

# Automated Topology Optimization of Flexible Components in Hybrid Finite Element Multibody Systems using ADAMS/Flex and MSC.Construct

Häußler, P.; Emmrich, D.; Müller, O.; Ilzhöfer, B.; Nowicki, L.; Albers A.

## Abstract

In the past Finite Element Analysis (FEA) and Multibody System Simulation (MBS) were two isolated approaches in the field of mechanical system simulation. While multibody analysis codes focused on the nonlinear dynamics of entire systems of interconnected rigid bodies, FEA solvers were used to investigate the elastic/plastic behaviour of single deformable components. In recent years different software products e.g. ADAMS/Flex have been introduced into market, that utilize sub-structuring techniques to combine advantages from both FEA and MBS.

In the field of multibody system simulation the intention is the realistic representation of component level flexibility. For FEA purposes this method can be used to derive complex dynamic loading conditions for these flexible components which could not be done manually in general. Particularly in the field of finite element based structural optimization the formulation of realistic boundary- and loading-conditions is of vital interest as these significantly influence the final design.

Since structural optimization implies a change of the components shape (i.e. the mass distribution) during each iteration, the dynamic inertia loads will change respectively. In traditional topology optimization constant loads and boundary conditions are used<sup>1</sup>. Taking these iteration-dependent load changes into account implies a further benefit from a coupled approach.

This article describes the topology optimization of dynamically loaded finite element flexible components embedded in a multibody system by means of an automated coupling of ADAMS and the MSC.Construct optimization package. This approach, developed by the authors, has been used successfully in the past for the shape optimization of flexible components [10]. The necessary batch process for the whole procedure is briefly described and the requirements when used for topology optimization are explained. A hybrid multibody system is shown and results from topology optimization are discussed. The results encourage developers and offer new opportunities in the field of structural optimization as well as multibody system simulation.

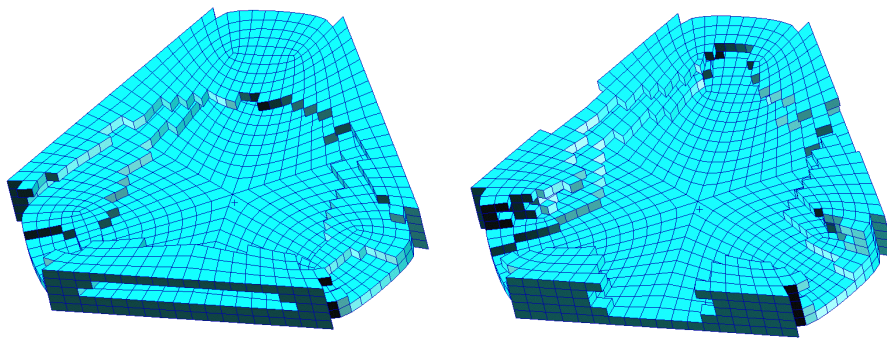


Figure 1: Comparison of results from a topology optimization. The component is a dynamically loaded plate with traditional static loads (left) and dynamically updated loads from ADAMS (right)

---

<sup>1</sup> Except in the case of simple body motion where accelerations can be formulated manually

# Introduction and Motivation

## Finite Element Analysis and Multibody System Simulation

In the past finite element analysis and multibody system simulation were two isolated tools in the field of computer aided numerical simulation. Both of them had (and still have) their specific fields of application and were developed to meet different needs.

The finite element analysis focuses on the simulation of component level elastic/plastic behaviour. Here, the main focus is the performance of a structure under certain applied loads. Typical results are stress/strain distributions, deflections or normal modes of vibration. Often linear models are used since deflections are small and within an elastic approximation. Non-linear transient analyses of FE models with time dependent loads are time consuming due to the generally large number of degrees of freedom.

Multibody system simulation focuses on the dynamic simulation of entire mechanical systems of bodies interconnected by various joints. Here, in general, the system equations of motion have a highly nonlinear nature. In most applications bodies are assumed to be rigid which leads to much fewer degrees of freedom compared to a typical finite element analysis. The main interest lies on the system's overall behaviour, not on the bodies themselves.

In recent years, new methodologies have been developed [3] [4] [16] [18] that combine aspects of both worlds, finite element analysis and multibody system simulation. By the use of sub-structuring techniques and component mode synthesis [3] a new approach for the representation of the elastic properties of a body within a multibody system has been integrated in several commercial MBS codes [13]. At first the aim here was to refine MBS models and to enhance MBS simulation capabilities. In the near past, also the finite element community became aware of certain benefits from this coupled approach.

## Topology Optimization

In this article we want to focus on structural optimization, namely finite element based topology optimization in the context of hybrid multibody systems, that means, systems with flexible bodies on a finite element basis.

Every finite element based model that is intended to be optimized in the sense of topology optimization needs a set of imposed loads and boundary conditions (BC). The optimization then leads to an improved model with respect to the loads and BC's defined before. The optimization itself is an iterative procedure where the bodies' geometrical structure is changed until an user defined objective is met (see figure 2).

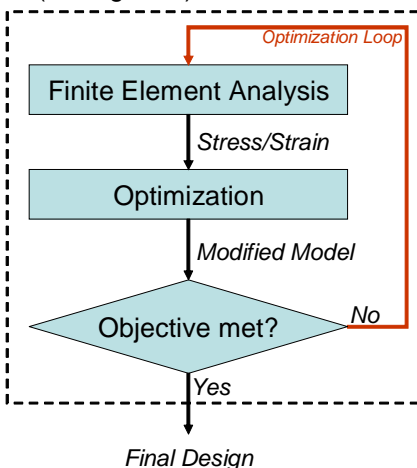


Figure 2: Basic optimization process

For statically loaded structural components, loads and BC's can be defined in a straight forward manner. Things get more complicated when dynamically loaded bodies are considered. For example fast moving machinery parts can be subject to complex inertia loads that can not be modelled by hand in general. Furthermore in traditional finite element based optimization the applied loads do not change during the iterations. Since the optimizer makes changes to the bodies spatial mass distribution, especially in the case of topology optimization the loads due to the bodies inertia will change during the optimization process. This change in the bodies mass distribution and inertia can lead to a completely different system overall behaviour which can then again effect the loads on the body.

To overcome the above mentioned difficulties a coupled approach has been developed by the authors with a hybrid multibody system for the automatic derivation of iteration dependent, complex loads on a flexible body in conjunction

with shape optimization in the past [10].

While the changes of the mass distribution, induced by a shape optimization, are small, the latter should be of significant importance when topology optimization is used. This article briefly describes the setup of this approach and presents results of an optimized plate under dynamic loads.

For the optimization the software CAOSS from the company FE-Design was used which is the optimization module behind MSC.Construct. The origin of CAOSS was at the Institute of Machine

## Review of Theory

### Representation of Flexible Bodies in Multibody Systems

In this section we present a brief introduction to the theory of flexible bodies in multibody systems. It is intended for readers who are not familiar with concepts like sub-structuring or modal superposition. A detailed discussion of these topics can be found in [3] [4] [18].

A body  $i$  in a multibody system is typically described using a body fixed coordinate system which we shall denote as BCS in what follows (see figure 3). The global position  $\vec{r}_P^i$  of a Point  $P$  on the body  $i$  at a time instant  $t$  can be described by the following common expression:

$$\vec{r}_P^i = \vec{R}^i + \mathbf{A}^i \vec{u}_P^i \quad (1)$$

Here  $\vec{R}^i$  denotes the origin of the body fixed coordinate system (represented in a global frame of reference) and  $\vec{u}_P^i$  is the position vector of point  $P$  given in the body fixed coordinate frame. The time dependent transformation matrix  $\mathbf{A}^i$  describes the transformation from the BCS to the global inertial system.

From (1) it is clear, that  $\vec{u}_P^i$  is a constant vector for a rigid body since the position of an arbitrary point  $P$  does not change with respect to a body fixed frame of reference.

If component level flexibility is introduced, the vector  $\vec{u}_P^i$  is no longer time invariant but rather depends on the actual deformation of the body:

$$\vec{r}_P^i = \vec{R}^i + \mathbf{A}^i \underbrace{(\vec{u}_0^i + \vec{u}_f^i)}_{\vec{u}_P^i} \quad (2)$$

Here  $\vec{u}_0^i$  is the (constant!) position of point  $P$  in an undeformed state of the body and  $\vec{u}_f^i$  is the time dependent vector of deformation.

### Finite Element Description

In this article we describe flexible bodies in terms of linear finite element models. In the preceding section, no description for the origin of the deformation vector  $\vec{u}_f^i$  was given. If we consider a finite element model we deal with a discrete structure with a certain number of nodes. These nodes have associated nodal masses<sup>2</sup>  $m_n$  and there is the finite element stiffness matrix made up out of the element stiffness matrices. This matrix describes the deformation of the body on a nodal level in the limits of small deformations that are in accordance with a linear model. The vector of deformation  $\vec{u}_f^i$  then describes the nodal displacement due to elastic deflection. (Note that we omit the identifier  $i$  which denotes the body  $i$  in what follows for simplicity). The deformation vector of node  $n$  can then be expressed in the following form:

$$\vec{u}_f^n = \mathbf{k}^{n-1} \vec{f}^n \quad (3)$$

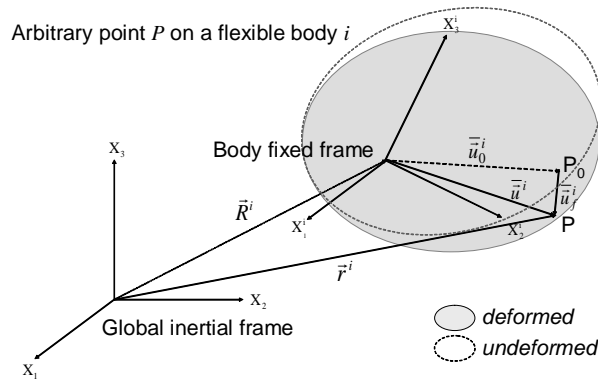


Figure 3: Location and orientation of points on a body

<sup>2</sup> A lumped mass formulation is assumed in this context

Here  $\mathbf{k}^{n-1}$  is the sub matrix of the inverted finite element stiffness matrix associated with node  $n$  and  $\vec{f}^n$  is the force, applied to the node. Equation (3) is a portion of the basic equation of linear static finite element analysis:  $\vec{F} = \mathbf{K}\vec{u}$ .

If we put this approach into the multibody system equations of motion it will result in a numerically inefficient model due to the very large number of degrees of freedom. Since today's typical FE models have up to millions of degrees of freedom a methodology for the reduction of these degrees of freedom has to be found.

### Modal Superposition

The formulation in (3) leads to a system with a large number of degrees of freedom. Due to the nature of small displacements that are within the limits of linear finite element analysis, a sophisticated and well used approach, the so called modal superposition can be utilized to dramatically reduce the number of degrees of freedom. The basic idea is to express the body deformation  $\vec{D}$  (that is, the nodal deformation vectors) as a weighted sum of a constant set of so called shape functions, or mode shapes  $\vec{\phi}$ :

$$\vec{D}(t) = \sum_{j=1}^{n_m} \vec{\phi}_j q_j(t) \quad (4)$$

Here the vector  $\vec{D}$  contains all the nodal deformation vectors  $\vec{u}_f^n$  and hence describes the overall deformation of the body at time  $t$ . A mode shape is a vector containing a complete displacement field of the body, that is deformation vectors for all nodes and is typically pre-computed during a finite element analysis. The important point here is the fact that the mode shapes  $\vec{\phi}$  are constant! For this reason, the complete time dependence is isolated in the so called "modal contribution factors"  $q_i(t)$ . Only these modal contribution factors remain as degrees of freedom to describe the bodies flexibility. It is clear, that (4) only represents an approximate description of the body deformation since a limited number  $n_c$  of shape functions is used. Therefore, the accuracy of the model significantly depends on the choice of shape functions [8] [17]. A widely used set of shape functions – also used in ADAMS/Flex – is the set suggested by CRAIG and BAMPTON in their 1969 AIAA paper [3]. This set consists of two basic types of mode shapes:

1. Normal modes of the constraint body (obtained from finite element analysis)
2. Static correction modes (obtained from finite element analysis)

While normal modes are common, the so called "static correction modes" may need some further explanation.

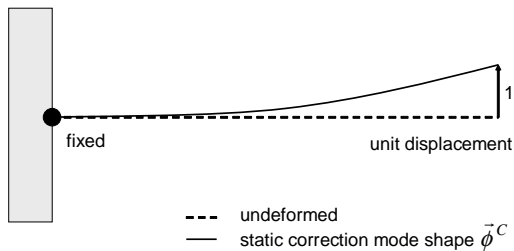


Figure 4: Static correction mode shape of a clamped beam

Assume a body that will be connected to other bodies in the MBS with joints at a certain set of nodes which we shall denote as interface nodes. A static correction mode shape  $\vec{\phi}_j^C$  is obtained by fixing all the interface nodes and applying a unit displacement/torque to one degree of freedom of only one of these interface nodes. The resulting deformation field is the associated static correction mode.

It is clear that these modes are very important since the interface nodes will be connected to joints in the MBS and therefore will be loaded with forces and torques. In a static MBS simulation these modes will primarily

contribute to the body deformation. If a dynamic simulation is performed, the normal modes will cover the dynamic part of the elastic deflections.

The bodies gross motion (i.e. the motion of the body fixed reference frame with respect to the global inertial frame) is described by adding six additional rigid body degrees of freedom to the flexible bodies set of generalized coordinates. This approach is often referred to as the "Floating Frame of Reference Method" [18].

The methodology of modal superposition leads to equations of motion with a significantly reduced number of degrees of freedom while preserving the full coupling between the rigid body motion and the bodies elastodynamics.

## Sub-Structuring/Component Mode Synthesis

The methods described above reduce the number of degrees of freedom in the system of equations that has to be solved for flexible bodies. A further improvement can be achieved by the use of sub-structuring techniques. In the methodology we presented so far there is still the complete finite element stiffness matrix  $\mathbf{K}$  contained. Since constraints (i.e. joints) are applied only to a certain set of interface nodes, the degrees of freedom can be partitioned into those of these interface or boundary nodes  $\bar{\bar{u}}_f^B$  and those of the interior nodes  $\bar{\bar{u}}_f^I$ . The basic equation of force balance can then be written:

$$\begin{Bmatrix} \bar{\bar{F}}^B \\ \bar{\bar{F}}^I \end{Bmatrix} = \begin{bmatrix} \mathbf{k}^{BB} & \mathbf{k}^{BI} \\ \mathbf{k}^{IB} & \mathbf{k}^{II} \end{bmatrix} \begin{Bmatrix} \bar{\bar{u}}_f^B \\ \bar{\bar{u}}_f^I \end{Bmatrix} \quad (5)$$

Recall that the static correction modes are the mode shapes for the interior degrees of freedom due to a unit deflection in each boundary degree of freedom. Therefore these constraint modes can be computed by setting the interior forces  $\bar{\bar{F}}^I$  in (5) to zero:

$$\begin{aligned} 0 &= \mathbf{k}^{IB} \bar{\bar{u}}_f^B + \mathbf{k}^{II} \bar{\bar{u}}_f^I \\ \bar{\bar{u}}_f^I &= \underbrace{-\mathbf{k}^{II^{-1}} \mathbf{k}^{IB}}_{\Phi^C} \bar{\bar{u}}_f^B \end{aligned} \quad (6)$$

Here the deflections of all the interior degrees of freedom are expressed by the exterior degrees of freedom by pre-multiplying  $\bar{\bar{u}}_f^B$  with the matrix of static correction modes  $\Phi^C$ . The physical coordinates  $\bar{\bar{u}}_f^I$  and  $\bar{\bar{u}}_f^B$  are then approximated by the weighted modal summation with the modal contribution factors  $q$ :

$$\begin{Bmatrix} \bar{\bar{u}}_f^B \\ \bar{\bar{u}}_f^I \end{Bmatrix} = \begin{bmatrix} \mathbf{I} & \mathbf{0} \\ \Phi^C & \Phi^N \end{bmatrix} \begin{Bmatrix} \bar{\bar{q}}^C \\ \bar{\bar{q}}^N \end{Bmatrix} \quad (7)$$

Using this modal superposition, a significantly simplified, generalized stiffness matrix  $\hat{\mathbf{K}}$  and mass matrix  $\hat{\mathbf{M}}$  can be derived to further improve the governing system of equations of motion.

A further discussion of these topics would lead to far within the framework of this article. For details, especially the implementation in ADAMS we refer to [3], [13] and [14].

## Topology Optimization

### Introduction

The aim of structural optimization is the optimal design of mechanical structures subject to certain boundary conditions to fulfill certain objectives as the maximization of the stiffness or the first natural frequency and others. Dependent on the nature of the design variables, it is possible to distinguish different fields of structural optimization. The following figure gives some examples for possible design variables.

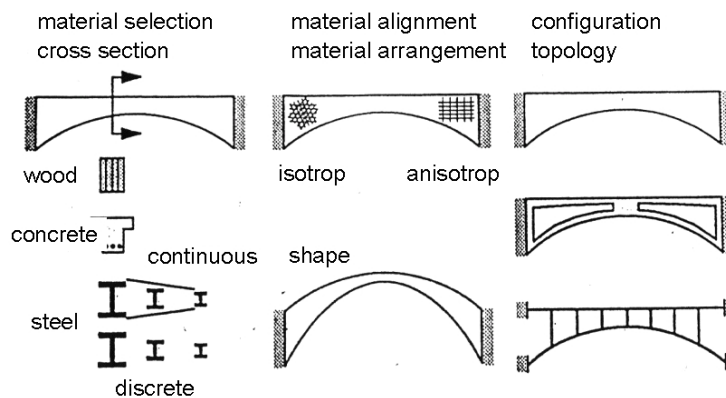


Figure 5: Fields of structural optimization [7]

Generally, the terms “sizing optimization”, “shape optimization” and “topology optimization” are used for classification.

The automated coupling of ADAMS and MSC.Construct for shape optimization was already addressed in [10]. This paper focuses on the automated coupling for topology optimization.

While sizing and shape optimization rely on an already existing initial design proposal, topology optimization tries to compute an optimum design of a structure in an available design space. The task is therefore to solve a material distribution problem.

The common approach for a optimization formulation is to define the design variables, an objective and the constraints.

### Design Variables for Topology Optimization

The following problem definition for topology optimization of an isotropic material is mainly based on a publication of BENDSØE and KIKUCHI [2].

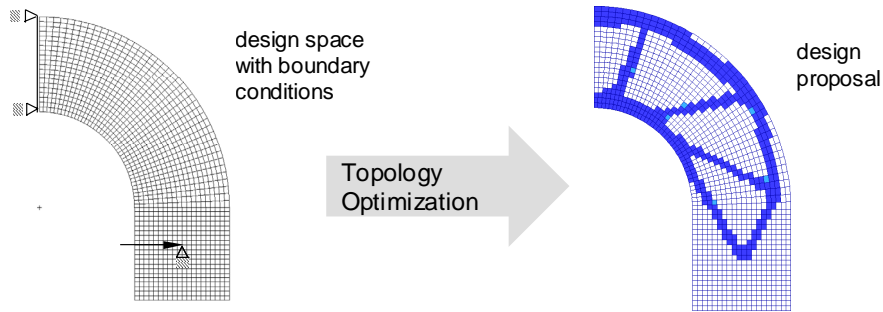


Figure 6: Topology optimization

The common approach is to start with a finite element model covering the complete available design space and the according boundary conditions. Then, the material property of each finite element is considered as a design variable. As final result, some of the elements are expected to have the properties of the material to be used, representing the structure, while the rest is expected to have zero density and Young’s modulus, in order to represent void space. Unfortunately, this integer optimization problem is not well posed, shows many local minima and is very costly to solve, especially since the number of design variables for topology optimization is usually very high. A workaround for this problem is to allow intermediate densities and Young’s moduli during the optimization process:

Assuming the resulting structure is to use 30% of the available space (to meet certain weight constraints), the initial density of all elements is set to 30% of the density of the massive material to be used. This results in an initial homogenous material distribution. For the according Young’s modulus of the elements, MLEJNEK [9] used the following relationship successfully:

$$E = \left( \frac{\rho}{\rho_0} \right)^k E_0 \quad (8)$$

where  $E_0$  and  $\rho_0$  are the properties of the massive material. For  $k$ , a value bigger than 1 is chosen. This way, intermediate densities are unfavorable. They are penalized due to their high cost of material with respect to their stiffness. (It should be mentioned here, that the exact value for  $k$  is not so important, as long as only static optimization neglecting weight are concerned. Of course, this is not the case for any dynamic or modal optimizations).

The design variable for each element for a topology optimization is now continuous:

$$\chi = \left( \frac{\rho}{\rho_0} \right) \quad (9)$$

Due to this definition, the result should mainly consist of elements with  $\chi = 0$  for the void space and elements with  $\chi = 1$  representing the structure itself.

One drawback of this discretized approach for topology optimization is the dependency of the final design on the chosen mesh. This leads to a limited “resolution” for the representation of the final

structure. In practice, this leads to a very large number of elements for the FE-models in order to keep the resolution high and each element introduces an additional design variable.

### Constraints

Typical constraints for a topology optimization are the available design space, usually taken into account by the limits of the FE-model. Additionally, often the fraction of the design space, allowed to be used by the final structure, is restricted. Rather than this, a limitation of the maximum allowed stress for each element seems to be a more reasonable constraint. However, amongst other difficulties, this produces as many constraints as design variables, which causes high computational costs. In some cases, it might also be required, to put limits on the deformation of the component. Side constraints, which are direct constraints on the design variables, are

$$\varepsilon < \chi \leq 1 \quad (10)$$

for all elements.  $\chi = 0$  is not permissible since it would cause singularities in the stiffness matrix of the FE-model. This can be prevented by setting the lower limit to a number very close to zero (e.g.  $\varepsilon = 10^{-6}$ ).

### Objective Functions

The range of the analysis types for topology optimization is mainly limited because of the difficulty or computational costs to obtain the sensitivities necessary for the optimization. State of the art is the application of linear static and modal analysis<sup>3</sup>. Therefore, the objective functions generally used are different stress hypotheses (e.g. Von Mises stress), the strain energy or natural frequencies.

### Optimization Strategy

The set up of the objective function, the design variables and the constraints are only one part of a complete optimization algorithm. Now, a optimizer using a certain strategy is necessary to compute a solution of the posed problem iteratively. Here two main different approaches can be distinguished: mathematical programming methods and optimum criteria methods.

Especially for topology optimization, the applied approach has to be selected, carefully, because of

- the large number of design variables (in practice, problems with 750.000 design variables have been optimized)
- computational costs of sensitivities for certain types of analyses and objective functions
- computational cost of each iteration.

So far, none of the both methods has proven to be overall better than the other one. Both have drawbacks and benefits.

MSC.Construct, the optimizer used for the coupled optimization introduced in this paper, is based on an optimum criteria method. This approach works very differently in comparison to the mathematical programming methods. Here, theses are formulated, which describe the properties, an optimum solution must have. A well-known and ascertained physical law is for instance the law of conservation of energy stating that a system always aims at the state of the minimum total energy.

For topology optimization, a possible thesis is, that a mass increase in regions under high stress and a mass reduction in lowly stressed regions leads to a homogenization of stresses of the component. This thesis can be used to implement a controller type optimizer which solves the above described problem [18].

---

<sup>3</sup> FE-Design has recently launched a program called "TOSCA", which can carry out topology optimization based on non linear analysis results of ABAQUS, a product of HKS.

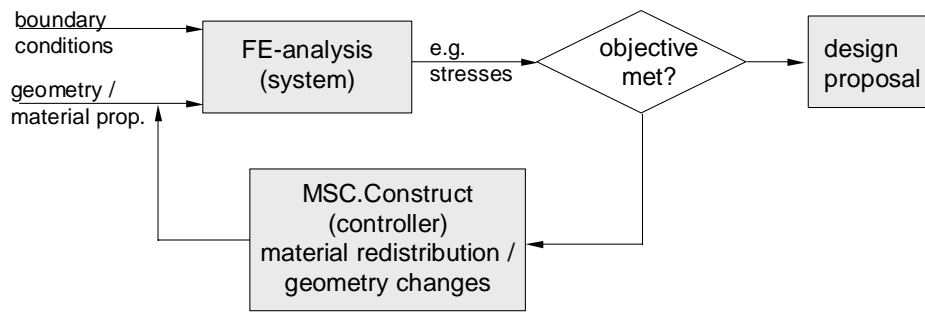


Figure 7: Optimizer as a “controller” of a component

A typical drawback of such a strategy is, that it is restricted to a certain class of problem, in this case to structural topology optimization. On the other hand, it has some very advantageous characteristics like:

- high speed of convergence
- almost independent of the starting values
- optimizer behaviour is easily observed and understood
- can easily deal with a high number of design variables
- no dependence on sensitivities
- an arbitrary FE-Analysis packet can be used, provided the necessary interfaces are available
- very stable optimization process

## Coupled Approach

### Topology Optimization on a System Level

In [10], a coupled setup was described in the context of shape optimization with MSC.Construct. One of the main differences of these two types of structural optimization is their effect on the bodies mass distribution and total mass and therefore the bodies inertia. Shape optimization makes only minor changes to the components surface, the topology remains untouched. Therefore the mass matrix and the inertia tensor are only slightly modified. For this reason also the body’s normal modes of vibration are not significantly influenced. This means that there are no major changes in the dynamic behaviour of the body.

A topology optimization starts typically with a design space that is completely filled with material. For MSC.Construct, the material’s density and Young’s modulus are dependent on the chosen objective: for the maximization of a certain natural frequency, where a modal analysis is required, the optimizer starts with an intermediate mass and density, as introduced in the review of the theory. If the optimization is based on results of a static FE-analysis, as it is here, the mass is not only redistributed during the optimization, but also reduced from a completely filled design space until the mass constraint is fulfilled. Both, the redistribution of mass as well as its reduction can have a serious impact on the dynamic properties of the body and possibly also on the system’s overall behaviour.

Furthermore, MSC.Construct changes the relationship between the stiffness and the density, what can be interpreted as an increase of the exponent  $k$  of equation (8). This way, MSC.Construct “forces” a faster convergence. This characteristic can be influenced by an optimization parameter of the software. Increasing  $k$  usually leads to a faster convergence, while decreasing this parameter stabilizes the optimization on the expense of computational effort.

The idea of the coupling is now, to adjust the boundary conditions and loads due to the changed component’s properties after each iteration. If the adjustments are small enough and the optimization process is very stable, then for each iteration, the boundary conditions can be considered as “quasi-static”. The optimization should still converge to a design proposal, which is now considering both, the changed component and system behaviour.

Two situations can be imagined from a mechanical point of view, where a convergence to a reasonable solution will not be given. Assuming that the applied thesis for the controller assumes, that an increase in stiffness and therefore mass leads to a decrease of the objective like stresses and vice versa. If the increase of mass leads to an increase of inertial forces, which again will increase the stresses, then the optimization might not converge to a lightweight solution. The other possibility is, that if a decrease of mass and stiffness reduces the objective, then the optimizer would try to “delete” the component completely in order to minimize the objective. However, this situation is by far less critical, as long as the component has two joints and transmits forces which are not purely a function of its own inertia. This force will ensure a minimum stress within the component and sooner or later the



optimizer should face a “normal” response of the structure. A lower mass constraint and therefore a lighter component design might be possible.

Of course, all these assumptions have to be subject to investigations.

## Software Setup and Dataflow

A “traditional” structural optimization with MSC.Construct is an iterative procedure basically using MSC.Nastran for the FE-Analysis and the optimization module(s) of MSC.Construct for the setup of the optimization problem and as optimizer. The complete batch-process is controlled by MSC.Construct (figure 8):

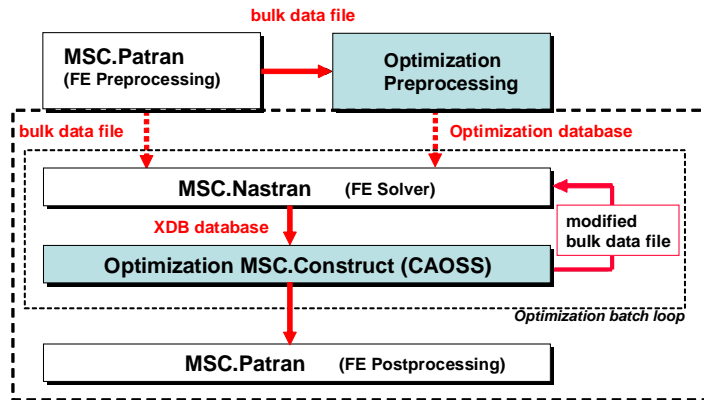


Figure 8: „Traditional“ optimization process with MSC.Construct

An ordinary finite element model containing loads and boundary conditions serves as input for the optimization preprocessor and later on for the finite element solver MSC.Nastran. The results from this finite element analysis, typically von Mises stresses are read by the optimization module in a next step. Here, the finite element structure is modified and a new input file for MSC.Nastran is written for the next iteration. Note that the loads and boundary conditions remain unchanged<sup>4</sup>. The process stops after the declared objectives are met.

In order to automatically generate dynamic loads and to cover the effect of the bodies changing inertia (including the feedback on the overall system) ADAMS is integrated in the process (figure 9):

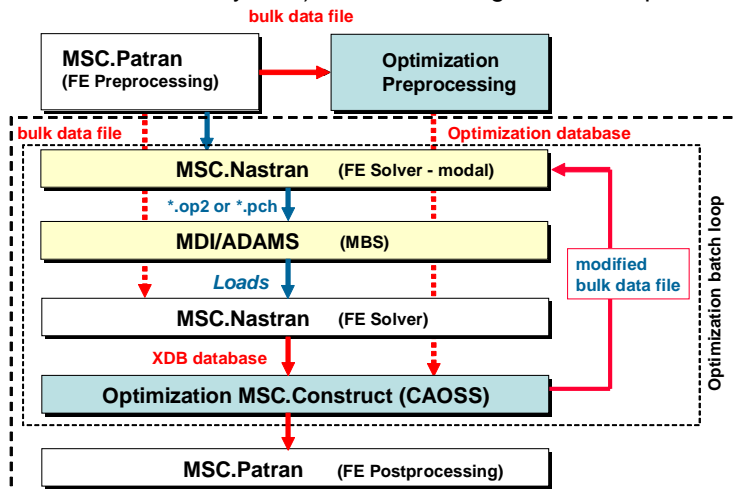


Figure 9: Coupled optimization process with ADAMS and MSC.Construct

Here, the finite element model does not contain loads and boundary conditions. They are provided by ADAMS after a dynamic simulation of the entire multibody system. This simulation is carried during each optimization iteration and new loads are being derived. Therefore the effect on the system (and body loads) due to the modifications of the optimizer are taken into account.

For the determination of loads in ADAMS the build in features (FEA loads export) are used. Since the optimization is an iterative procedure this export had to be automated for the use in a batch framework. For a more detailed description of the whole process we refer to [10].

<sup>4</sup> Except manually defined combinations of rotations and/or linear accelerations which lead to mass matrix dependent nodal forces. But as pointed out in the introduction these can not be easily derived for complex body motion.

## A simple Tripod Model

In order to study the effects explained in the prior sections a simple model of a tripod (figure 10) was set up. This model contains a flexible plate (tool holder) that is driven by three rods. While the rods are rigid bodies the plate is a finite element model containing three interface nodes to be attached to the ends of the rods with spherical joints. At the opposite end the two blue rods are fixed to ground while the red rod is linked to an additional (yellow) rod by a revolute joint. This additional rod is constrained with a translational joint to move in vertical direction (with respect to the global frame of reference) and serves as driver.

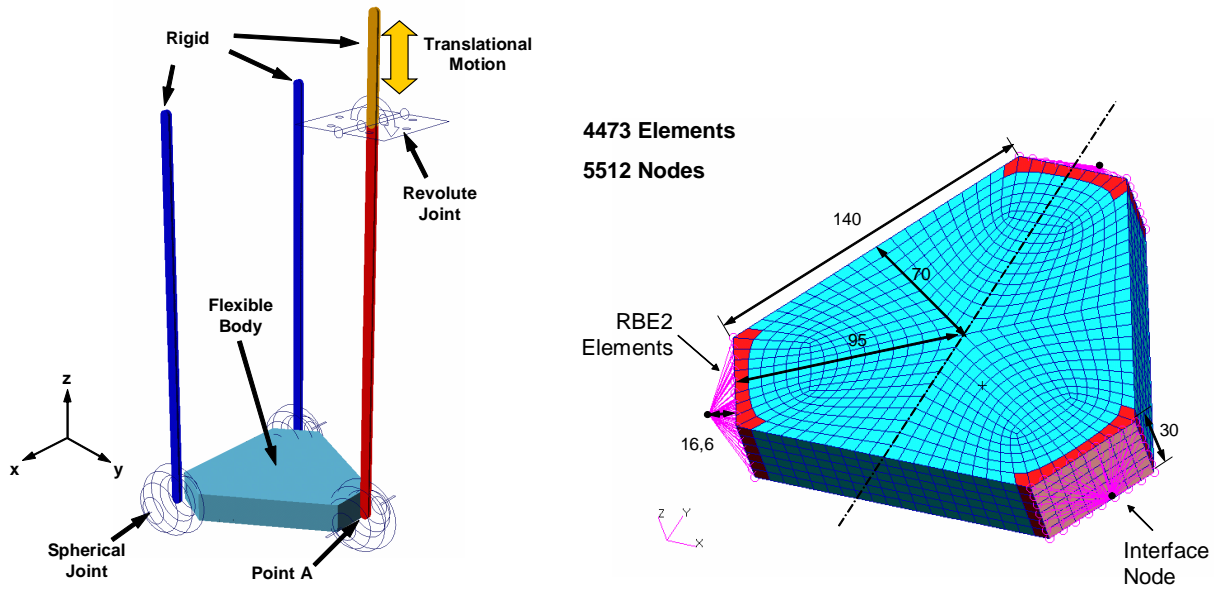


Figure 10: ADAMS model (left) and FE model with interface nodes and RBE2 elements (right)

The finite element model of the plate is made up of 4473 hexaeder elements (MSC.Nastran CHEXA) and contains three rigid bar elements (MSC.Nastran RBE2) that respectively connect one node in front of the center of each of the small sides of the plate to all surface nodes. This is done to distribute the loads on the surface for an accurate force application. Recall that these interface nodes are the boundary degrees of freedom  $\bar{u}_f^B$ . The red shaded elements (see figure 10, right) make up the group of so called “frozen elements”. The elements in this group will remain unchanged by the optimizer. This is necessary to avoid a disadvantageous force application. All the blue shaded elements form the design elements that can be subject to modifications due to the optimization module.

## Simulation

### ADAMS Dynamic Simulation

In the investigations presented here a simple model was setup in order to be able to reproduce results with analytical methods if necessary. Furthermore the resulting rotational motion could be modeled “by hand” with MSC.Nastran for comparisons.

As excitation a simple sinusoidal forced translational motion was applied to the yellow rod in the global  $z$  direction:

$$u_z(t) = A \sin(\omega t) \quad (11)$$

In order to investigate the dependency of the loads on the angular frequency  $\omega$ , two different simulations have been carried out:

	$A$	$\omega$	$\hat{u}_z$
Case 1 (1 Hz)	100 mm	$6.28 \frac{1}{s}$	0.4 g
Case 2 (3 Hz)	100 mm	$18.8 \frac{1}{s}$	3.6 g

Table 1: Parameters for the two Dynamic Simulations

Here,  $\hat{u}_z$  denotes the maximum acceleration of point A (see figure 10, left) in the global  $z$  direction and  $g$  is the gravity constant of the earth. The selected amplitude  $A$  leads to a tilt of the plate around the global  $x$  direction with a magnitude of approximately  $\pm 45$  degree.

The following diagram shows the  $z$  and  $y$  components of the forces arising in the spherical joint at point A:

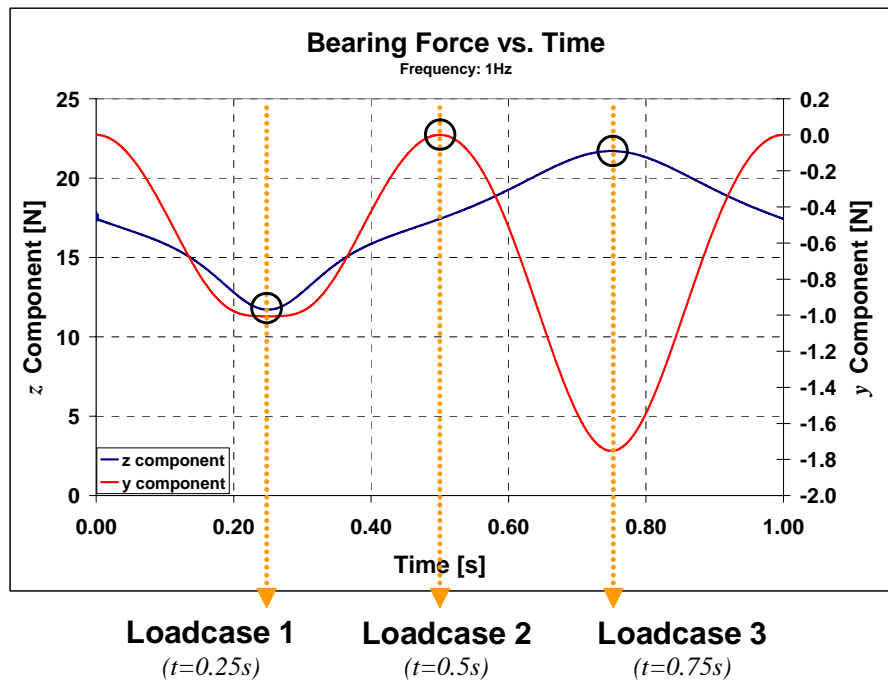


Figure 11: Bearing forces at a 1Hz excitation, measured at the spherical joint at point A (see figure 10, left)

From this diagram three time steps with characteristic values for the two force components were chosen (see figure 11):

Time Step 1	Minimum in the $z$ component
Time Step 2	Zero crossing in the $y$ component pure $z$ loading
Time Step 3	Maximum in the $z$ component

Table 2: Time steps with characteristic values in the  $y$  and  $z$  components of the bearing force

For the three time steps shown above, loads were exported by ADAMS to be applied to the plate for the finite element analysis.

### Topology Optimization Model

For the optimization MSC.Construct V.4.0 was used. As objective function the maximization of the global stiffness was formulated together with a constraint on the mass which had to be reduced by 50%. The maximization of the stiffness corresponds to a homogenization of the von Mises stresses.

The finite element analyses with the load cases generated by ADAMS were carried out with MSC.Nastran version 70.7.

In order to compare the coupled approach to the “traditional” technique and to study the influence of the load update also optimizations with constant loads were carried out. For this, the loads from one ADAMS simulation with the original plate model were used and afterwards the process shown in figure 8 was applied, keeping the initial loads.

## Results

### Bearing Forces at Point A

The following two diagrams show the  $y$  and  $z$  component of the bearing forces in the spherical joint at point A before and after the optimization for both frequencies (figure 12):

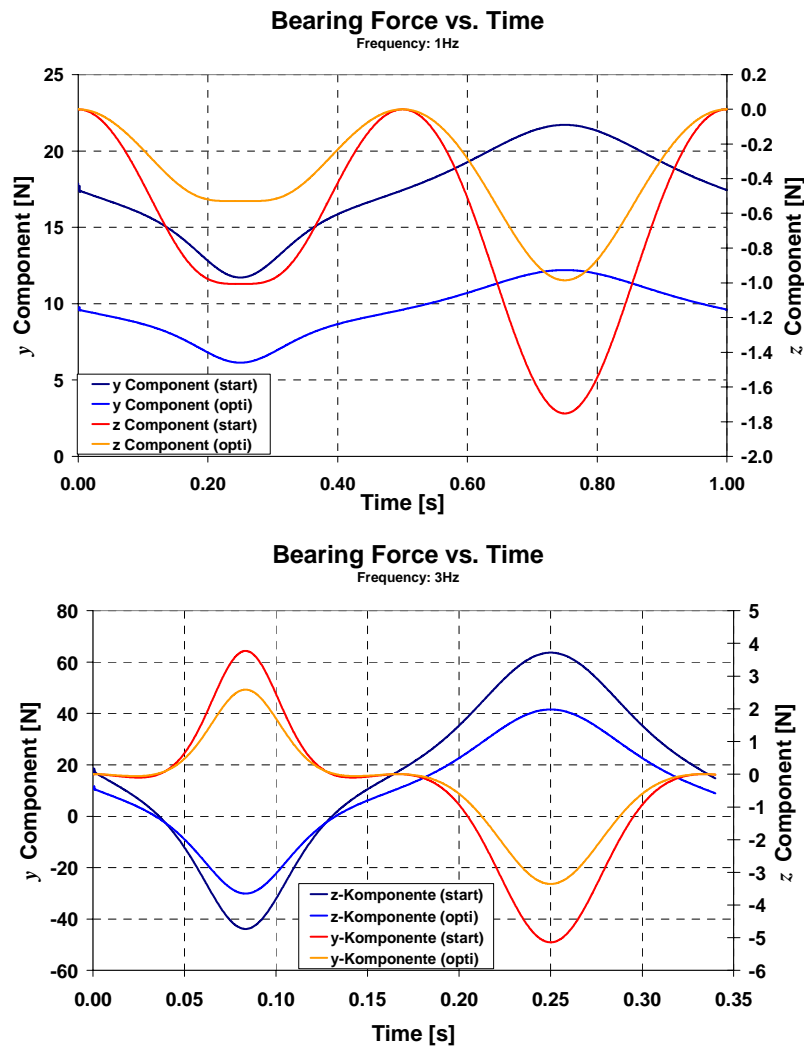


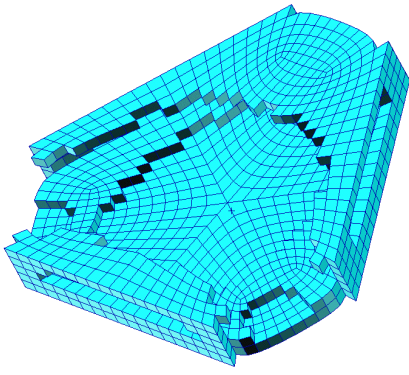
Figure 12: Bearing force components before and after the optimization for 1 Hz excitation (top) and 3 Hz (bottom)

In figure 12, the quantitative difference in the force components for the 1 Hz and 3 Hz excitations can be seen clearly. The  $z$  component additionally changes its slope significantly in the first half of the simulated time frame. The reason for this effect is the plate's inertia, that takes more influence on the forces in the 3 Hz simulation due to the higher accelerations. This example demonstrates the force's dependence on the effective accelerations. Unfortunately, the effects of mass reduction and mass redistribution can not be distinguished, since both happen at the same time.

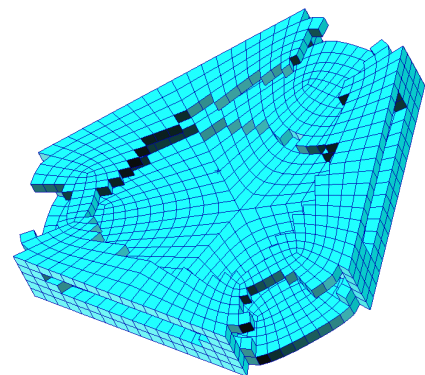
A further result here is the improvement of the overall system which is indicated by the lowered bearing forces of the optimized model. Furthermore this shows clearly the change in the loads during the iterations due to the modifications of the plates total mass and inertia. The coupled approach with ADAMS in the loop can take these effects into account.

### Topology Design Proposals

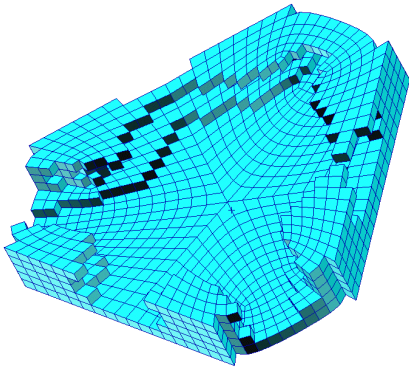
The following series of images show the results from the topology optimization of the coupled model with ADAMS in the loop (i.e. updated loads). In order to be able to see the changes inside the plate, all images are shown with one element layer removed (from top and bottom respectively).



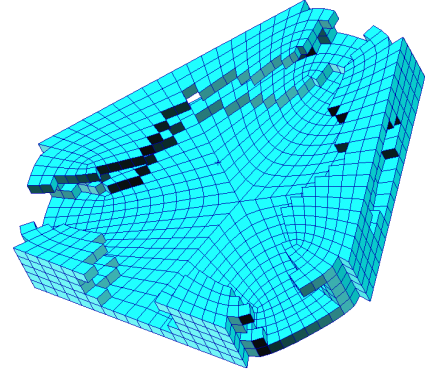
*“traditional”*: top view, upper element layer removed



bottom view, upper element layer removed

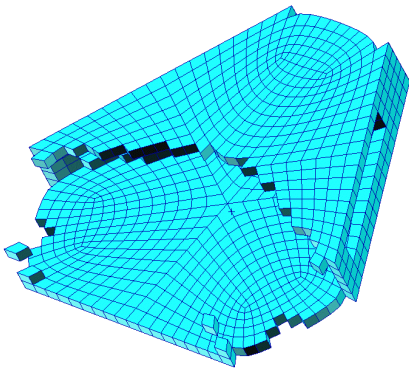


*coupled*: top view, upper element layer removed

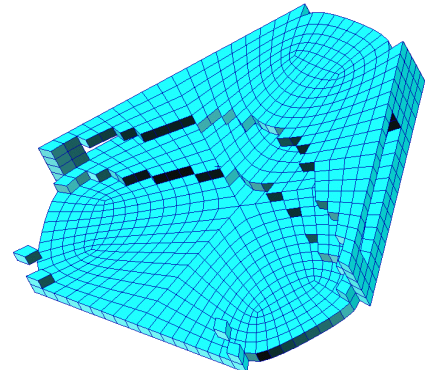


bottom view, upper element layer removed

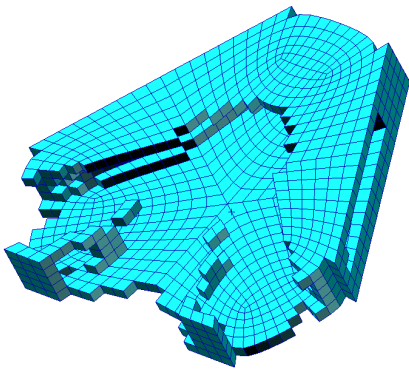
Figure 13: Topology optimization results for 1 Hz



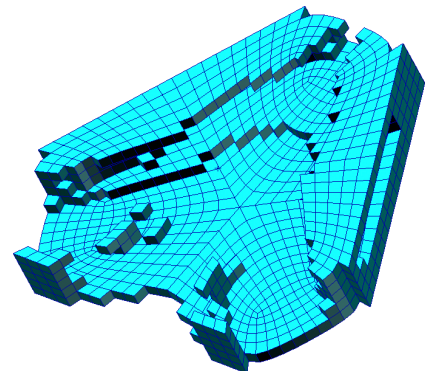
*“traditional”*: top view, upper element layer removed



bottom view, upper element layer removed



*coupled*: top view, upper element layer removed



bottom view, upper element layer removed

Figure 14: Topology optimization results for 3 Hz

Comparing the design proposals of the two optimizations based on the excitation with 1Hz, the topological differences can already be spotted easily. Changing the frequency to 3Hz and therefore increasing the acceleration shows, that also the topological differences increase. This is a clear indication of the also increasing influence of the inertia forces.

A close look at the design proposals shows small deviations of an axis symmetric solution that could have been expected. The reason for this are numerical round off errors in conjunction with the coarse finite element mesh of the plate. Trials have also shown, that the symmetry of the designs can be improved if the forced convergence is slowed down.

To be able to compare the occurring stresses of the design proposals, an additional ADAMS run with the results of the traditional optimization has been carried out to obtain the “real” loads on these components. With these loads a final FE-Analysis has been computed. The results are shown in *figure 15*. The traditional optimization shows a more homogeneous stress distribution which comes closer to a “fully stressed design” as the coupled approach. However, for large regions the stresses of the coupled approach are significantly lower.

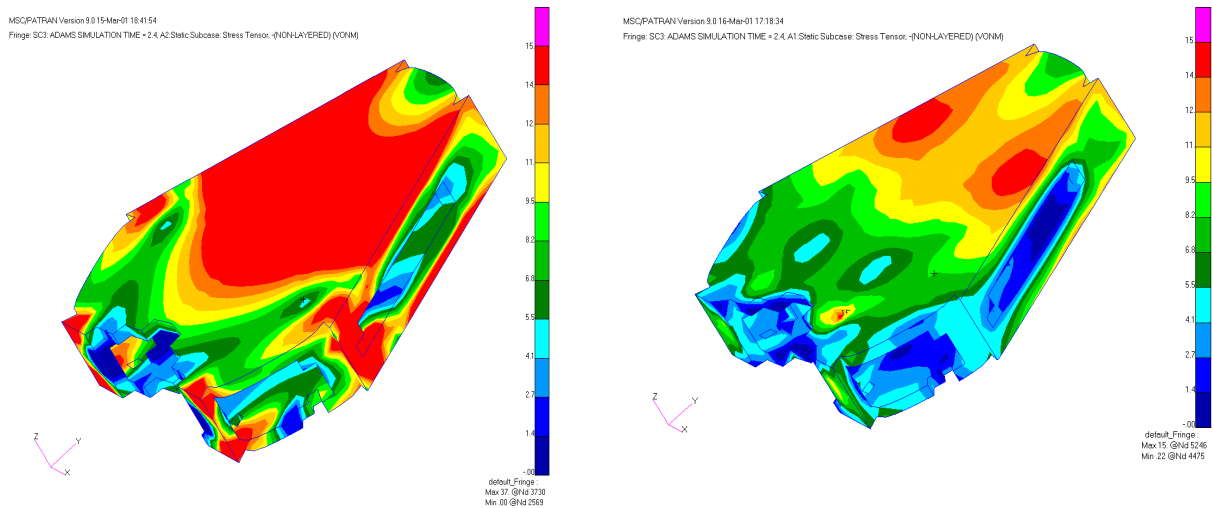


Figure 15: Von Mises stress distribution of the final design proposal of the traditional optimization (left) and the coupled optimization (right) for 3 Hz, load case 3

Also, the maximum occurring stress has been reduced by more the 50% compared to the stress maximum of the original model.

Another consequence of the coupled optimization is the increase of the first natural frequency for 1 and 3 Hz from 2 to 2.7 kHz and 1.8 to 2.6 kHz respectively. For many application, this can reduce disturbing vibrations within mechanical systems (cp. *figure 16*).

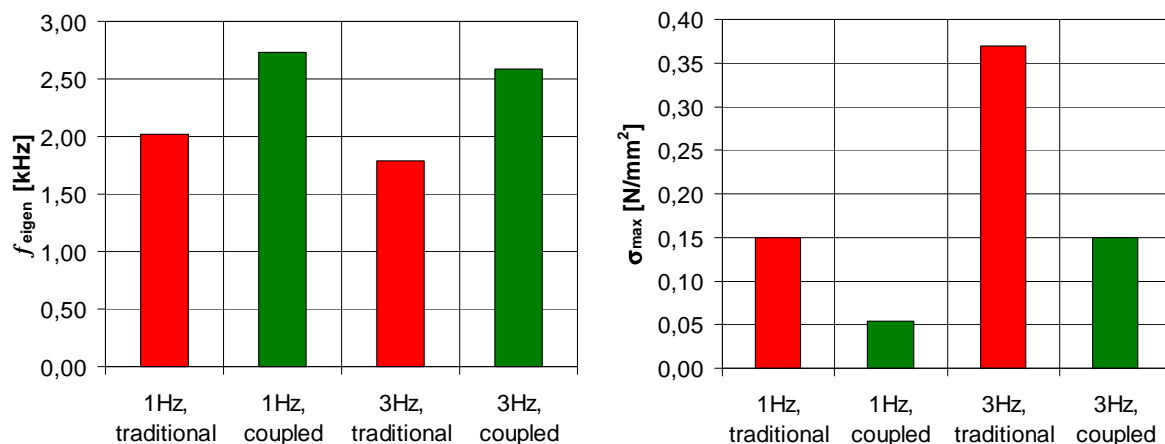


Figure 16: First natural frequencies and maximum von Mises stresses of the final design proposals. Investigations of the intermediate iterations of the optimization process showed that the applied method was robust and the optimizer converged reliable. Further investigations on complex models with more demanding dynamic properties will have to be carried out in order to validate the reliability of the method in general and its limits.

## **Conclusions**

In this article a brief review of the theory of hybrid multibody systems and finite element based topology optimization was presented. A methodology and implementation for a coupled optimization setup containing ADAMS, MSC.Construct and MSC.Nastran were explained and the benefits from such a setup were discussed.

The tripod example shows, that for a topology optimization of accelerated components within a mechanical system, it is important to consider the changes of the boundary conditions and system behaviour during the optimization since these are subject to significant changes. This can be reached by the introduction of a MBS simulation into the optimization loop to update the loads for the static FE analysis after each iteration. The resulting optimization process has computed a design proposal with significantly improved mechanical properties in comparison to the traditional optimization with constant boundary conditions within the framework of the example presented here.

The reliability and enhancement of the method as well as the convergence behaviour of the optimizer will be subject of further investigations.

## **Acknowledgements**

These investigations are part of the priority program “machine tools using parallel kinematics” funded by the DFG (Deutsche Forschungsgemeinschaft).

## References

- [1] **Allinger, P.; Bakhtiary, N. et al.**, *A New Approach for Sizing, Shape and Topology Optimization*, SAE International, International Congress & Exposition 1996, February 26-29, Detroit, Michigan, SAE Technical Paper Series No. 960814.
- [2] **Bendsøe, M.P., Kikuchi, N.**, *Generating optimal topologies in structural design using a homogenisation method*, Computer Methods in Applied Mechanics and Engineering, Vol.71, pp. 197-224, 1988
- [3] **Craig, R. R.; Bampton, M. C. C.**, *Coupling of Substructures for Dynamic Analyses*, AIAA Journal Vol. 6, No. 7, 1968, pp. 1313 ff.
- [4] **Geradin, M.; Cardona, A.**, *Flexible Multibody Dynamics*, John Wiley and Sons, 2000
- [5] **Hurty, W. C.**, *Dynamic Analysis of Structural Systems Using Component Modes*, AIAA Journal Vol. 3, No. 4, 1965, pp. 678 ff.
- [6] **Ilzhöfer, B.; Müller, O.; Häußler, P.; Allinger, P.**, *Shape Optimization Based on Parameters from Life Time Prediction*, NAFEMS-Seminar: Betriebsfestigkeit, Lebensdauer, 8.-9. November 2000, Wiesbaden.
- [7] **Kimmich, S.**, *Strukturoptimierung und Sensibilitätsanalyse mit finiten Elementen*, Dissertation. Universität Stuttgart, 1990
- [8] **Melzer, F.**, *Symbolisch-numerische Modellbildung elastischer Mehrkörpersysteme mit Anwendung auf rechnerische Lebensdauervorhersagen*, VDI Fortschritt-Berichte Reihe 20 Nr. 139, 1994
- [9] **Mlejnek, H.P.**, *Some aspects of the genesis of structures*, Structural Optimization Vol. 5, 64-69, 1992
- [10] **Müller, O.; Häußler, P. et al.**, *Automated Coupling of MDI/ADAMS and MSC.Construct for the Topology and Shape Optimization of Flexible Mechanical Systems*, 1999 International ADAMS Users' Conference. November 17-19, 1999. Berlin, Germany.
- [11] **Müller, O.; Allinger, P. et al.**, *Topology Optimization of Large Real World Structures*, NAFEMS World Congress '99 - Newport, Rhode Island, USA, April 25-28, Proceedings ISBN 1-874376-25-5, Volume 2, pp 827-839, 1999.
- [12] **Müller, O.**, *MSC/CONSTRUCT - Topology and Shape Optimization of Large Real World Structures Using the Distributed Parallel MSC/NASTRAN*, MSC Users' Conferences in Paris and Birmingham, June 15-18, 1998.
- [13] **Óttarsson, G.**, *Modal Flexibility Implementation in ADAMS/Flex*
- [14] **Óttarsson, G.; Moore, G.; Minen, D.**, *MDI/ADAMS-MSC/NASTRAN Integration Using Component Mode Synthesis*, Americas User's Conference, MSC.Software, 1998
- [15] **Sauter, J.; Mulfinger, F.; Müller, O.**, *Neue Entwicklungen im Bereich der Gestalt- und Topologieoptimierung*, ANSYS-USERS-MEETING, Tagungsband (23 Seiten), Arolsen, Oktober 1992.
- [16] **Schwertassek R.; Wallrapp O.**, *Dynamik flexibler Mehrkörpersysteme*, Vieweg & Sohn Verlagsgesellschaft, 1999
- [17] **Schwertassek R.; Wallrapp O.; Shabana, A.**, *Flexible Multibody Simulation and Choice of Shape Functions*, Nonlinear Dynamics Vol. 20, pp. 361-380, 1999
- [18] **Shabana, A. A.**, *Dynamics of Multibody Systems*, Cambridge University Press, 1998

### Contact

Dipl.-Phys. Pascal Häußler  
Institute of Machine Design  
University of Karlsruhe  
Kaiserstraße 12

76131 Karlsruhe  
Germany

Email: [cae@mkl.uni-karlsruhe.de](mailto:cae@mkl.uni-karlsruhe.de)  
Internet: <http://www.mkl.uni-karlsruhe.de/>