Determination of Joint Stiffness of a Three Story Steel Frame by Finite Element Model Updating

M. Mahdavi, M.R. Davoodi & A. Mostafavian
Babol University of Technology, Iran

SUMMARY:
Due to some experimental and analytical uncertainties, there are differences between the results of the finite element models and experimental outcomes. To address this issue, finite element model updating methods are usually used. In this article, the structure was a three story steel frame with bolted flange joints and one bay in each direction. Tests and results of analyses showed that the results are different and the model had to be updated. In this specific structure, joint stiffness was one of the sources of uncertainty. So, in the updating procedure this factor had to be used as the variable. First of all, an impact modal testing was utilized. The test was performed with an impact hammer, a spectrum analyzer and an accelometer. Furthermore, model updating method was used to update the finite element model of the structure with the experimental results, and finally by this procedure the stiffness of the joints was determined and the model was updated.

Keywords: Flange joint, Model updating, Experimental modal analysis, Natural frequencies, Genetic algorithm

1. INTRODUCTION

Many steel structures are built around the world. All of them are analyzed and designed to meet regulations. But in most of the cases real behavior of the structure is different from the analytical model. The difference in behavior originates from the difference between characteristics of the analytical model and the actual structure elements. These differences which are called uncertainties of the model can be both mechanical and geometrical. The structure of this study is built with bolted flange joints and the sophisticated semi-rigid behavior of these kinds of joints can be one of the sources of uncertainty in any structure including one under investigation. So in this article and based on previous studies, joint stiffness was assumed to be the main origin of uncertainty of the structure. Furthermore, to determine the behavior of the bolted flange joints of this specific structure, the method of inverse problem was utilized. In this method, by updating the analytical model which means making convergence between the analytical and experimental outcomes, the actual characteristics of the joints was determined.

To update a finite element model, it is essential that certain responses of the structure are selected and the difference between these responses in experimental and analytical states is minimized through tuning of certain parameters in the model. Modal properties of the structure can be used for this purpose. The most common modal properties which are usually used to update the finite element models are natural frequencies and mode shapes. Wu and Li [1] used a two-stage finite element model updating method to update and validate the analytical model which was used in damage detection procedure. Same as the current work, joints stiffness and modulus of elasticity was selected to be the variable of the optimization procedure. In the mentioned article, five natural frequencies and the corresponding mode shapes were used to update the finite element model of the IASC-ASCE structural health monitoring benchmark steel frame. Chellini [2] et al. used vibration frequencies of the first four modes and relative mode shapes for updating a finite element model of a high ductile steel-concrete composite frame and determined connections and bracing stiffness, to perform damage assessment. Dilena et al. [3] used few first experimental natural frequencies to update the model of single span bridge to calculate the boundary conditions and estimate the crack effect on the structure. Mojtahedi et al. [4] used finite element model updating method which was based on first four modes to predict variations in dynamic properties of an experimental model of an offshore jacket platform to determine brace and support stiffness.

During the finite element model updating procedure, it is common to pair the modes by the use of mode shape data [5]. But it is not always that easy to gain good data for the experimental mode shapes. For this purpose the experimental and analytical DOFs should be equated and data should have a good quality of measurement. Since in this specific structure, there are many degrees of freedom and in some parts it is impossible to measure the responses, then the best way would be model updating strategies that directly don't need mode shapes to be performed.

In the present study, to determine the behavior of the structure under actual conditions, the method of finite element model updating has been used. Natural frequencies of the grid were considered the only responses used in the updating process. Then, modal testing was performed and the natural frequencies were obtained via experimental modal analysis. Furthermore, the finite element model of the frame was updated by minimizing the difference between experimental and analytical natural frequencies and as a result the stiffness of the bolted flange joints was obtained.

2. THREE STOREY STEEL FRAME:

In this article, the structure under investigation is a three-story steel frame with one bay in each direction. One side of structure is single braced and the roofs are x-braced. Fig1. is an overall view of the frame. Sections used in the model are exactly the same of the actual structure.

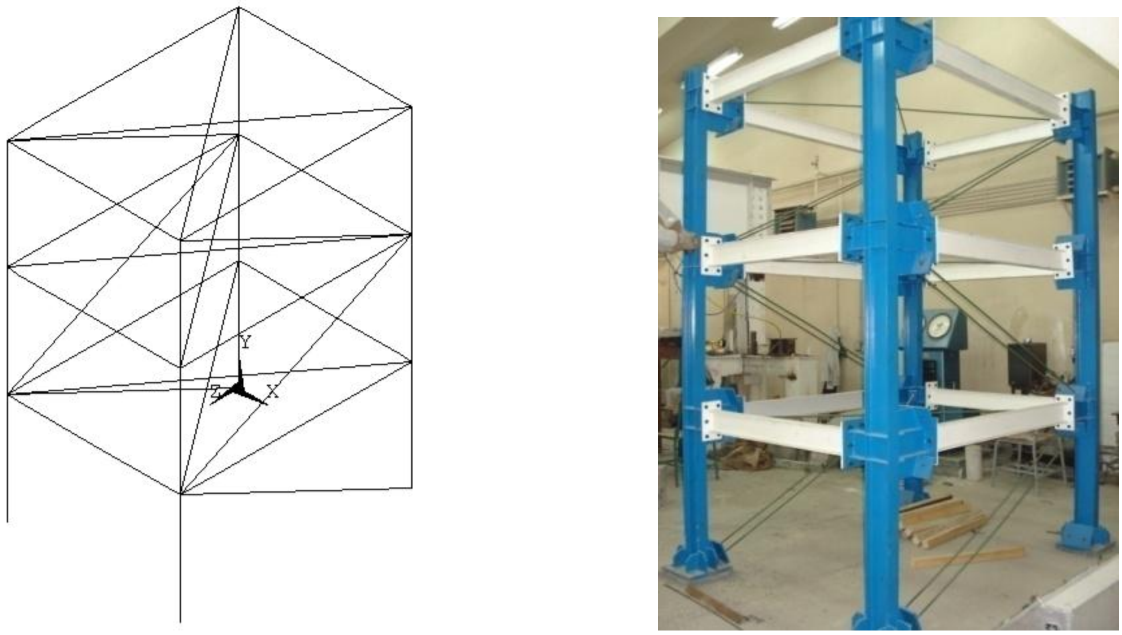
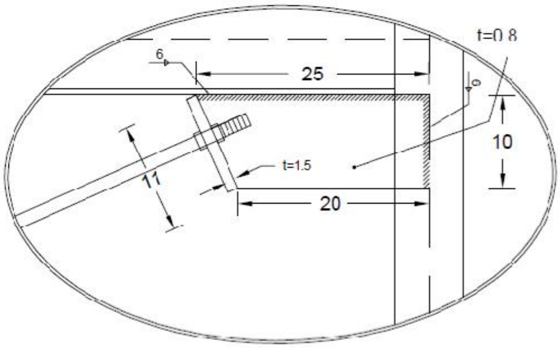


Figure1. Three-story steel frame (a) FEM model (b) actual structure

Lateral and roof braces as shown in fig2.a & b are steel rod with the diameter of 10 mm. The braces are fixed at their place with nuts on both sides of the brace plate. The braces connection plates to the structure were designed as shown in fig2.b. The plates are 11 × 20 cm with the thickness of 15 mm and the connection plate is a trapezoidal with the dimensions shown in fig2.c.

(a)

(b)



(c)

Figure2. (a) Roof braces (b) side braces and connection plates (c) connection plates detail

Beams are IPE120 and the columns are made of IPE140 sections, see fig.3.a. Each story has a 0.95 m height and the columns are fixed to the concrete slab of the laboratory with bolts and base plate system as shown in figure3.b.

(a)

(b)

Figure3. (a) Beams and columns (b) bolts and base plates

Finally the structure parts were connected together with bolted flange joints. The connecting parts were designed based on the Iran regulations for steel structures design. And as a result, the flanges of the joints are 13×22×1.5cm plates with 6 bolt holes and connectors are fully threaded Φ12 bolts (bolts diameter are 12mm). Due to the IPE140 flange size limitations, the joints are not exactly placed at the beam to column connections. In the braced side the flanged joints are located at the distance of 30cm of the beam to column connection zone, see fig4, and on the other side this distance is 10cm.

Figure4. Bolted flange joints

After all the members of the frame have been assembled, the bolts at each joint were tightened in a series of steps, so that the nonlinear effects of the joint are reduced. If the bolts are not tight enough, the nonlinear effects will be considerable even in low excitation levels and this will lead to difficulties in measurements and experimental modal analysis.

A bolted flange joint has a complex behavior and this would cause significant differences between experimental an analytical results. Since, there are many uncertainties in a single connection and there are differences between whole connections in the structure e.g. gaps, level of tightness of flanges and bolts, the connections should be investigated in an actual structure and a single, isolated bolted flange joint would not result in a accurate outcomes.

3. FINITE ELEMENT MODELLING OF FRAME

The finite element model of the structure which would be used as a base for model updating process should have maximum similarity to the actual structure, in dimensions, materials and mechanical characteristics. For this purpose, a finite element model of the frame was made in ANSYS [6] program and to enhance the initial model, further considerations should be applied.

One of the most important issues that should be modeled carefully is the mass of the members and parts of the structure. As mentioned before, there are some structural elements like beam, column and braces which their mass would be modeled carefully in the modeling procedure. In ANSYS beams and columns were modeled with the Beam4 element type and the braces were modeled with Link8. But there are some other parts like brace and joints plates, stiffeners and column extensions that their mass couldn't be modeled at the first stage. To solve this problem, for each of the mentioned parts a point mass was added to the model. The mass added to the parts are listed in table 1.

Table1. Point mass applied to the FEM

Mass type	Joints	Middle story brace plate	1st story brace plate	3rd story brace plate and column extension
Point mass 10^{-3} kg.s^2/m	2.75	18.00	3.00	11.50

After adding masses, there is another issue that should be carefully treated. As mentioned before, the joints stiffness is not specified and it should be modeled in a way that could be changed during the updating procedure. To apply these conditions to the joints, three springs were modeled at the joints place. The springs were created with Combin14 element type. Two of the springs were placed to make flexural strength about two axes of the beams at the connections and another one makes torsional strength for the joints. Finally, the modeling procedure resulted in a model which is shown in figure 5.

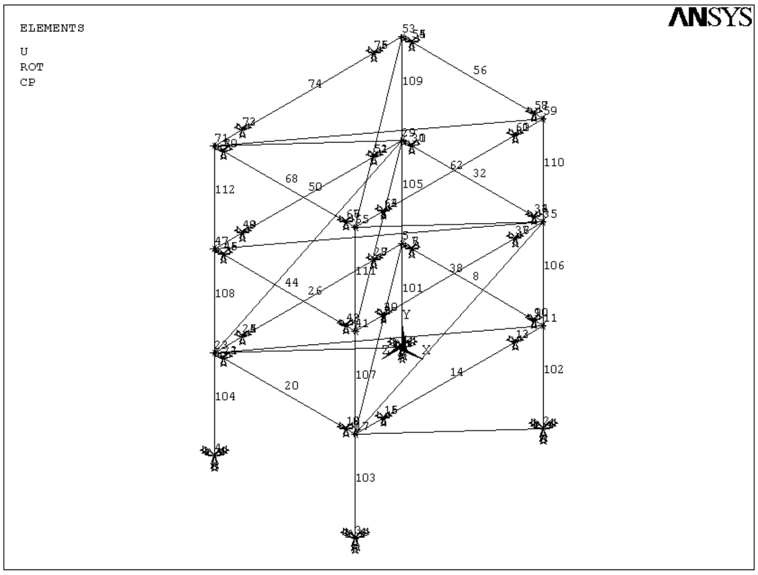


Figure5. Finite element model of the frame

4. MODAL TESTING

Modal testing was utilized to determine the structures modal properties and the resulted FRFs (Frequency Response Functions) were used in the model updating procedure. Among modal testing methods, the most convenient and time saving one which is impact testing for this case, was applied. The equipments which were used in this study are listed in table 2. Excitation was applied through an impulse hammer integrated with a force transducer and the resulting response of the frame was measured through two attached accelerometers in horizontal X and horizontal Y directions. The input signals were recorded by a four channel spectrum analyzer for T=4 s and processed to calculate frequency response functions with a frequency resolution of f =0.025 Hz. The test setup and transducer placements are shown in figure 6. With the setup used in this case, two rows of the FRF matrix will be resulted. As a sample, one of the experimentally measured FRFs is shown in figure 7. Figure 7.a is a point FRF of node 3 in x direction and fig.7.b is the corresponding coherence graph.

Table2. Measurement equipments for modal testing

Spectrum Analyzer	Impulse hammer	Accelerometers
B&K PULSE 3560C (3109+7533 Modules)	AP Tech AU02	DJB A/120/V

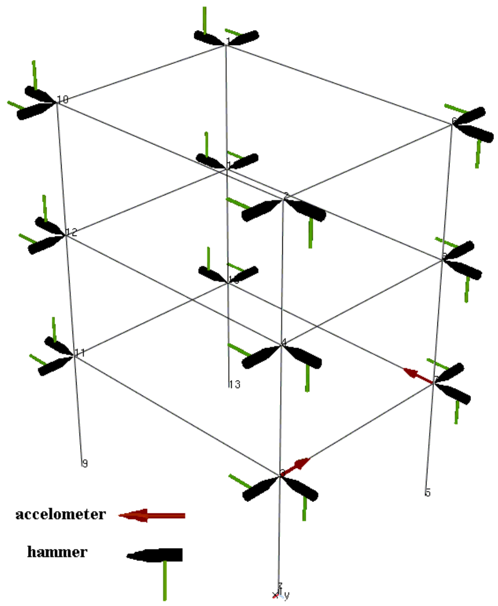
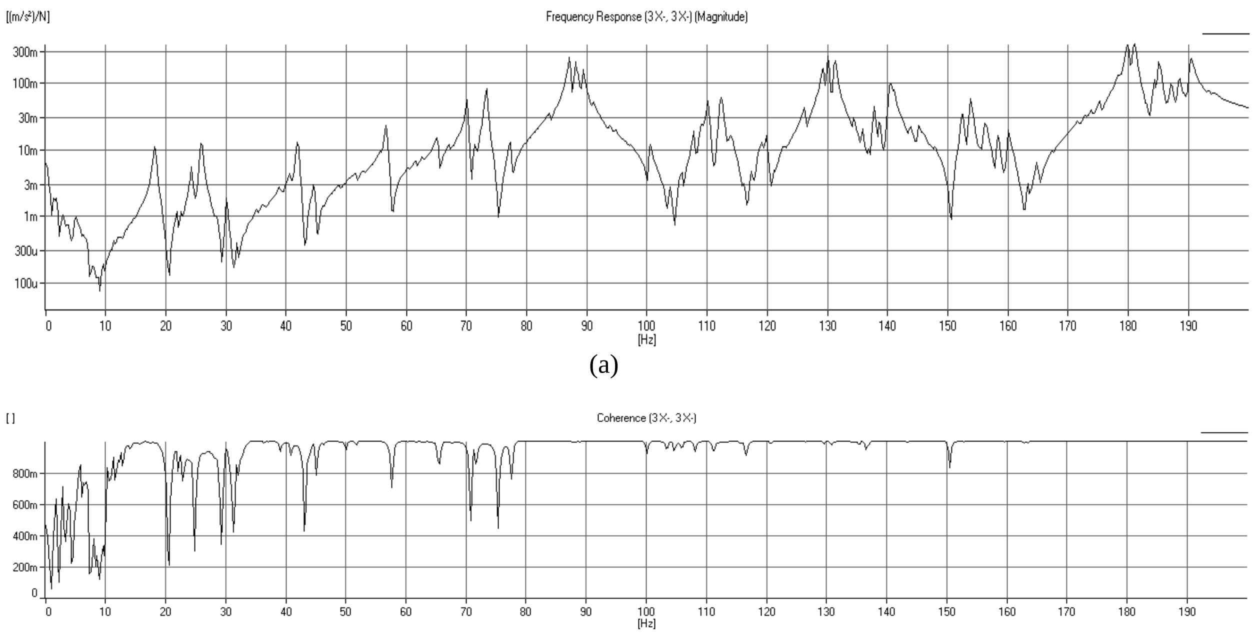


Figure6. Test measurement setup



(a)

(b)

Figure7. Sample FRF and Coherence graphs

5. FINITE ELEMENT MODEL UPDATING

Finite Element Model Updating (FEMU) methods can generally be divided into direct methods, indirect methods and machine learning methods. In direct methods, individual elements of the system matrices stiffness and mass) of FE model are tuned directly to obtain experimental results [7].In indirect methods, which are most popular FEMU procedures, the values of some selected parameters of finite element model are modified to minimize the difference between the analytical and experimental responses of the structure. As the procedure of indirect methods shows, they are based on optimization techniques [8].Machine-learning methods such as neural networks, wavelet analysis and fuzzy theory are another branch of methods in which FEMU is performed through autonomously learning identification of complicated patterns and taking intelligent decisions based on data [9]. The method used in this article is an indirect one which is based on genetic algorithm optimization technique.

5.1. Selection of Responses and Parameters

As mentioned before, in this study natural frequencies are selected to be used for optimization process. For this reason, using 48 obtained FRFs and ICATS-MODENT [10], first ten natural frequencies were determined. The method which was used for the experimental modal analysis is Global-M [11]. It is a multi DOF, multi FRF method. So, the concluded natural frequencies are listed in table 3.

Table3. Frequencies determined by experimental modal analysis, Global-M method

Frequency number	1	2	3	4	5	6	7	8	9	10
Frequency (Hz)	18.1	24.48	25.86	29.97	41.85	55.59	56.45	65.19	70.01	73.2

Furthermore, to make the optimization process more time saving and as it seam unnecessary to use all the ten obtained natural frequencies, six first natural frequencies were selected to be used in optimization process.

On the other hand, to obtain the best answers and make the model updated, some few characteristics of the FEM model should be selected to be tuned during the optimization process. In this article, based on the complex behavior of the bolted flanged joints which is the most influential source of uncertainty in this specific structure, stiffness of the joints about three main axes which are modeled with Combin14 element type in ANSYS, were selected to be used as the optimization parameter. It should be mentioned that in this procedure, it is assumed that the stiffness of all joints are identical.

5.2. Optimization Problem

5.2.1 Objective function and optimization variables

Considering the selected responses for the FEMU, which are the first six natural frequencies of the frame, the objective function includes the difference between experimental and corresponding analytical natural frequencies. One way to express the objective function is the least squares approach which is very efficient and has become the common way to solve updating problems [12]. In the present study, the objective function was defined as:

$$h(P) = \sum_{i=1}^{6}\left(\frac{f_i^{exp}-f_i^{ana}(P)}{f_i^{exp}}\right)^2 \tag{5.1}$$

Where *h(P)* is the objective function to be minimized, f_i^{exp} is the i-th experimental natural frequency of the frame, $f_i^{ana}(P)$ is the i-th analytical natural frequency of the frame and P is the vector of updating parameters. As Eqn. 5.1. shows, the experimental and analytical vibration modes are paired sequentially.

The optimization parameters which are selected in this article are two bending stiffness about x and z-axes and on torsional stiffness about y-axis. In this optimization procedure if we use a large ratio of upper bound and lower bound, then the optimization problem would be difficult to solve. For this reason, this ratio was selected to be 100 based on try and error. After some tries an initial guess of the upper and lower bounds for each of the parameter were selected as follows. For stiffness of joint about its strong axis the upper bound was 2.16×10^{10} kgf/mm^2 and the lower bound was 2.16×10^{8} kgf/mm^2. For the stiffness about the weak axis of the joint the upper bound was 4.42×10^{6} kgf/mm^2 and the lower bound was 4.42×10^{4} kgf/mm^2 and finally for the torsional stiffness the upper bound 1.88×10^{8} was and the lower bound was selected to be 1.88×10^{6}.

5.2.2 Optimization algorithm

To carry out the FEMU for the three-story steel frame and determine the updated parameters values, the objective function in Eqn. 5.1 must be minimized. In this article, GA was used as the optimization method and the reason of this choice is because of the complexities like indeterminacy of explicit mathematical from of the objective function, the necessity of obtaining the global minimum of the objective function and the lack of appropriate initial values for the optimization variables. As a result, an evolutionary optimization method should be utilized and GA is one of the best in this case [13]. Because genetic algorithms are based on the survival-of-the-fittest principle of nature, they try to maximize a function called the fitness function. The commonly used transformation to convert an unconstrained minimization problem to a fitness function is given by [14]:

$$H(P) = \frac{1}{1+h(P)} \tag{5.2}$$

In which *H(P)* is the fitness function and *h(P)* is the objective function. Fitness value approaches 1 if the analytical and experimental natural frequencies of the frame near each other, otherwise it approaches 0.

5.3. Overall Steps of Model Updating

Overall steps followed to carry out the indirect FEMU using GA have been summarized in the flowchart of Figure 8. At first, an initial population of the updating parameters vectors was produced randomly, taking into account the upper and lower bounds of the updating parameters. Population size was considered three hundred times the number of updating parameters. Thus, the population has 900 members, each of which is a three entry vector of the updating parameters P = {MX, MY, TZ}. For each member of the population, analytical natural frequencies were obtained through modal analysis of the finite element model and using experimental natural frequencies, fitness value for each member was evaluated via Eqn.5.2. Since termination criterion is not satisfied at this point, the next step is the evolution cycle in which two new members (two new updating parameters vectors) are produced and replace two members with the least fitness values in the population. The steps of evolution cycle, modal analysis and fitness function evaluation will be repeated until termination criterion is satisfied.

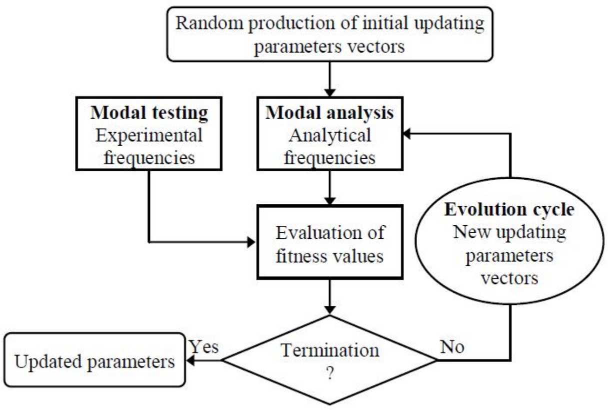


Figure8. Overall steps involved in the FEMU

6. Results And Discussion

In the present work, stiffness of joints in a three-story steel frame is determined by FEMU of the frame by reduction of errors in analytical natural frequencies. For this purpose, a GA optimization computer code was provided in ANSYS with APDL (ANSYS Parametric Design Language). The updating process was run under the same conditions several times and the run giving the highest fitness value was chosen.

Finally, the stiffness parameters of the joints were determined. The stiffness resulted in this FEMU procedure is presented in table 4.

Table4. Stiffness parameters of the frame joints

Stiffness type	Strong axis	Weak axis	Torsional
Stiffness determined (kgf/mm^2)	530552162.966	8003059	231758.132

The determined stiffness was used in the initial model and the frequencies were extracted. The frequencies which are resulted sequentially from both experimental and updated analytical model are listed in table 5 and the error is added in this table too.

Table5. Experimental and updated analytical model natural frequencies

Frequency number	1	2	3	4	5	6
Experimental Frequency (Hz)	18.1	24.48	25.86	29.97	41.85	55.59
Analytical Frequency (Hz)	17.11	27.63	29.80	30.28	30.84	33.35
Error (%)	-5.47	12.87	15.24	1.03	-26.31	-40.01

On the other hand, comparing the corresponding mode shapes of the concluded natural frequencies qualitatively, shows that the only mode shapes which are paired successfully based on updating the model with natural frequencies are the first translational mode in x-axis direction and the first torsional mode shape of the structure which are shown in figure 9. Since these first modes usually influence more than 90% of the structures behavior, the results are acceptable for this stage of the updating.

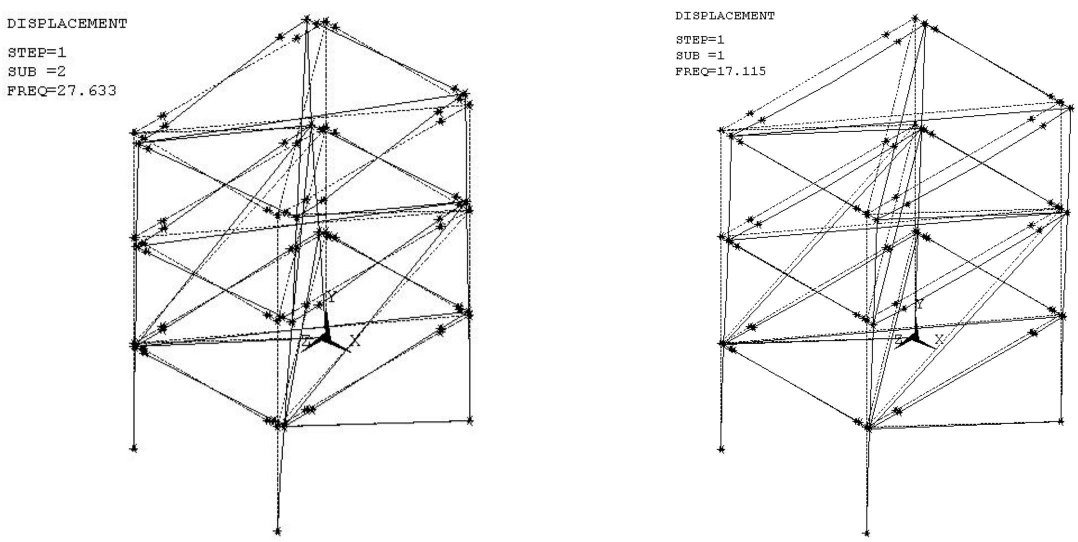


Figure9. Two first paired modes

Finally, as the results show there is not a complete agreement between the updated model and the experimental results and this may be an outcome of poor updating parameters. As previously mentioned, the only parameter which was used to verify the results convergence was the natural frequencies and this was because of insufficient experimental data to determine the corresponding mode shapes. So, if a proper data is available and the natural frequencies are paired in respect of their MACs (modal assurance criterion), which is a criterion to check how much similar two mode shapes are, then there would be better results by the same updating strategy.

REFERENCES

Wu, j.R and Li, Q.S. (2006). Structural Parameter Identification and Damage Detection for a Steel Structure Using a Two Stage Finite Element Model Updating Method. *Journal of Constructional Steel Research*. **62**, 231-239.

Chellini, G. Roeck, G.D., Nardini, L., and Salvatore, W. (2010). Damage analysis of a steel-concrete composite frame by finite element model updating. *Journal of Constructional Steel Research*. **66**:398-411.

Dilena, M., Morassi, A., Perin, M. (2011). Dynamic identification of a reinforced concrete damaged bridge. *Mechanical Systems and Signal Processing.* **25**:2990-3009.

Mojtahedi, A., Lotfollahi Yaghin, M.A., Hassanzadeh, Y., Ettefagh, M.M., Aminfar, M.H. and Aghdam, A.B. (2011) Developing a robust SHM method for offshore jacket platform using model updating and fuzzy logic system. *Applied Ocean Research* **33**:398-411.

Alkhfaji, S.S. and Garvey, S.D. (2011). Modal correlation approaches for general second-order systems: Matching mode pairs and an application to Campbell diagrams. *Journal of Sound and Vibration.* **330**:5615-27.

ANSYS. Version 5.4. SAS IP Inc. www.ansys.com;1997.

Yang, Y.B. and Chen, Y.J. (2009). A new direct method for updating structural models based on measured modal data. *Engineering Structures* **65**:32-42.

Bakhtiari-Nejad, F., Rahaib, A. (2005). and Esfandiarib, A. A structural damage detection method using static noisy data. *Engineering Structures* **27**:1784-93.

Lu, Y. and Tu, Z. (2004). A two-level neural network approach for dynamic FE model updating including damping. *Journal of Sound and Vibration* **275**:931-52.

ICATS. www.icats.co.uk;2002.

Ewins, D.J. (2000). Modal Testing: Theory, Practice and Application, John Wiley & Sons.

Teughels, A. (2003). Inverse modeling of civil engineering structures based on operational modal data, Ph.D. Thesis. Katholieke Universiteit Leuven.

Marvala, T. (2010). Finite-element-model Updating Using Computational Intelligence Techniques, Springer

Rao, S.S. (2009). Engineering Optimization-Theory and Practice, John Wiley & Sons.