Automated Coupling of MDI/ADAMS and MSC.CONSTRUCT for the Topology and Shape Optimization of Flexible Mechanical Systems

Müller, O.; Häußler, P.; Lux, R.; Ilzhöfer, B.; Albers, A.

Abstract

Today's implementations of structural optimization lack the capability to consider load changes caused by the structural optimization itself. The loads and boundary conditions maintain the same during the entire optimization.

This article describes a new approach of an automated coupling of ADAMS and the MSC.CONSTRUCT structural optimization package. A batch-oriented closed-loop scheme was set up which allows ADAMS system-simulation with the body-shapes determined by the optimization and loads export during every iteration of the optimization procedure. After geometry modifications by MSC.CONSTRUCT the new body shape is imported into the ADAMS model for a new system simulation.

This approach enables the automated determination of complex loading-conditions for bodies in mechanical systems and the use of dynamically updated, i.e. iteration-dependent boundary-conditions for the structural optimization. When the dynamic analysis of ADAMS is used, even complex loads due to rotation and acceleration which depend on the mass distribution can be considered within the optimization.

This new and batch-oriented process is explained in this article and first optimization results will be presented. The results encourage further developments and offer major new opportunities for both the users of MBS as well as the optimization community.

1 Introduction

In the last years both, Finite Element Analysis (FEA) and Multi Body System Simulation (MBS) have evolved to powerful tools supporting engineers in various fields of product development and research. In addition programs for structural optimization are available from many vendors of FEA packages. With the continuous increasing computational capabilities these optimization techniques become more and more important for the effective development of competitive products. In this field the authors contribute to the development of the optimization code CAOSS (Computer Aided Optimization System Sauter) which is behind MSC.CONSTRUCT.

The formulation of realistic boundary and loading-conditions is of vital interest for structural optimizations as these give the physical structural loading and therefore determine the best shape. To find the correct loadings can be a challenging task when dealing with complex dynamical systems. Since mechanical system simulation programs, integrating flexible bodies, came to market, their use for determining loading-condition provide new possibilities for both FEA and structural optimization. MDI offers the ADAMS/Flex package which allows the integration of flexible bodies in mechanical systems, e.g. by interfacing with MSC.NASTRAN.

2 Basic Methods

Within this work a couple of standard simulation tools were used: Finite Element Analysis to describe the components flexibility and other FEA results, MBS to simulate the mechanical system behavior, incorporating the flexible component. Finally the optimization software which modifies the component shape and therefore modifies the mechanical behavior which is analyzed in both, the FEA and the MBS software.

Following some notes which should help the reader to get some insights into the wide field of structural optimization.

2.1 Structural Optimization

The field of structural optimization techniques can be seen in different ways depending on the background of the reader. In general there are three views: The one is separating the structural optimization types from the application field where it might be used for: Then we speak from sizing, shape or topology optimization. This is visualized in Figure 1.
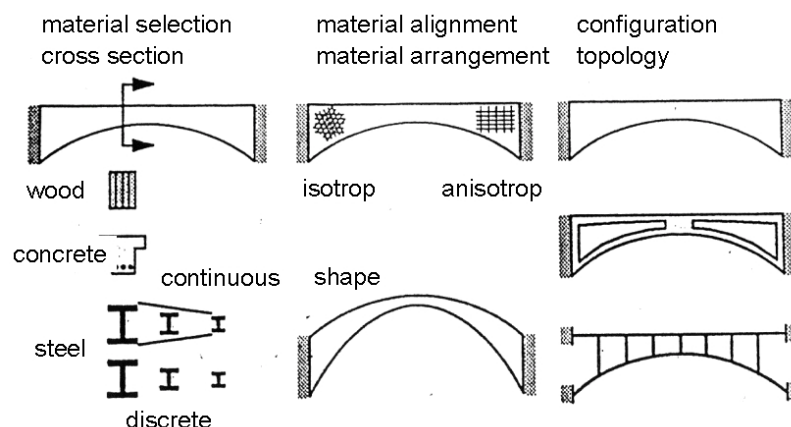

Figure 1: Optimization Types

The second one is emphasizing the difference among the approaches solving the optimization problem: mathematical approaches or optimum criteria approaches. The third one is separating parametric from nonparametric optimization. All of the above introduced views may be mixed in reality.

For the work described in this article the FEA based optimization package MSC.CONSTRUCT is used. With this package both, parameter free shape and topology optimization is possible [1] [3] based on optimum criteria. The authors contribute to the development of CAOSS which is the core behind MSC.CONSTRUCT.

2.1.1 Classification by Application

The following gives an overview of the main differences among sizing, shape and topology optimization. Both the numerical and the user-specific characteristics are discussed shortly.

2.1.1.1 Sizing Optimization

For optimization purposes using „Sizing Variables" i.e. the cross section and thickness of finite elements, many mathematical programming approaches were tested and implemented into finite element programs (e.g. MSC/NASTRAN, PERMAS, COSMOS/M) and special optimization programs (e.g. MBB-LAGRANGE, STARS, ADS). During the optimization, the element properties are modified on the FE level. Due to the easy calculation of the sensitivities for sizing optimization purposes even realistic problems can be handled. Today these approaches can be considered as state-of-the-art.

2.1.1.2 Shape Optimization

Compared with the sizing optimization the shape optimization is more complex. For the shape optimization two approaches are used:

Shape optimization based on FE models

The coordinates of the surface nodes are regarded as design variables which will be modified during the optimization. This usually leads to a large number of design variables which might cause considerable mathematical difficulties. Using suitable couplings of node displacements to define basis vectors, complex geometry changes can be described in the Solution 200 of MSC/NASTRAN with only few design variables.

Shape optimization based on geometry models

Using the shape optimization method based on geometry models, the linkage of an FE model and a geometry model is maintained. As in this case the parameters of the geometry model are the design variables, the geometry model has to be fully parametric. Therefore the use of an efficient solid modeler is necessary. Each parameter modification of the geometry model also results in changes of the FE model. Within each optimization loop the entire FE model has to be set up anew according to the modifications of the geometry model parameters considering the boundary conditions. This method is used e.g. in the programs ANSYS, COSMOS/M and IDEAS. Many interfaces are put on the integration of an FE solver and a CAD system (e.g. Pro/ENGINEER). The selection of the design variables is left to the user. In general they differ due to his experiences and creativity. The results of the optimization essentially depend on the number and selection of the design variables. If free form surfaces are allowed, the selection of the design variables is very difficult. Using mathematical programming methods, the number of design variables must not be to large too, because this causes the numerical efforts to increase drastically.

The main difficulty with shape optimization is to transfer the surface changes to the FE mesh. Most programs avoid this transfer by an automatic remeshing in each optimization loop. Hence the original element topology (meshing) is destroyed and often models with only tetrahedral elements are created. Only few programs are capable of such a transfer in a way that the mesh modification is analyzed starting from the modification of the surface while maintaining the element topology. To avoid the difficulties with remeshing some programs use p-elements for the shape optimization. Still there is the problem of selecting suitable design variables. For large models the numerical efforts are extremely high.

2.1.1.3 Topology Optimization

Both for sizing and shape optimization a first design proposal, which is used as the start design, exists. The objective of general structural optimization methods is to compute even this first design proposal. Therefore an area (2D or 3D) with a homogeneous material distribution is used. Subsequently the functionally required boundary conditions (e.g. node constraints, nodal loads) are applied. The efforts for the modeling and preparation is extremely low. The optimum structural shape with the appropriate topology is issued as design proposal. The originally homogeneous material distribution becomes highly inhomogeneous. Areas arise with no more mass at all (openings and holes) and areas which contain high density mass (bars and struts).

Compared with the sizing and shape optimization the numerical efforts strongly increases. The number of design variables is typically far beyond 200.000. Therefore large efforts are necessary for the sensitivity analysis. So far no method can be considered as a standard for calculating the optimum topology due to the above mentioned difficulties regarding the mathematical approaches. Essentially because of the expensive calculation efforts these approaches only can handle extremely simplified models. Commercially only few programs are available. Many FE developers work in this field. Because of the development of powerful iterative solvers and the more and more increasing computer capacities, topology optimization will be state-of-the-art very soon.

2.1.2 Classification by the Underlying Approach

Two approaches to the solution of a problem predominate over optimization. On the one hand these are the mathematical programming methods. They replace the mechanical model by a parametric mathematical substitute model which will then be studied with strictly mathematical methods. The other approach is based on the optimum criteria methods. For that reason requirements are formulated which are valid for the optimum design. If the optimum criteria can be applied to the certain task, the solution converges rapidly.

2.1.2.1 Mathematical Programming Methods

The structural optimization problem will be opened to the mathematical programming methods by formulating a substitute problem. The problem formulation looks as follows:

$$\min f(x)$$
$$g_i(x) = 0; j = 1,...,m$$
$$x \in IR^n$$
$$x_i \leq x \leq x_u$$

In this case $f(x)$ is the target function (e.g. structural weight, deformation, stress, etc.). The function $g_i(x)$ represents mostly nonlinear boundary conditions, which constrain the solution.

The calculation of the solution is done in two steps, the calculation of the search direction and the increment γ. With this the vector γd_v is determined, which leads from the current vector x_v to the next solution $x_{v+1} = x_v + \gamma d_v$. For the calculation of the search direction and the increment many approaches exist, according to the various types of target functions and boundary conditions. Finally the solution produced by the analysis program must be verified. If the start values are „disadvantageous", it might occur that the optimizer only identifies a local minimum as the solution. Therefore it is always recommendable to modify the start values. As the initial problem is transformed into a substitute model, the mathematical programming methods can generally be used. But they require an enormous amount of computing time which will even increase if the number of design variables and active restrictions is increased.

2.1.2.2 Optimum Criteria

In contrast to the mathematical programming methods, the optimum criteria methods take advantage of the knowledge on the physics and mechanics of the respective problem set. Theses will be postulated describing the optimum.

A well-known and ascertained physical law relating to structural mechanics is for instance the Fully Stressed Design which can actually only be applied to statically determined structures. An important mathematical optimum criterion is the Kuhn-Tucker condition which is for convex optimization purposes fulfilled necessarily and adequate in the optimum. The theses on stress homogenization and stress minimization are optimum criteria, too.

Regarding the optimum criteria methods, these criteria and the response behavior of modifications of the physical model are implemented into the algorithm. With suitable redesign rules, a convergence behavior is achieved which cannot be attained with mathematical optimizers. Applying this particular physical and mechanical knowledge, the optimum criteria methods remain limited to the certain application areas. Applying this knowledge makes the individual optimization steps comprehensible. The remaining potential to the optimum, which is known from the optimum criteria, can easily and exactly be estimated.

The optimum criteria are particularly well proven for shape and topology optimization where a large number of design variables is required. The convergence speed is independent of the number of design variables.

2.2 Multi Body System Simulation

The simulation of multi body systems is successfully applied for many years in a great variety of fields. The most of these systems are systems of rigid bodies connected by different kinds of joint and force elements.

In recent days general purpose multi body simulations packages came to market, that are able to deal with complex flexible bodies. The typical approach is to interface with an FEA package where the bodies are represented in so called modal coordinates. It is therefore necessary to perform a normal mode analysis in the FEA package. The theory behind this technique is the component mode synthesis [4] and is also used by ADAMS/Flex [5]. The incorporation of such general flexible bodies gives way to major new possibilities for both the simulation of multi body systems as well as the FE analysis of structural components.

3 New Approach for System Based Optimization

Today's FEA based structural optimization suffer from different bottlenecks. These are:

- The formulation of the correct loadings can be a challenging task when dealing with complex dynamical systems
- Limitation to component based optimization due to the lack of the inclusion of system responses
- Optimization techniques are based on "static" loading conditions i.e. loading conditions that are not updated during the iterations of the optimization procedure
- $\Rightarrow$ System response can not be considered during the optimization. The body mass as well as the normal mode frequencies change during the optimization process. This influences the systems dynamic behavior and therefore the loading conditions of the body

With the possibility to incorporate flexible bodies into the multi body system simulation and export loading conditions for different FEA packages it is a natural next step to extend the procedure to the optimization of structural components.

3.1 Setup

To achieve an automated optimization of a system embedded flexible structural part the authors coupled MSC.CONSTRUCT and ADAMS in a way that ADAMS performs system simulation and acts as a load set "generator" for MSC.NASTRAN. MSC.NASTRAN was used as the FEA engine behind MSC.CONSTRUCT.

3.1.1 MSC.CONSTRUCT Optimization

For a better understanding of the coupling it is necessary to first get familiar with the "traditional" optimization procedure of MSC.CONSTRUCT.

The optimization modules of MSC.CONSTRUCT make use of an external FE solver such as MSC.NASTRAN which was also used by the authors. The body to be optimized was modeled for a FE analysis and a parameter file describing the optimization problem was written. This preprocessing is fully supported from within MSC.PATRAN. When using MSC.NASTRAN the model describing the whole FE model of the body is stored in a so called bulk data file (.bdf) which is also used by MSC.CONSTRUCT. This file includes geometrical information as well as boundary and loading conditions. The iterative optimization procedure is stated in the following diagram:
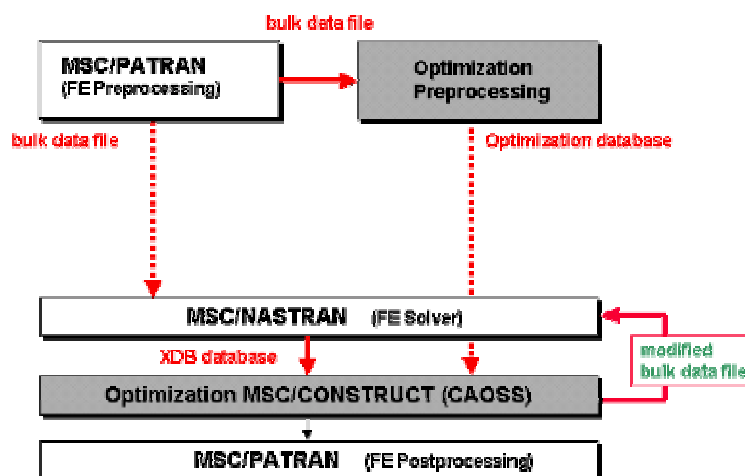

Figure 2: "Traditional" optimization with MSC.CONSTRUCT

In each iteration of the optimization the bulk data file is read by MSC.CONSTRUCT together with the stress results from the prior MSC.NASTRAN run (XDB database). The optimization module changes the bulk data file (containing the model) for the next analysis with MSC.NASTRAN. After the process has converged (the user can define a variety of different target functions) the resulting body with resulting stress distributions can be post processed.

3.1.2 Incorporating ADAMS

To use ADAMS in the above described way it has to be placed prior to the FE analysis in order to generate the loading conditions of the body (see Figure 3). Therefore the loading conditions need not to be modeled in advance with a preprocessor.
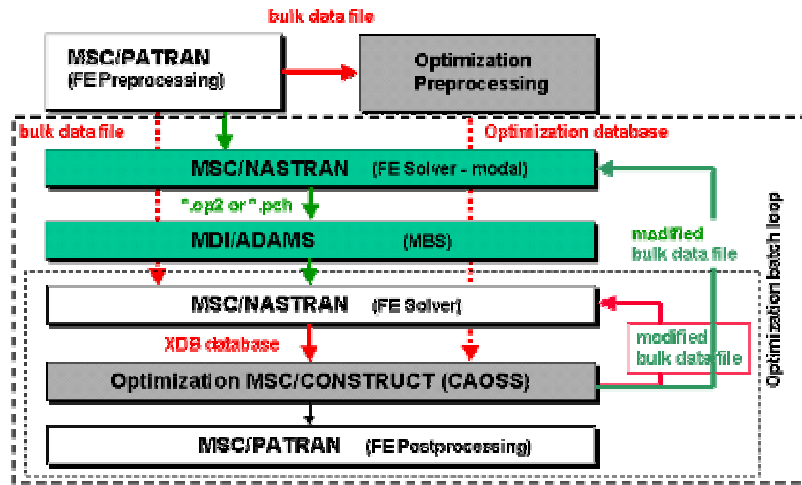
Figure 3: Coupled approach with ADAMS

Notice the additional MSC.NASTRAN run (normal mode analysis) before ADAMS. It is needed to produce a modal neutral file (.mnf) for the ADAMS simulation with the latest shape of the body. After each run of the optimization module the flexible body data is updated by this way.

3.2 mkl/SYSSHAPE

To realize the above described process it is necessary to automate the complete interfacing and data management. For this purpose the script package mkl/SYSSHAPE was developed by the authors to allow a fully batch oriented optimization. The user has to accommodate a template initialization file for mkl/SYSSHAPE with information about the model and job such as ADAMS database name as well as FE model information.

Afterwards mkl/SYSSHAPE can be used in several different so called run modes. One mode prepares the bulk data file for normal mode analysis, runs MSC.NASTRAN and the translator program to generate a modal neutral file. This file can be imported into ADAMS for the multi body system model.

The other run modes are intended to be called by the control program of MSC.CONSTRUCT during different steps of the optimization process.

Despite of the data management and interfacing mkl/SYSSHAPE performs some essential tasks to the FE model of the body. One of these tasks is the formulation of boundary conditions in order to perform a successful FE analysis. These boundary conditions are not supplied by ADAMS. What seemed to be a challenging task exposed to be very simple since the nature of the dynamic load sets generated from ADAMS. ADAMS writes a load set containing bearing forces on the one hand and a set of statements covering the inertia loads on the other. Typically (Newton's law) these forces balance each other in such a way that the resulting force is zero. The body should not move at all (and therefore one could assume that a constraint is not necessary) but the limited numerical accuracy of FE solvers gives rise to a weak resulting force. Because of the presence of this force a constraining mechanism has to

be applied to the body. A very simple technique is to add spring elements to three arbitrary nodes of the FE model to prevent the body from rigid body motions. The resulting force is easily compensated without constraining possible body deformations.

3.3 Verification

Due to the nature of the MSC.NASTRAN load sets written by ADAMS it is difficult to estimate whether the formulation of the loadings is correct since ADAMS writes a combination of MSC.NASTRAN statements (set of RFORCEs and GRAVs). These include several statements that together form the inertia loads; this is also the case if the body performs a simple motion (e.g. a rotational motion along one axis).

To verify these load sets two test models were set up and compared. One with a simple linear accelerated beam and one with a beam rotating around an axis. Both motions can be easily modeled with MSC.NASTRAN manually. One expects that the "pure" MSC.NASTRAN models lead to the same results as the models with load sets generated by ADAMS.

3.3.1 Beam FEM Model

As test model we used a simple cantilever beam with an RBE2 element at one end (right side in Figure 4) connecting the surface grid points to one node:
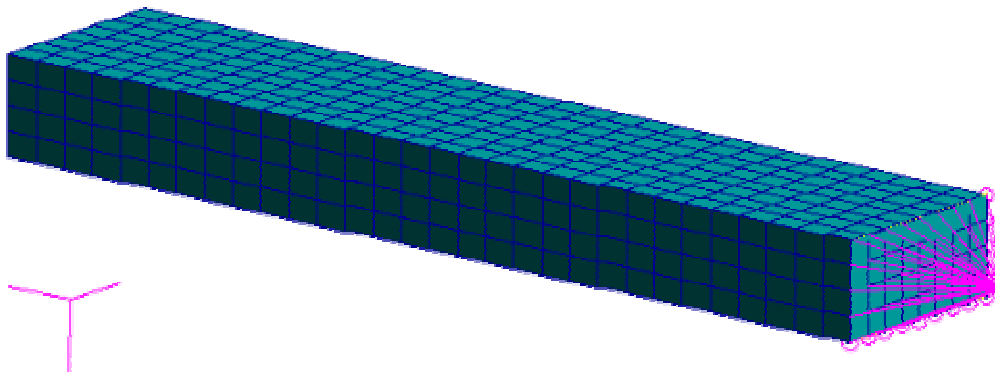
Figure 4: Cantilever beam FE model with interface point

This node is the only exterior point of the superelement and is used in ADAMS as interface point. Furthermore an evaluation path was defined along one side of the beam. "Path" means a set of nodes in line. The stress results at these nodes were exported during FE analysis to be evaluated in diagrams (see below).

3.3.2 Linear Acceleration

A single, body fixed constant force $\vec{F}$ was applied to the interface point (see above) leading to an linear acceleration $\vec{a} = \vec{F}/m$ of the beam (with the beam mass m). In MSC.NASTRAN the linear acceleration is modeled with one GRAV statement.

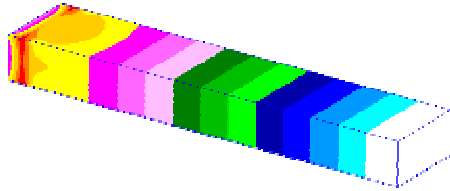Stress results for the linear accelerated beam (von Mises):
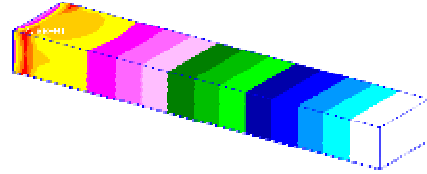

Figure 5: Loads from ADAMS

Figure 6: Single GRAV statement

The stress distribution as well as the minimum and maximum values of the two models coincide. The same is true for the nodal stress results along the analysis path. For this reason we only present one chart:
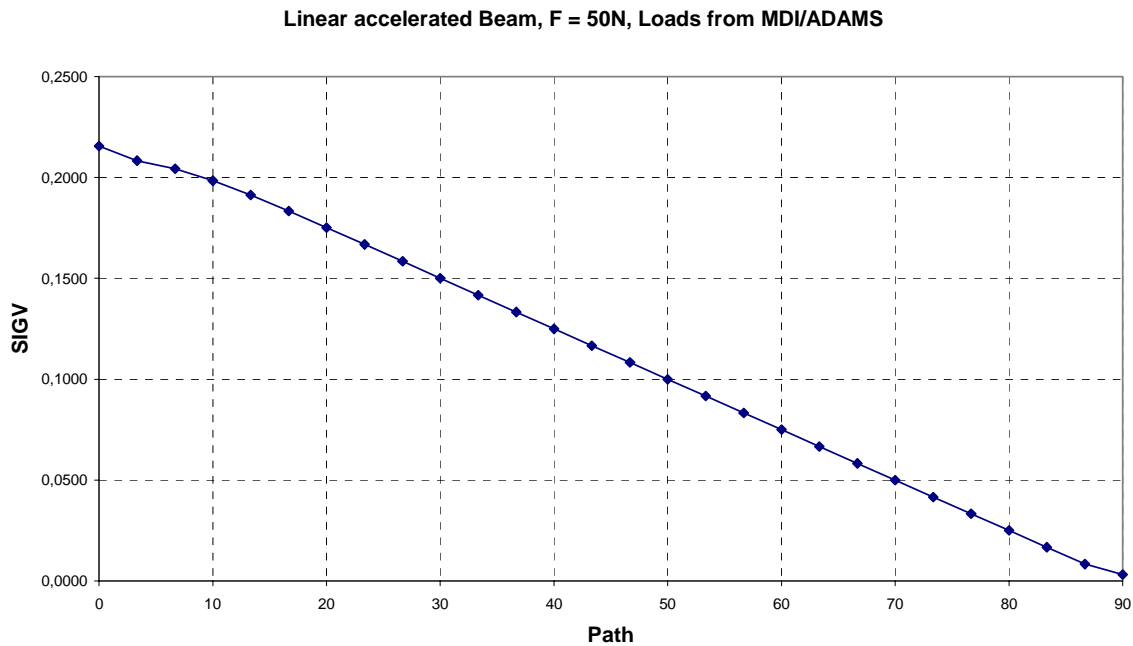
Figure 7: Nodal stress along path (von Mises)

3.3.3 Rotational Motion

To apply a rotational motion to the beam it was necessary to incorporate a second body since motion generators can not be applied to flexible bodies directly. Therefore a rigid cylinder was included and fixed to the interface point of the beam. The rotational motion with a constant angular velocity $\vec{\omega}$ then was applied to the cylinder forcing the beam to a centrifugal motion. In MSC.NASTRAN the rotational motion was modeled with one RFORCE statement.
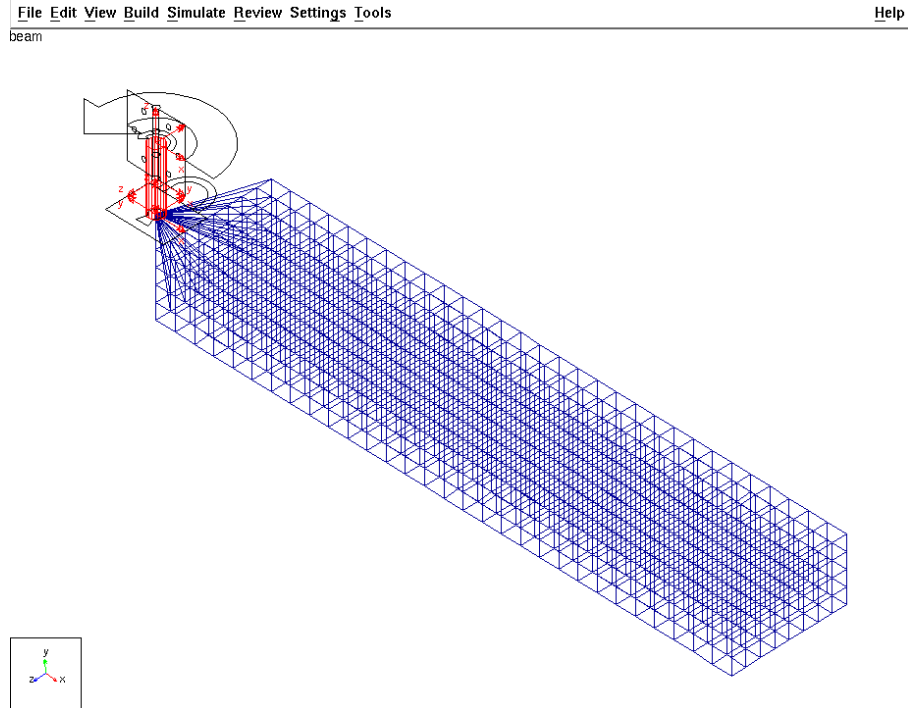

Figure 8: ADAMS model with flexible beam

With this model we also performed a shape optimization to investigate optimizer behavior. The surface nodes at both sides of the beam where defined as design nodes. Notice that these are the nodes that can be moved by the optimization module in a direction perpendicular to the beam surface. The first three layers of elements on the left side of the beam did not belong to the design. Stress results for the rotating beam (von Mises) after six optimization iterations of MSC.CONSTRUCT:
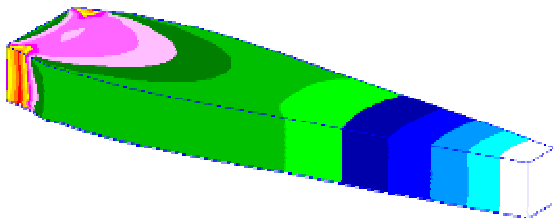

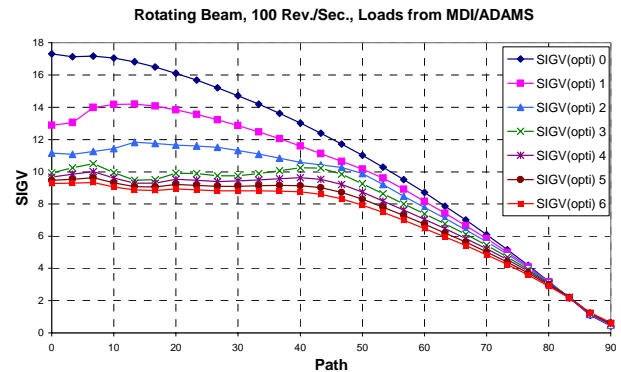Figure 9: Stress distribution after optimization

Figure 10: Path results

The stress homogenization of the design area can be seen clearly on both pictures. The comparison of these results with an optimization on loadings from a manually written RFORCE statement again led to precise coincidence.

The two test models demonstrated that the batch oriented optimization process with automatically updated load sets from ADAMS was successfully realized. Together with the reasonable results this encourages the consideration of more complex models.

4 First Results

4.1 Model

As a natural next step investigated a body subjected to a combination of the two principal motions of our test model: linear acceleration and rotational motion. For this reason we modeled a simple crank mechanism with a flexible connection rod. All additional bodies were modeled rigid.
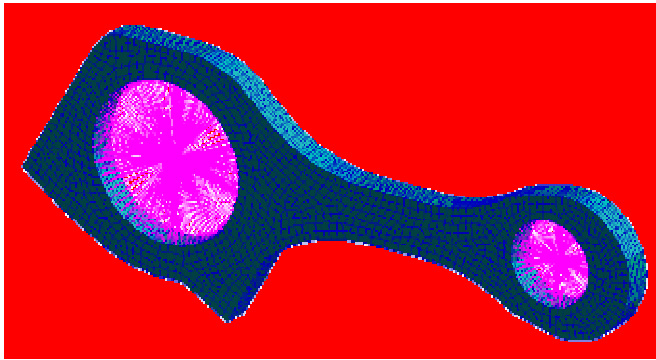

Figure 11: Conrod FE model with two interface points

Figure 12: ADAMS model

To keep the model simple for a first experiment we again used a rotational motion generator. The design nodes are the surface nodes along the shaft of the con-rod (on the upper and lower side in the figure above). Again we additionally defined an analysis path along one side of the shaft to investigate nodal stress results on the design area.

4.2 Simulation

Load sets for MSC.NASTRAN were exported at nine time steps during one cycle of the crank mechanism. The corresponding conrod position at these nine times is shown in Figure 13:
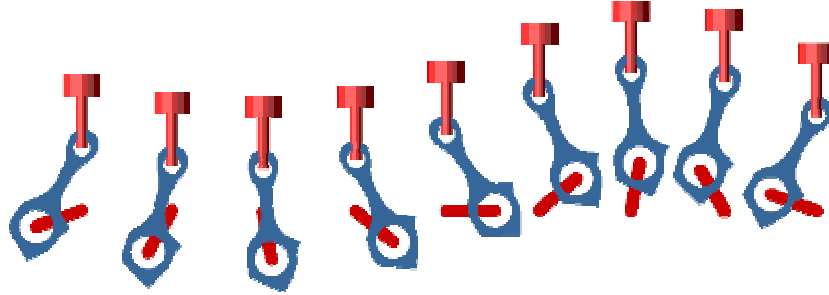

Figure 13: Load case generation time steps

Each of these export times produces one MSC.NASTRAN subcase. For a better understanding of the actually applied loads the following picture shows the force distribution applied to the conrod:
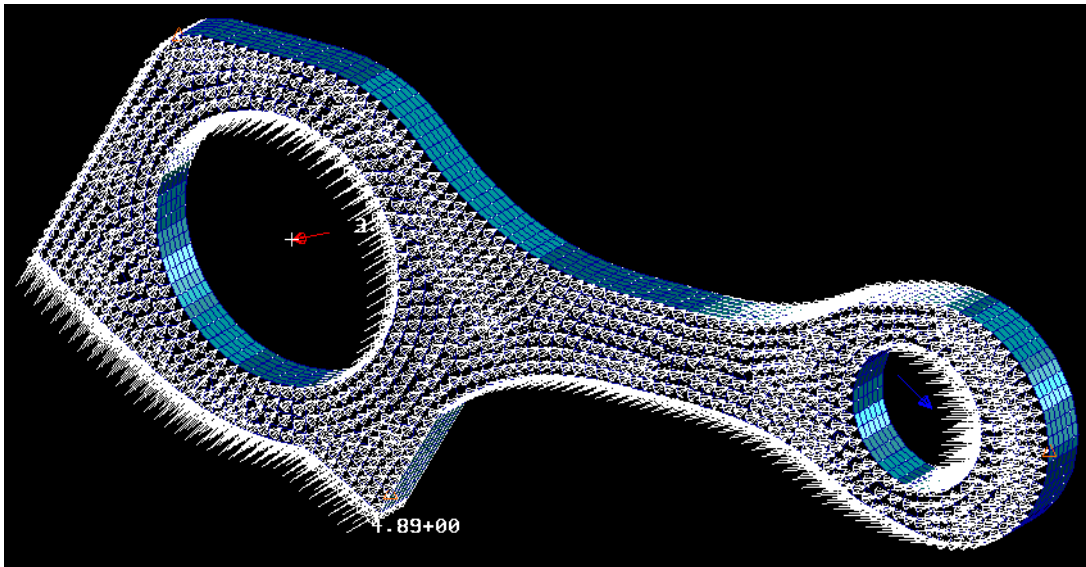

Figure 14: Applied loads (forces on nodes)

Bearing forces (red and blue arrow at the bearing centers) are balanced by the sum of the inertia loads distributed over all the nodes of the entire model. The picture above shows the dynamically loaded conrod in "static equilibrium".

4.3 Results

After nine iterations the optimization process reached it's target (the homogenization of the stresses at the level of a given stress maximum in the design area). The stress results before and after optimization (von Mises) are shown in Figure 15:
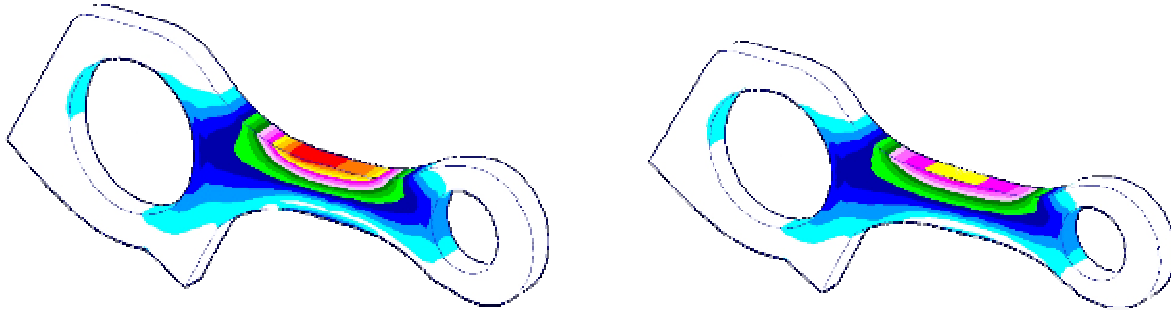

Figure 15: Stress distribution before (left) and after (right) optimization

The reduced stress maximum can clearly be seen. The maximum was reduced from 1.020 N/mm^2 to 591 N/mm^2, which is a reduction of 42 %. For a more detailed quantitative study we put the analysis path results together into one x-y plot:
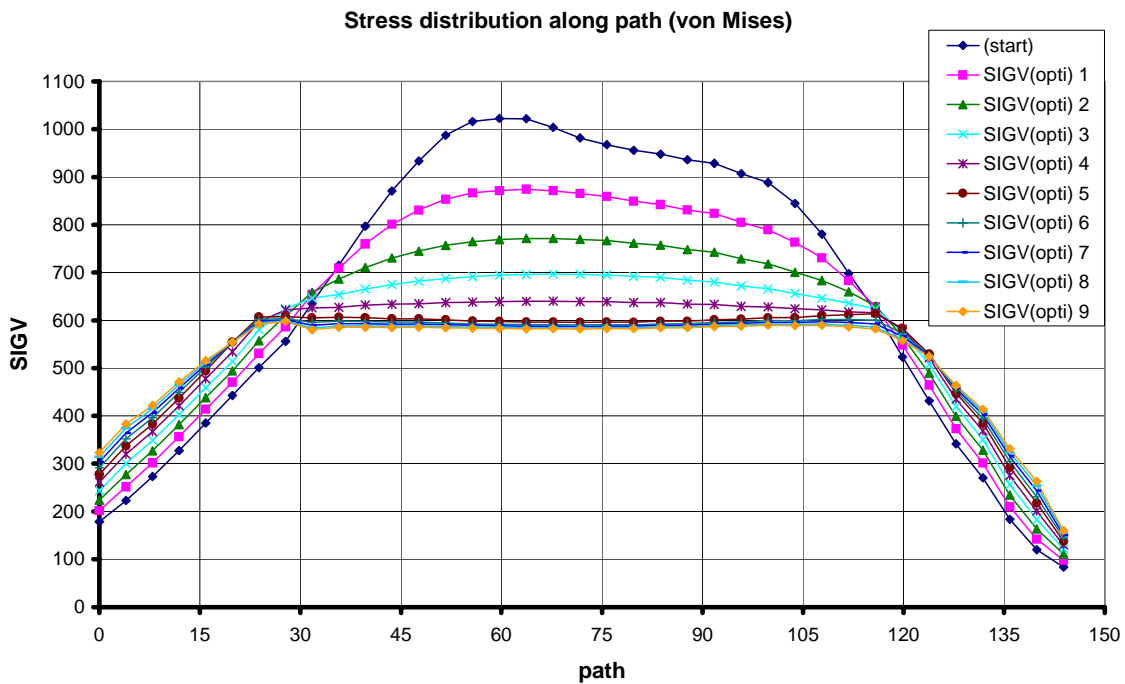

Figure 16: Nodal stress along path (von Mises)

The decrease of the stress maximum at the evaluation path for each iteration can easily be seen in the above Figure 16. At the end of the optimization process a homogeneous stress distribution on the design area can be determined.

5 Conclusion

In the presented paper it was shown that the coupling of Multi Body System Simulation Software and Finite Element Technology can be successfully applied to the structural optimization of systems of flexible bodies. For the realization the commercial available software tools MDI/ADAMS, MSC.CONSTRUCT and MSC.NASTRAN were used.

At the Institute of Machine Design of the University of Karlsruhe, a fully automated procedure was developed which handles the data management and the management of the various tools. Verification examples showed that the load export from ADAMS as well as the routines work properly.

To demonstrate the capability of the new procedure a shape optimization of a conrod which is part of a crank mechanism was successfully performed. This example showed the challenging opportunities which are now opened for the structural shape and topology optimization of entire mechanical systems.

6 References

[1] Müller, O., Albers, A., Allinger, P., Sauter, J.: 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.

[2] Allinger, P., Bakhtiary, N., Friedrich, M., Müller, O., Mulfinger, F., Puchinger, M., Sauter, J.: *A New Approach for Sizing, Shape and Topology Optimization*, SAE International, International Congress & Exposition 1996, February 26-29, Detroit, Michigan

[3] 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.

[4] Craig, R. R.: Bampton, M. C. C.: *Coupling of Substructures for Dynamic Analyses*, AIAA Journal Volume 6, No. 7, July 1968

[5] Óttarsson, G.: *Modal Flexibility Implementation in ADAMS/FEA*, Mechanical Dynamics, Marburg, 1997