APPLICATION NOTE

Information on this document

Contents

This document is the fourth in a series of four application notes on structural dynamics. The document includes information on how to compare results from different structural dynamics studies, as well as a brief introduction to the Finite Element Method.

Target group

This document serves as an introduction to structural dynamics and is intended for acousticians who need information on the comparison of measured and simulated data and the MAC value.

Questions?

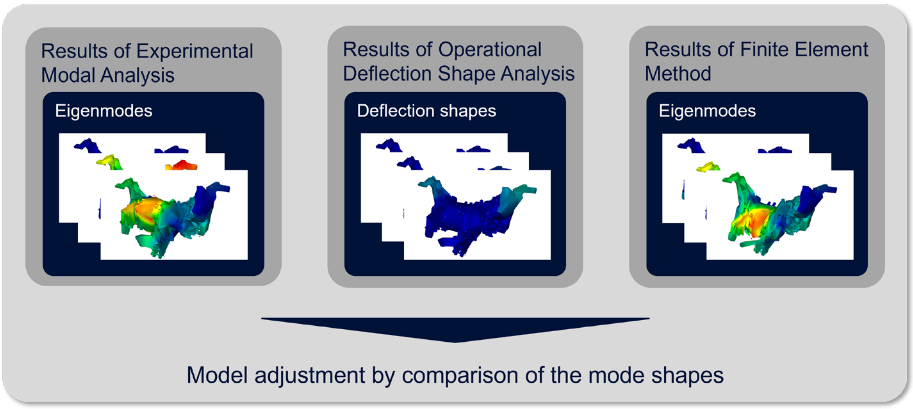
Do you have questions? Your feedback is appreciated!
For questions on the content of this document: Imke.Hauswirth@head-acoustics.com
For technical questions on our products: SVP-Support@head-acoustics.com

Structural dynamics – Part 4 (Comparison of results)

1. Introduction

Comparison of different analysis results

There are various approaches for determining the structural dynamic behavior of a test object. These include Experimental Modal Analysis (see Structural Analysis – Part 2), Operational Deflection Shape Analysis (see Structural Analysis – Part 3) and Finite Element Method (see box on the next page). Each of these approaches has its strengths and weaknesses. For this reason, it should be advisable to combine the different methods and compare the results in order to gain a thorough understanding of the vibration behavior of a test object. The basis for the comparison of results is the comparison of the mode shapes obtained.



It is not only useful to compare between measured and calculated models, but also between two calculated or two measured models. Furthermore, a model can be compared with itself in order to check the orthogonality of the found eigenmodes or mode shapes.

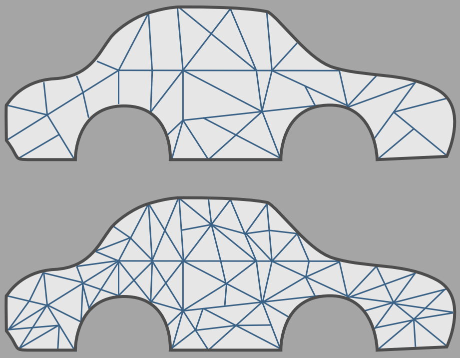
Finite Element Method (FEM)

Finite Element Method

The Finite Element Method is a general numerical method for calculating a wide range of physical problems. In the field of structural dynamics, this method is used to determine the modal properties of a structure. However, it can also be used to pre-calculate stresses in components in order to predict possible failure, or to perform temperature simulations. Modal analysis using the FE method consists of three steps: modeling, calculation and evaluation; also referred to as pre-processing, solving and post-processing.

Modeling

In the modeling step, the geometry of the existing structure is first transferred to the FE program, and an FE substitute model is generated (idealization). During discretization, the substitute model is divided into a defined number of finite elements and structure nodes. This process is also referred to as meshing. The structure nodes of the FE mesh along with their directions of movement represent a degree of freedom (DOF) of the FE model. Material properties, such as elasticity modulus and density of the underlying structure, are assigned to the elements between the nodes. As with experimental modal analysis, the position and number of structure points is also important for the quality of the results in modal analysis with FEM. The desired deflection shapes can only be calculated and displayed by means of a sufficient number of structure nodes. However, a high number of structure points not only enhances the accuracy of the results, but also causes a sharp increase in calculation time.



Calculation

In the calculation step, the parameters of the existing structure are first transferred to the FE substitute model. In addition, the structure's loads such as force or pressure are specified (which is useful for non-linear models). The modal parameters are calculated by a computer program, for which various solution methods (so-called „solvers“) are available.

Evaluation

During evaluation, the natural frequencies are read out. In addition, the eigenmodes of the FE substitute model can be visualized.

Applications

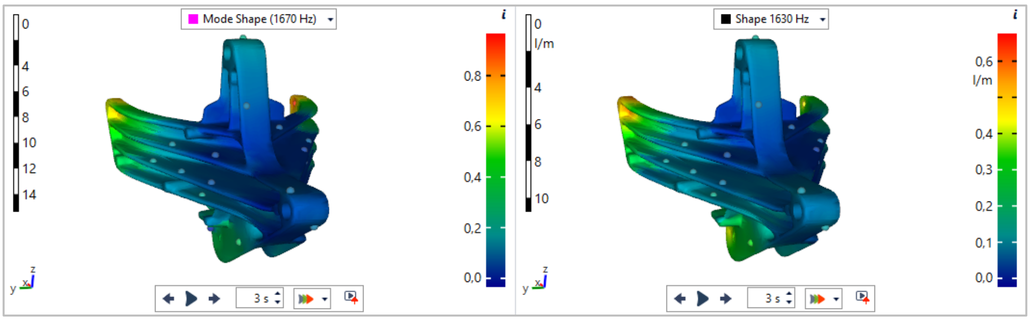
One advantage of numerical modal analysis is that the finite element model can be created and tested before a physical prototype is available. Furthermore, the effects of a modification on a finite element model are usually less complicated, less costly and faster to test than on a physical component. However, there are also some disadvantages: the generation of a finite element model is not easy and, depending on its size, also time-consuming and costly. The model may not be accurate enough, so it is often necessary to validate the results of the theoretical prediction with measured data from a modal analysis.

2. Mode Shape Comparison

There are different methods to compare the models. Two of them are described in more detail below.

Visual check

Visual check of the individual modes: For this purpose, the animated eigenmodes or mode shapes of the models to be compared are displayed either side by side or superimposed and checked visually.

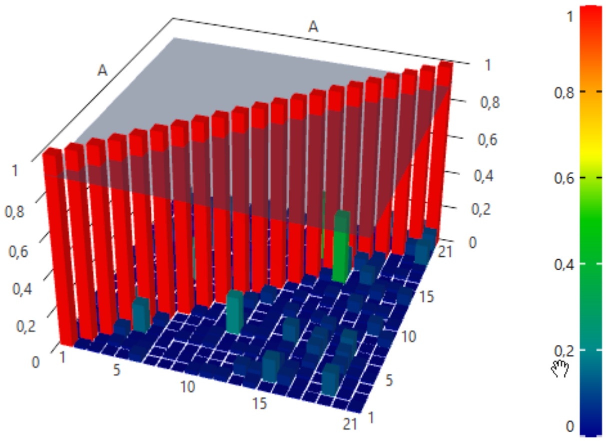


Due to the high expenditure of time during evaluation and the risk of confusion, especially with complex structures, this form of comparison is only reasonably applicable for a small number of mode shapes and for models with a smaller number of degrees of freedom.

MAC value

Modal Assurance Criterion (MAC): Modal Assurance Criterion (MAC) is a mathematical comparison method based on the evaluation of vibrations in the form of vectors. If the results are in the form of eigenmodes (as in modal analysis), their eigenvectors are determined for this purpose; in the case of mode shapes from operational deflection shape analysis, the comparison is based on the mode shape vectors. When comparing two models, a rectangular matrix is created which has as many columns as there are vectors in the first model and as many rows as there are vectors in the second model. The matrix elements represent the value of a standardized scalar product of a vector of the first model and a vector of the second model, hence the orthogonality between these two vectors is evaluated. Thus, each matrix element is a measure of the agreement between the vectors. If both vectors are the same, the matrix element is equal to 1. If the two vectors are orthogonal to each other, it is 0. In practical application, the matrix elements take values between 0 and 1. Basically, if the value is close to 0, it may be assumed that the vectors are not correlated, whereas if the value is close to 1, the corresponding vectors are very likely to be the same. Thus, at best, all values of the matrix that are not on the diagonal are close to 0, i.e., the mode shapes found are orthogonal to each other.

AutoMAC



If a model is compared with itself and the corresponding MAC values are calculated, these are referred to as AutoMAC values. The AutoMAC matrix is used to evaluate the quality of the determined mode shapes. In an AutoMAC matrix, the values of the diagonals always equal 1 since the respective vector is compared with itself. The values off the diagonals should be close to 0. If high values occur off the diagonals, this indicates for example that the spatial resolution is insufficient due to the selected measurement points and that the information on the vibration response could only be insufficiently recorded.

The AutoMAC matrix is also suitable for separating closely spaced eigenmodes or mode shapes. This is useful, for example, in higher frequency ranges or for complex structures where eigenmodes with closely spaced frequencies occur due to increasing mode density. The AutoMAC value can be used to check in a very easy and effective way whether the modes found in modal analysis are actually orthogonal.

Shape Comparison Project

For the comparison, ArtemiS SUITE provides the Shape Comparison Project. The basis for the comparison is the Shape Table. Such a table can be created with a Modal Analysis Project as well as with an Operating Deflection Shape Project. In addition, the mode shapes can be imported from numerical simulations. Various formats of different simulation programs are supported. In addition to the mode shapes, the shape table also contains, among other things, a reference to the Measurement Point Library used. Before a comparison, it must be ensured that the models used are in the same coordinate system. In addition, each degree of freedom of one model must be assigned to a degree of freedom of the other model.

In Auto Mode, the Shape Comparison Project compares the results of a model with itself. For this purpose, the corresponding AutoMAC matrix is automatically calculated and displayed as a bar chart. If two models have been loaded, the user can also switch to Group Mode, in which case the MAC matrix for the loaded models is determined and displayed. Furthermore, ArtemiS SUITE evaluates the mode shapes and compares matching mode shapes in tabular form.