LINKING FINITE ELEMENT ANALYSIS AND TEST: CASE STUDY IN AUTOMOTIVE INDUSTRY

Eddy Dascotte
Dynamic Engineering
Ambachtenlaan 21
3001 Heverlee (Leuven)
Belgium

ABSTRACT

The accuracy of vibration and noise analysis of vehicle parts like a cylinder block or gear box is mainly determined by the modal parameters that are used. The validation of finite element models, used to compute these modal parameters, requires linking and comparison with experimental modal analysis results. The application of sensitivity analysis and model updating techniques allows the analyst to improve the quality of finite element models.

A integrated software package for performing these tasks is discussed in this paper, putting emphasis on key features and problem areas. A number of typical applications from automotive industry are presented as well as a case study of a cylinder block model in the presence of test data that show unsatisfactory agreement with the results from analytical modal analysis.

1. INTRODUCTION

Over the years, the finite element (FE) method has matured and many excellent analysis packages are commercially available today. More and more companies have been able to take advantage of the considerable benefits available to FE users. However, applying FE techniques requires careful preparation and a critical examination of results. The advantages and disadvantages of the application of the method in structural dynamics are known to their users.

The dynamic response of the model often differs substantially from that of the real structure. This can be due to errors in estimated physical element parameters or boundary conditions, but also because the finite element discretisation does not approximate the real world situation well enough. The stiffness and mass modeling can be inadequate because of the use of an insufficient number of elements or because of the underlying element formulations.

Because of the many assumptions, simplifications and the limited number of degrees of freedom that are used to predict the structural behavior, it is accepted that the results of FE analysis will be only valid in a given frequency domain.

Techniques have been developed to validate the finite element solution and to automatically improve the quality of the model. Validation involves the calculation of a correlation criterion [1] to identify the similarities and differences between the analytical modal analysis results and a set of corresponding experimental data. Model updating methods are mostly based on a sensitivity formulation. These methods require the computation of a sensitivity matrix [S] by considering the partial derivatives of modal parameters and other reference responses with respect to structural parameters via a truncated Taylor's expansion. The resulting matrix equation is of the form

$$\{\Delta q\} = [S]\,\{\Delta p\} \qquad (1)$$

where the elements of $\{\Delta p\}$ are the unknown adjustments in structural elements that are required to produce the difference $\{\Delta q\}$ between the reference response vector and the actual system responses. Sensitivity-based methods are most popular because of their ability to reproduce the correct measured natural frequencies, mode shapes and any other reference response like total mass. Although the basic formulation looks simple, the practical application can be tedious and there are many problems to solve underway.

The nature of the method is such that if one analyses equation (1), it will be clear that in most practical applications, this system of equations will be highly under-determined. In an initial model updating phase, each element of the vector $\{\Delta p\}$ represents a correction to a possible modeling error or adjustment because of inadequate element performance. There are, however, many possible parameters in a FE model. Because it would be impractical to include all possible parameters simultaneously, the challenge will be to make distinction between the following error types:

- errors due to the finite element formulation that is used
- errors on the input values of mass or stiffness related physical element properties
- errors due to geometrical simplification (mesh density)

Once the parameters that are in error are localized, it is relatively easy to correct them in the FE model. The correction can be calculated from equation (1) by inversion of the sensitivity matrix [S]. However, this is a rectangular, fully populated matrix. Good results have been obtained with the Bayesian method to compute a gain matrix and iteratively minimize an error function [1]. This method uses statistics in terms of covariance matrices to express confidence in responses and initial parameter estimations. An advantage of the Bayesian method is the user-controllability of the iterative procedure through parameter constraints and confidences. On the other hand, the user needs to decide on the values to enter for these control variables which requires information on the modeling assumptions and on the measurement conditions. The more parameters and reference response that are included, the more of these decisions have to be made. In general, the purpose of the FE model and the analysis will guide the analyst in the selection of parameters and responses.

It must be clear that many decisions have to be made during the operations and access to various data management and analysis tools is required. With this in mind, a specialized software toolbox (SYSTUNE) was developed [1,2,7].

The philosophy behind the toolbox approach is to cover all the different possible situations related to

- purpose of the FE model
- information available on the modeling assumptions
- the number and type of analytical and experimental analysis results that are available

With respect to model updating, the toolbox approach must facilitate the selection of parameters and responses for different kind of analysis results. In figure 1, this is visualized as a number of consecutive model updating runs for mass analysis results, static analysis and dynamic analysis. Force updating can be added to complete the validation of the analytical model, including acting forces.

The analyst should have many dedicated tools at his disposal for operations like error localization, sensitivity analysis, correlation analysis, geometry mapping, mode shape pairing, generalized objective function calculation and minimization, etc., all to be tailored to the specific model characteristics. The development of a single, generally applicable model updating algorithm, allowing a high level of automation and removing engineering skill, is very unlikely. Error localization and model updating is interactive and knowledge- and decision-based. The more knowledge is available, both from the analytical and the experimental side, the easier justified decisions can be made to enhance the performance of the FE model. The supporting software design must serve the use and development of engineering skill. This also includes state-of-the-art graphical user interfaces and visualization tools. As a result, a dedicated engineering workstation, combining CPU power with extended graphical capabilities, is the most suitable platform for this kind of software.

2. APPLICATIONS

The SYSTUNE program has been successfully applied on FE models up to 100,000 DOF and provides interfaces with most industry-standard FEA codes. It is a powerful postprocessor tool that, in the hands of an experienced analyst, can help one to meet design objectives faster. Some applications of this software that were recently completed will be shortly presented hereafter.

A lot of effort is currently put in the design of parts like for example bumpers and turbo compressors, made of reinforced plastics to replace metallic components. To optimize the design of these parts, a reliable FE model is required. The material properties, however, are often only approximately known. If these material properties are selected as the parameters in the updating algorithm, corrected values will be obtained for the initial estimates [3].

For assembled structures, model updating should sometimes be done in several steps: first separate updating of properties of subcomponents and next tuning of stiffness of joints or connections between the subcomponents. An acoustical radiation study showed that a noise problem which arises during powered working conditions of an alternator is due to interaction of magnetic forces with three resonance frequencies of the alternator housing between 1 kHz and 2 kHz. In order to solve the noise problem, structural modifications which shift these resonance frequencies out of the excited frequency domain are necessary [4]. Figure 2 shows the FE model of the hinge mount alternator. It is a complex component that was divided into three substructures which are represented on figure 3: the slip ring end frame on the top, the stator and the drive end on the bottom. The global Young's modulus was adjusted for each of these components in order to better approach the experimental resonance frequencies, measured for each component separately. The cylindrical subcomponents slightly overlap and are bolted together in three locations with axial bolts. In the second analysis phase, the stiffness of these bolts, modeled as springs was adjusted. The resonance frequencies of the assembled structure now served as reference. The updated model was used to examine the influence of different structural modifications.

In [5], the updating of joint stiffness of a car body structure (figure 4) showed that rotational spring stiffness reductions were required to obtain satisfactory correlation. A number of these stiffness even needed to be reduced to zero. From this, it could be concluded that some rigid joints actually had to be modeled as pin joints.

Sometimes the analyst is only interested in results of sensitivity analysis. In the example of a plastic air inlet pipe (figure 5) some thickness variance was observed due to the extrusion of the pipe [6]. With resonance sensitivities for shell thickness perturbation, the effect of this variance could be quantified.

Another interesting application is that of geometrical model reduction. Coarse finite element models can be derived from fine meshes with the same dynamic behavior. This capability generates important time savings in CPU time and disk space when detailed analysis is not required and only limited resources are available. In [6], this application was demonstrated on an oilpan model for acoustic analysis. The coarse FE model (figure 6a) that was used did not include all geometrical details like stiffeners as they were modeled in the fine mesh (figure 6b). Using the shell thickness of all elements as parameters, only a few iterations were required to determine the equivalent thicknesses that were required in order to obtain the same resonance frequencies and mode shapes as obtained with the fine model.

3. CASE STUDY: PASSENGER CAR CYLINDER BLOCK

The finite element model of a passenger car, 4 cylinder-in-line, engine was supplied as an MSC/NASTRAN bulk data file. The model consists of 1785 nodes and 1548 elements (both shell and volume elements) totaling 10548 DOF (figure 7). Since geometrical simplification was necessary to build the FE model, local stiffness and masses could be under- or overestimated. Structural parts like stiffening webs, molding points, certain embossments or cooling circuit channeling were coarsely modeled or omitted. The engine block is made from a single casting except for the 5 bearing caps. Each of the 5 bearing caps is fastened to the block by 2 bolts. The stiffness modeling of the connections probably is overestimated. Doubts also exist on mass distribution. Total mass, however, is correctly modeled. The mesh density was considered satisfactory to represent the first global modes (torsion and bending) and a number of local bearing local modes.

Experimental modal analysis was done on the freely suspended structure. Triaxial sensors were used for data acquisition at 78 locations. The most characteristic global modes are measured between 0-1000Hz. The first global mode shape is a torsion mode at 462.27 Hz, the second is a global bending mode at 743.74 Hz. The accuracy of the test results is considered to be high.

The resonant frequencies and mode shapes obtained from analysis and test are compared by calculating the global Modal Assurance Criterion (MAC). The global MAC uses all modal displacements corresponding with the measurement points and FEM nodes pairs which are identified by spatial correlation. The mode pairs that have the highest MAC values are listed in Table 1. It can be seen that correlation between initial FEM calculations and experimental data is generally bad. By looking at MAC values only, FEM mode shapes 1 and 3 can be safely paired with experimental modes. This is confirmed after visual comparison. If only node/point pairs on the bearings are used for MAC computation, local correlation can be investigated. Columns 5 and 6 of table 1 show the correlated mode pairs and MAC results for this case. Two additional paired bearing modes can be identified.

From table 1 it can be seen that in general the MAC-values are low and thus no good mode shape correlation is possible except for the global modes. The number of calculated modes in the 0-1000 Hz frequency band (10) is much higher than the number of experimental modes (3).

For the correlated global mode shapes (bold), the error on the resonance frequency is too high to be caused by local stiffness and mass modeling errors only. Given the characteristics of the structure (mono-bloc, cast iron), it seems impossible to explain errors of this magnitude by local or global physical parameter corrections (on Young's modulus, mass density or shell thickness) within acceptable bounds (like 30%), even if corrections for geometrical simplifications are required. Another cause needs to be identified.

By default, MSC/NASTRAN does not take into account the normal rotational stiffness of shell elements and suppresses the corresponding equations. Several test runs using alternative element formulations that take into account this rotational stiffness into account by adding a fictive rigidity, showed that this DOF has an important influence on the modal parameters of the structure.

In order to demonstrate this phenomenon, the following experiment was carried out using the SYSTUNE model updating software:

a. The in-plane rotational stiffness is computed as the average value of the other rotational stiffness terms that are on the diagonal of the element stiffness matrix multiplied with a factor f.

b. Eigenfrequencies were computed for different values of f.

The solution using an added in-plane rotational stiffness converges to the MSC/NASTRAN solution for zero value of f (see table 2). If the resonance frequencies and mode shapes corresponding with f = 100 values are compared against the experimental values, the overall correlation does improve. It was therefore decided that further analysis should be carried out on the FEM using these added fictive stiffness for the shell element normal rotational degrees of freedom.

Table 3 shows the correlation analysis results using the altered shell element formulation. Correspondence between the first two analytical modes and the corresponding experimental modes is confirmed. Moreover, three bearing modes can be paired. Visual comparison showed, however, that mode pair 10 can not be retained since the mode shapes do not sufficiently correspond. The relatively high MAC value (73.8) is probably caused by spatial aliasing.

The reference responses that are selected for tuning are the experimental modes 1, 2, 7 and 8. Mode 1 is a global torsion mode, mode 2 a global bending mode, and modes 7 and 8 are local modes of the bearings. Since the structure is analyzed free/free, no boundary conditions have to be investigated. Young's moduli and densities of all elements were selected as local updating variables. This is justified by the type of modeling errors (geometric simplifications) which have both local stiffness and mass effects.

Model updating was done using the selected responses and parameter selections. Mode shape pairing is repeated with each iteration. This is required because it is possible that the mode sequence will change during iteration. The Bayesian parameter estimation technique was applied for all iterations using a total mass constraint and upper and lower bounds for the parameter modifications (30%). Table 4 summarizes the correlation level before and after model updating (5 iterations). The average MAC value has clearly improved. Especially the MAC values for the local bearing modes now show much better results. All remaining errors on resonant frequencies are less than 1% which is within the tolerance margin.

4. CONCLUSIONS

The examples included in this paper have shown various applications of test/analysis integration on a wide range of structures. The software that is used is only the first step of a ongoing integration of methods and utilities in order to provide the structural dynamics analysts with a specialized postprocessor that is a complimentary tool to the general purpose finite element codes and test software on the market.

5. REFERENCES

[1] E. Dascotte, P. Vanhonacker, 'Development of an Automatic Mathematical Model Updating Program', Proceedings of the 7th International Modal Analysis Conference, Las Vegas, Nevada, February 1989.

[2] Dynamic Engineering, 'SYSTUNE V3.1 User's Manual', 1991.

[3] E. Dascotte, 'Material Identification Of Composite Structures From The Combined Use Of Finite Element Analysis And Experimental Modal Analysis', Proceedings of the 10th International Modal Analysis Conference, February 1992, San Diego, California.

[4] P. Vanhonacker, 'A Methodology For Combining Experimental And Mathematical Modal Analysis With Application To An Automotive Component', Proceedings of the 6th International Modal Analysis Conference, February 1988, London, England.

[5] E. Dascotte, 'Application of Sensitivity Analysis and Finite Element Model Tuning On a Car Body', Proceedings of the 15th International Seminar On Modal Analysis and Structural Dynamics, September 1990, Leuven, Belgium,

[6] E. Dascotte, 'Practical Applications of Finite Element Model Tuning Using Experimental Modal Data', Proceedings of the 8th International Modal Analysis Conference, February 1990, Orlando, Florida.

[7] C.F. McCulloch, E. Dascotte, 'A Robust Methodology For Updating Finite Element Models', Proceedings AUTOTECH 91, November 1991, Birmingham.

6. TABLES

Mode #	FEA [Hz]	EMA [Hz]	MAC (global) [%]	EMA [Hz]	MAC (local) [%]
1	**324.7**	**462.2**	**91.8**	**462.2**	**94.2**
2	481.2	1598.9	63.0	1738.5	44.5
3	**546.0**	**743.7**	**71.4**	**743.7**	**95.8**
4	624.3	1143.4	14.9	2215.5	47.2
5	685.7	1057.4	52.3	1231.4	5.9
6	740.5	1093.9	60.6	1964.2	25.1
7	874.1	1016.6	44.5	**1016.6**	**66.3**
8	927.6	988.6	40.9	**988.6**	**60.5**
9	954.0	1407.6	23.4	1057.4	59.2
10	992.1	2007.1	25.6	1143.4	41.8

Table 1.

NASTRAN	f = 0.01	f = 0.1	f = 1	f = 10	f = 100
324.78	328.5	349.9	399.8	450.7	494.8
481.25	484.1	504.3	563.9	613.6	675.4
546.07	546.9	552.8	591.1	654.2	714.4
624.36	624.6	626.1	632.6	690.7	772.8
685.71	687.3	695.9	728.7	781.5	841.6
740.59	741.2	745.2	764.2	804.4	890.9

Table 2

Mode #	FEA [Hz]	EMA [Hz]	MAC (global) [%]	EMA [Hz]	MAC (local) [%]
1	**494.8**	**462.2**	**92.7**	**462.2**	**94.2**
2	**675.4**	**743.7**	**81.1**	**743.7**	**96.0**
3	714.4	1093.9	29.9	2215.5	59.3
4	772.8	1598.9	54.2	1964.2	13.8
5	841.6	1057.4	27.7	1738.5	15.1
6	890.9	-	<10.0	1057.4	52.9
7	946.8	1016.6	42.4	**1016.6**	**61.9**
8	1007.1	988.6	40.9	**988.6**	**54.7**
9	1048.6	1143.4	26.2	1143.4	42.9
10	1170.9	1407.6	39.5	1407.6	73.8

Table 3

EMA [Hz]	FEM (initial) [Hz	Diff. [%]	MAC [%]	FEM (new) [%]	Diff. [%]	MAC [%]
462.2	494.8	7.0	92.7	466.3	0.8	87.4
743.7	675.4	-9.1	96.0	736.3	-0.9	93.5
988.6	946.8	-6.8	61.9	990.1	0.1	88.7
1016.6	1007.1	1.8	54.7	1017.7	0.1	82.9

Table 4.

7. FIGURES

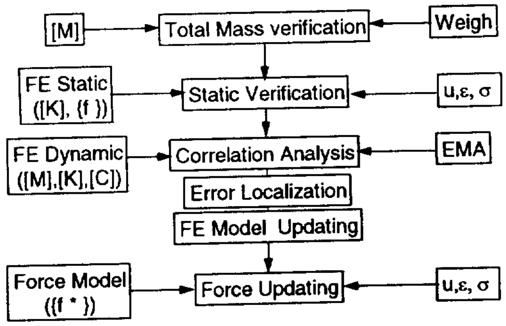

Figure 1. Validation and Model Updating Procedure.

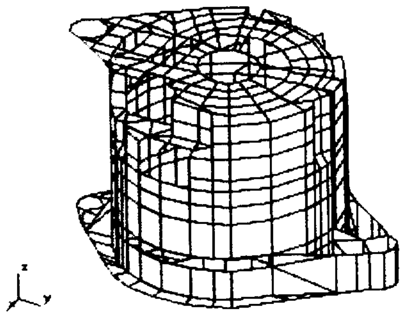

Figure 2. FE Model of a Hinge Mount Alternator.

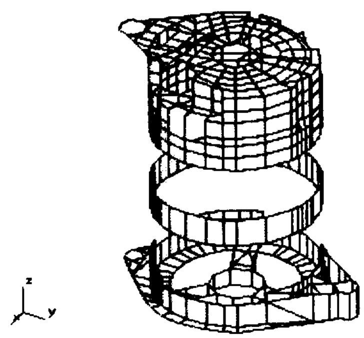

Figure 3. Slip Ring End, Stator and Drive End.

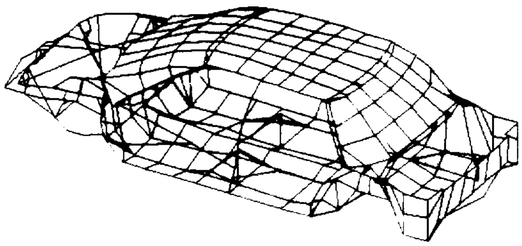

Figure 4. Car Body Structure.

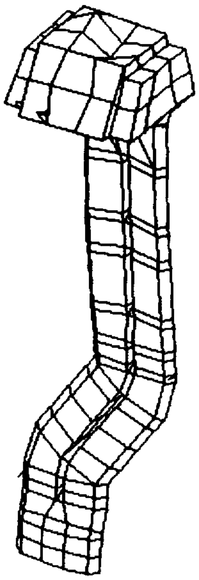

Figure 5. Air Inlet Pipe.

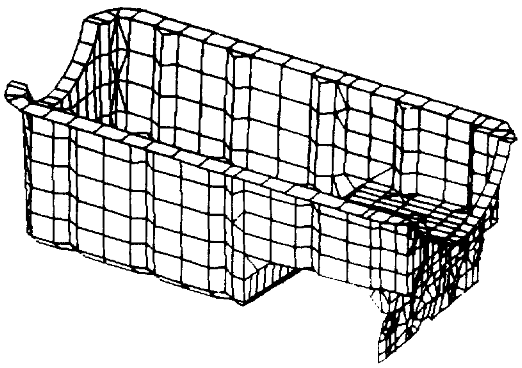

Figure 6a. Coarse FE Model of an Oilpan.

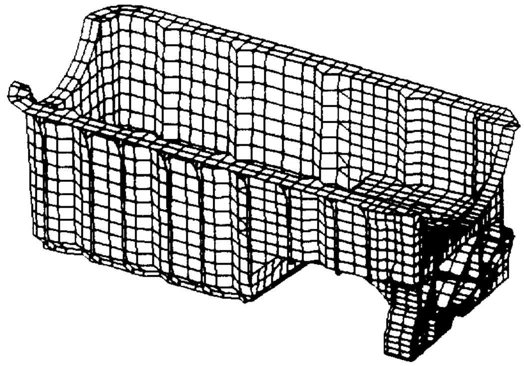

Figure 6b. Fine FE Model of an Oilpan.

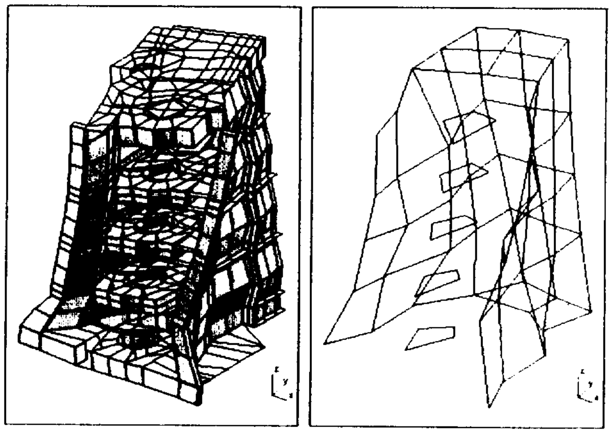

Figure 7. Torsion Mode of Engine Block (FEM and Test).