

Damage detection on beam structures by means of VCUPDATE

A. Mordini & H. Wenzel

VCE - Vienna Consulting Engineers

Email: Mordini@vce.at

ABSTRACT: One of the most promising computational methods in the field of Structural Health Monitoring is the Finite Element Model Updating. The VCUPDATE solution presented in this work is a Scilab code performing the iterative updating algorithm interfaced with a Finite Element code (OpenSees or ANSYS) executing the numerical analysis. The code is applied to the damage detection on beam structures. At first, a simple case numerically generated by OpenSees is investigated. Subsequently, using experimental test data, the code is applied to a real structure by using OpenSees as well as ANSYS.

1 INTRODUCTION

Every Finite Element (FE) model is a numerical approximation of the real structure by several assumptions which can lead to heavy differences between the model and the real structure. Through the comparison of experimental data from dynamic testing and numerical data, these differences can be usually well highlighted.

The Finite Element Model Updating (FEMU) is a numerical technique used to minimize the differences between the real structure and the FE model. In fact, the FE model properties (e.g. mass, stiffness, boundary conditions) can be modified by the updating procedure and the resulting structure presents a better dynamic agreement with the physical reality.

Consequently, the FEMU can be employed also to the purpose of damage detection. In fact, by using experimental data from a damaged structure and applying the updating procedure, the resulting structural parameter distributions (e.g. elastic modulus) can point out a deficiency in the structural properties which means the occurrence of damage. In this optic, the FEMU can be proficiently applied in the field of Structural Health Monitoring.

This contribution presents the FEMU solution VCUPDATE, an iterative updating algorithm written in Scilab (Scilab Consortium 2006) and interfaced with the FE codes OpenSees (PEER 2006) and ANSYS (SAS 2005). The code has been already applied with good results to several different structures (Mordini et al 2007, 2008, Mordini & Wenzel 2007).

2 THEORETICAL BASICS

The FEMU can be performed using direct or iterative methods (Friswell & Mottershead 1995). The former has the advantages of not requiring iteration and of reproducing measured data exactly. However, if the measured data are inaccurate, a FE model with no physical meaning could be obtained. On the contrary, within iterative methods, a non-linear penalty function is minimized through subsequent linear steps and therefore, more computational time is required. In this work, a sensitivity-based iterative method is used. The main steps of the code are shown in Figure 1.

The software framework is controlled by the Scilab code. A graphical front-end has been developed in order to facilitate the input phase. The parameters are computed by the Scilab code and are subsequently passed-in to the FE code which performs the numerical analysis computing the eigenfrequencies and the mode shapes. Afterwards, this data are transferred to the Scilab code by using external files and they are used to compute the new parameter values. The program flow is controlled by a convergence criterion which stops, if the case, the iterative procedure.

There is no virtual limit to the parameter number and to the problem complexity to be investigated. Obviously, the computational time has to be taken into account in the FE model creation as well as in the parameter choice. In particular, the analyst should always remember that an iterative algorithm

can involve a high number of iterations and that the parameters should be chosen according to the investigated problem with engineering insight.

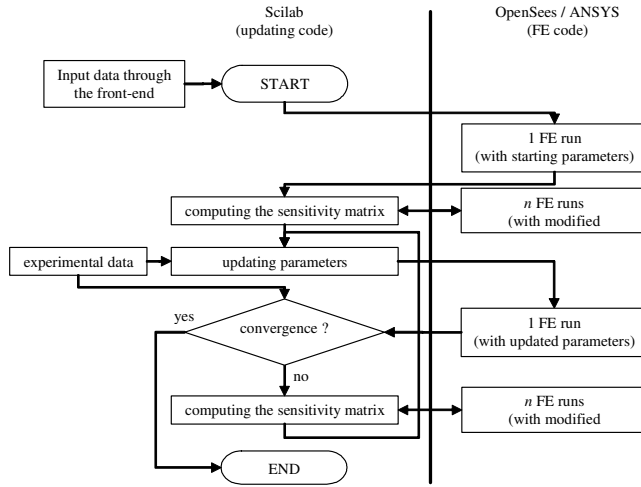


Figure 1. VCUPDATE architecture.

The first version of VCUPDATE was developed by using OpenSees. Subsequently, the connection with ANSYS was realized in order to provide the system with better modeling capabilities. Moreover, ANSYS allows the graphical visualization of the model and of the results, including the outcomes from the updating procedure. It has been observed, however, that the updating procedures by using OpenSees are much faster. Due to this advantage, the use of OpenSees is in general preferred.

Finally, both Scilab and OpenSees are open source and free of charge software. This aspect can be decisive for a wide range of engineering companies which are interesting in employing the FEMU applications but can not afford the high licensing prices of the commercial solutions available on the market.

2.1 Sensitivity analysis

The sensitivity analysis is performed to select the most sensitive parameters for the FE model. Moreover, the sensitivity matrix is used in the updating algorithm too.

At j^{th} iteration, the sensitivity matrix can be written as

$$S_j = \frac{\partial d_j}{\partial p_j} \quad (1)$$

where ∂p_j is the perturbation in the parameters and ∂d_j is the change between measured and numerical data.

The matrix S can be computed by analytical methods (direct derivation) as well as by numerical methods (perturbation techniques). If direct derivation is chosen, the differentiation of the structural eigenvalue problem with respect to the parameters must be performed, where the system stiffness and mass matrices are required. On the contrary, perturbation techniques can be carried out using results from multiple FE analyses, without knowledge of the system matrices.

In this work, the perturbation technique approach is chosen: the sensitivity matrix is calculated using the forward difference of the function with respect to each parameter. In particular, if the problem has n parameters, $n+1$ FE runs are required: the first one with the starting values for the parameters and n different runs perturbing one parameter only in each of them.

The forward difference method has the form (Jai-shi & Ren 2005)

$$S = \frac{\partial d}{\partial p} \approx \frac{\Delta d}{\Delta p} = \frac{d(p + \Delta p) - d(p)}{\Delta p}, \quad (2)$$

$$\text{with } \Delta p_i = \frac{\Delta D}{100} (\bar{p}_i - \underline{p}_i), \quad (3)$$

where Δp_i is the perturbation in the i^{th} parameter, ΔD is the forward difference step size and $(\bar{p}_i - \underline{p}_i)$ is the difference between the upper and lower bound for the i^{th} parameter. Each vector $\Delta d / \Delta p$ gives one column of the sensitivity matrix.

2.2 Sensitivity matrix and mode shape scaling

Using different structural properties with very different values as parameters may lead to numerical problems in the iterative solution (Dascotte et al. 1995). This problem arises from the ill-conditioning of the sensitivity matrix and is more probable when both frequencies and mode shapes are used as data. An appropriate scaling of the sensitivity matrix can improve the stability and speed up the convergence.

In this work, the sensitivity matrix is scaled by normalizing parameters and data by their initial values. The scaling matrices are defined as

$$D_p = \begin{bmatrix} \frac{1}{p_1} & & 0 \\ & \dots & \\ 0 & & \frac{1}{p_n} \end{bmatrix} \text{ and } D_d = \begin{bmatrix} \frac{1}{d_1} & & 0 \\ & \dots & \\ 0 & & \frac{1}{d_n} \end{bmatrix}. \quad (4)$$

The scaled parameters and scaled data can be written as

$$\Delta \bar{p} = D_p \Delta p \text{ and } \Delta \bar{d} = D_d \Delta d, \quad (5)$$

respectively. Therefore, the scaled sensitivity matrix is

$$\bar{S} = D_d S D_p^{-1}. \quad (6)$$

This method can lead to numerical problems when the mode shape values are very small or zero. In this case, a different way to scale the sensitivity matrix has to be used.

Experimental and numerical mode shapes should be scaled consistently to properly perform the updating algorithm. Moreover, experimental and numerical mode shapes can be 180 deg out of phase. To solve these problems, the Modal Scale Factor (*MSF*) can be used. In particular, within VCUPDATE, the i^{th} numerical mode is rescaled to the i^{th} experimental one by multiplying it by a *MSF* calculated as (Friswell & Mottershead 1995, Allemang & Brown, 1982):

$$MSF(\varphi_{exp,i}, \varphi_{num,i}) = \frac{\varphi_{exp,i}^T \varphi_{num,i}}{\varphi_{num,i}^T \varphi_{num,i}}. \quad (7)$$

2.3 Updating algorithm

The updating problem can be represented as the minimization of a penalty function $J(\Delta p)$ subjected to the constraint $\Delta d = S \Delta p$ (Friswell & Mottershead 1995). For each parameter, an upper and a lower bound can be specified as $p_i \leq p_i \leq p_i$.

Inserting the weighting matrices W_d and W_p for data and parameters respectively, the penalty function can be written as

$$J(\Delta p) = \varepsilon^T W_d \varepsilon + \Delta p^T W_p \Delta p. \quad (8)$$

Substituting the error $\varepsilon = \Delta d - S \Delta p$ and minimizing $J(\Delta p)$ with respect to Δp , gives the updated parameter values as

$$p_{j+1} = p_j + [S_j^T W_d S_j + W_p]^{-1} S_j^T W_d \Delta d_j; \quad (9)$$

where $\Delta d_j = d_0 - d_j$ is the difference between experimental and numerical data.

The sensitivity matrix should be computed at each iteration, but this can lead to an excessive computational time. VCUPDATE allows the user to choose the frequency with whom the sensitivity matrix is updated. For some extremely time consuming cases, the initial matrix can be used for the entire analysis.

According to experience, not all the experimental data are measured with the same accuracy. Usually, lower frequency data are more reliable than higher

ones. In order to express the user confidence in measured data, weighting matrices are used. The parameters can also be weighted separately. Using weighting matrices is very powerful, but engineering insight is required.

Five different ways to create the W_p matrix according to Dascotte et al. (1995) are implemented in VCUPDATE. Several different tests were performed in order to verify the methods: in particular, some of them should be used carefully, since they provide, in some cases, an increased convergence speed but, in other cases, the final results are worse.

2.4 Convergence criteria

The iterative scheme presented above is repeated until a convergence criterion is satisfied. The criterion can be based on parameters or on data.

Two different convergence criteria are implemented within VCUPDATE according to Dascotte & Vanhonacker (1989). The first is based on the frequency deviation:

$$CC_{abs,j} = \frac{1}{n} \sum_{i=1}^n \frac{|f_{exp,i} - f_{j,i}|}{f_{exp,i}} \text{ (at } j^{th} \text{ iteration),} \quad (10)$$

where $f_{exp,i}$ is the i^{th} experimental frequency and $f_{j,i}$ is the i^{th} numerical frequency at j^{th} iteration.

A different way to check the convergence is to include the mode shape correlation too. The most important indexes for this purpose are the Modal Assurance Criterion (*MAC*) (Allemang & Brown 1982) and the Normalized Modal Difference (*NMD*) (Maya & Silva 1997):

$$MAC(\varphi_j, \varphi_k) = \frac{|\varphi_j^T \varphi_k|^2}{(\varphi_k^T \varphi_k)(\varphi_j^T \varphi_j)} \quad (11)$$

and

$$NMD(\varphi_j, \varphi_k) = \sqrt{\frac{1 - MAC(\varphi_j, \varphi_k)}{MAC(\varphi_j, \varphi_k)}}; \quad (12)$$

The closer the *MAC* is to one, the better the correlation is. Since the *NMD* is much more sensitive than the *MAC* to the differences in similar vectors, it can be used when the mode shapes are highly correlated, while it is less useful for uncorrelated vectors. The closer the *NMD* is to zero, the better the correlation is.

The second convergence criterion implemented in VCUPDATE includes the *MAC* index:

$$CC_{tot,j} = \frac{1}{2nW_{max}} \sum_{i=1}^n \left(W_f \frac{|f_{exp,i} - f_{j,i}|}{f_{exp,i}} + W_\phi (1 - MAC_i) \right)$$

(at j^{th} iteration), (13)

where W_f and W_ϕ are the weights for the frequency and MAC deviation respectively and $W_{max} = \max(W_f, W_\phi)$. By using the weights, the balance between the importance of frequencies and mode shapes can be directly established.

The closer the selected CC approaches to zero, the better the agreement is between experimental and numerical data. The convergence is achieved if $CC \leq \varepsilon$ (point A in Figure 2). During the analysis all the information (updated data, updating parameters, CC) are stored and if the convergence is not attained, the updating procedure stops when a maximum number of iterations is reached. Then, the information corresponding to the iteration with the minimum value of CC (point B) is used as output.

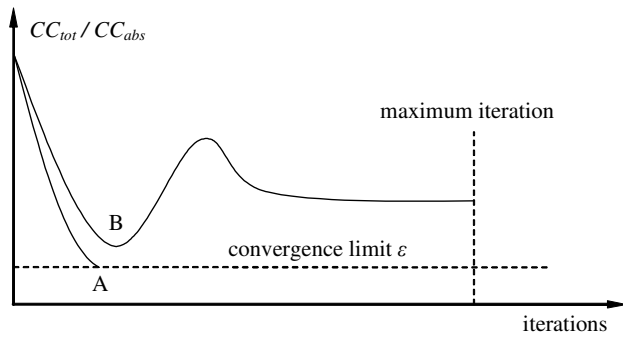


Figure 2. Convergence criterion.

2.5 Modal Expansion

In experimental tests, only a small number of locations can be instrumented and measured while the results from the FE model can be obtained virtually in any location. In order to estimate the eigenvectors in the unmeasured locations, a modal expansion method is implemented in VCUPDATE. The required data are estimated using the modal data from the FE model (Friswell & Mottershead 1995). Starting from the measured modes Φ_m and the numerical modes in the measured locations $\Phi_{n,m}$, a transformation T is defined assuming the measured modes to be a linear combination of the analytical modes:

$$\Phi_m = \Phi_{n,m} T \longrightarrow T = \Phi_{n,m}^+ \Phi_m, \quad (14)$$

where $^+$ denotes the pseudoinverse operation.

Then, the experimental data in the unmeasured locations Φ_u are computed from the numerical data in the unmeasured locations $\Phi_{n,u}$:

$$\Phi_u = \Phi_{n,u} T.$$

3 APPLICATIONS

3.1 Simply supported beam with simulated damage

In order to verify the implementation of the updating procedures, a simply supported beam is investigated by means of VCUPDATE. In this case, the experimental data are taken from an OpenSees run with modified structural parameters. The possible effect of noise is not considered. In the following paragraphs, “Numerical Experimental” is referred to the numerically generated data. The example, shown in Figure 3, is taken from Jaishi & Ren (2005).

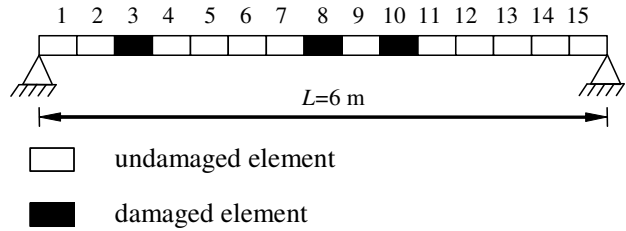


Figure 3. FE model of the simply supported beam with the damage locations.

The structural properties are: length $L=6$ m, elastic modulus $E=32000$ MPa, cross-sectional area $A=0.05$ m², moment of inertia $J=1.66 \times 10^{-4}$ m⁴ and mass per unit volume $M=2500$ kg/m³. The beam is modeled in OpenSees with 15 elements and the damage is simulated by imposing reduced material properties E and J for elements 3, 8 and 10 by 20, 50 and 30% respectively. A value of 0.0005 has been chosen for the convergence limit.

Table 1. Results of updating procedures for the simply supported beam.

n. of mode	exper. freq. f_0 [Hz]	initial model			case 1, updated		case 2, updated		case 3, updated	
		freq. f [Hz]	$\frac{f_i - f_{0,i}}{f_{0,i}}$	NMD	freq. f [Hz]	$\frac{f_i - f_{0,i}}{f_{0,i}}$	freq. f [Hz]	$\frac{f_i - f_{0,i}}{f_{0,i}}$	freq. f [Hz]	$\frac{f_i - f_{0,i}}{f_{0,i}}$
1	7.26	8.99	23.88%	0.03	7.26	0.00%	7.26	-0.03%	7.26	-0.04%
2	33.74	35.98	6.65%	0.05	33.74	0.00%	33.72	-0.06%	33.72	-0.04%
3	68.99	80.94	17.33%	0.11	68.99	0.00%	68.93	-0.08%	68.96	-0.05%
4	131.82	143.86	9.13%	0.09	131.82	0.00%	131.74	-0.06%	131.77	-0.04%
5	195.76	224.63	14.75%	0.20	195.76	0.00%	195.68	-0.04%	195.68	-0.04%
6	306.02	323.01	5.55%	0.13	306.02	0.00%	305.97	-0.02%	305.89	-0.04%
7	389.53	438.47	12.56%	0.23	389.53	0.00%	389.44	-0.02%	389.37	-0.04%
8	515.73	569.88	10.50%	0.24	515.73	0.00%	515.51	-0.04%	515.51	-0.04%
9	652.01	715.06	9.67%	0.23	652.00	0.00%	651.69	-0.05%	651.73	-0.04%
10	761.24	869.98	14.28%	0.44	761.24	0.00%	760.89	-0.05%	760.92	-0.04%
		CC_{abs}	0.1243		CC_{abs}	0.0000	CC_{abs}	0.0005	CC_{abs}	0.0004

The updating procedures are performed in order to evaluate the damage using the first 10 flexural frequencies and corresponding mode shapes as numerical experimental data. Three different cases are investigated.

- Case 1, 6 parameters. E and J of elements 3, 8 and 10 are used as updating parameters. The parameters are less than the considered modes. It is assumed that the damage locations are exactly known.
- Case 2, 10 parameters. E and J of elements 3, 5, 8, 10, 13 are used as parameters. The parameters are equal to the considered modes. It is assumed that the damage locations are partially known.
- Case 3, 30 parameters. E and J of all elements are used as parameters. The parameters are more than the considered modes. It is assumed that the damage locations are unknown.

The case of undamaged beam is the starting point for all the updating calculations. As the modes shapes are highly correlated, the NMD instead of the MAC is used.

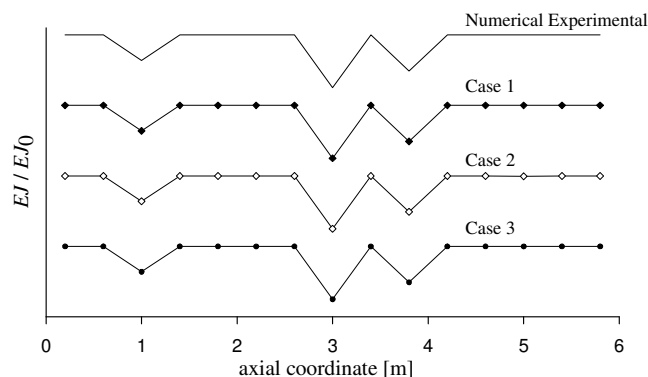


Figure 4. Damage localization.

The results of the updating procedures are shown in Table 1. The final values of NMD are approximately zero for all cases. The updating procedures provide almost perfect results and allow exactly determining the damage locations as shown in Figure 4 where the ratio of the updated bending stiffness to the initial one is reported for all the cases and compared to the numerical experimental one.

In order to verify the implementation of the modal expansion method implemented in VCUPDATE, the case 3 is repeated using only the eigenvectors of the nodes in the central part of the beam (from node 4 to 13). Then, the modal data for other nodes are estimated from the FE model. The case of undamaged beam is the starting point also for this analysis. In this case, since the modes shapes are scarcely correlated, the MAC is used. This procedure requires a higher number of iterations and therefore, more computational time with respect to the previous analyses.

Due to lack of space, not all the results can be here extensively reported. However, the agreement in terms of frequencies and damage detection is very high. The updating procedure provides a great improvement in the MAC index as can be seen from Figure 5. If the analysis is repeated using less numerical experimental data, good results are obtained for frequencies but the mode shape correlation is unacceptable.

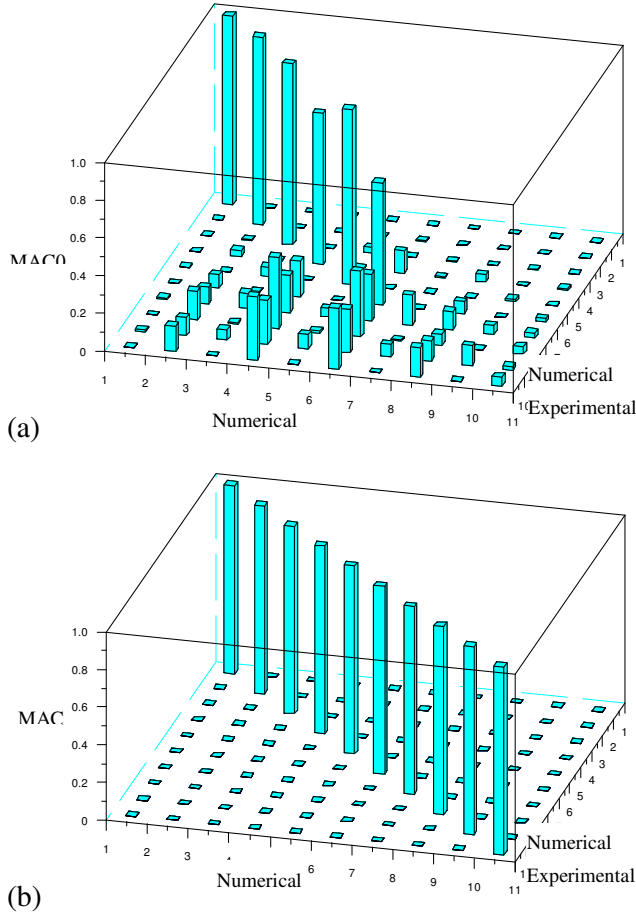


Figure 5. Initial (a) and updated (b) MAC matrix.

3.2 Reinforced concrete beam

A reinforced concrete beam tested in laboratory is investigated. The beam is part of an extensive laboratory program carried out on five identical beams with different damage location (Maeck 2003).

The beam has already been numerically investigated by the team of Professor Guido De Roeck (Teughels et al. 2003, Teughels et al 2002.).

The beam is indicated in the experimental program with the number 4. The beam was artificially damaged by applying a vertical static load at $x=4$ m in a three point bending configuration. Then, the modal test was performed in a free-free boundary condition configuration. This means that the beam supports were very flexible allowing free modal displacements.

This procedure was repeated for different load levels: in this work, the experimental data related to the step number 5 are considered. The corresponding imposed load is 25 kN.

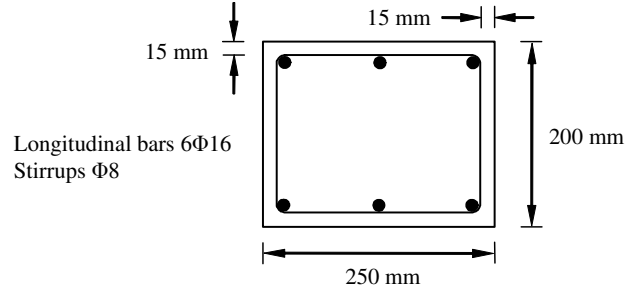


Figure 6. The investigated beam cross-section.

The structural properties are: length $L=6$ m, cross-sectional area $A=0.05$ m², moment of inertia $J=1.93 \times 10^{-4}$ m⁴ (considering the steel contribute, Figure 6) and elastic modulus $E=37500$ MPa.

To record the dynamics outcomes, 31 accelerometers are placed at both sides of the upper surface of the beam, for a total number of 62 sensors. The values from each couple of sensors are averaged to obtain only 31 values. The first 4 frequencies and mode shapes are recorded. Due to the damage, a reduction in frequencies and significant changes in mode shapes are observed. In the first mode, a slight torsional component is recorded but it cannot be observed in the modes shapes as the average value from the sensors with same axial coordinate is reported.

The beam is modeled using 30 2D beam elements and the elastic modulus for each element is taken as parameter. In order to reproduce the experimental conditions, the beam is supported by very flexible springs. This lead to the first four mode shapes shown in Figure 7 where no points with zero displacement common for all the modes are present.

The experimental setups for the damaging steps and the dynamic testing tests are shown in Figure 8a and Figure 8b respectively.

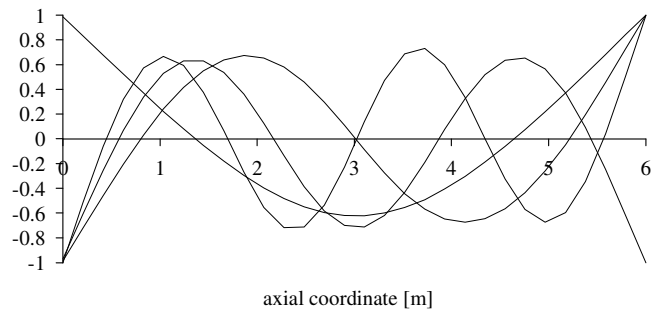


Figure 7. First four mode shapes.

Two different updating processes are performed, namely the initial-to-undamaged and the undamaged-to-damaged step. In the first, the initial FE model is updated to the undamaged beam in order to

eliminate the initial modeling errors. Then, in the second, this undamaged model is updated to the damage state. A value of 0.0005 for the convergence limit is used.

The convergence ratio is very fast: for the first updating process, two iterations only are necessary whereas for the second the convergence is achieved in three iterations only.

The results in terms of frequencies and *NMD* are reported in Table 2. The improvement in frequency values is very high and the FE model can be taken very close to the experimental model. This can be seen also from the updated CC_{abs} values which are almost zero.

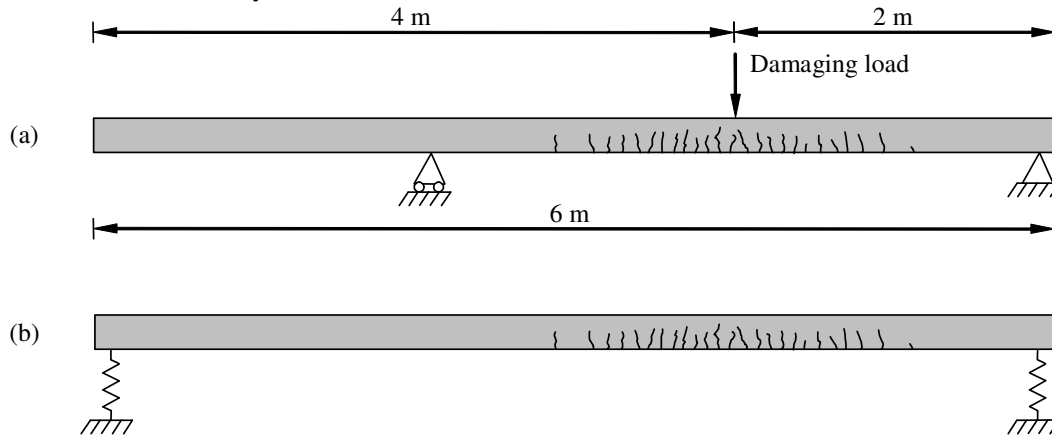


Figure 8. Static (a) and dynamic (b) test configuration with experimental crack pattern.

The improvement of the updating procedures on mode shapes can be seen in Figure 9 and Figure 10. The correction in the mode shapes is evident. The worst case is the first mode, probably due to the effect of the torsional component that is here neglected.

It can be concluded, therefore, that VCUPDATE is able to bring the numerical model to be almost coincident with the real structure in terms of dynamic behavior.

The comparison between the elastic modulus distribution obtained by VCUPDATE and a previous study (Teughels 2003) is reported in Figure 11. The reported elastic modulus ratio is strictly related to the damage since the loss of stiffness due to the progressive external load can be properly modeled as an elastic modulus decrement. This damage distribution can be as well directly compared to the experimentally obtained crack pattern reported in Figure 8b.

The damage can be successfully located with a smooth distribution. The initial-to-undamaged step is characterized by an elastic modulus reduction in the central part of the beam, probably due to the crack pattern induced by the self weight. On the contrary, in the undamaged-to-damaged step, the maximum change is located close to the applied load as can be seen from Figure 8. A parameter decrement can be recognized near the first support ($x=2$ m): this could be due to the effect of the beam self weight.

The damage distribution provided by VCUPDATE is in good agreement with the previous study where a more sophisticated algorithm is implemented. In fact, in the work of Teughels, the FE properties are not separately corrected, but a damage function obtained combining seven triangular shape functions is used. This procedure should avoid a non-realistic stiffness pattern with many peaks instead of a smooth distribution and other numerical problems. Moreover, the optimization problem has only seven variables and this should lead to save computational time. However, the VCUPDATE procedures are as well very effective since the solution is achieved in few iterations.

The same structure is investigated by VCUPDATE by means of the FE code ANSYS. In this case, a single step analysis is considered: the starting model is directly updated to the damaged beam. In the practical assessment of safety levels for Structural Health Monitoring purposes, in fact, this is the standard way to evaluate the existing structures when no previous test data are available.

The use of VCUPDATE in conjunction with ANSYS provides the user with increased modeling capabilities. The FE mesh can be freely chosen and there is no necessity to place a node of the FE model in the same location of a sensor where the eigendata are recorded. In this way, the best FE model can be chosen without taking care of the sensor position. The code requires the position of each sensor and

subsequently, the numerical mode shapes are computed in the sensor positions as a function of the mode shapes of the entire FE model.

Both a Beam and Plane stress model are used. The first contains the 2-node, 2D elastic beam BEAM3 element. It is a uniaxial element with tension, compression, and bending capabilities, with three degrees of freedom (two translations and one rotation) for each node. The Plane stress model is

based on the 4-node, 2D structural solid, PLANE42 element. It is a plane element used in plane stress case with two translational degrees of freedom for each node. For both analyses, the springs are modeled with the spring-damper COMBIN14 element. In order to test the mesh independence of VCUPDATE, several different mesh refinements are used in the calculations, for both Beam and Plane stress models.

Table 2. Updating procedure results for the OpenSees analysis.

n. of mode	experimental freq. f_0 [Hz]	starting model			updated model		
		frequency f [Hz]	$\frac{f_i - f_{0i}}{f_{0i}}$	NMD	frequency f [Hz]	$\frac{f_i - f_{0i}}{f_{0i}}$	NMD
<i>initial-to-undamaged step</i>							
1	22.02	23.72	7.72%	0.057	22.03	0.05%	0.054
2	63.44	65.20	2.78%	0.035	63.43	-0.01%	0.030
3	123.27	127.51	3.44%	0.060	123.26	-0.01%	0.030
4	201.92	210.25	4.13%	0.084	201.91	0.00%	0.048
		CC_{abs}	0.0451		CC_{abs}	0.0002	
<i>undamaged-to-damaged step</i>							
1	19.35	22.03	13.86%	0.094	19.35	0.02%	0.065
2	56.90	63.43	11.48%	0.104	56.89	-0.03%	0.043
3	111.64	123.26	10.41%	0.130	111.62	-0.02%	0.031
4	185.22	201.91	9.01%	0.171	185.18	-0.02%	0.038
		CC_{abs}	0.1119		CC_{abs}	0.0002	

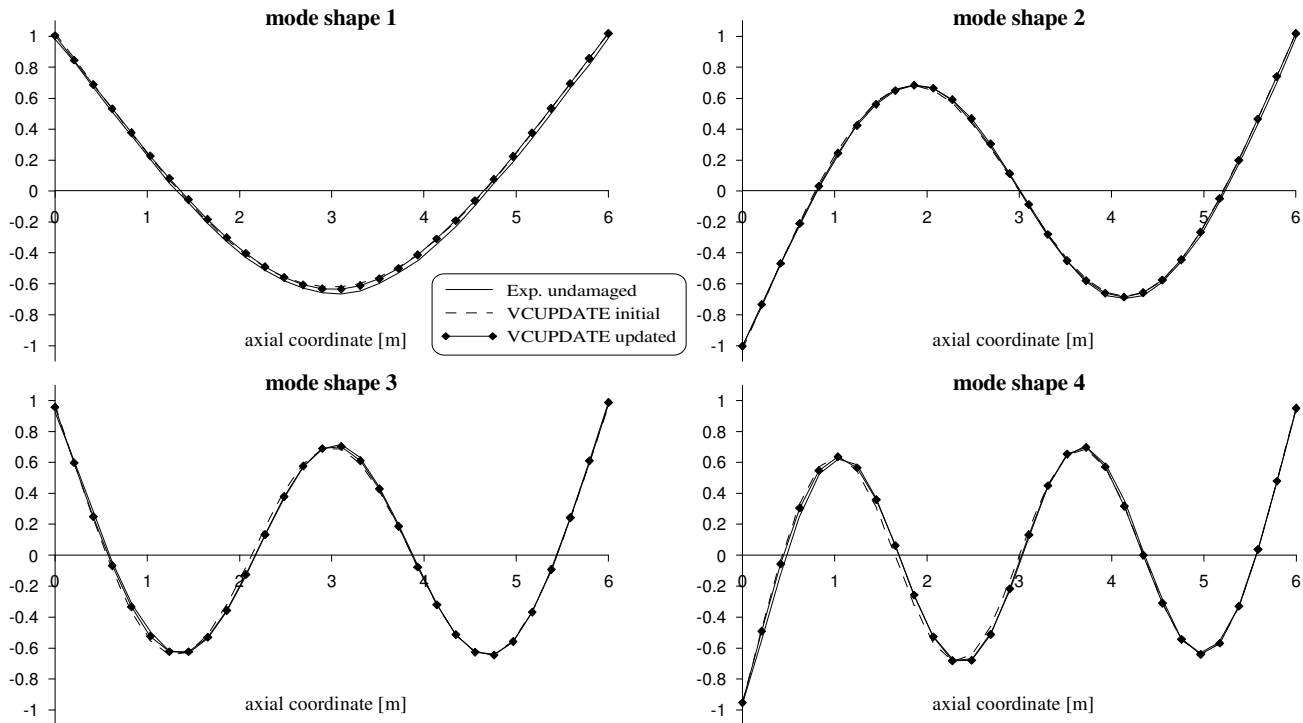


Figure 9. Experimental and numerical mode shapes for the *initial-to-undamaged* step.

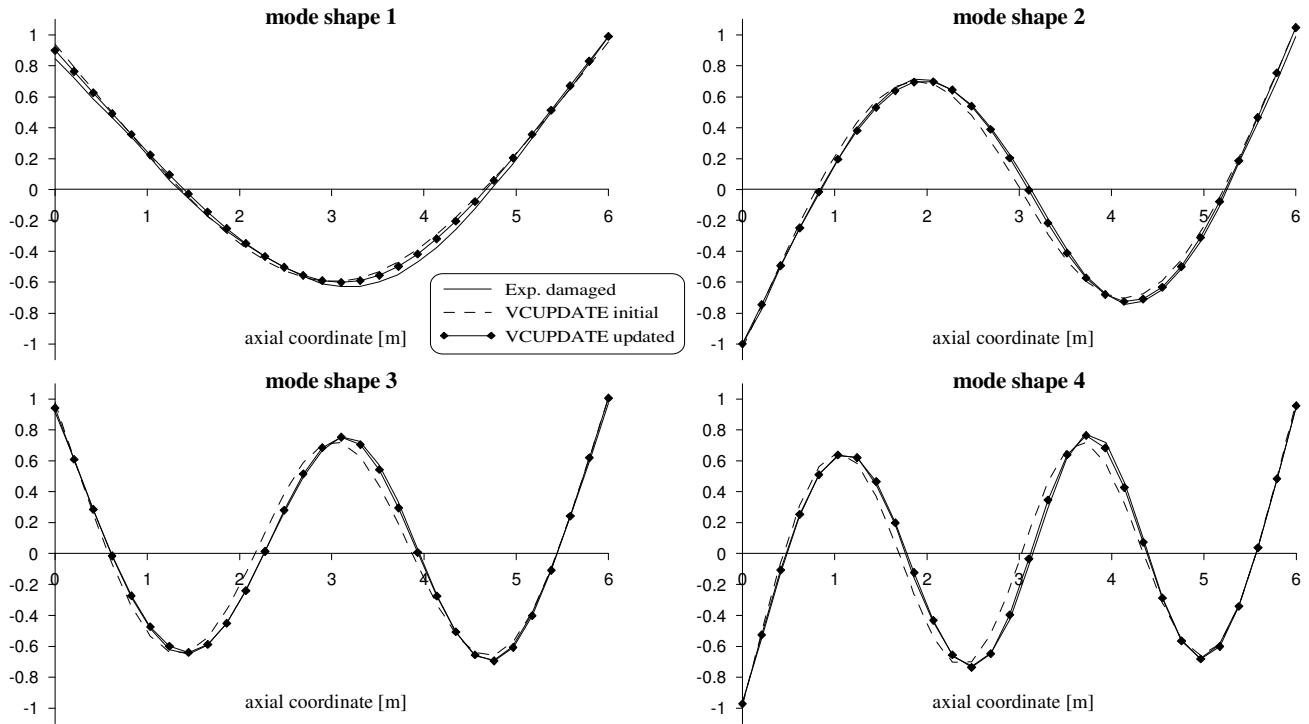


Figure 10. Experimental and numerical mode shapes for the *undamaged-to-damaged* step.

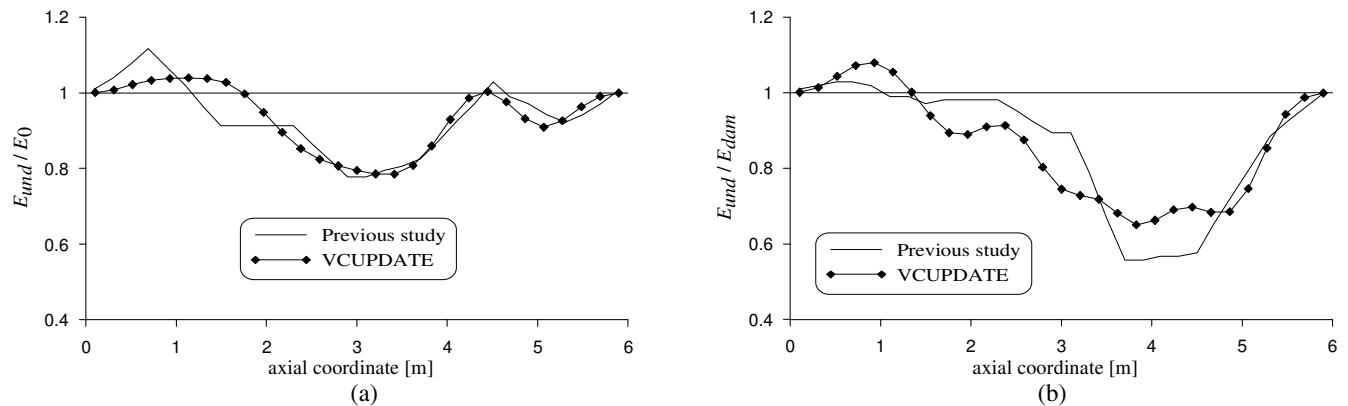


Figure 11. Damage distribution for *initial-to-undamaged* (a) and *undamaged-to-damaged* (b) step.

In Figure 12, the analysis outcomes in terms of elastic modulus distribution for the Beam model with two different meshes are shown. Analogously, for the Plane stress model, the results are shown in Figure 13 for an element size of 0.4 and 0.2 m respectively.

The damage distribution of these figures can be compared with the one from the OpenSees analysis (Figure 11). It can be noted that the agreement is, in general, very satisfactory. The results in terms of eigenfrequencies, *NMD* values and mode shapes are as well very good but they cannot be extensively reported here due to lack of space.

In general, both models and all the meshes can capture very well the physical structural behavior. Moreover, it can be noted from the pictures, that the employed mesh has a negligible influence on the outcomes. Obviously, it is always required that the adopted mesh can effectively describe the structural response.

In Figure 12, it can be noted that the left part of the beam presents an elastic modulus higher than the starting one. This is, in principle, not possible, but the same phenomenon is revealed also in the OpenSees calculation and in the previous study (Figure 11). Therefore, it is possible that this discrepancy is

due to the experimental data or to a real non-homogeneous distribution of the structural stiffness due, for example, to a local different concrete drying process. Furthermore, it is clear that the effect is less remarkable in the Beam model with the refined mesh. This is probably due to the fact that the fine mesh can better describe the structural behavior. Moreover, the higher the number of elements, the higher the number of parameters in the updating procedure and therefore, the better is the possibility to reach the minimum of the objective function. In the Plane stress model (Figure 13), the same problem is even more pronounced. But, again, in case of

more refined mesh, this phenomenon is limited to a very few elements.

It has to be mentioned that no numerical error could be addressed to mode shape expansion since in this case this procedure is not used.

Moreover, no errors can be addressed to concrete non-linearities since the level of the excitation during the dynamic investigation was very low. This means that, even in the damaged case, the structure remained in a limited range where no remarkable change of stiffness is recorded.

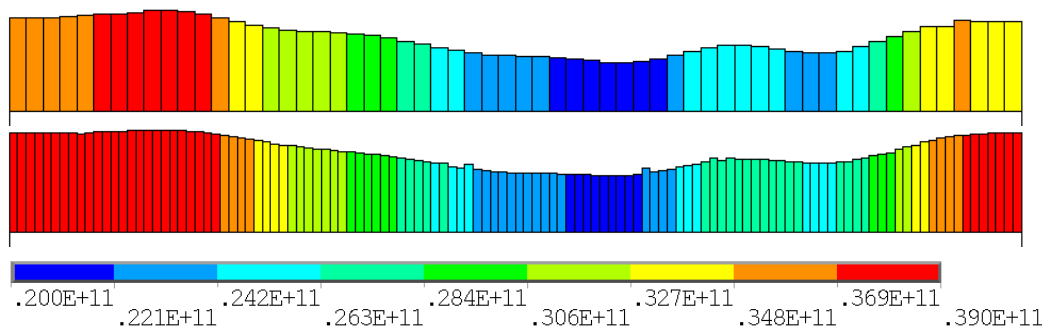


Figure 12. Elastic modulus distribution [N/m²] for the Beam model with element size 0.1 and 0.05 m.

4 CONCLUSIONS

In this contribution, the application of Finite Element Model Updating to the damage detection on beam structures is presented. The updating procedures are implemented in VCUPDATE, a Scilab code interfaced with the Finite Element codes OpenSees and ANSYS. The code is here applied to damage detection on beam structures.

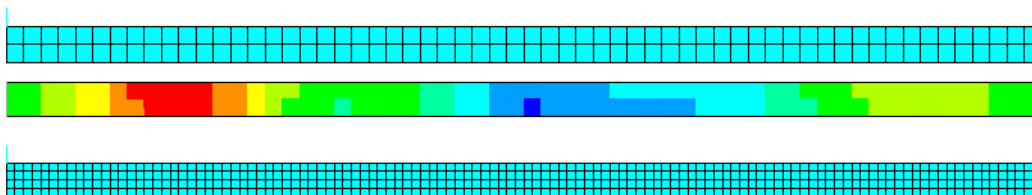
At first, a simply supported beam with simulated damage is investigated. This means that the experimental eigendata are numerically generated by OpenSees. By using frequencies and mode shapes, it is possible to effectively locate and quantify the damage with high accuracy.

Subsequently, the real case of a reinforced concrete beam is examined. This structure was artificial-

ly damaged in a laboratory test inducing an asymmetrical damage distribution. By combining frequencies and mode shapes, it is possible to accurately locate and quantify the damage. When possible, the results were compared with outcomes from other studies.

In this optic, it must be underlined that, even if VCUPDATE is based on simple theoretical basics, it provides very good results even when compared with other codes based on more advanced theoretical formulation.

In all the applications, the results provided by VCUPDATE are very satisfactory obtaining a very high accuracy in the modal behavior description as well as in the damage location. All the reported outcomes are deeply and critically evaluated and discussed.



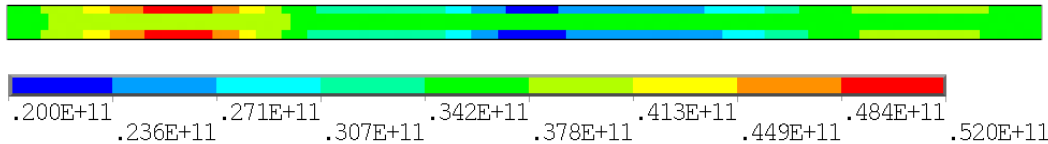


Figure 13. Mesh and elastic modulus distribution [N/m²] for the Plane stress model with element size 0.01 and 0.05 m.

Teughels A, Maeck J, De Roeck G, "Damage assessment by FE model updating using damage functions" *Computers and Structures*, 80 (25), p.1869-1879, Sep 2002.

REFERENCES

- Allemang RJ, Brown DL., "A Correlation Coefficient for Modal Vector Analysis", *Proceedings of the 1st International Modal Analysis Conference (IMAC)*, Orlando, Florida, 1982.
- Dascotte E, Strobbe J, Hua H., "Sensitivity-based model updating using multiple types of simultaneous state variables", *Proceedings of the 13th International Modal Analysis Conference (IMAC)*, Nashville, Tennessee, 1995.
- Dascotte E, Vanhonacker P., "Development of an automatic mathematical model updating program", *Proceedings of the 7th International Modal Analysis Conference (IMAC)*, Las Vegas, Nevada, 1989.
- Friswell MI, Mottershead JE., *Finite Element Model Updating in Structural Dynamics*, Kluwer Academic, Dordrecht, The Netherlands, 1995.
- Jaishi B, Ren W., "Structural Finite Element Model Updating Using Ambient Vibration Test Results", *ASCE Journal of Structural Engineering*, 4, 2005.
- Maeck J., *Damage Assessment of Civil Engineering Structures by Vibration Monitoring*, Ph.D. dissertation, Katholieke Universiteit Leuven, 2003.
- Maya NMM, Silva JMM (Eds), *Theoretical and experimental modal analysis*, Research Studies Press Ltd, Taunton, England, 1997.
- Mordini A, Savov K, Wenzel H, "The Finite Element Model Updating: a powerful tool for Structural Health Monitoring", *Structural Engineering International - IABSE*, Volume 17, Number 4, 2007.
- Mordini A, Savov K, Wenzel H., "Damage detection on stay cables using an open source-based framework for Finite Element Model Updating", *Structural Health Monitoring*, Jun 2008; vol. 7: pp. 91 - 102.
- Mordini A, Wenzel H., "VCUPDATE, a numerical tool for Finite Element Model Updating", *Proceedings of the 2nd International Operational Modal Analysis Conference IOMAC*, Copenhagen 2007.
- PEER Pacific Earthquake Engineering Research Center, *OpenSees, Open System for Earthquake Engineering Simulation*, version 1.7.3, 2006, <http://opensees.berkeley.edu>
- SAS IP Inc. ANSYS, version 10.0, 2005, <http://www.ansys.com>
- Scilab Consortium, *Scilab, A Free Scientific Software Package*, version 4.1, 2006, <http://www.scilab.org>
- Teughels A., *Inverse modeling of civil engineering structures based on operational modal data*, Ph.D. dissertation, Katholieke Universiteit Leuven, 2003.
- Teughels A, De Roeck G, Suykens J.A.K, "Global optimization by coupled local minimizers and its application to FE model updating" *Computers and Structures*, 81 (24), p.2337-2351, Sep 2003.