes in response due to time history analysis after model updating

Response Type	Change (%)	Response Type	Change (%)	Response Type	Change (%)
N1 Base V2	31.8	**N1 Base M2**	25.7	**N1 UX**	68.8
S1 Base V2	26.1	**S1 Base M2**	28.4	**S1 UX**	40.6
N2 Base V2	17.2	**N2 Base M2**	24.4	**N1 UY**	43.3
S2 Base V2	31.4	**S2 Base M2**	20.8	**S1 UY**	40.3
N1 Base V3	25.8	**N1 Base M3**	26.1	**Deck UX**	78.6
S1 Base V3	31.5	**S1 Base M3**	23.6	**Deck UY**	47.9
N2 Base V3	24.0	**N2 Base M3**	40.6	**Deck UZ**	50.4
S2 Base V3	29.2	**S2 Base M3**	31.6		

5. Conclusions

Being able to predict how a structure is actually going to behave allows engineers and other stakeholders to make important decisions. Many designs are based off of finite element results that have not been put into perspective; models are created to the best of the engineer or researcher's knowledge, but this can often still have significant discrepancies with the actual measured response. While true validation of a finite element model is not exactly possible, applying engineering judgment and using a benchmark experimental model allows calibration of a model that can be considered much more reliable and accurate than what was originally created; it allows the designer to have more confidence in their design.

The Port Mann Bridge, one of the largest cable-stayed bridges in North America, is monitored with sensors in real-time, allowing the streaming of data which can be imported to carry out an operational modal analysis. The results of the experimental data acted as the benchmark dynamic properties with which to compare the finite element model. After modelling the bridge in SAP2000, the results were correlated with the experimental results using the program FEMTools. Once the two models were prepared, a detailed investigation into parameter selection was carried out to determine suitable properties to change. A number of parameters were chosen, and the model was calibrated by manually varying these parameters. In the end, an updated model was created which had an average frequency difference of 0.6% and an average MAC value of 89%. Following the model updating, some main conclusions can be made:

- The finite element model is sensitive to changes in material and section properties of the steel deck girders, concrete main towers, cables, and longitudinal stiffness of the approach connection.
- To maintain a physically meaningful model, parameters must be chosen and modified with engineering insight, and great care should be taken in the amount and distribution of their variation.

The linear time history analysis was used to demonstrate the significant of model updating. Overall, the updated model had some large and some insignificant variations, such as a 79% increase in transverse displacement at the mid-span from the Chi-Chi ground motion; and only a 0.2% increase decrease from the Olympia ground motion. This also shows the importance of a broad suite of ground motions, because one ground motion might not be affected as much by the frequency shifts in the updated model, whereas another ground motion could have been largely affected.

With the current state of art in model updating, it is important to note that a finite element model cannot be truly validated against the experimental results. This places a large responsibility on the engineer or researcher to exercise educated judgment in the modification of parameters. In the art of model updating, there is also no one solution. With the vast amount of parameters to choose from, many combinations of parameter changes can converge to a similar result.

There is much that still needs to be investigated for this project in the future. The scope of the current work included simplifications and limitations which can be expanded upon. They can be summarized as follows:

- Expand model to include approach spans and nonlinear properties.
- Explore alternative cable modelling techniques.
- Include other experimental data from cables, foundations, and approach piers.
- Update model using more rigorous, automated updating procedure in FEMTools
- Include parameters at a component level, rather than only a global level

5. Acknowledgements

I would like to thank Dr. Ventura for his generous support and motivation for this research. His encouragement was greatly appreciated throughout the process. I would also like to thank Dr. Yavuz Kaya who facilitated the process of obtaining the experimental data. Lastly, Dr. Martin Turek was also very helpful in providing me with the design information that I needed to complete the work.

6. Copyrights

7. References

[1] Mottershead, J. E., Link, M., & Friswell, M. I. (2011). The sensitivity method in finite element model updating: A tutorial. *Mechanical Systems and Signal Processing*, 2275-2296.

[2] Marwala, T. (2010). Finite-element-model Updating Using Computational Intelligence Techniques. Springer-Verlag London Limited.

[3] Structural Vibration Solutions, Inc. (2015). *ARTeMIS Modal Software*. Retrieved from Structural Vibration Solutions Website: http://www.svibs.com/

[4] McDonald, Steve (2016): Operational modal analysis, model updating, and seismic analysis of a cable-stayed bridge. *Master's Thesis.* The University of British Columbia, Vancouver, Canada.

[5] ADINA R & D, Inc. (2015). *ADINA Software*. Retrieved from http://www.adina.com/

[6] Computers and Structures Inc. (2014). *SAP2000 V15*. Retrieved from http://www.csiamerica.com/products/sap2000

[7] Dynamic Design Solutions NV. (2014). Retrieved from FEMTools Software: www.femtools.com