

Experimental Modal Analysis and Model Validation of Antenna Structures

B.R. Potgieter, G. Venter

Abstract—Numerical design optimization is a powerful tool that can be used by engineers during any stage of the design process. There are many different applications for structural optimization. A specific application that will be discussed in the following paper is experimental data matching. Data obtained through tests on a physical structure will be matched with data from a numerical model of that same structure. The data of interest will be the dynamic characteristics of an antenna structure focusing on the mode shapes and modal frequencies. The structure used was a scaled and simplified model of the Karoo Array Telescope-7 (KAT-7) antenna structure.

This kind of data matching is a complex and difficult task. This paper discusses how optimization can assist an engineer during the process of correlating a finite element model with vibration test data.

Keywords—Finite Element Model (FEM), Karoo Array Telescope (KAT-7), modal frequencies, mode shapes, optimization, shape optimization, size optimization, vibration tests

I. INTRODUCTION

THIS paper will discuss how numerical design optimization can assist an engineer during any stage of a design process. This type of optimization performs iterative tasks, leaving the engineer free to concentrate on providing the correct input as well as evaluating and interpreting the output. It furthermore provides increased insight into a problem and reduces design time. Some of the most common structural optimization applications are mass minimization of a structure, concept generation, concept evaluation, structure failure prevention and data matching. This paper only deals with the specific application of experimental data matching. The data of interest is the dynamic characteristics, specifically the modal frequencies and mode shapes, of an antenna structure.

Mundt and Quinn [1] describe the correlation of test data with a finite element model (FEM) as being a difficult and laborious task, which has generally been performed manually. Data matching usually requires a comparison of the numerical and test data. On the basis of this comparison, an assessment is made of which parameters need to be changed and to what degree. These changes are then made by manually editing the input data and rerunning the finite element analysis. This iterative process continues until the analyst has tuned the FEM, essentially one mode at a time.

The antenna structure used was a simplified model of the Karoo Array Telescope-7 (KAT-7) currently being constructed

BR Potgieter is a Masters student at the Mechanical and Mechatronic Engineering Department, University of Stellenbosch, Private Bag X1, Matieland 7602, South Africa (email: 14596709@sun.ac.za)

Prof. G. Venter is with the Mechanical and Mechatronic Engineering Department, University of Stellenbosch, Private Bag X1, Matieland 7602, South Africa (email: gventer@sun.ac.za)

Thank to SKA South Africa for providing the funding for this project.

in South Africa as part of its bid for the Square Kilometer Array (SKA) project. This model was used in order to perform vibration tests which provided a set of frequencies and mode shapes. A FEM of the antenna was generated and then subjected to optimization techniques in order to match its data with the test data. GENESIS, a structural optimization program, was used to apply the optimization techniques.

Optimization allows for a more accurate FEM, which is important as these FEMs are used by analysts to gain a better understanding of a structure's response. In this case, the response of an antenna to environmental conditions is of concern. Antennas are used to collect signals and by understanding the structure's dynamic behavior, an analyst can ensure that the antenna is collecting the signal correctly.

II. OPTIMIZATION

Structural design optimization has been developed to automate the design process by removing the need to repeat unnecessary finite element analyses. According to work by Schmit and Miura [2] numerical optimization is essentially mathematical programming which provides a very general framework for scarce resource allocation. The problem statement for numerical design optimization is very closely related to the problem statement of traditional engineering problems. Due to this, the design tasks to which it can be applied are inexhaustible. Structural optimization makes use of an approximation of the original problem [2]. This approximation is solved by the optimizer and reduces the overall cost of structural design as it is no longer necessary to repeatedly call the finite element analysis during the actual optimization process.

VRAND [3] gives the most general form of an optimization problem: the goal is to find a set of design variables X_i , $i = 1, l$ contained in vector \mathbf{X} that will

$$\text{Minimize } F(\mathbf{X}) \quad (1)$$

Subject to:

$$g_j(\mathbf{X}) \leq 0 \quad j = 1, m \quad (2)$$

$$h_k(\mathbf{X}) = 0 \quad k = 1, n \quad (3)$$

$$X_i^L \leq X_i \leq X_i^U \quad i = 1, l \quad (4)$$

where

$$\mathbf{X} = X_1, X_2, \dots, X_l \quad (5)$$

Equation (1) is the objective function. The optimizer tries to minimize this function by changing the design variables in (5). Inequality (2) and equality constraints (3) can be implemented and must be satisfied by the final set of design variables. Upper and lower bounds (4), which will limit the search area, can be placed on the design variables.

GENESIS has two methods for matching data, namely the Least Squares Method and the Beta Method. The Least Squares Method minimizes the sum of the differences between experimental and numerical data. The Beta Method minimizes the maximum difference between the experimental and numerical data. The Least Squares Method was used in this project. GENESIS's mode tracking algorithms allow for the tracking of modes regardless of frequency, which in turn assists in matching all required modes simultaneously.

GENESIS has 5 different optimization techniques, namely shape, size, topology, topometry and topography. Only shape, size and topometry techniques were used, a brief summary of which follows.

Shape optimization, as the name suggests, involves the changing of the shape of a FEM. The latter is achieved by shifting the grid locations of a FEM. A shape domain wherein grids can be shifted is defined. Vectors are used in this shape domain to guide the shape changes. These vectors are associated with design variables that control their magnitude.

Leiva [4] states that, in sizing optimization, the element cross-sectional dimensions are given as design variables. This allows the optimizer to change element properties by altering the elements' cross sectional dimensions (height, thickness, etc). The sizing optimization is applied to specified domains and all the elements in that domain are assigned the same design variable.

Leiva [4] refers to topometry optimization as a specialized sizing optimization technique. In sizing optimization, all elements are associated with a given property group and are thus designed identically. Topometry optimization designs the elements on an element level rather than a property level, which allows for each element to be designed individually.

III. PHYSICAL MODEL

The physical model was designed as a scaled and simplified model of the KAT-7. The latter is part of a larger project called the MeerKAT, which is part of South Africa's bid to host the SKA project - the world's largest radio telescope. The SKA project will consist of about 3,000 antennas with a diameter of ten to fifteen meters each. Half of the antennas will be concentrated in a 5 km diameter central region, and the rest will be distributed 3,000 km from the central region. These antennas will form a radio telescope with an extremely large collecting area, making it 50 times more sensitive and able to survey the sky 10,000 times faster than any radio telescope array built previously [5]. The KAT-7 will consist of the first seven 12 m antennas of the MeerKAT, which will eventually consist of 80 antennas. The MeerKAT will be used to show South Africa's commitment to and capability of hosting the SKA project. It will also contribute to the development of the technology needed for the SKA.

A FEM of the KAT-7 has already been generated and is being used to gain a better understanding of the structure's response to environmental conditions [6]. Through matching experimental data from the physical structure, an analyst could ensure that the FEM is an accurate model. This would allow for confidence in the model's representation of the physical structure. A simplified model of the KAT-7 will be used to illustrate how optimization can assist an analyst in performing the experimental data matching.

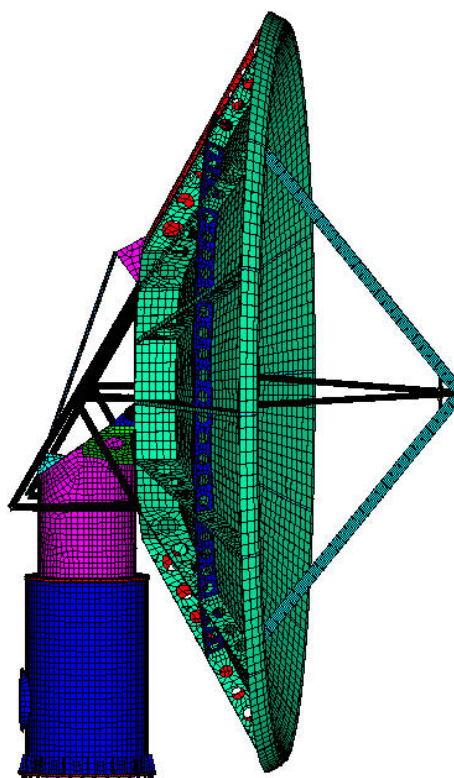


Fig. 1: KAT-7 Antenna

A. Simplified Model

The simplified model has a 1.2 m diameter dish and is made up of two separate pieces, namely the pedestal and the dish. The latter can be mounted onto the pedestal and is able to rotate through different angles of elevation. It has four support arms that all meet above it at a 'feed piece', which represents the focal point. The dish was made from a flat piece of sheet metal, rolled so that the two ends could be welded together. The support arms were also welded onto the dish. This model was used to perform vibration tests.

IV. NUMERICAL MODEL

A. KAT-7

A FEM of the KAT-7 was generated by the SKA project team in order to gain an understanding of the response of the antenna to various environmental conditions. The more

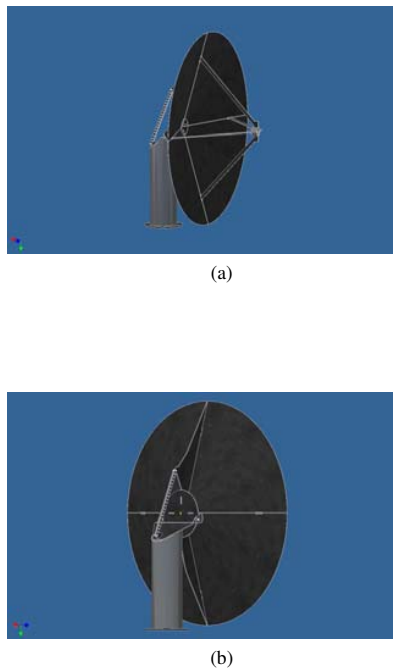


Fig. 2: Simplified Model

accurate the FEM, the better their understanding of the structure will be - optimization can assist the analyst in improving the FEM. Using the simple model, this paper will show how optimization can be used to improve a FEM. MMS [6] states that the FEM of the KAT-7 is modeled as follows: the pedestal and the yoke, being steel plate structures, are modeled with shell elements. The counterbalance structure is modeled with beam elements and the counterbalances modeled as mass elements. The antenna dish is modeled as a sandwich, employing layered shell elements. The feed legs are modeled as beam elements and the feed itself is modeled as a 1.5 m diameter disc with a mass of 100 kg.

B. Simplified Model

The FEM of the simplified model consists of 10871 shell elements and 285 rigid body elements. Normal modes analysis was performed by the finite element analysis and five modes were extracted. This simple model will be used to show how optimization can be used to improve a FEM, allowing for a better understanding of the true structure.

V. VIBRATION TESTS

The vibration tests were performed on the physical model in order to gain its modal frequencies and corresponding mode shapes. Only the first five modes were extracted from the recorded data.

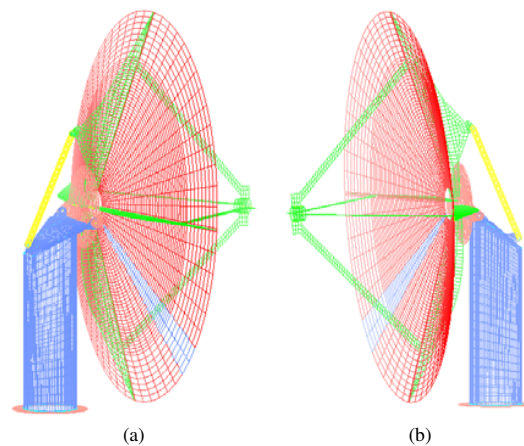


Fig. 3: FEM of the Simplified Model

A. Test Set-up

The vibration tests were performed on the dish separate from its pedestal, as the interest lay in how the dish deforms and how the feed piece moves relative to the dish. This deformation and movement is what will influence the collection of the incoming signals. The pedestal's first bending mode was at 622 Hz, whereas the dish's highest mode of interest had a frequency of 42 Hz. It can consequently be seen that the pedestal will have a very small influence on the frequency range of interest. This also simplified testing as the dish could be tested using soft supports instead of a fixed support set-up. Soft supports are preferred to fixed supports. Ewins [7] states that fixed supports are difficult to implement in practice - it is hard to provide a base or foundation for the test structure that is sufficiently rigid to provide the necessary grounding.



Fig. 4: Test Setup

Ewins [7] states that soft supports are used to model free-free support condition. The test structure is not attached to the ground at any of its coordinates and is, in effect, freely suspended in space. In this condition the structure will only have 6 rigid body modes that are determined solely by its mass and inertia properties and in which there are no bending or flexing modes. These rigid body modes are all at a frequency of 0 Hz.

In practice, however, this is not possible as the structure must be supported in some way. This condition can be approximated by using a suspension system, e.g. hanging the structure from bungy cords or resting it on a tire tube. The suspension system will cause the rigid body mode's frequencies to become non-zero, but they will still be very low in relation to the bending modes of the structure. Ewins [7] therefore states that the highest rigid body mode's frequency must be less than 10-20% of that of the lowest bending mode. The free-free condition was simulated by resting the dish on a tire tube as shown in Fig. 4.

B. Data Collection

An electromagnetic shaker was used to excite the dish, whilst 14 accelerometers were used to measure its response. The input force was measured by a load cell attached to the top of the stinger that carries the load from the shaker to the dish. Eight accelerometers were placed on the rim of the dish to obtain its mode shapes. In addition, each support arm had an accelerometer, whilst the feed piece had two in order to track the movement in an x-y plane. The signal received from the measuring equipment was converted into the frequency domain and then represented as a frequency response function (FRF). Inman [8] states that the natural frequencies, damping ratios and modal amplitudes can be calculated from each peak of the measured FRF. The data collection and processing was done using LMS Test.Lab.

VI. DATA MATCHING

Before optimization can be used to correlate the two models, the data of each first needs to be comparable. The output of the FEM is set up so that the displacements of the nodes that represent the accelerometer measurement positions will be in the same orientation. This allows for direct comparison of the degrees of freedom measured and calculated from the respective models. The comparison and correlation of frequencies and mode shapes can now begin.

Table I: Initial FEM vs Test

Mode	Initial FEM	Test	MAC
1	13.73 Hz	13.93 Hz	0.997
2	24.62 Hz	23.61 Hz	0.209
3	24.83 Hz	26.15 Hz	0.241
4	34.62 Hz	32.06 Hz	0.908
5	42.75 Hz	42.13 Hz	0.881

A. Data Comparison

The comparison of the mode shapes was achieved in two ways. Firstly, an external python script was written in order to plot the mode shapes for visual comparison. Secondly, the Modal Assurance Criteria (MAC) were calculated. The MAC values were used as a measure of how well the mode shapes match. The data obtained from the two models were the structure's first five bending modes. The results are shown in Table I and Fig. 5, illustrating the MAC between the two models as well as the respective modes' frequencies.

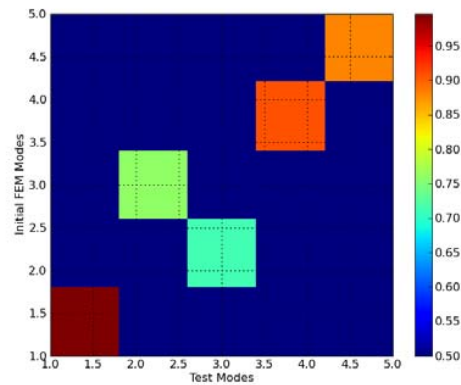


Fig. 5: Initial FEM vs Test

As shown, the frequencies are fairly close but the mode shapes do not correlate. From the MAC plot it is clear that the second and third modes are not in the correct order. The comparison shows that the tuning process should concentrate on correcting the order of the modes as well as improving the frequency and mode shape correlation.

B. Optimization Set-up

Objective Function

The objective function was to minimize the least squares error between the mode shapes. This was done by calculating the displacements of the grid points that would correspond to the measurement positions of the accelerometers on the dish itself. These displacements were normalized by using the grid with the largest displacement in each mode.

Constraints

Equality constraints were applied. The frequencies were constrained to the values measured from the vibration tests.

Design Variables

Shape optimization was used to adjust the shape of the dish. This was done with perturbation vectors that were each assigned their own design variables. Sizing optimization was used to design the element thicknesses that represented the welded part of the dish.

C. Methodology

The choice of perturbation vectors was not made at random; they were carefully selected and implemented in relevant domains. Two methods were used to identify the correct vectors, the first being a visual comparison between the FEM and the Physical model and the second the use of topometry.

Visual Comparison

The FEM is an exact replica of the designed model, therefore it has no imperfections or any variation from the design. Manufactured models do, however, often vary from the intended design. Therefore, a few differences can be found by visually comparing the two models. The support arms were, e.g. not exactly 90 degrees to one other. For this reason, we can implement vectors that adjust the positions of the support arms until they represent that of the physical model.

The FEM does not have a representation of the welded section either. Welding will cause a change in the structural properties around welded sections. By applying sizing optimization to the elements that represent the welded section, the optimizer can try to match the change in structural properties.

Topometry

As mentioned earlier, topometry is a special kind of size optimization that changes each individual shell element's thickness. By applying the same object function and constraints as in the original problem, the optimizer will solve the problem by adjusting the thickness of all the individual elements. The result will show which area of the dish has a higher or lower stiffness and give the user an indication of how to adjust the shape accordingly. The analysis showed, e.g. that the elements between two support arms had increased in thickness more than elsewhere. This part of the dish therefore probably had a higher stiffness than other sections. This resulted in a vector pushing the two arms closer together on the FEM, as was later shown to be correct after measurements were done on the physical model.

VII. RESULTS

After optimization was applied to the initial FEM, a comparison of the data was again performed in order to determine whether the optimizer had achieved its objective. As before, the comparison consisted of visual comparisons and the calculation of MAC values. This data is presented in Table II and Fig. 6, showing that the mode shapes are now in the correct order and the error in the frequency values decreased.

Table II: Test vs Updated FEM

Mode	Updated FEM	Test	MAC
1	13.70 Hz	13.93 Hz	0.996
2	23.76 Hz	23.61 Hz	0.892
3	25.47 Hz	26.15 Hz	0.913
4	34.59 Hz	32.06 Hz	0.912
5	42.65 Hz	42.13 Hz	0.892

VIII. CONCLUSION

Through optimization, the FEM was updated to be a better representation of the physical structure, especially the dynamic characteristics thereof. This is advantageous to analysts trying to gain understanding of a structure's responses - in this case the response of the antenna to environmental conditions during operation. Through optimization, the process of matching frequencies and mode shapes was automated and all modes

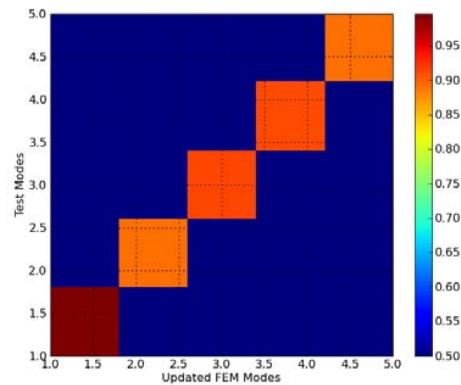


Fig. 6: Test vs Updated FEM

could be matched simultaneously. This would have been very difficult to achieve through traditional methods.

Although the optimizer has produced a good result, it requires a lot of input and insight from the user. The random application of vectors will not lead to the optimizer producing correct results. Optimization is a mathematical process - it has the potential to find a result that satisfies the objective function, but this may not make sense in engineering terms. Engineering insight has to be applied to ensure that the result is valid. This can be achieved by, e.g. using constraints, reasonable bounds on design variables and carefully selected vectors.

Although the work presented in this paper has been conducted on a simplified model, the same techniques could be applied to the actual KAT-7 antenna structure. The scope of the SKA project is immense, encompassing the eventual erection of 3,000 antennas in total. If the FEM models of the first 7 antenna are accurate, a full understanding of these structures can be gained before the commencement of the larger project. Optimization can thus be of significant value to this project.

REFERENCES

- [1] C. Mundt and G. Quinn, *Test-analysis correlation with design optimization*, Long Beach Convention Center, Long Beach, CA, 2005.
- [2] L. Schmit and H. Miura, *Approximation concepts for efficient structural synthesis*. NASA CR-2552, 1976.
- [3] *Genesis design manual, version 10.1*, Vanderplaats Research and Development, Inc, Colorado Springs, 2008.
- [4] J. Leiva, *Topometry optimization: a new capability to perform element by element sizing optimization of structures*, 2nd ed. MI, USA: Vanderplaats Research and Development, Inc, 2008.
- [5] *Ready to host the SKA*, SKA South Africa, Johannesburg, South Africa, 2008.
- [6] *KAT-7 antenna structure design and analysis*, MMS Technology (Pty)Ltd, Centurion, RSA, 2008.
- [7] D. Ewins, *Modal Testing: Theory and Practise*, 2nd ed. Research Studies Press Ltd, 1984.
- [8] D. Inman, *Engineering Vibration*, 2nd ed. Prentice Hall International, Inc, 2001.