Simulation of various damage scenarios using finite element modelling for structural health monitoring systems

V.V. Nguyen, J. Li, Y. Yu, U. Dackermann
Centre for Built Infrastructure Research, University of Technology Sydney, Sydney, Australia

M. M. Alamdari
Smart Infrastructure, Data61, CSIRO, Sydney, Australia

ABSTRACT: Structural health monitoring (SHM) is a developing technology for asset management of structures including bridge assets. A crucial benefit of SHM is its ability to monitor the health status of structures using continuous measurements. As a key in SHM, the application of damage detection algorithms to assess the condition of a structure using vibration measurements can be enhanced by providing structural information under various damaged scenarios, which can be obtained from updated numerical models that realistically represent the in-situ structure. However, the dynamic characteristics of a structure are sensitive to uncertainties of various parameters, including material properties and boundary conditions, which require updating in the finite element (FE) model to ensure that the model replicates the actual structure. This study focuses on the development of an FE model for the accurate simulation of a jack arch replica structure of the Sydney Harbour Bridge. An experimental jack arch replica is produced to simulate various damage scenarios for laboratory testing. A matching FE model of the jack arch replica is generated and updated using genetic algorithm (GA) based on experimental measurements. Damage is simulated in the updated model and the results are validated using the experimental test results. The successful simulation of damage using updated FE models enables the generation of a large number of damage cases that can be trained into an SHM system to improve its damage detection capabilities.

1 INTRODUCTION

In an effort to ensure the safety of the general public, bridge asset managers are investing in more reliable methods that continuously monitor the condition of their structures. Traditionally, visual inspection has been used in Australia to assess the condition of these assets; however, due to budgetary constraints and the need for consistency in the assessments, it is impractical to collect data on the condition of these assets at the desired time intervals (Kotze, Ngo and Seskis 2015). Structural health monitoring (SHM) has been implemented in bridges in the past couple of decades. Originally used in the aerospace, automotive and general mechanical engineering field, health monitoring has been gaining activity in the civil engineering field. Ko and Ni (2005) highlighted SHM implementation in China and explored potential uses for a range of different sensors in detecting damage and assessing the health of a structure. Hsieh et al. (2006) provided some examples of such applications in Salt Lake City where SHM was implemented on I-80 Flyover Bridge, South Temple Bridge and I-215 Curved Girder Overpass Bridge to monitor and record the dynamic behaviour of the bridge, develop non-destructive testing methods and to determine the feasibility of using SHM for structural condition assessments.

This study investigates the application of SHM on a replicated jack arch component of the Sydney Harbour Bridge (SHB). Damage is incrementally inflicted into this specimen using a circular saw. Experimental modal analysis is conducted on the specimen before damage is inflicted and after each damage increment. A finite element model is produced and updated to a develop knowledge based model whereby the modal parameters determined experimentally are synchronised with those determined numerically by optimising predefined updating parameters. Damage is subsequently simulated in the updated model and the results are verified with experimental data.

2 EXPERIMENTAL SETUP

The specimen used in this investigation is a replica of a jack arch component from the bus lane of the SHB. The specimen is a cantilever beam that is 1 m wide, 2 m long and has a height of 375 mm. The specimen is fixed to a steel clamp on one end and is supported by a mechanical jack in the middle section. A steel I-beam is embedded in the concrete specimen with 50 mm of concrete cover on both sides. The tip of the specimen is shown in Figure 1.

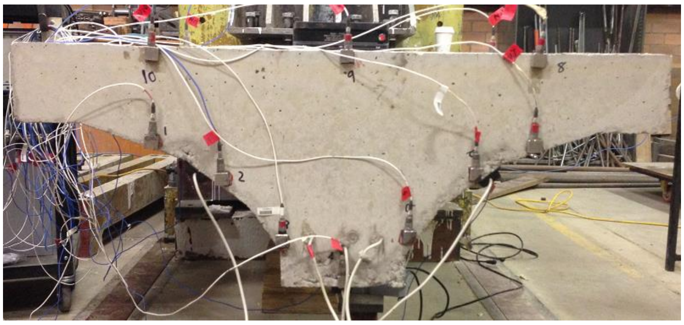


Figure 1: Experimental specimen of jack arch replica.

Damage was simulated in the specimen by using a circular saw to cut the specimen diagonally across its tip to simulate a crack, which occurred in one of the jack arches on the SHB as described by Mustapha et al. (2015). The crack has since been repaired. The damage was inflicted to the specimen in four increments to simulate different severities of damage as shown in Figure 2. These five condition cases are subsequently used to analyse algorithms to detect the presence and severity of damage.

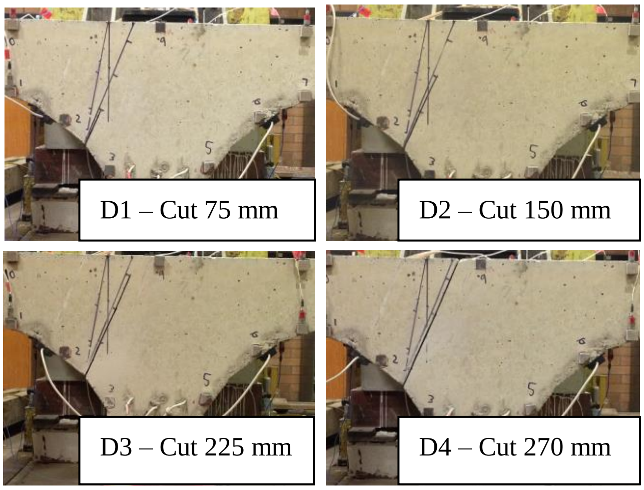


Figure 2: Four damage cases of specimen.

Experimental modal analysis was conducted on the specimen to identify its modal parameters. An impact hammer was used as reference force and 15 accelerometers were used to measure the response. The configuration of the accelerometers is shown in Figure 3. The impact hammer was used to excite the specimen near the corner 50 mm from its tip and 50 mm from the right of the specimen to excite a range of vibrational modes including bending and torsional modes. The response of the structure was measured for two seconds at a sampling rate of 8 kHz for each impact. This modal testing was repeated 20 times for all condition cases.

The time history of the reference and the response were processed to determine the frequency response functions (FRFs) using the H1 estimator. That is, by dividing the cross spectrum of the response and reference by the auto spectrum of the reference. Then, for each condition case, a representative FRF was calculated by averaging the 20 determined FRFs - thereby maintaining their repeatable features while removing the non-repeatable features that were likely attributed to random noise.

Modal parameter identification was then carried out using the commercial software Siemens LMS Modal Analysis. The Time Multiple Degree of Freedom Module was used to conduct the modal parameter identification, producing a stabilisation diagram from which vibrational modes could be identified. The stabilisation diagram is shown in Figure 4.

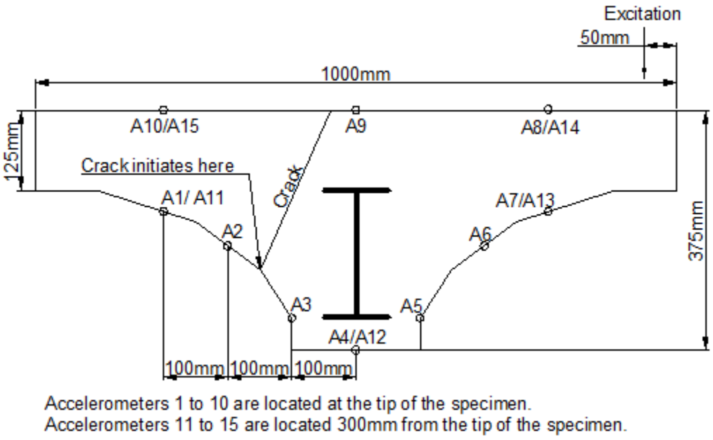


Figure 3: Accelerometer locations for modal analysis.

The vibrational modes were selected based on the stable poles in the stabilisation diagram and the peaks identified in the FRFs. The mode shapes were then processed to extract the modal displacement vector at the sensor locations and identify the mode shape types for each mode. This procedure was repeated for all damage cases to identify any incremental changes in the natural frequency (NF) caused by damage. The results are summarised in Table 1.

Table 1. Mode shape types and natural frequencies identified using experimental modal analysis.

Mode	Mode Type	Healthy NF (Hz)	D1 NF (Hz)	D2 NF (Hz)	D3 NF (Hz)	D4 NF (Hz)
1	Vertical bending	45.1	45.1	45.1	45.1	45.0
2	Torsional	180.2	180.3	180.1	179.6	179.2
3	Vertical bending	263.0	263.4	262.9	262.8	262.3
4	Torsional	572.3	572.3	571.8	571.7	571.1
5	In-plane bending	905.4	904.2	902.1	900.8	898.1

It can be observed from Table 1 that the changes in the natural frequencies identified in the specimen as a result of inflicting damage incrementally are most predominant in mode 5. While all modes show a general trend of natural frequencies decreasing with increasing damage, this trend is clearer in mode 5. This indicates that the cut that was inflicted to the specimen can be more easily identified when observing the in-plane bending mode.

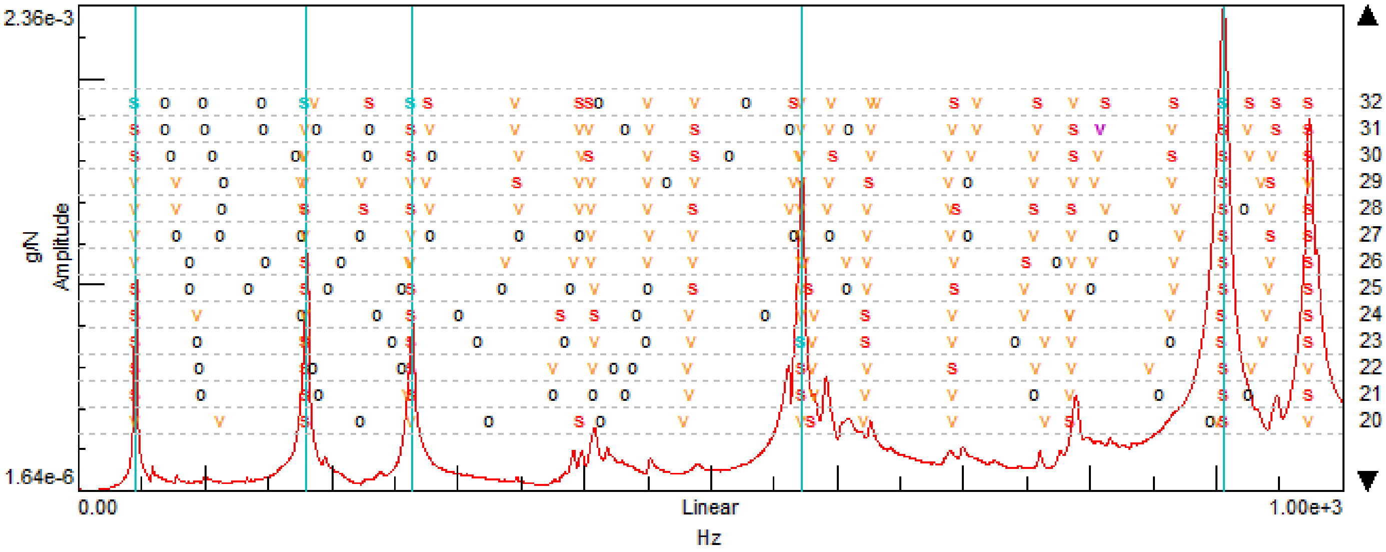


Figure 4: Stabilisation diagram

3 FINITE ELEMENT MODELLING

3.1 *Initial Finite Element Model*

A finite element model (FEM) of the experimental specimen was produced using the commercial software ANSYS APDL to simulate the vibrational response of the specimen. The concrete was modelled using SOLID65 elements and the material properties were based on the static chord modulus of elasticity (MOE) tests conducted on cylinders from the specimen. The concrete had an MOE of 35 GPa and a Poisson's ratio of 0.223. Its density was measured as 2320 kg/m^3. The steel I-beam embedded within the specimen was modelled using SOLID185 elements with standard material properties. It was modelled as having an MOE of 200 GPa and a Poisson's ratio of 0.3. Its density was modelled as being 7850 kg/m^3. The clamp supports at the fixed end of the beam was modelled with the same material properties as the steel I-beam. Perfect bonding was assumed between the support fixtures and the specimen. The model is illustrated in Figure 5.

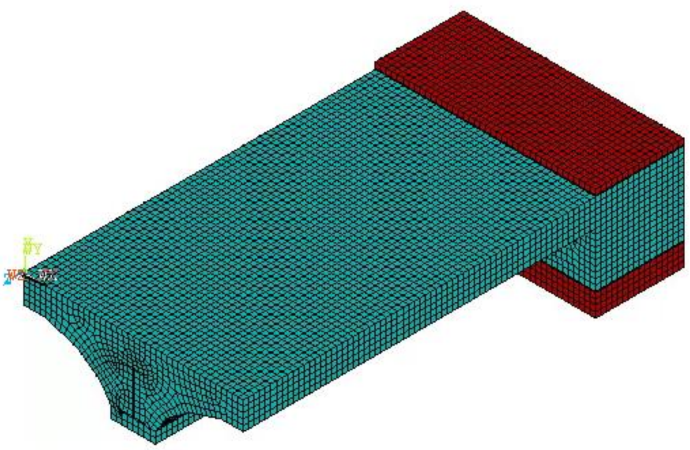


Figure 5: Initial finite element model.

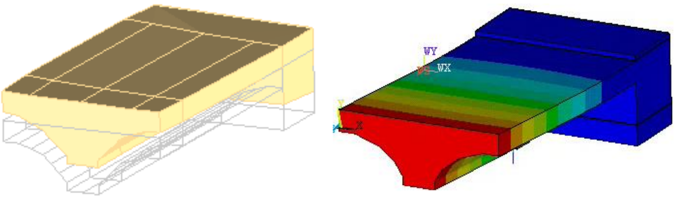
The natural frequencies and mode shapes of the model were extracted using the Block Lanczos eigenvalue analysis method (Montgomery 1995). In order to compare the modes identified in the finite element model to the ones identified through experimental modal analysis, it was necessary to ensure that the modes were paired. Furthermore, this procedure needed to be automated so that an optimisation procedure could clearly correlate the inputs into the eigenvalue analysis with its outputs without human intervention. Brehm, Zabel and Bucher (2010) have highlighted that the modal assurance criterion (MAC), shown in Equation 1, is a very meaningful mode pairing criteria.

$$MAC_{jk} = \frac{\left|\phi_{mj}^T \phi_{ak}\right|^2}{\left(\phi_{ak}^T \phi_{ak}\right)\left(\phi_{mj}^T \phi_{mj}\right)} \qquad (1)$$

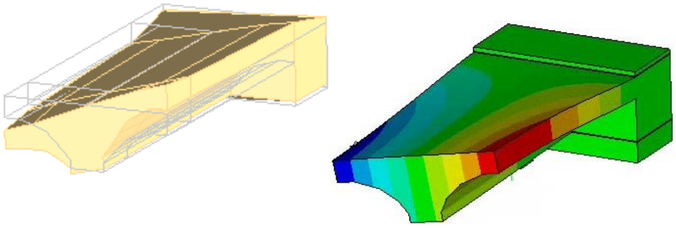
where ϕ_{mj} is the measured modal vector of the jth mode and ϕ_{ak} is the analytical modal vector of the kth mode. A MAC matrix comparing the finite element modes to the experimental modes was created to determine how well the experimental modes matched with the analytical modes. The highest MAC value was identified for each experimental mode and the corresponding finite element mode was paired with that particular experimental mode. This was a necessary procedure, as the change in the parameters of the model can alter the mode shape and the order of the natural frequencies. In addition, some of the modes that were identified in the eigenvalue analysis of the finite element model were not detected in the experimental modal analysis procedure. For example, the second mode that was identified in the eigenvalue analysis was a lateral bending mode which was not picked up in the experimental modal analysis, as the accelerometers were positioned to capture vertical acceleration.

A comparison between the mode shapes and natural frequencies (NF) of the experimental structure (left) and the numerical model (right) is summarised in Figure 6.

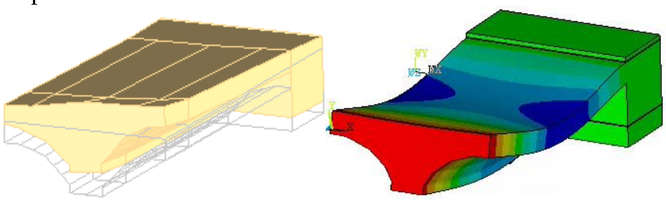
Mode 1 –Vertical Bending MAC 0.999
Experimental NF 45.05 Hz Numerical NF 73.29 Hz



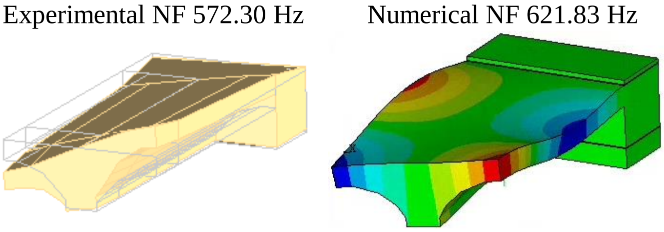
Mode 2 – Torsion MAC 0.993
Experimental NF 180.20 Hz Numerical NF 202.59 Hz



Mode 3 – Vertical Bending MAC 0.926
Experimental NF 262.99 Hz Numerical NF 374.10 Hz



Mode 4 – Torsion MAC 0.992
Experimental NF 572.30 Hz Numerical NF 621.83 Hz



Mode 5 – In-plane bending MAC 0.934
Experimental NF 905.43 Hz Numerical NF 939.10 Hz

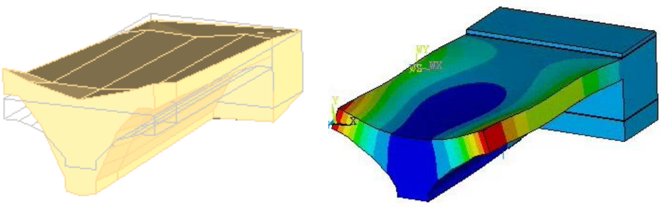


Figure 6: Pairing of first five identified modes.

3.2 *Model Updating Parameter Selection*

In order to produce a more realistic and accurate FE model of the specimen, it was refined implementing the following features. The initial model considered the supports to be fully fixed, whereas in reality, the specimen was placed inside a steel clamp with plaster used as adhesive to bond the specimen to the support. Further, a mechanical jack was used to provide additional support to the specimen. The materials tested from the specimen may only reflect the material properties from a concentrated area and may not represent the material properties of the specimen as a whole. The bonding of the steel I-beam and concrete also needs to be considered. All these features influence the vibrational response of the structural component. The implemented refinements are illustrated in Figure 7, which shows the specimen sliced along the middle.

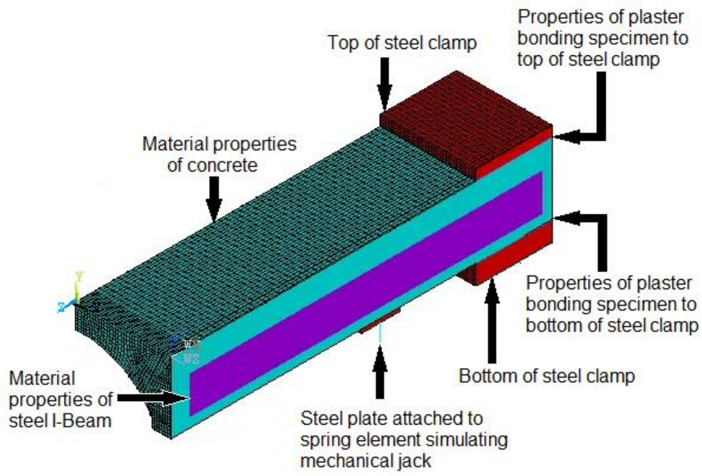


Figure 7: Updated parameters of FE model.

The first set of parameters to be identified in the FE model are the material properties of concrete and steel, which make up the composite structural component. A range of ±5% is considered for the material properties to account for their potential variability. The fixed end of the specimen was assumed to be fully fixed in the initial FE model. However, further investigation of the specimen indicated that this was not the case. Imperfections were observed when inspecting the fixed support of the specimen and gaps between the concrete and the steel clamp support could be identified. In addition, the plaster used to bond the specimen to the support was not as stiff as the specimen or the steel support. This was implemented in the model by smearing the imperfections of the layer of plaster that was used as adhesive to bond the specimen to the clamp. In addition, a steel plate was modelled with a spring element to simulate the mechanical jack that was used to provide additional support to the specimen. The spring element was modelled with a varying stiffness to be updated.

3.3 *Model Updating using GA Optimisation*

There are two main approaches of updating an FE model, i.e. direct methods and iterative methods (Friswell and Mottershead 1995). The direct methods update the entire stiffness and mass matrices in a single non-iterative solution. Whilst this method is able to reproduce the measured data exactly, any physical meaning from the initial FE model is lost in the updating process. The iterative methods use updating parameters with physical meaning to simulate the structure in more detail. This leads to having knowledge-based models.

Optimisation was used to update the FE model using the iterative approach to achieve a more realistic representation of the specimen. In this investigation, Genetic Algorithm (GA) was used to optimise the parameters of the model due to its higher probability of converging to a global optimal solution than a gradient-based method (Marwala 2010).

The optimisation of the FE model was achieved by interfacing ANSYS with MATLAB. ANSYS is a general purpose FE analysis software package and while it excels in allowing the user to define a model and extract the modal parameters from the model, the optimisation algorithms provided by the software was found to be inadequate in finding a solution for the global optimisation problem. A GA optimisation procedure was prepared to update the parameters of the model to ensure that the numerical model represented the real life specimen. The detailed optimisation procedure can be divided into the following steps:

Step 1. Create initial FE model. Initialize the algorithm parameters, including chromosome number N=10, maximum iteration number T=50, crossover probability p_{cr}=0.7, mutation probability p_m=0.01 and variation ranges of inputs. To avoid possible problems of ill-conditioning, the updating parameters should be kept small (Nguyen 2015). A Spearman's rank correlation matrix was produced to determine the input parameters that had the highest correlation to the modal parameters of the model. Parameters that were found to be insignificant were removed from the optimised procedure. The reduced parameters and their limits are shown in Table 2.

Table 2. Reduced updating parameters

Parameter	Minimum	Maximum
Concrete Modulus of Elasticity (MPa)	31500	38500
Concrete Poisson's Ratio	0.2	0.24
Concrete Density (kg/m3)	2280	2520
Plaster layer bonding bottom of specimen to clamp (MPa)	1	500
Plaster layer bonding top of specimen to steel clamp (MPa)	1	500

Step 2. Set initial iteration number t=1 and randomly generate initial chromosome.

Step 3. Launch ANSYS software and set up the FE model based on the current chromosome.

Step 4. Conduct eigenvalue analysis and calculate the fitness value based on the objective function shown in Equation 2. The fitness value is the inverse of the objective function.

$$Objective\ Fuction = \sum_{i=1}^{N} \left(\frac{\omega_i^m - \omega_i^{calc}}{\omega_i^m} \right)^2 \qquad (2)$$

Step 5. Adopt the roulette wheel strategy to choose part of the chromosome for the generation of new chromosomes.

Step 6. Conduct the crossover and mutation operations to generate new chromosomes.

Step 7. Judge the stopping criterion. In this study, the maximum iteration was selected as the termination condition of GA. When the current iteration arrives at its maximum value (T=50), the GA will be terminated and output the final FE model. Otherwise, t=t+1 and go to Step 3.

A flow chart summarising the procedure used for the GA is shown in Figure 8.

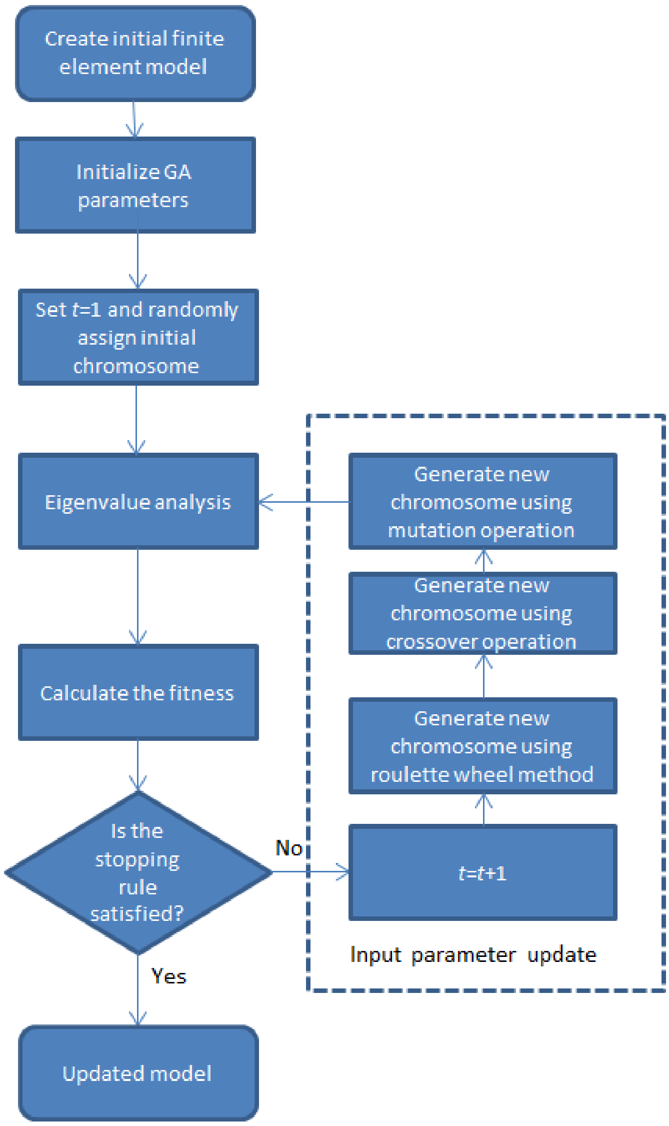


Figure 8: Flow chart of optimisation procedure

The optimisation procedure was run to update the FE model. Figure 9 shows the increase in the fitness value with each of the iterations in the genetic algorithm. A comparison between modal parameters of the updated FE model and the experimental specimen is shown in Table 3. The results show that the genetic algorithm was successful in optimising the updating parameters to closely match the FE model to the physical specimen.

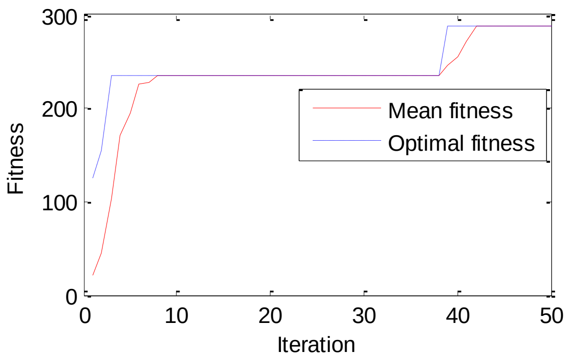


Figure 9: Genetic algorithm fitness value for each iteration.

Table 3. Comparison between the natural frequencies of the specimen and updated FE model.

	Experimental NF (Hz)	Updated FEA NF (Hz)	Error
Mode 1	45.05	45.45	0.88%
Mode 2	180.20	181.98	0.99%
Mode 3	262.99	274.17	4.25%
Mode 4	572.30	586.19	2.43%
Mode 5	905.43	932.52	2.99%

4 DAMAGE SIMULATION

A focus of this study is the use of modal analysis to detect damage in a structure and the potential to realistically model the damage. As shown in Table 1, the in-plane bending mode was found to be more sensitive to the saw cut than the other modes. The damage detection component of this study therefore focuses on this mode. The FE model was updated a second time, focusing on optimising the in-plane bending mode. The four incremental saw cuts that were inflicted to the specimen were simulated in the model by removing elements with the same geometrical space as the saw cut. A comparison between the natural frequencies (NF) identified in the experimental specimen and the FE model for all damage cases is shown in Table 4.

Table 4. Changes in natural frequency of fifth mode.

Crack Length (mm)	0	75	150	225	270
Experimental model NF (Hz)	905.4	904.2	902.1	900.8	898.1
Updated FE model NF (Hz)	905.2	903.4	902.9	902.6	901.2

The results of the experimental and the numerical model show that the natural frequency of the in-plane bending mode decreases as damage increases. This trend indicates that damage can be detected from observing the changes in the natural frequency of the mode that is most sensitive to damage.

5 DISCUSSION AND FUTURE WORK

In the past couple of decades, model updating has been extensively researched for civil engineering applications. This paper explored the potential to update an FE model with the results from vibration testing on a structural component inflicted with local damage (the cracking at the tip of the specimen). The success of this study has the potential to lead to further investigations into damage detection algorithms for SHM systems on actual bridges. While the results from this study showed that both the modal analyses conducted on the updated FE model and the experimental specimen were able to capture the indicative changes from the in-plane bending mode for the special type of damage, further investigation needs to be conducted to capitalise on such findings. For example, it is recommended that the change in the natural frequency of a mode needs to be at least 5% for it to be a reliable indication of damage (Salawu 1997). In this study, the changes in natural frequencies due to the damage might be small for the practical application of the method. In next stage of this research, besides natural frequencies, more parameters will be considered for detecting the local damage including an exploration of mode shapes and modal strain energies, thorough investigation of model updating parameters, more robust mode pairing techniques and optimisation algorithms to generate an FE model as representative of the physical structure as possible. The initial results of this study are promising but further work is required.

6 REFERENCES

Brehm, M., Zabel, V., Bucher, C. 2010. An automatic mode pairing strategy using an enhanced modal assurance criterion based on modal strain energies. *Journal of Sound and Vibration* 329: 5375-5392.

Friswell, M.I. & Mottershead J.E. 1995. *Finite element model updating in structural dynamics*. Swansea: Springer.

Hsieh, K.H., Halling, M. W. & Barr, P.,J. 2006. Overview of vibrational structural health monitoring with representative case studies. *Journal of Bridge Engineering* 11: 707-715.

Ko, J.M. & Ni, Y.Q. 2005. Technology developments in structural health monitoring of large-scale bridges. *Engineering Structures* 27 (12): 1715-1725.

Kotze, R., Ngo, H. & Seskis, J. 2015. *Improved bridge deterioration models, predictive tools and costs*. Sydney: Austroads.

Marwala, T. 2010. Finite-element-model updating using computational intelligence techniques. London: Springer.

Montgomery P.L. 1995. A Block Lanczos algorithm for finding dependencies over GF(2). *Lecture Notes in Computer Science* 921: 106-120.

Mustapha, S. et al. 2015. Pattern recognition based on time series analysis using vibration data for structural health monitoring in civil structures. *Electronic Journal of Structural Engineering* 14(1): 106-115.

Nguyen, K., Freytag, B., Ralbovsky, M. & Rio, O. 2015. Assessment of serviceability limit stage of vibrations in the UHPFRC-Wild bridge through an updated FEM using vehicle-bridge interation. *Computers & Structures*: 29-41.

Salawu, O.S. 1997. Detection of structural damage through changes in frequency: a review. *Engineering Structures* 19 (9): 718-723.