

# Validation and Updating of ANSYS Finite Element Models Using Experimental Modal Data

Eddy Dascotte

Dynamic Design Solutions (DDS)  
Interleuvenlaan 64, 3001 Leuven, Belgium

## Abstract

A procedure for validation and updating of finite element models by integrating results from experimental modal analysis (EMA) and normal modes analysis in ANSYS is presented.

Using correlation analysis, sensitivity analysis and a Bayesian parameter estimation algorithm, physical element properties like for example material properties, beam cross sectional properties, boundary conditions or concentrated masses are modified in such a way that an objective function, describing the 'error' between the experimental and numerical database, is minimized. The resulting finite element model can be used with more confidence for further dynamic analysis.

The proposed procedure has been implemented in a dedicated postprocessor program (SYSTUNE) which can access ANSYS database files. The concepts of this program are outlined with special emphasis on the integration capabilities with ANSYS.

An application example is included.

## Introduction

The finite element (FE) method has matured over the past two or three decades to a point where design, meshing, analysis and postprocessing are highly integrated and automated. However, in view of the higher complexity, applying FE for structural dynamics requires the user to know the limitations of the methods he is using, and to examine the results critically.

There is a need to assess and understand why the response of the model differs from that of the real structure, as it often does quite substantially. This can be due to a multitude of errors like errors in the physical element parameters (material, thickness, . . .) or boundary conditions, or because the FE discretisation is an inadequate approximation to the real world. The modeling of stiffness and mass can be inaccurate because of an insufficient number of elements, or because of the underlying element formulations.

In figure 1, the proposed validation procedure is illustrated as a sequence of updating runs, in which mass, static behavior and dynamic behavior are corrected. Adding the updating of applied forces completes the validation of the analytical model.

From software development side, three approaches to implement such a validation procedure can be considered:

1. Add diagnostic tools in finite element analysis programs;
2. Add modules to test systems;
3. Stand-alone, dedicated, toolbox.

Each of these approaches has its advantages and disadvantages and is targeted to a specific user group. The first approach has the advantage that extensive use can be made of existing analysis features and that no interfacing of FE data is required. However, the internal database of a FEA program is not specifically designed to accommodate for test data. This approach will be chosen by FE analysts who have an occasional need for correlation and sensitivity analysis and rely on design optimization to update a model to test data. The second approach is more targeted to test engineers with a specific need for pre-test and correlation analysis only and who did not received a special training in finite element analysis.

The alternative of using a dedicated toolbox program that is highly interfaced with analysis and test databases, combines the advantages of the first two other approaches.

## **The SYSTUNE Test - FEA analysis Integration System**

The SYSTUNE program has been developed to be used with ANSYS and most test data formats [1-3]. The main characteristics of this program are:

- o Integration of diagnostics and analysis tools in a single program;
- o Integrated data interfaces with many external FE and test database formats;
- o An interactive command language or graphical user interface especially designed to assist the FEA-oriented user as well as the test engineer. This provides maximum user-friendliness and productivity;
- o Integration of a graphics postprocessor adapted to inspect the database and to visualize analysis results;
- o Possibility to use internal element formulations, sensitivities and solvers for seamless integration with external analysis tools. The program takes control of data interfacing and execution of the external tools so that user interaction is minimized;
- o Stand-alone operation for specific applications (pre-test planning, material identification, quality control, . . .) providing portability and tailoring to specific needs;

- Advantages related to maintenance and continued development independent of the release schedule of the entire FE code or test analysis system.

The methodology that is used was designed to cover all possible situations related to the purpose of the FE model, knowledge of modeling assumptions and the number and type of analytical and experimental results that are available.

The toolbox includes routines for database management (direct interface programs, coordinate system transformations,...), correlation analysis (geometry mapping, modal assurance criterion, eigenvector orthogonality checking, mode shape expansion, model reduction,...), sensitivity analysis and model updating.

Effective methods for model updating are generally based on a sensitivity formulation [4], using a truncated Taylor series expansion. The resulting matrix equation is of the form:

$$\{\Delta R\} = [S]\{\Delta P\} \quad (1)$$

where the elements of  $\{\Delta P\}$  are the unknown adjustments to design variables that are required to produce the changes  $\{\Delta R\}$  between the reference response vector and the actual system responses. Methods based on sensitivity are most popular because of their ability to reproduce the correct measured natural frequencies, mode shapes and any other reference response such as mass.

The sensitivity matrix  $[S]$  contains the gradients of responses with respect to design variables:

$$[S] = S_{ij} = \frac{\partial R_i}{\partial P_j} \quad (2)$$

These sensitivities can be computed for all physical element properties (material properties - isotropic as well as orthotropic -, geometrical properties, boundary conditions, masses and spring stiffness) by using direct derivation or perturbation techniques depending on whether mass and stiffness show a proportional or non-proportional behavior with respect to the property.

From equation (1),  $\{\Delta P\}$  can be computed. The most general estimation procedure is the Bayesian parameter estimation algorithm in which relative confidences in initial estimates of the parameters are taken into account together with confidences in the test data [5]. The Bayesian view originates from statistical parameter estimation methods: the responses, as well as the model parameters, are not considered as deterministic values, but as stochastic variables with a certain probability to have the correct value. The confidences in the different measured test values and the confidences in initial

parameter estimations, can be expressed with weighting matrices for both the response and parameter vectors.

In a Bayesian parameter estimation procedure, the discrepancy between initial model predictions and the test data is resolved by minimizing a weighted error E:

$$E = (\{R\} - \{R^e\})^T [C_R] (\{R\} - \{R^e\}) + (\{P\} - \{P^o\})^T [C_p] (\{P\} - \{P^o\}) \quad (3)$$

with  $\{R\}, \{R^e\}$  respectively the analytical response and test data vectors;  
 $\{P\}, \{P^o\}$  respectively the parameter vectors of the final and original model;  
 $[C_R], [C_p]$  respectively the weighting matrices expressing confidences in test data and model parameters.

This error can be minimized by taking the partial derivative of (3) with respect to  $P_j$  equal to zero. This leads to:

$$\{P\} = \{P^o\} + [G] (\{R^e\} - \{R\}) \quad (4)$$

with  $[G]$  the gain matrix computed as:

$$[G] = [C_p] [S]^T ([C_R'] + [S] [C_p] [S]^T)^{-1} \quad (5)$$

In an iterative procedure (figure 2), an error function is verified to control convergence. A general form of such error function can be:

$$E = \frac{1}{N} \sum_{i=1}^N \frac{|\Delta R_i|}{R_i} \quad (6)$$

or any other user-defined function.

## Interfacing with ANSYS Rev. 5.x

Interfacing between the SYSTUNE internal database and ANSYS 5.x files is done at different levels:

- **FE Model Description.** The complete FE model description is read from a .CDB file that is created in ANSYS with the command CDWRITE.
- **Element Matrices.** The non-assembled element stiffness and mass matrices, defined in the global coordinate system, are required for using SYSTUNE dynamic analysis and sensitivity analysis. They are imported from ANSYS database files with extension .EMAT;
- **Normal Modes.** These are read from a database file with extension .RST and are used for correlation and sensitivity analysis;

Figure 3 shows how these data are used by the different analysis tools in SYSTUNE.

## **Model Updating vs. Design Optimization**

Model updating is typically something that is done before design optimization. An 'error function' in model updating expresses differences between calculated and measured reference data, thus calibrating the design variables, usually referred to as parameters in model tuning, to improve the capability of the finite element model to correctly predict dynamic behavior.

The 'objective' function in design optimization usually expresses the quality of the design in terms of cost, weight, reliability, etc. It is possible to use design optimization as an alternative to Bayesian parameter estimation. Sensitivity-based error function minimization (2), however, as implemented in SYSTUNE, is a more direct and cost-effective approach that is well-suited for the purpose of model updating. This method has been refined to also include parameter constraints (like minimum or maximum allowable values) and parameter linking as is also often used in design optimization. The use of weighting factors to express confidences in parameters and test data is inherent to the Bayesian procedure that is used to minimize the error function.

The key to model updating is e parameter selection. Once this is done correctly, finding the required parameter modifications is relatively easy. In design optimization, the selection of variables is merely based on manufacturing considerations. Selection of variables for model updating, however, requires a good deal of knowledge about the assumptions made in the finite element model and the possible error sources in the analysis. Diagnostics tools are available to guide the engineer and to pinpoint potential modeling deficiencies.

The entire updating cycle has been automated to the point where no user interaction is required to interface between ANSYS and the toolbox so that ANSYS can be used for eigenvalue re-analysis. In fact, SYSTUNE can be seen as a master program that uses ANSYS and possibly other analysis programs as subroutines to perform certain tasks. This leads to a very flexible and "open" system that permits maximum user-controllability and general applicability.

## **Applications**

The SYSTUNE program has been used successfully on FE models of more than 100.000 degrees of freedom. it provides the competent analyst with the sort of "toolbox" which has been described above. Apart from the basic applications (correlation, analysis, sensitivity analysis, model tuning), the program can also be used for:

**MATERIAL PROPERTIES IDENTIFICATION** - There is an increasing desire (mainly with weight reduction as the overall aim) to replace metallic parts with

components made from reinforced plastics, for example for vehicle bumpers, body and mechanical components. The design of these parts can be optimized using FE modeling, but only if the parts can be reliably modeled. Material properties are often not sufficiently well known. If material properties are selected as the parameters in an updating procedure, corrected values are obtained.

**STRUCTURAL ASSEMBLIES** - When a structure is composed of several sub-structures assembled together, updating is often most effective when carried out in several steps. First, a separate updating of the properties of the components should be performed. Then, a tuning of the stiffness of joints or other connections between them can be carried out.

**MODEL MESH REDUCTION** - Coarse FE models can be derived from fine models and forced to adopt the same dynamic behavior, saving computer time and space in further calculations of dynamics and acoustics. An example of such application is demonstrated hereafter.

**CASE STUDY: OIL PAN**

The structure that is used is an oilpan originally modelled using 2308 shell elements (ANSYS element type 63) and 680 volume elements (ANSYS element type 45) with 3252 nodes (Figure 4). This model takes into account all stiffeners and models the geometry correctly. A new FE model was constructed using only 900 shell elements (ANSYS element type 63) (Figure 5). This reduced model will be used to calculate the acoustic radiation of the different modes of the oilpan. Stiffener ribs were not modeled since they do not contribute to the acoustic radiation. However, they contribute to the mass and stiffness of the oilpan. The problem here was to calculate the equivalent plate thicknesses to be used with the reduced model in order to obtain the same modal properties as obtained with the initial fine mesh. IN fact, no experimental data was used as reference data but the results obtained from a reliable model.

The following summarizes the resonance frequencies of the new model, before and after the tuning operation:

		Fine Mesh	Original Coarse Mesh		Updated Coarse Mesh	
		Freauencv	Freauencv	Difference	Frequency	Difference
Mode	No.	Hz	Hz	%	Hz	%
	1	182.3	125.4	-31.2	180.3	-1.1
	2	335.6	311.7	-7.1	335.0	-0.2
	3	515.7	434.5	-15.7	518.0	0.4
	4	728.7	689.5	-5.4	735.0	0.8
	5	844.5	782.2	-7.4	844.5	0.0
	6	946.3	862.3	-8.8	945.0	-0.1
	7	999.6	1034.4	3.5	1000.7	0.1
	8	1014.6	907.2	-10.6	1008.4	-0.6
	9	1131.8	999.1	-11.7	1136.0	0.4
	10	1331.4	1218.7	-8.5	1328.9	-0.2

**Table 1. Frequency Comparisons.**

Figure 6 shows the Modal Assurance Criterion results before (left) and after updating (right). Figure 7a shows the convergence curve showing how the average error on the resonance frequencies was reduced from 11.0 % to 0.4 % in 5 iterations. Figure 7b shows the resulting plate thickness variation (in %), used by the updated model, with respect to the estimated equivalent thickness. This starting value is derived from the exact plate thickness, the mass of the oilpan and estimations of the influence of the stiffener ribs. Figure 8 compares the first torsion modes of the original model and the updated coarse model.

## Conclusions

A stand-alone toolbox program to be used with ANSYS for integration of test and analysis data was described. The program imports ANSYS normal mode analysis and test data for a number of applications related to finite element model validation and calibration. Extensive use can be made of ANSYS analysis capabilities for iterative updating of a finite element model.

It seems very unlikely that a single, general-purpose algorithm for model updating will be developed, which also removes the high level of engineering skill that is required to select parameters and interprets results. Localizing errors is still very dependent on knowledge and decision-making. The more knowledge is available, from both the analytical and experimental sides, the easier judgments can be justified.

Integration of data interfaces and different alternative diagnostics tools in a single dedicated postprocessor program, using graphical user interfaces and visualization tools is a productive alternative to classical trial-and-error approaches.

## References

- [1] 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.
- [2] C.F. McCulloch, E. Dascotte, 'A Robust Methodology for Updating Finite Element Models', Proceedings AUTOTECH 91, November 1991, Birmingham.
- [3] 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.
- [4] M. Imregun, W.J. Visser, 'A review of Model Updating Techniques', The Shock and Vibration Digest, Volume 23, No. 1, January 1991.
- [5] J. Collins, G. Hart, T. Hasselman, B. Kennedy, 'Statistical Identification of Structures', AIAA journal, 12/2, Feb. 1974, 185-190.

## Figures

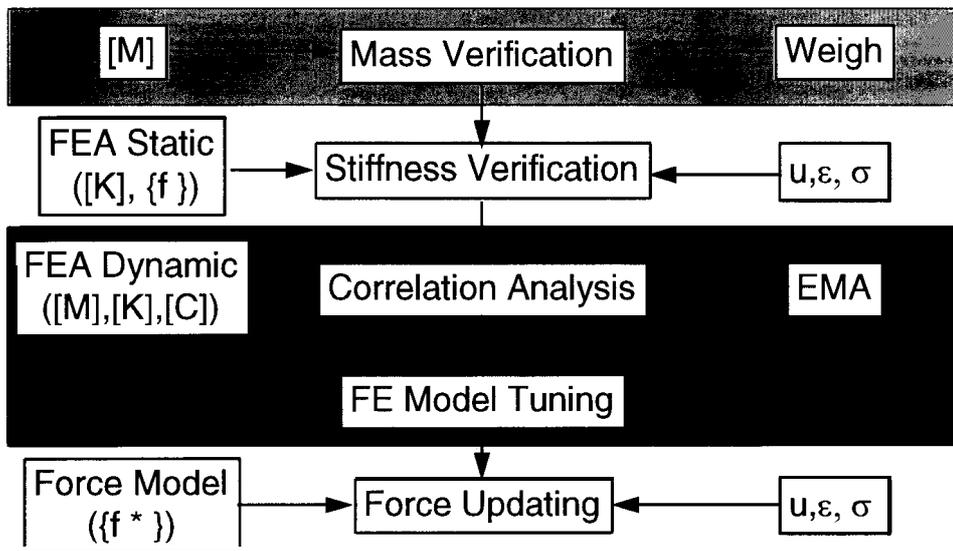


Figure 1. Finite Element Model Validation Procedure.

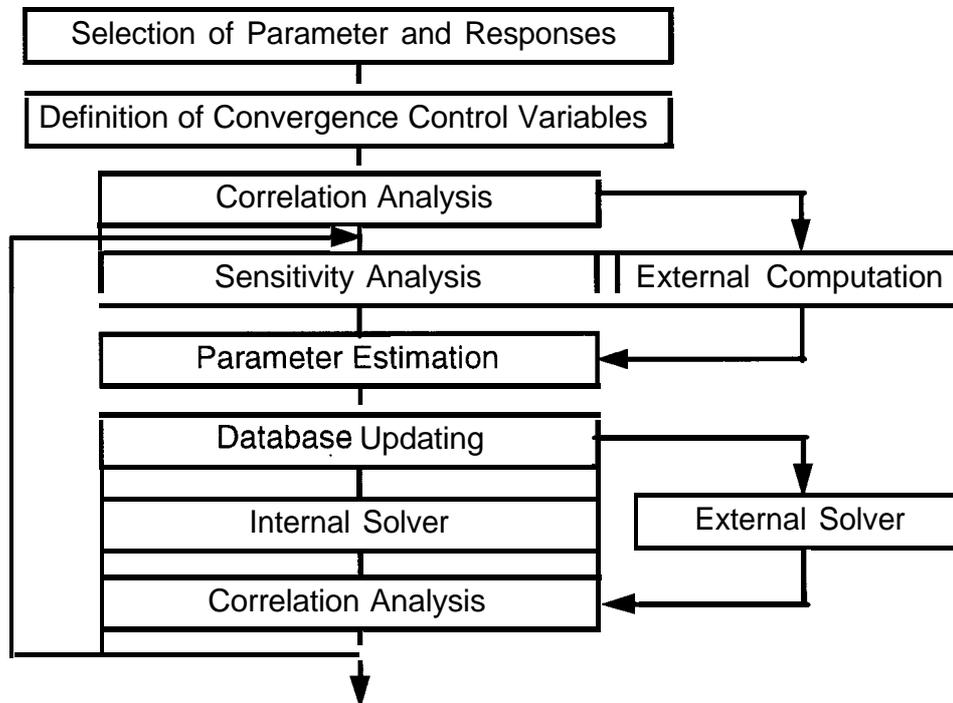


Figure 2. Finite Element Model Tuning Flow Chart.

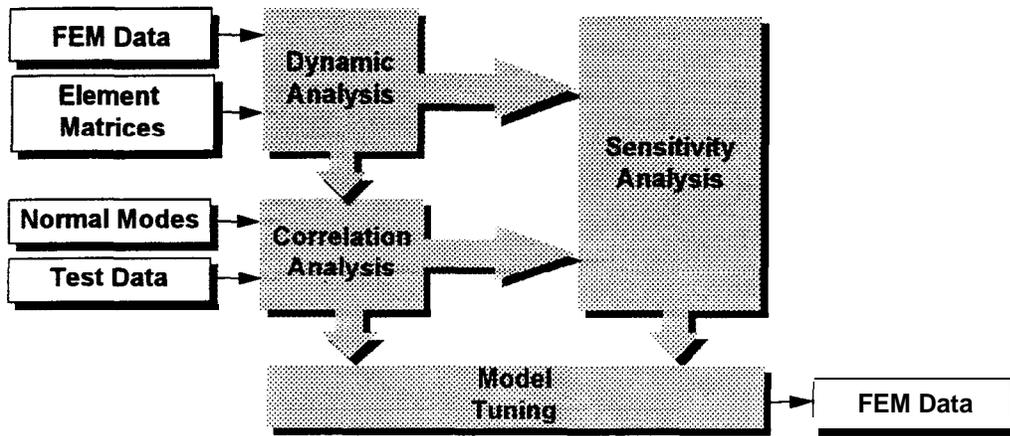


Figure 3. Data Flow Chart.

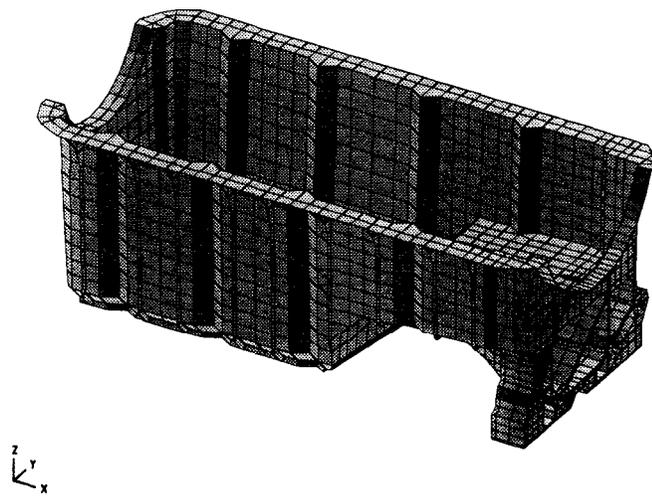


Figure 4. Original Fine Mesh Model.

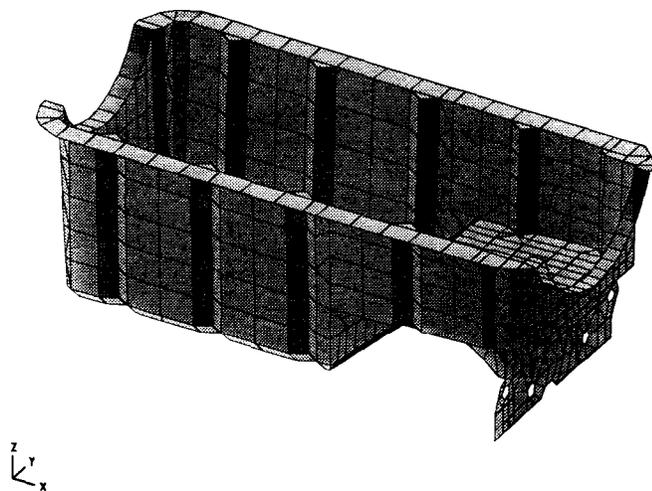


Figure 5. Coarse Mesh Model

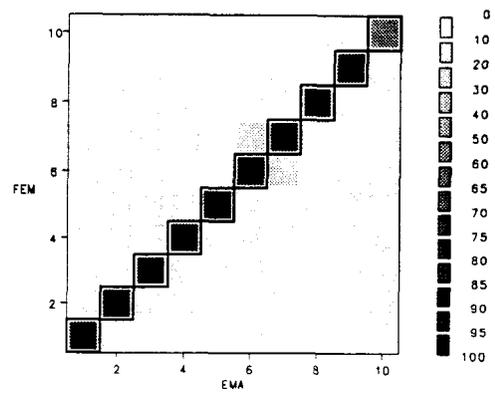
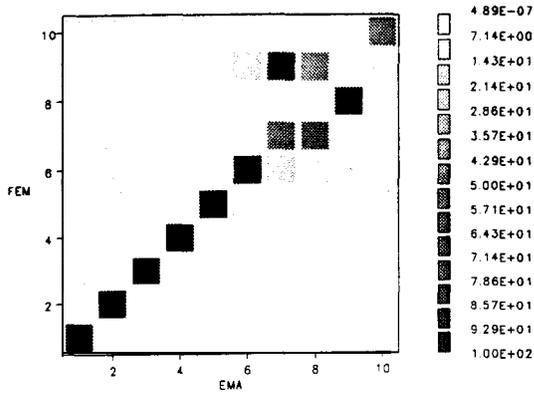


Figure 6. MAC-Matrix Before and After Model Updating.

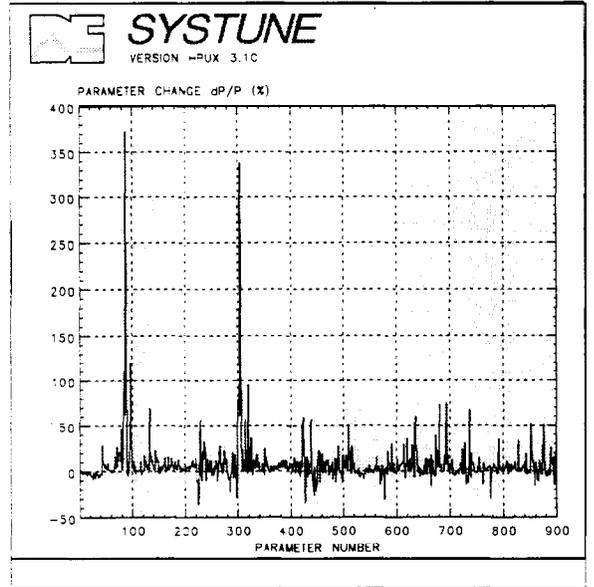
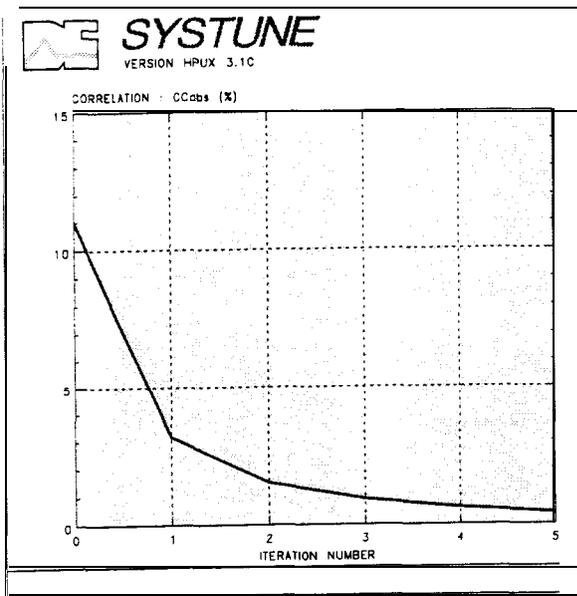


Figure 7. Convergence Curve and Parameter Modification.

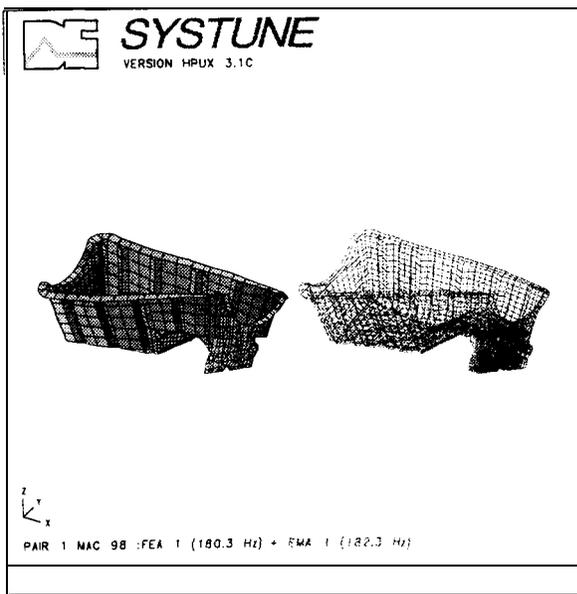


Figure 8. Torsion Mode After Model Updating