

# The use of MSC/NASTRAN sol200 in a test-analysis integration process applied to connection-weld modelling.

*Tom Van Langenhove*<sup>1</sup>,  
*Dennis Turner*<sup>2</sup>

<sup>1</sup> *LMS International, Interleuvenlaan 68, 3001 Leuven, Belgium*

*e-mail : tom.vanlangenhove@lms.be*

<sup>2</sup> *Matra Bae Dynamics, P.O. Box 19, Six Hills Way, Stevenage Hertfordshire SG1 2DA*

## **Abstract**

When building finite element models, a lot of simplifying assumptions and estimates have to be made. Idealisation, discretisation and parameter evaluation are all possible error sources. If these models have to be used in lifetime estimations, optimisation processes or system synthesis computations, they have to be a valid representation of reality. In this paper it is discussed how MSC/NASTRAN sol200 is used in the process of validation and verification of dynamic finite element models.

This paper focuses on a frame support. It shows the different steps taken at both the experimental and the analytical side and describes the complete process based on the 'actual' application problems. The main features of this support frame are the welded joints, the connections between solid and shell elements and the modeling of a solid base by means of shell elements with a certain thickness. Using powerful functionality's such as correlation, sensitivity analysis and physical as well as proportional updating, a well-correlated FE model is obtained and the welded joints and the thickness of the base are identified as 'hot spots'.

# 1. Introduction.

The requirements for higher performance and efficiency force the analysis engineer to validate the quality of his mathematical model by means of experimental data and different correlation tools. Lack of correlation demands the creation of an updated FE model, using experimental data like resonance frequencies, before any meaningful analysis can be done. A modal updating case evolves over several steps as indicated in fig. 1.

The first step aims at the development of a synergetic data platform that contains both test and analysis data in a coherent format. It contains the interface between the FE program and the modal analysis package on one side, and the updating package on the other side. In this paper, the LMS CAE Gateway/ Correlation/ Sensitivity/ Updating software is used for this purpose.

The second step deals with the matching of the analysis nodes and test nodes, the different coordinate systems (local and global), the different name giving in test and FE and the development of a common wireframe representation on which both experimental and analytical data can be visualised.

Once this step is completed successfully, one can start to compare the dynamic behaviour of both test and FE model. Different tools can be used. Some of them are based on modal information (modal deformation shapes and resonance frequencies), others are based on frequency information (frequency response functions).

If sufficient correlation exists, the fine-tuning of the FE model itself can start. If not, there is probably a fundamental error in the analytical model or in the test model. As a rule of thumb, the application of modal updating algorithms becomes relevant when the MAC values between FE-mode shapes and experimental mode shapes approach a level of 85%.

These advanced model tuning tools will bring the experimental and analytical results into agreement by modifying parts of the FE model. The test results are used as targets. This procedure is explained in this paper and illustrated by means of a real case study.

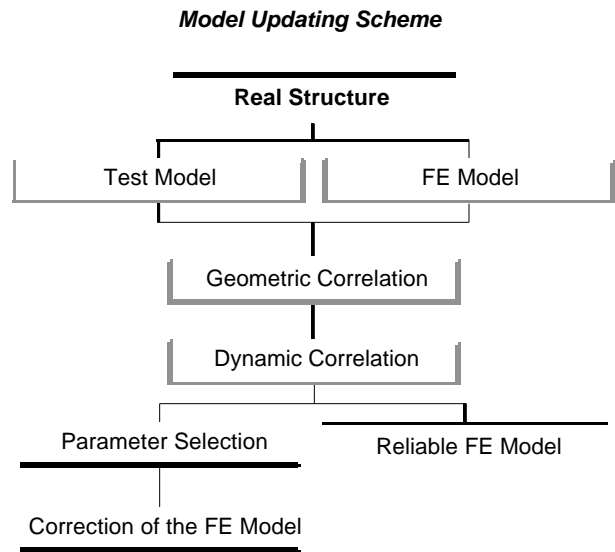


Figure 1 : Model Updating Scheme

# 2. Problem definition.

The model under investigation is a little support frame. The main features are the connections between the solid (top, base) and the shell (triangular part) elements, the welded joints and the modelling of the solid base by means of shells, see Figure 2. Following the modal updating scheme, the correlation between the finite element model and the test model is first investigated and based on the outcome, the finite element model is further fine-tuned.

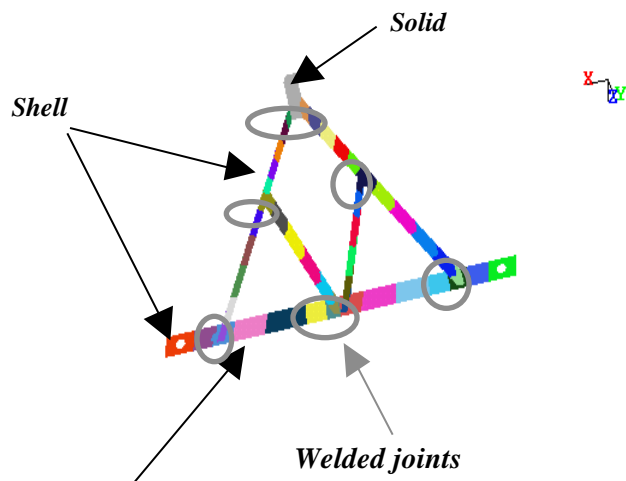


Figure 2 : Frame Model with indication of the main features. The different colours represent different element groups (see § 5.1) or different properties (see § 5.2)

The width and the height of the structure are respectively 82 cm and the 47,4 cm, the average distance between two FE nodes is approximately 0.3 à 0.6 cm.

## 2.1. Modal Analysis.

The frame is dynamically characterised in free-free conditions by means of an experimental modal analysis, as part of standard testing procedures. The test geometry (= wireframe) consists of 17 test points : 1 reference point and 17 response points. Frequency response functions were measured between the response DOFs (2 DOFs/point) and one excitation DOF, yielding in 34 FRFs. The modal analysis resulted in 9 deformation mode shapes in the frequency range 0-900Hz.

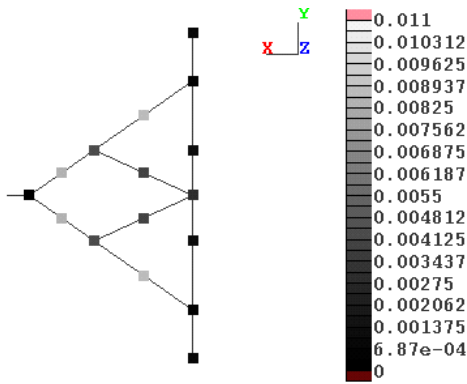


Figure 3 : Test wireframe and correlated nodes. The grey scale gives an indication of the distance (m) between corresponding test/FE nodes.

## 2.2. Finite Element Model.

The structural dynamics FE model is an MSC/NASTRAN model, and the modal parameters (resonance frequencies and mode shapes) were calculated in the frequency range from 0 to 750 Hz, in free-free conditions. Although the structure operates in reality in non-free conditions, a separate, subsequent tuning of structural and 'clamping' parameters simplifies the overall updating effort. In this paper, the focus is on the structural parameter tuning.

The initial FE model contains 3604 nodes and 3330 elements (shells and solids). There are 3 property cards : one solid property card for the top, one shell property card for the base and one shell property card for the remaining part, see also Fig. 2. No special actions were taken to model the joints or the connections between shell and solid elements. It is the intention to check the

validity of these assumptions using different model updating techniques. The FE analysis yields 13 modes (including the 6 rigid body modes) in the selected frequency range.

## 3. Geometrical correlation.

The first step in the correlation consists of the topological matching of the mesh of measurement points with the mesh of finite element nodes. This will solve the incompatibility between both meshes, not only due to the difference in name definition and mesh density, but also due to a possible difference between both global and/or local axis systems.

To align both global axis systems, it is necessary to apply to one of the models a scaling, a rotation about the origin and a translation. Normally, these transformations are applied to the test model, since the FE model is the model to update. This node matching will result in a node pair table containing the corresponding experimental and analytical nodes.

It is also necessary to check that DOFs as well as nodes correlate. Each nodal displacement is represented in its own nodal axis system which implies that even after applying the estimated transformations on the test geometry, the nodal information for two correlated nodes can still not be compared. A test displacement vector, expressed in test nodal co-ordinates will be translated to FE nodal co-ordinates by means of the following actions :

- definition the vector in global test co-ordinates.
- definition the vector in global FE co-ordinates.
- definition the vector in nodal FE co-ordinates.

Mathematically, this is expressed as [4] :

$$\{U^{corr}\} = [Euler^{FE}] [R]^{-1} [Euler^{test}] \{U^{test}\} \quad (1)$$

Now the following situations can occur with respect to the dimension of the test vector  $\{U^{test}\}$  :

- Dimension = 3 : the test vector can be translated correctly into FE nodal coordinates by means of formula (1). The dimension of the translated vector equals then 3.
- Dimension = 1 or 2 : the test vector can only be translated correctly if each measured test direction coincides with an FE local direction. The dimension of the translated vector equals then 1 or 2. If this condition is not fulfilled, the transformation cannot be done due to the matrix incompatibility.

The only way to perform the translation in all other cases is to assume that the unmeasured DOFs equal zero, thus that the displacement in the unmeasured direction is negligible.

When the appropriate node pair table exists, linking the experimental and the analytical nodes, then mode shapes, response functions and wireframe can be transferred between the test and the FE model. Upon completion, one disposes of an experimental and a numerical modal vector set with mode shapes that can be jointly animated on a single, unique wireframe representation of the structure.

### 3.1 Test to FE transformation.

The geometry transformation parameters are estimated based on an interactive selection of three sets of corresponding nodes in both models. This leads to a test node correlation percentage of 94.1%, thus all test nodes do correlate except for the test node at the top, see Fig. 3 and Fig. 4.

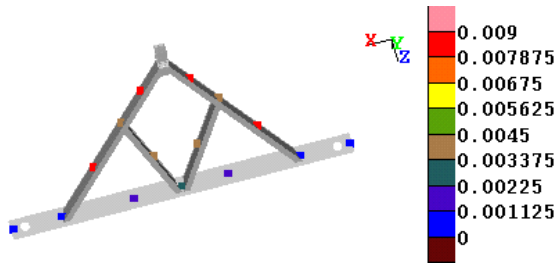


Figure 4 : FE geometry with the correlated nodes on top.

To perform the DOF transformation, we have to set the third, non-measured, direction equal to zero. This will only be correct if the displacement in this non-measured direction is indeed negligible. In all other cases, we do lose some dynamic information.

### 3.2. FE to test transformation.

By projecting the FE model onto the test model, it is possible to transform the DOF information correctly and to investigate thus the importance of the assumption about the displacement of the non-measured direction by means of a dynamic correlation study between the two sets of modal vectors in both cases (Test->FE and FE->Test).

Comparing the MAC-values (see equation (3)) calculated between

- the transformed test modes and the original FE

modes, filtered with a DOF group which contains all correlated nodes (node pair table) and 3 DOFs/node (i.e. translational DOFs).

- the transformed FE modes and the original test modes, filtered with a DOF group which contains all correlated nodes (same node pair table) and 2 DOFs/node.

shows us that in general, the difference is relatively small except for the last two mode pairs, see Table 1. This gives us enough confidence to proceed with the test model projected on the FE model.

EMA f. Hz	FEM f. Hz	Diff Hz	Diff (%)	MAC Fe/Test	MAC Test/Fe
186.41	235.76	49.35	26.47	0.85	0.80
330.04	344.38	14.34	4.34	0.88	0.86
371.51	384.84	13.33	3.59	0.70	0.68
360.58	419.60	59.01	16.37	0.81	0.81
385.95	428.98	43.03	11.15	0.98	0.98
520.91	548.49	27.58	5.30	0.88	0.71
690.98	723.00	32.02	4.63	0.82	0.74

Table 1 : MAC values when projecting FE on test and test on FE

## 4. Dynamic correlation

### 4.1. Modal based correlation.

Now a complete, consistent data set has been set up, an evaluation of the agreement between the FE and experimental analysis can be undertaken using different correlation techniques, such as a visual comparison of the mode shapes on the common wireframe of the structure, a Modal Assurance Criterion (MAC) calculation, a MACCo, etc. [5]. A mode pair table is automatically generated after selection of a threshold MAC value, see Table 1. (threshold MAC = 65%).

Inspection of the MAC-values, see Fig. 5, and the MPT shows us that :

- the deformation modes 3&4 (test) and 9&10 (FE) are switched in frequency.
- 7 mode pairs are found, with relatively high MAC-values, but rather high frequency shifts.
- FE modal frequencies are consistently higher than test modal frequencies, indicating a global stiffness over-estimation or a global under-estimation of the total mass of the FE model.

However the total mass of the finite element model equals physical mass of the structure. The outcome of this dynamic correlation is thus

65 acceptable and the FE model is suitable for updating.

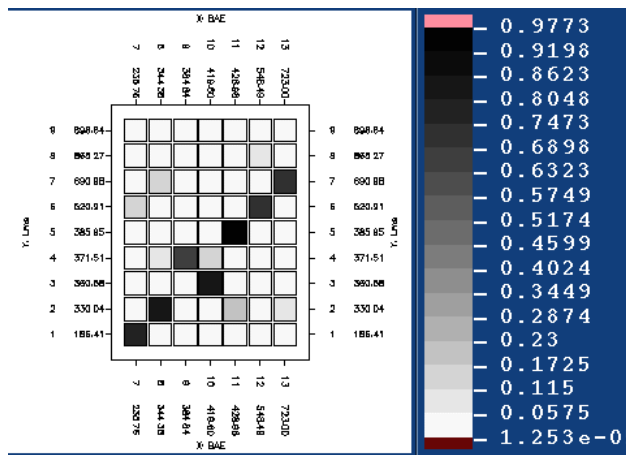


Figure 5 : MAC. The x-axis represents the FE modal frequencies, the y-axis the test ones.

The driving-point FRAC, see Fig. 5, shows us that decreasing the global stiffness would improve our correlation results. We could have made the same conclusion from the mode pair table.

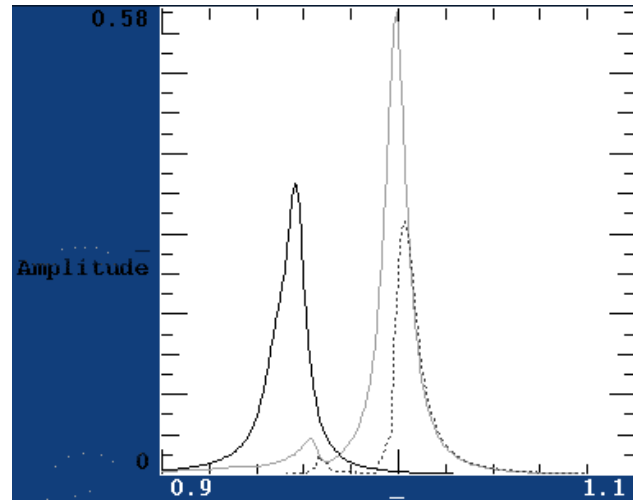


Figure 6 : FRAC for different frequency factors. Original (solid black), After link updating (dashed black), After SOL200 updating (solid grey)

## 4.2. Frequency function based correlation.

This section groups a further set of techniques, such as visual comparison of the FRF functions, FRAC or RVAC calculations, that can be used to evaluate the agreement between the reference (test) and verification (FE) model. The experimental FRF's are the measured ones, the analytical FRF's on the other hand are synthesised within the LMS CAE/Gateway software using modal superposition. To have analytical FRF's with a reasonable amount of damping, the damping ratio's of the experimental model can be used using the generated mode pair table.

The Frequency Response Assurance Criterion (FRAC) expresses the comparison between experimental and analytical functions as a single number between zero (no correlation) and one. Frequency response functions with similar appearance can however show low FRAC values. It can be shown that a global under or over estimate of the stiffness or mass of an FE structure, can give rise to a global frequency shift, resulting in poor FRAC values even when the only difference appears to be a frequency shift. If a stiffness factor of  $k$  is applied to the structure, then this will result in a frequency shift of  $\sqrt{k}$  and an amplitude shift of  $1/\sqrt{k}$ . By defining a frequency factor  $f = \sqrt{k}$ , and computing the FRAC values for a range of values of  $f$ , a maximum value in this range can be determined. This value indicates the frequency shift required and hence the global stiffness factor that needs to be applied in order to improve the correlation between the analytical and experimental frequency response functions.

## 5. Correction of the FE model.

In a model updating project, three of the preparatory phases are the correlation study, the selection of the updating parameters and the updating targets (in this case the test modal frequencies).

The selection of the FE model parameters which are subject to change, is usually based on the interpretation of the correlation results. It is important that these parameters do have sufficient influence on the FE modal frequencies. This can be investigated by means of a 'sensitivity analysis' (sensitivity of the modal frequencies with respect to parameter changes). Apart from the visualisation of critical areas, the sensitivity results are used as direct input for the optimisation process. In fact, the sensitivity coefficients are an essential part of the mathematical formulation of the error minimisation function. The correction step will try to minimise the difference between test and FE modal frequencies.

The kind of updating parameters is dependant on the actual approach used to update the finite element model based on the experimental results. In this paper, two approaches are used [1],[3] :

- The first approach is a one-step, non-iterative approach in which parameters are corrected by solving a system of sensitivity equations. This updating approach will be referred to as the 'Link Solution'.

- The second approach is an iterative method based on a sensitivity formulation. The definition of the updating problem will be transformed into an optimisation problem in MSC/NASTRAN. This updating approach will be referred as the ‘MSC/NASTRAN Sol200 Solver’.

### 5.1. Link Solver Updating.

Based on a simple straightforward spatial subdivision of the FE model in element groups, see Fig. 2, the ‘proportional’ parameters  $\{p\}$  which are subject to change are defined by assigning a ‘proportional’ stiffness parameter to each group of elements. The parameters are proportional to the mass and stiffness matrices (such as Young’s modulus and mass density). The solution of the updating problem is then found in a single step by the (constrained) solution of the following set of equations:

$$[S]\{p\} = \{r\} \tag{2}$$

This direct solution scheme is fully independent of the FE code. No communication between the FE code and the updating (and sensitivity) code is required when using this approach. Once the ‘optimal’ solution has been reached, the MSC/NASTRAN input deck is corrected with the new parameter values and a new ‘normal modes’ analysis is performed to obtain the corrected FE model results.

The residues  $\{r\}$  are estimators of the ‘distance’ between the FE model and the experimental model, and are in this case the differences between the FE resonance frequencies and the corresponding experimental resonance frequencies (which are the targets). The matrix  $[S]$  is a sensitivity matrix, and contains the sensitivities of all residues to all parameters (proportional sensitivity results). This matrix is the outcome of a proportional sensitivity analysis. This type of sensitivity analysis can be performed without any knowledge of the element definitions, and is therefore quick and easy to use. The main observations of the proportional stiffness sensitivity analysis are :

- All sensitivity coefficients are positive.
- The groups around the welded joints and the groups of the base are the most sensitive.
- Sensitivity values increase as the modal frequency increases.

Two cases were investigated :

- In the first case,  $\{r\}$  contains the full mode pair table, see Table 1, and no constraints are applied to the updating parameters. All parameters have the same ‘weight’ in the cost functions. Using this set-up, we can check if the Link Solver identifies the welded joints and the base as ‘hot spots’.

The updated values for the proportional stiffness parameters can be visualised on top of the geometry, see Fig. 7.

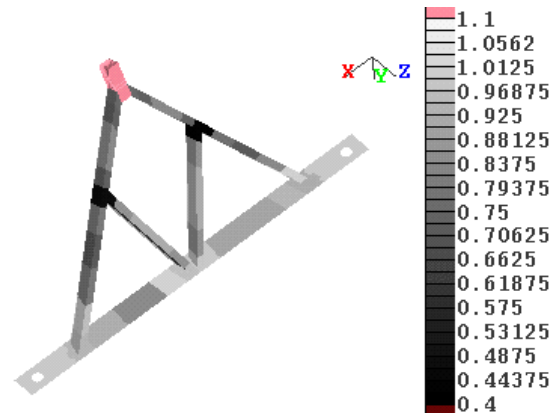


Figure 7 : Parameter changes after ‘Link updating’. The default values for all parameters before updating were one.

One will notice that the values of all proportional stiffness parameters do decrease and that especially the groups around the welded joints are identified as ‘hot spots’. The proportional stiffness change of the elements of the base however was minor.

Applying these updated parameters to the FE model results in an ‘updated’ FE model valid in a frequency bandwidth between 0-750Hz. The outcome of a new correlation analysis between the original test model and the corrected FE model is given in Table 2.

EMA f. Hz	FEM f Hz	Diff Hz	Diff (%)	MAC before Test/FE	MAC after Test/FE
186.41	212.26	25.85	13.87	0.80	0.80
330.04	329.01	-1.03	-0.31	0.86	0.94
360.58	356.31	-4.28	-1.19	0.81	0.87
371.51	367.59	-3.92	-1.05	0.68	0.71
385.95	384.14	-1.82	-0.47	0.98	0.97
520.91	518.97	-1.93	-0.37	0.71	0.74
690.98	704.35	13.37	1.94	0.74	0.78

Table 2 : Mode pair table after ‘Link updating’.

The frequency mismatch is successfully tackled with exception of the first mode and the new MAC values are satisfactory. The proportional stiffness updating as used in the Link Solver is not

able to fully correct the FE results and is thus not able to provide a model which is valid in the bandwidth 0-750Hz. However, the mode switch has disappeared and the FRAC peaks at a frequency factor value of almost one, see Fig. 6.

- In the second case, { } contains only the first mode pair and no constraints are applied to the updating parameters.

The updated values for the proportional stiffness parameters are shown in Fig. 8. The groups around the welded joints AND the groups of the base are now identified as ‘hot spots’.

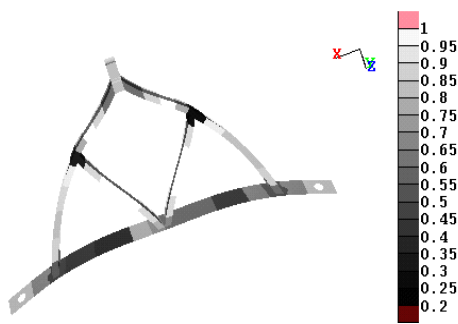


Figure 8 : Parameter changes after 'Link updating'.

The difference between the first mode of the corrected FE model and the first test frequency is now only 8Hz with a MAC value of 0.80. However, this corrected FE model is only valid in the frequency range around the first FE modal frequency.

The drawback of this method however is that the results are not physically relevant nor interpretable since the individual stiffness matrix elements can alter without any relation to the physical model from which they originate.

## 5.2. MSC/NASTRAN Sol200 updating.

The updating parameters in this case are physical variables. These parameters are directly related to shell properties (shell thickness as design variable), beam properties (cross section as design variable) or other FE property cards. This approach offers thus more of an insight into what modelling errors have been made rather than just their location.

Since the original input deck contained only 2 shell property cards and thus only two design parameters, a very low spatial distribution for updating is available. Therefore, the input deck was modified to obtain more property cards, see Fig. 2, this of course without changing the dynamic behaviour of the structure. For each of

these property cards, the shell thickness is used as design variable. The possible evolution of the numerical value of these variables has been constrained between 50% and 150% of the initial value. The residues are in this case the differences between analytical and experimental modal frequencies and the full mode pair table is taken into account.

The optimisation problem is translated into an MSC/NASTRAN SOL200 optimisation problem, see equation 3. The objective function is solved in an iterative way using a Bayesian parameter estimation technique.

$$W_r \|\{ \} - [S] \{ dp \} \|^2 \quad (3)$$

The outcome of this optimisation analysis contains the parameter evolution, the updated thickness values and the modes of the ‘corrected’ finite element model. The ‘optimal’ solution using this approach is reached by increasing the thickness of the shell elements on the base and decreasing the thickness of the elements around the welded joints, see Fig. 9.

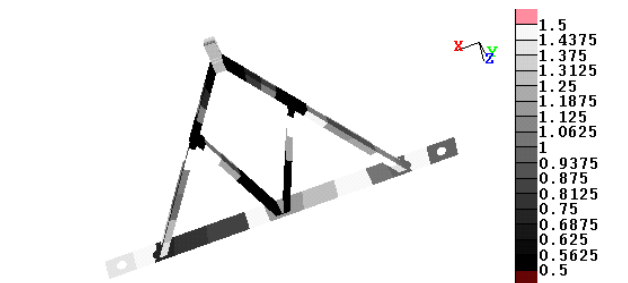


Figure 9 : Parameter changes after 'Sol200' updating.

This result points directly to poor weld penetration and to the modelling of the solid base by means of shells. This is a valuable insight into not only where the modelling error is but also what the error is. This new FE model is valid in the frequency bandwidth between 0-750Hz.

A new correlation analysis results in the following mode pair table, see Table 3. The mode switch disappeared, and the frequency differences are minimised, including the first pair of corresponding modes. The MAC values are somewhat lower than in the case of the ‘Link Solver’.

EMA f. Hz	FEM f Hz	Diff Hz	Diff (%)	MAC before Test/Fe	MAC after Test/Fe
186.41	188.13	1.72	0.92	0.80	0.80
330.04	329.85	-0.19	-0.06	0.86	0.82
360.58	359.15	-1.43	-0.40	0.81	0.88
371.51	367.94	-3.57	-0.96	0.68	0.71
385.95	388.26	2.31	0.60	0.98	0.95

520.91	518.98	-1.92	-0.37	0.71	0.62
690.98	692.42	1.44	0.21	0.74	0.58

Table 3 : Mode pair table after 'Sol200 updating'.

## 6. Conclusions.

The updating process was illustrated in this paper on a small frame structure. The outcome of the initial dynamic correlation was sufficiently high to proceed with an automatic updating procedure. Although we already knew in advance the possible problem zones of this support frame, the updating was based on a straightforward spatial subdivision of the model. Two approaches were used, namely the Link Solver and the MSC/NASTRAN SOL200 optimizer. Both solvers achieved to tackle the mode switch and the frequency differences between the analytical and the experimental resonance frequencies with success, and the thickness of the base and the welded joints were identified as 'hot spots'. *The Sol200 approach has the advantage that it yields physically relevant and interpretable results and thus offers more of an insight into not only where the modeling error is but also what the error is.* Using this information, the engineer can choose whether he uses this information to re-model (in this case) the welding or to proceed with this 'updated' FE model.

A second step in the analysis could be to launch an optimization wherein only the shell thickness of the elements around the welded joints (remember the welding is NOT modeled in this case) is subject to change, in order to include the effect of the welding in the FE model. *In this case, Sol200 updating is rather used to tackle specific modeling (design) issues on a detailed level of the structure.*

The fact that 'updating' in this case is used on a relatively small structure does not exclude its usefulness for larger structures, since the modeling issues are concentrated mainly on detailed level.

## 7. References.

### MSC PRODUCTS

(3) *MSC/NASTRAN Design Sensitivity and Optimization V.68*, The MacNeal-Schwendler Corporation, USA, 1994.

(6) *MSC/NASTRAN Basic Dynamic Analysis V.68*, The MacNeal-Schwendler Corporation, USA, 1994.

### LMS PRODUCTS

(4) *LMS CADA-X Link Manual Rev. 3.4.05*, LMS International, Leuven, Belgium, 1997.

### CONFERENCE PAPERS

(1) N. Roy, A. Girard, L. Bugeat and J. Brica, *A survey of finite element model updating methods*, Proceedings of the international Symposium on Environmental Testing for Space Programs - Test Facilities and Methods, ESTEC Noordwijk, June 1990.

(2) M. Brughmans, F. Lembregts, *Using experimental normal modes for analytical model updating*, proceedings of the eight international modal analysis conference, Kissimmee, Florida, 1990.

(5) M. Brughmans, J. Leuridan, K. Blauwkamp, *The application of FEM-EMA correlation and validation techniques on a body-in-white*, Proceedings of the International Conference on structural Dynamics Modeling, Test, Analysis & Correlation, UK, pp 269-282, 1993.