Model Update in Structural Dynamics for Electric Powertrains

Elias Gasser, Christian Petruzzino and Dr. techn. Franz-Josef Falkner (supervisor)

Abstract—**This paper investigates model updating techniques in structural dynamics, specifically focusing on the application of Finite Element Analysis (FEA) to estimate noise, vibra- tion, and harshness (NVH) in electric powertrains. The primary objective is to enhance the accuracy of Finite Element models through the integration of experimental modal analysis data obtained via impact testing. To achieve this, a methodology is developed using Python programming within the MARC/Mentat environment to conduct sensitivity analyses on the FEA. This approach enables the identification and prioritization of critical simulation parameters essential for accurate frequency and mode shape calculation. Additionally, an automated optimiza- tion framework is developed to iteratively update the simulation parameters, enhancing model fidelity. The proposed methodology is validated through comprehensive testing on various components and assemblies within electric powertrains. Evaluation of the simulation optimizations is performed using metrics such as Modal Assurance Criterion (MAC) matrices and frequency error assessments. The outcomes of this research contribute to advance the effectiveness and reliability of Finite Element models for NVH prediction in electric powertrain applications. The developed methodologies provide a systematic approach to integrate experimental data into simulation models, leading to better predictions of noise, vibration, and harshness in the design phase of various electric powertrains.**

Index Terms—**Model Updating, Impact Testing, Simulation Optimization, Sensitivity Analysis, Electric Powertrain.**

I. Introduction

The automotive market is rapidly evolving, with a significant shift towards electric vehicles (EVs) driven by environmental concerns and advancements in technology. As this transition unfolds, it brings forth new challenges, one of which revolves around understanding and mitigating the noise, vibration, and harshness (NVH) characteristics of electric powertrains.

In traditional internal combustion engine (ICE) vehicles, noise generation predominantly stemmed from the mechanical complexities of the engine, including moving and unbalanced parts like the crankshaft. However, in electric powertrains, the dynamics are notably different. Here, the kinematics primarily consist of rotational motion, with electric motors operating at significantly higher rotational speeds than their ICE counterparts. As a consequence, achieving perfect balance in components such as the rotor and gear steps becomes imperative. Even slight imbalances can introduce excitation forces that propagate throughout the powertrain, resulting in high-frequency noises, which are not only undesirable but also potentially harmful to the overall driving experience and customer perception.

In response to the growing demand for quieter electric vehicles, automakers impose stringent noise level requirements on electric powertrains. These requirements necessitate accurate noise predictions early in the development process, prompting the utilization of advanced Computer-Aided Engineering (CAE) tools such as SMT MASTA at automotive companies like GKN.

However, despite the sophistication of these simulation tools, discrepancies between predicted and measured noise levels persist, often attributed to uncertainties inherent in the Finite Element (FE) models used. These uncertainties primarily arise from challenges in accurately predicting material properties and contact stiffness parameters without physical prototypes.

This gap between simulation and reality underscores the need for a systematic approach to enhance FE models using experimental modal analysis. By incorporating real-world data into the simulation process, it becomes possible to refine model parameters and improve the accuracy of noise predictions for electric powertrains.

Currently, the numerical modal analysis is updated by hand, going through several iterations by changing the simulation parameters. In the development phase, time is usually short and going through these iterations takes a lot of time. This process has a great potential for automation, which would also come with a time saving.

Ultimately, the goal is to enhance the predictive capabilities of FE models for existing electric powertrains, enabling more accurate noise simulations, by bridging the gap between simulation and reality.

II. State of the Art

A. Experimental Modal Analysis

Experimental Modal Analysis serves the crucial purpose of identifying modal parameters, including eigenfrequencies, damping ratios, and mode shapes for various components. One widely utilized method, also employed at GKN, involves impact testing. This method entails exciting the structure through controlled hammer impacts, while simultaneously measuring the input force exerted by the hammer. The response of the structure is captured using one or multiple accelerometers strategically placed on the structure. In addition, the structure is suspended with a rubber band or placed on a thick foam plate whose stiffness is much lower than the stiffness of the structure itself. This eliminates the influence of the structure supporting the tested part.

Subsequently, the acquired data is transformed into a frequency response function (FRF), which serves as the foundation for calculating modal parameters. This process typically involves employing various techniques such as curve fitting or peak picking.

At GKN the state of the art software for the experimental modal analysis is Simcenter Testlab. In Simcenter Terstlab Impact Testing, the most common way to estimate the modal parameter is to use the PolyMAX add-on. PolyMAX is a modal curve-fitter that uses curve fitting techniques to fit the measured FRF to a synthetic one by adjusting the modal parameters. [1]

B. Numerical Modal Analysis

Finite Element Modal Analysis (FEMA) is a method of approximating the eigenfrequencies and mode shapes of a structure. These parameters are crucial for understanding how a structure will respond to dynamic loads. The finite element method (FEM) is a numerical technique that approximates the behavior of physical systems, making it possible to perform modal analysis on complex structures that are difficult to analyse analytically.

Resonance occurs when the frequency of external forces matches one of the natural frequencies of a structure, leading to large amplitude vibrations. FEMA assists in identifying these natural frequencies and designing structures to avoid resonance.

The primary objective of the FEMA is to address a generalized eigenvalue problem derived from the following equations. To begin, we consider the general equation of motion of a dynamic system. [2]

$$\boldsymbol{M}\ddot{\boldsymbol{q}}(t) + \boldsymbol{C}\dot{\boldsymbol{q}}(t) + \boldsymbol{K}\boldsymbol{q}(t) = \boldsymbol{0} \tag{1}$$

Where $\boldsymbol{M}$ is the mass matrix $\boldsymbol{C}$ is the damping matrix and $\boldsymbol{K}$ is the stiffness matrix. First, damping is not considered in modal analysis. In metal structures, damping is usually very small, so it doesn't affect eigenfrequencies and mode shapes. Equation (1) is simplified to

$$\boldsymbol{M}\ddot{\boldsymbol{q}}(t) + \boldsymbol{K}\boldsymbol{q}(t) = \boldsymbol{0}. \tag{2}$$

The response of the system $\boldsymbol{q}(t)$ under a specific initial condition is assumed to be harmonic and is described by this equation

$$\boldsymbol{q}(t) = \boldsymbol{\varphi} e^{i\omega t}. \tag{3}$$

Where $\boldsymbol{\varphi}$ is the deformation pattern or mode shape and ω is the corresponding natural frequency [2]. Taking the second derivative of equation (3) and substituting it into (2) gives the formulation of the generalized eigenvalue problem,

$$\boldsymbol{K}\boldsymbol{\varphi}_i = \omega_i^2 \boldsymbol{M}\boldsymbol{\varphi}_i \tag{4}$$

where i stands for the different modes. The generalized eigenvalue problem can be transformed into a standard eigenvalue problem, where the eigenvalues are the natural frequencies squared and the eigenvectors are the mode shapes. Hence, a Cholesky decomposition on the mass matrix is needed:

$$\boldsymbol{M} = \boldsymbol{S}\boldsymbol{S}^{\boldsymbol{T}}. \tag{5}$$

Inserted into equation (4) gives,

$$\boldsymbol{K}\boldsymbol{\varphi}_i = \omega_i^2 \boldsymbol{S}\boldsymbol{S}^{\boldsymbol{T}}\boldsymbol{\varphi}_i. \tag{6}$$

Next, both sides of the equation are multiplied by $\boldsymbol{S}^{-1}$ and $\boldsymbol{S}^{-T}\boldsymbol{S}^{T}$ is inserted on the left side. As a result, the formulation for the standard eigenvalue problem,

$$\tilde{\boldsymbol{K}}\tilde{\boldsymbol{\varphi}}_i = \omega_i^2 \tilde{\boldsymbol{\varphi}}_i, \tag{7}$$

is created. With

$$\tilde{\boldsymbol{\varphi}}_i = \boldsymbol{S}^{\boldsymbol{T}}\boldsymbol{\varphi}_i, \tag{8}$$

and

$$\tilde{\boldsymbol{K}} = \boldsymbol{S}^{-1}\boldsymbol{K}\boldsymbol{S}^{-T}, \tag{9}$$

[3]

This transformation allows the generalized eigenvalue problem to be converted into a standard eigenvalue problem, which can then be solved using the iterative algorithm Lanczos. This algorithm is also employed in Marc/Mentat.

In this paper, the reason for using FEMA is to adjust the stiffness of the FE models by looking at the eigenfrequencies. The previous chapter described how the eigenfrequencies are determined experimentally. With this knowledge, the stiffness of the FE model, represented by the stiffness matrix $\boldsymbol{K}$, can be adjusted so that the experiment matches the simulation. At GKN ePowertrain, the FE solver software used is Marc/Mentat.

C. Modal Assurance Criterion

Section II-B describes numerical modal analysis, while II-A describes experimental modal analysis, now an interface between the two worlds is needed. It is not always easy to match the eigenfrequencies found in the experiment with those found in the simulation. It is also possible that the frequencies calculated in the simulation are not found in the experiment. In most cases, the mode shapes are used to pair the eigenfrequencies of the simulation with those of the experiment. For simple components, this is often done intuitively by simply looking at the mode shapes of the simulation and trying to find similarities in the mode shapes of the experiment. However, a better way to pair the eigenfrequencies is to calculate the Modal Assurance Criterion (MAC). The MAC matrix helps pair the simulation and experiments by comparing the mode shapes and giving an indication of how similar they are. The formula for this is given by the equation 10. [4]

$$MAC(i,j) = \frac{(\varphi_{EMA,j}^T \cdot \varphi_{FE,i})^2}{(\varphi_{EMA,j}^T \cdot \varphi_{EMA,j})(\varphi_{FE,i}^T \cdot \varphi_{FE,i})} \tag{10}$$

The result of this equation is the MAC matrix. Values close to 1 indicate a high similarity between the mode shapes, while a value close to 0 indicates the opposite.

D. Model Updating

Now that the mode shapes of the EMA and the numerical modal analysis can be paired by the MAC, the next step is to adjust the numerical modal analysis so that the eigenfrequencies match the ones from the EMA. This process is called model updating. The current approach to model updating is manual. The parameters of the numerical modal analysis are adjusted through several iterations by hand until the eigenfrequencies match those of the experiment. The process is shown in the figure 1. Doing this type of process manually is very time-consuming, and it becomes even more difficult when the number of uncertain parameters and eigenfrequencies is large. In addition, one can never be sure that the optimal parameters have been chosen.

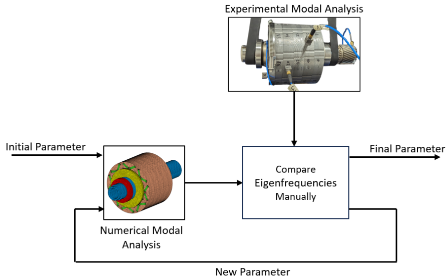

Fig. 1. Flow chart of the manual update process.

E. Sensitivity Analysis

It is not always easy to decide which parameters to change in the model updating process. Not every parameter has the same effect on the eigenfrequencies, and often different parameters influence different eigenfrequencies of the numerical modal analysis. When the number of uncertain parameters is large, it is difficult to get an overview of the importance of each parameter, so a systematic approach is needed to find the critical parameters. This is where sensitivity analysis comes in.

There are many ways to perform a sensitivity analysis. But in each case, some samples of the simulation are needed. This means that the numerical modal analysis should be calculated with different combinations of uncertain parameters. Since numerical modal analysis often requires a lot of computation, it is important to generate as few samples as possible, but still enough to get reasonable results. There are several approaches to generate such a set of parameters that gets the most information out of as few samples as possible. One of the most popular is Latin Hypercube Sampling (LHS). [5]

Now that the parameter samples are defined, the corresponding simulation eigenfrequencies can be calculated and stored for each sample. To get an estimate of how much a parameter correlates with a particular eigenfrequency, the Pearson correlation coefficient is calculated. The Pearson correlation coefficient is a good indicator of the linear relationship between two data vectors. Suppose the vector x represents the samples of a parameter and y is the corresponding result vector of an eigenfrequency. The Pearson correlation coefficient between these two parameters can be calculated according to the equation 11, where m_x and m_y are the mean values of x and y. [6]

$$r = \frac{\sum(x - m_x)(y - m_y)}{\sqrt{\sum(x - m_x)^2 \sum(y - m_y)^2}} \tag{11}$$

A Pearson correlation coefficient close to 1 or -1 indicates a good correlation, while a value close to zero indicates a poor correlation.

F. Optimization Algorithm and Target Function

To create an automated optimization process a target function that describes the error of the optimization process is needed. The success of the result of a manual or automated optimization process is commonly described by the target function 12.

$$F = \sum_{i=1}^{n} \left(\frac{f_{i,EMA} - f_{i,FE}}{f_{i,EMA}} \right)^2 \tag{12}$$

takes the eigenfrequency i of the EMA and subtracts from it the corresponding numerically calculated eigenfrequency i, resulting in the frequency error. Additionally, the result is divided by the physical eigenfrequency to get the relative error, since the frequency error is always calculated in $\%$ and not in Hz. The relative error is then squared to make sure the result is positive and to give more weight to a large error. Finally, the errors of all modes are summed up to get a single value for the objective function F, which can then be minimized.

There are several minimization algorithms that can be used for a global, nonlinear, constraint minimization problem. [7] uses Bayesian Optimization (BO) to minimize an expensive objective function that solves an FE model for each evaluation. BO is a machine learning based global optimization technique that uses as few iterations as possible to find the minimum. BO uses samples to train a Gaussian process (GP) to approximate the objective function. The samples for this could be, for example, Latin hyper cube samples. The Gaussian process approximates the objective function using a probabilistic model. [7]

G. Linear Regression

Often the objective function is time-consuming to evaluate because of the need to solve a numerical modal analysis at each iteration. For larger problems, this can mean that the optimization process can take several days. The idea of this section is to speed up the optimization by creating a surrogate model of the finite element model that can reproduce the same results but much faster. This would reduce the optimization time to a few seconds. The first step in optimizing a simulation is always to perform a sensitivity analysis, which requires samples. The same samples can be used to train the surrogate model, for example by using linear regression. The trained model can then be used to predict the eigenfrequencies in no time.

III. Objectives

The primary objective of this paper is to develop a methodology for updating the finite element modal analysis of various components and sub-assemblies of electric powertrains using data acquired from experimental modal analysis. This methodology will encompass the development of a structured workflow for automatic simulation updates based on experimental data inputs within Marc/Mentat. One goal is to automate the direct optimization approach by solving the simulation in every iteration. Furthermore, a systematic approach to perform an optimization with a trained surrogate model is developed. To know how to set up the optimization a method to perform sensitivity analysis within Marc Mentat is created. The effectiveness of the proposed methodology will be evaluated through extensive testing on various components.

IV. Concept

A. Equipment and Software for the EMA

The hammer used for Impact Testing is the PCB 086C03. PCB says that the applications for this hammer are medium structures such as car frames, and small electric motors.

The hammer can be equipped with different tips to excite different frequency ranges on the structure. It is possible to set up the hammer with a hard, medium, soft or very soft tip. At GKN ePowertrain, the frequencies of interest are between 0 Hz and 5000 Hz, so the hard tip is used most of the time. [8]

The accelerometers used are also from PCB and are of type 356A43. The frequency range is from 0Hz to 7000Hz with an accuracy of 5 %, which is sufficient as it is larger than the frequency range of interest. With an accuracy of 10 % the accepted frequency range is up to 10 kHz. The acceleration range of the sensor is $\pm$ 500 g. [9]

The data acquisition system (DAQ) used is the Siemens SCADAS Mobile 01. Equipped with the V24 card, the DAQ can handle up to 24 channels at 24-bit resolution and a sampling frequency of 51, 6 kHz.[10]

The software used at GKN for the experiments, measurements and data processing is "Simcenter Testlab Impact Testing". The software automatically connects to the DAQ system and there is the possibility to include different add-ins to the software and adapt it to the measurements and data processing one is going to do. For more information on how to use Simcenter Testlab Impact Testing [11] or [12] is recommended.

B. Concept of the Optimization Process

The general workflow for optimizing a numerical modal analysis should begin with a sensitivity analysis. The user should become familiar with the model and be able to identify the most important parameters. Once the most important parameters are known, the data generated by the sensitivity analysis should be used to train a surrogate model of the numerical modal analysis. The next step is to perform optimization on the surrogate model to obtain a rough estimate of the optimal parameters. To further improve the numerical modal analysis, a further optimization can be performed by iterating with an optimizer directly over the numerical modal analysis. In this case, however, the bounds of the parameters can already be tightly constrained. In the next section, these steps are explained in more detail.

The optimization process is created in Python scripts that can be run in Marc/Mentat. Marc basically provides two modules in Python: "`py_mentat`" and "`py_post`". `Py_mentat` can be used to create models, modify models, set model parameters, run models, and so on. While `py_post` is used to open the t16 result files and extract result data such as mode shapes and eigenfrequencies. [13]

`Py_mentat` offers the possibility to send Marc commands with the command `py_send()`, in the brackets the command can be set as a string, this offers the possibility to have full control over a Marc/Mentat model. [13]

The direct minimization algorithm is based on Bayesian optimization and Gaussian processes as described in section II-F. The Bayesion optimization function is taken from the Scikit-Optimized. The function `gp_minimize` takes as input the name of the objective function, the initial parameters `x0` and the parameter bounds `bnds`.

The linear regression model is taken from the module `sklearn.linear_model`. The function used in the specified module is called `LinearRegression()`.[14]

V. Realization

A. Super-NE Rotor Assembly

The Super-NE rotor shown in figure 2 is basically made up of 4 different components. The rotor shaft is made of steel and is also the input shaft for the gearbox. Therefore, a spline is machined on one end of the shaft. The blue parts are called end rings and are also made of steel. They are pressed onto the rotor shaft to compress the stacks. The stacks are the gray parts between the end rings. The assembly also contains a ball bearing on the opposite side of the spline.

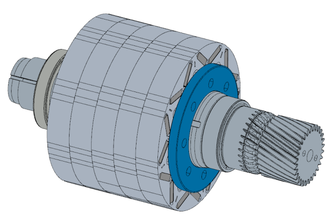

Fig. 2. The Super-NE rotor assembly.

B. EMA of the Rotor

The geometry of the measurement is created in Simcenter Testalb. In this case, many more measuring points are used as it can be seen in the figure 3. In total, the geometry is defined by 60 nodes distributed over the outside diameter of the stack, on the end rings and on the rotor shaft. The sensors are distributed at 45 deg intervals around the circumference of the stack and the end rings.

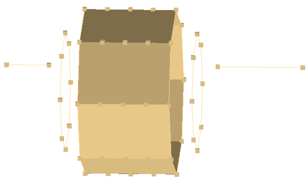

Fig. 3. Geometry used for the EMA of the rotor.

As for the stack, the rotor is also suspended, but with stiffer elastic bands. The setup of the rotor test can be seen in the figure 4. The geometry points are marked on the rotor surface and the accelerometers are glued to the rotor with superglue.

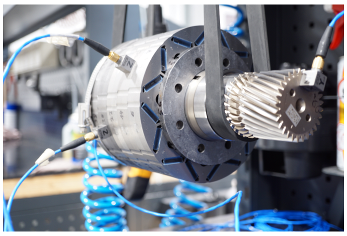

Fig. 4. Set up of the experimental modal analysis of the rotor.

For the test, 4 impact points are selected. Two in radial direction on the shaft, more precisely on the ends of the shaft. Another point is chosen in tangential direction on the stack. This is done by fastening a pipe clamp around the stack, then the screw of the clap can be hit with a hammer to get an impact in tangential direction. Another impact is applied to the end rings in the axial direction.

The mode shapes and eigenfrequencies are then estimated using the PolyMAX add-in. The result of the mode shapes can be seen in the figure 5. The mode shapes are named in this case to distinguish them. Five rotors are measured to get an averaged result of the eigenfrequencies, the results of the different samples are shown in the table I.

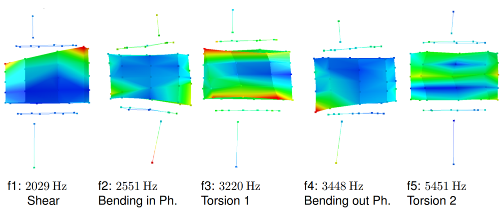

Fig. 5. Modes shapes of the rotor EMA.

Sample Nr.	f1 / Hz	f2 / Hz	f3 / Hz	f4 / Hz	f5 / Hz
1	2019	2520	3157	3562	5391
2	2038	2521	3251	3494	5575
3	2104	2527	3145	3405	5408
4	1936	2614	3211	3495	5446
5	2040	2544	3273	3400	5378
Average	2029	2551	3220	3448	5451

TABLE I
RESULTS AND AVERAGE OF THE ROTOR EMA.

C. Model of the Super-NE Rotor

The simplified model shown in 6 is created under the following assumptions. This model does not take into account the contact stiffness between the different components. All contacts are modeled as glued contacts. The stacks are also further simplified. First, the magnets are filled with stack material and the density of the stack material is changed accordingly. Furthermore, the stack is modeled as one body instead of five separate stacks. An orthotropic material law in cylindrical coordinates is used for the stack material. The model of the stack can be seen in figure 6 on the right. The idea of this model is to adjust the stack material parameters to compensate for the simplifications made.

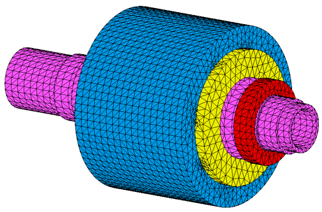

Fig. 6. Mesh of the simplified Super-NE rotor model.

D. Model Update of the Super-NE Rotor

Since the simplified Super-NE model is a light model, direct optimization is performed here as described before. To visualize the effect of the different parameters on the eigenfrequencies, a sensitivity analysis is performed with 30 Latin hypercube samples. All parameters of the stack material are analyzed in the sensitivity analysis. The result is shown in figure 7 The parameters that are optimized are again the parameters of the stack material, except those that are similar to steel, more precisely, parameters E3, G23 and G31 are optimized. It can also be seen in figure 7 that with these three parameters all eigenfrequencies of interest can be influenced.

	E1-E2	E3	G12	G23	G31
Shear	0.09	0.23	-0.02	0.54	0.90
Bending i.	-0.11	0.98	0.08	0.11	0.14
Torsion	0.14	0.05	0.13	1.00	0.21
Bending o.	-0.04	0.78	0.00	0.27	0.64

Fig. 7. Sensitivity analysis of the stack material of the Super-NE simplified model.

The Bayesian optimization process is started by feeding the sensitivity analysis data into the algorithm. The next

30 direct optimization iterations are performed, yielding the results shown in the table II.

EMA / Hz				
Mode 1	Mode 2	Mode 3	Mode 4	Mode 5
2029	2551	3220	3448	5451
Initial NMA / Hz				
1674	1891	3193	3334	5710
17.5 %	25.9 %	0.8 %	3.3 %	4.8 %
Optimized NMA / Hz				
2003	2502	3156	3569	5634
1.3 %	1.9 %	2.0 %	3.5 %	3.4 %

TABLE II
RESULT OF THE MODEL UPDATE OF THE SUPER-NE ROTOR SIMPLIFIED MODEL.

The final parameters of the Super-NE rotor simplified model are listed in table III. It becomes visible that the shear moduli are similar to the one of the complex model.

Stack Material:	Value / $\mathrm{N/mm^2}$
E1	200000
E2	200000
E3	948
G12	72000
G23	942
G31	808

TABLE III
OPTIMIZED PARAMETER OF THE SUPER-NE ROTOR SIMPLIFIED MODEL

VI. MODEL UPDATING OF THE K5 ROTOR

As a second component, the K5 rotor will be modeled and optimized using experimental modal analysis. For this, a simplified model of this rotor is created with similar characteristics to the simplified Super-NE rotor. The experimental modal analysis data is already available, so only some basic evaluation is performed.

A. Rotor Assembly

The K5 rotor has significant design improvements over the Super-NE rotor. The assembly of the K5 rotor is shown in figure 8. The most significant design change is the nut and shoulder design. In the case of the K5 rotor, instead of the stacks being compressed by the endrings through a press fit, the endrings are compressed by a nut on the shaft and on the other side of the shaft, a shoulder supports the opposite endring. In this way the compression of the stacks can be adjusted more precisely by tightening the nut with a certain torque and compressing the stacks with a certain force beforehand. Another design change on the K5 rotor are the endrings. The outer diameter of the endrings has been increased to nearly match the outer diameter of the stacks. This is done to better distribute the compression of the stacks. The stack design remains the same as the Super-NE, with the addition of one stack compared to the Super-NE rotor. Another difference from the Super-NE rotor is that the endrings are made of an aluminum alloy.

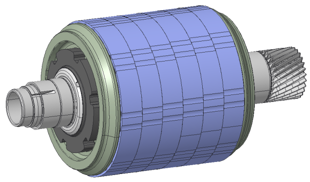

Fig. 8. CAD model of the K5 rotor.

B. EMA of the K5 Rotor

As mentioned, the data of the EMA already exists. What is missing is the evaluation of the data by estimating the mode shapes and the frequencies. This is again done as described in the examples before. The results of averaging the eigenfrequencies are shown in table IV.

Mode Name	f_n / Hz
Shear	2128
Bending in Phase	2469
Torsion 1	3036
Bending out of Phase	4147

TABLE IV
RESULTS AND AVERAGE OF THE K5 ROTOR EMA.

The modes of the K5 rotor are similar to the ones of the Super-NE, and can be differentiated by the names in table IV.

C. Numerical Modal Analysis of the K5 Rotor

The K5 rotor model makes similar simplifications to the simplified Super-NE rotor model. The mesh is shown in the figure 9. This model is also designed to be simple, small, and efficient so that it can be easily implemented in SMT MASTA. The stack is modeled in the same way as for the Super-NE model, with one body that combines all 6 stacks. The contact stiffness is neglected and the contact areas are modeled as bonded by combining the nodes of the contact bodies. The nut, which is new to this design, is also modeled with bonded contacts to the shaft and the endring. The spline of the rotor shaft is modeled as a cylinder with equivalent mass.

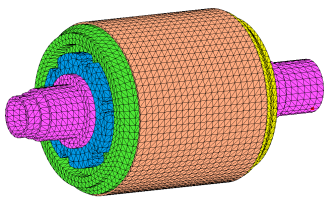

Fig. 9. Mesh of the K5 rotor model.

As a first attempt, the parameters of the simplified Super-NE model are selected for the stack material. The shaft is modeled as steel with a Young's modulus of $208000\,\mathrm{N/mm^2}$. The endrings are modeled as aluminum with a Young's modulus of $72000\,\mathrm{N/mm^2}$.

The mode shapes of the numerical modal analysis can be seen in figure 10.

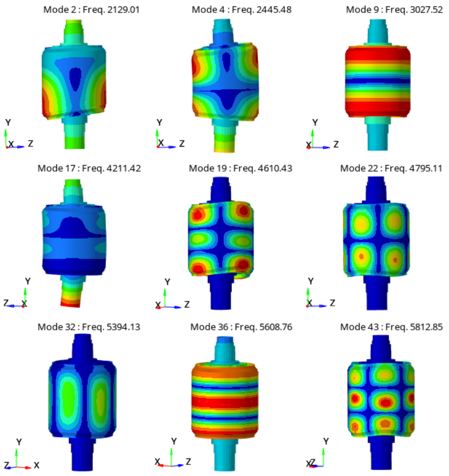

Fig. 10. Mode shapes of the K5 rotor model.

D. Model Updating of the K5 Rotor

Using the parameters of the Super-NE rotor stack for the simplified model proved to be a very rough approximation. Therefore, an optimization attempt using the direct approach is started. In this case, sensitivity analysis is not necessary because the parameters and the influence of the parameters are basically the same as for the Super-NE. The parameters to be adjusted are the stack material parameters, except for those similar to steel. The mode shapes used for optimization are listed in the table V.

EMA	NMA
Shear	Mode 2,3
Bending in Phase	Mode 4,5
Torsion 1	Mode 9
Bending out of Phase	Mode 17,18

TABLE V
PAIRING OF THE NUMERICAL MODAL ANALYSIS WITH THE EMA OF THE K5 ROTOR.

In this case, the optimization is performed by initializing the optimization with 10 latin hypercube samples generated directly by the optimization algorithm. Then 20 more Bayesian optimization steps are performed. A plot of the objective function is shown in figure 11. The plot shows that the objective function is random for the first 10 iterations, but then the algorithm converges quite quickly. The peaks in the 20 optimization steps are some attempts of the algorithm to find a lower minimum, but it becomes clear that the lowest minimum is found quite early.

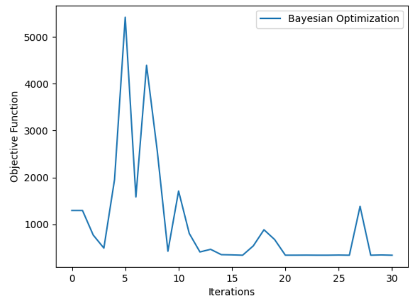

Fig. 11. Plot of the objective function for every iteration of the optimization process.

The results of the optimization process are listed in the table VI. In this case, the errors are all below 2 %. The initial numerical modal analysis is computed with the parameters of the simplified Super-NE model.

	Mode 1	Mode 2	Mode 3	Mode 4
EMA	2128	2469	3036	4147
Initial NMA	2080	2318	2722	4053
Error	2.3 %	6.2 %	10.3 %	2.3 %
Optimized NMA	2129	2445	3027	4211
Error	0.0 %	1.0 %	0.3 %	1.5 %

TABLE VI
EIGENFREQUENCIES OF THE MODEL UPDATE OF THE K5 ROTOR MODEL IN HZ AND THE ERRORS IN %.

The optimized parameters can be found in the table VII. It can be seen that the parameters change quite a lot, and it is surprising that parameter G31 decreased quite a lot where E3 and G23 increased.

Stack Material:	Value / N/mm^2
E1	200000
E2	200000
E3	1319
G12	72000
G23	2109
G31	291

TABLE VII
OPTIMIZED PARAMETER OF THE K5 ROTOR SIMPLIFIED MODEL.

In conclusion, the Super-NE parameters cannot be used for the K5 rotor, although they are a good first guess. This is probably due to the design changes made to the K5 rotor. The optimization workflow applied to the K5 rotor was very effective and yielded good results.

VII. RESULTS

In conclusion, it can be said that the created workflows have been successfully implemented on the different components tested. Starting with the sensitivity analysis script and workflow, it can be said that it is a very useful tool and helped a lot to reduce the number of parameters used for optimisation.

Regarding the optimisation with a surrogate model using linear regression, it can be concluded that the implementation for the complex Super-NE model worked perfectly. The model is trained using existing data from the sensitivity analysis, which saves a lot of time. In addition, the linear regression optimisation is useful when the limits of the sensitivity analysis are too narrow. The surrogate model is able to extrapolate the frequency data, so wider limits can be chosen for the optimization if needed.

Concerning the direct optimization workflow, it can be said that it worked well for the simpler models, but is not suitable for large simulation models. The efficiency of the Bayesian optimisation algorithm is good, as not so many samples are needed to find the global minimum. The disadvantage of this method is the need to find suitable bounds. However, looking at the results of the sensitivity analysis often helps to adjust the bounds to appropriate ones.

To get a measure of the success of the optimisation process, the average of the frequency error over all estimated eigenfrequencies of all components is calculated. The results are listed in the table VIII. Looking at the averaged errors, it can be said that all components are updated successfully, regardless of the optimization method. In general, a frequency error below 5 % is acceptable, and fortunately all errors in this experiments are within this range.

Component	Error
Stack	2.9 %
Super-NE Rotor Advanced Model	1.8 %
Super-NE Rotor Simple Model	2.4 %
K5 Rotor	0.7 %
PIT Rotor	1.2 %
Average	1.8 %

TABLE VIII
AVERAGED FREQUENCY ERROR FOR THE CORRESPONDING COMPONENTS MODEL.

VIII. SUMMARY AND OUTLOOK

In summary, the optimization workflow can save some time in finding the optimal model parameters, but it is not always easy to find the appropriate initial values and bounds. The advantage of the automated process is that once it is set up, one can run the script and do something else while the calculations are running. Sometimes setting up the optimization takes a bit of time and thought, but in general it takes less time than manually iterating over a certain component and changing the parameter each time. Especially when the number of unknown parameters is high.

It turns out that the optimization algorithm drastically reduces the errors for each tested component. However, one should not underestimate the power of manual model updating for simple models. A sensitivity analysis combined with manual tuning can be more efficient in some simple cases than setting up a whole optimization script where bounds and initial parameters are unknown. Nevertheless, a combination of some manual tuning to get the rough bounds and additionally updating the simulation with one of the suggested optimization methods will lead to the best results.

The next step will be to implement the optimized components in MASTA. The MASTA model could then be tested against a real acceleration measurement. One could then see if some peaks in the spectrum match better than before in the MASTA model.

To further improve the MASTA model, other components should be optimized, such as the stator or the stator in combination with the housing.

REFERENCES

[1] Bart Peeters, Herman Van der Auweraer, Patrick Guillaume and Jan Leuridan, *The PolyMAX frequency-domain method: a new standard for modal parameter estimation?* LMS International, Interleuvenlaan 68, B-3001 Leuven, Belgium; Department of Mechanical Engineering, Vrije Universiteit Brussel, Pleinlaan 2, B-1050 Brussel, Belgium, 2004.

[2] Franz-Josef Falkner, “Machine Dynamics,” Script, Februar 2021, Management Center Innsbruck.

[3] C. Schenk, “Mechanical Simulation,” Script, September 2023, Management Center Innsbruck.

[4] Miroslav Pastor, Michal Binda, Tomáš Hararik, *Modal Assurance Criterion*. Technical University of Košice, Faculty of Mechanical Engineering, Letná 9, 042 00 Košice, Slovak Republic, 2012.

[5] Statistics How To, “Latin Hypercube Sampling: Simple Definition,” [Online], Available: https://www.statisticshowto.com/latin-hypercube-sampling/.

[6] Newcastle University, “Strength of Correlation,” [Online], https://www.ncl.ac.uk/webtemplate/ask-assets/external/maths-resources/statistics/regression-and-correlation/strength-of-correlation.html.

[7] Su Guvenir Torun, Hakki M. Torun, Hendrik H.G. Hansen, Chris L. de Korte, Antonius F.W. van der Steen, Frank J.H. Gijsen, Ali C. Akyildiz, *Multicomponent material property characterization of atherosclerotic human carotid arteries through a Bayesian Optimization based inverse finite element approach*. Journal of the Mechanical Behavior of Biomedical Materials, 2021.

[8] PCB PIEZOTRONICS, “Hammer Datasheet,” [Online], Available: https://www.pcb.com/contentstore/MktgContent/LinkedDocuments/Force-Torque/TM-FRC-Impact-Hammers_Lowres.pdf.

[9] PCB PIEZOTRONICS, “Accelerometers Datasheet,” [Online], Available: https://www.pcb.com/contentstore/MktgContent/LinkedDocuments/Vibration/TM-VIB-356A43-44-45_lowres.pdf.

[10] Siemens, “Simcenter SCADAS Mobile and SCADAS Recorder,” [Online], Available: https://community.sw.siemens.com/s/article/simcenter-scadas-mobile-and-scadas-recorderref.

[11] Elia Osti, “Anleitung Simcenter Testlab Impact Testing,” Internal Manual, July 2023, GKN Driveline.

[12] Siemens, “Simcenter Community,” [Online], Available: https://community.sw.siemens.com/s/.

[13] Marc 2023.1, “Python: Tutorial and Reference Manual,” [Documentation], .

[14] Scikit-Learn, “LinearRegression,” [Online], https://scikit-learn.org/stable/modules/generated/sklearn.linear_model.LinearRegression.html.

Elias Gasser is a Master’s student of the Mechatronics program at MCI Innsbruck, Austria. He worked in the MCI’s structural dynamics laboratory as a research assistant and was also a tutor in the bachelor’s program for mechanics.