Automated Structural Optimization of Flexible Components using MSC.Adams/Flex and MSC.Nastran Sol200

Albers, A.; Emmrich, D.; Häußler, P.

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 behavior of single deformable components. In recent years different software products e.g. ADAMS/Flex have come into the market, that utilize sub-structuring techniques to combine the benefits of 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 cannot 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 and the components' dynamical properties will change accordingly. In traditional structural optimization, usually constant loads and boundary conditions are used[1]. A coupled MBS-FEA optimization approach opens up the possibility to take these iteration-dependent load changes into account while optimizing the component. This leads to an improved design of the considered component and shorter product development time.

The article describes the structural optimization of dynamically loaded finite element flexible components embedded in a multibody system by means of an automated coupling of MSC.ADAMS with MSC.Nastran Sol200 as optimizer. The approach is presented and the requirements for such a system-based optimization are explained. An example of shape optimization using different possibilities of MSC.Nastran Sol200 on the basis of a simple crank drive mechanism is shown and the optimization results are discussed.

The presented approach offers new opportunities in the field of structural optimization as well as multibody system simulation by combining different software products of MSC.Software.

[1] Except in the case of simple body motion where accelerations can be formulated manually

Introduction and Motivation

Finite Element Analysis and Multibody System Simulation

The aim of multibody system simulation is the dynamic simulation of mechanical systems consisting of mostly rigid bodies. The equations of motion for multibody systems are usually highly non-linear. It is therefore necessary to keep the degrees of freedom of such a simulation as low as possible to reduce the computational effort. This is usually possible since the main focus of such a simulation is the system's overall behaviour rather than the individual bodys'.

The traditional application of finite element analysis in structural engineering is the investigation of the behaviour of individual mechanical bodies under load. Therefore, the loads on these components have to be determined before such an investigation can be carried out. The determination of the loads can be done by experiments or by calculation, e.g. by a multibody system simulation. A typical characteristic of such a simulation is a high number of degrees of freedom to represent the body with its stresses and deformations as accurately as possible. Linear solutions for such systems are usually computed within some hours, while non-linear solutions may require days. This becomes especially important, if the task is the optimisation of a component. The optimisation process usually requires several subsequent analyses. This often makes the application of nonlinear solutions too inefficient for a fast development process.

In the last years, efforts have been made to combine the advantages of both types of simulations, resulting in software products such as MSC.Adams/Flex and MSC.Adams/Autoflex. The aim was a multibody simulation closer to reality, not only consisting of rigid bodies, but also representing their flexible behaviour under the occurring loads. This was achieved by a modal representation of the flexibility of the bodies calculated by FEM analysis. This was not only an improvement for the MBS simulation, but also for the FEM simulation. These so called hybrid multibody systems made it possible, to determine loads on flexible components for FEM analyses to a very high accuracy.

Another benefit of the combination of FEM and MBS simulation is the possibility, to use FEM-based structural optimisation using calculated loads of a MBS simulation and reimport the improved FEM model to investigate the influences of the changes to the component on the whole system and the arising loads for the component itself. It is of growing importance to consider these effects, especially for dynamic systems, where the components loads are influenced by its inertia.

For highly dynamic systems and for large changes of the component's mass distribution caused by the optimisation, it is even beneficial to automatically update the acting loads on the component during the optimisation process. Possible scenarios of such an optimisation set-up using the optimality criteria based optimiser MSC.Construct have already been presented [Mül-99][Häu-01].

Here, the possibilities of the gradient based FEM optimiser MSC.Nastran Sol200 for the "coupled" optimisation are shown. The chosen example of a simple crank drive mechanism doesn't represent a real mechanism but is still well suited to show the set-up of such an optimisation and point-out some important effects which need to be considered.

Model Setup

Adams Model

The Adams Model consists of a simple crank drive mechanism. Since a demo FEM model of the connecting rod was used (see next chapter), the rest of the dimensions were chosen to represent a reasonable crank drive mechanism.

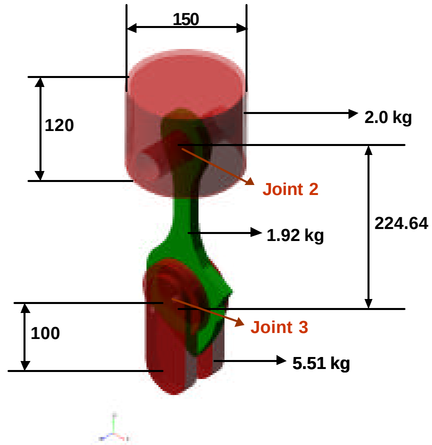


Figure 1: Setup of the Adams Model

Its basic dimensions can be taken out of figure 1.
The multibody system simulates a crankshaft turning with 1500rpm. Additionally, a force, representing a pressure, is acting on top of the piston. Starting at the upper dead centre, a take-in process with a maximum pressure of 5 bar is simulated. After 360° crank angle, the piston is moving to the bottom dead centre without pressure. Then, a compression process for again 5 bar is simulated.

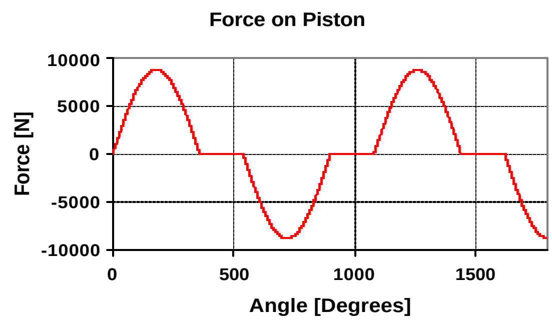


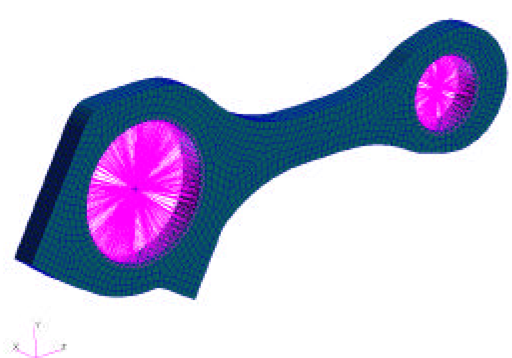
Figure 2: Force on the piston for take-in and compression

FEM Model

The model of the connecting rod is derived from the demo FEM model shipping with Adams. It is small enough for short computation times on a PC (5916 Elements, 8017 Nodes without "bearings", see later), but large enough to demonstrate the optimisation methodology applied in this paper.

Component Mode Synthesis

For the modal analysis which is needed for the flexible representation within Adams by means of a component mode synthesis, the nodes of the bearing seats are connected to the centre of rotation of the bearings using MSC.Nastran's RBE2 elements. This means a rigid coupling of all the nodes's dof to the nodes of the centre points.



length: 355mm
mass: 1.922kg

Figure 3: Model of Connecting Rod for Modal Analysis

For the modal analysis, the first 12 natural eigenfrequencies have been computed which are between 3.4kHz and 18kHz. Additionally, the 6 Craig-Bampton static correction modes per bearing node have been computed by the Adams DMAP for MSC.Nastran. This sums up to 24 modes, including 6 rigid body modes for the bodies' representation in Adams.
Note: This way of modelling the bearings leads to an artificially stiffer behaviour of the rod within Adams, since the RBE2s will transmit not only compression, as a contact would do, but also tension. For this part of the model, this seemed to be acceptable.

For details, especially the implementation of flexible bodies in ADAMS we refer to [Cra-68], [Ött] and [Ött-98].

Static Analysis

After the MBS simulation carried out with Adams, the points of time producing the critical loads on the conrod need to be determined. Then the export function of Adams can be used to generate the loads acting on the flexible body in MSC.Nastran format. These loads can then be used for a static analysis of the body to obtain the stress distribution. The loads exported by Adams are in a dynamic equilibrium. Which means that the forces at the supports compensate the inertia forces. However, this is only fulfilled to a certain numerical accuracy. Since there are initially no fixed nodes in the model, something have to be done so that the equilibrium is fulfilled exactly. There are different way to do this

compensation which are well described in [McC-01]. Here, the equilibrium was achieved by the usage of the so called inertia relief. There are mainly two options for inertia relief in MSC.Nastran: the "manual option" and the "automatic option". For the normal option, the user has to define 6 support (SUPORT) entries, which statically define the model. The FE-solver will then generate the necessary accelerations and numerically very small forces at these supports to force an equilibrium. In the automatic option, the FE-solver uses all grid points which are connected to mass, to produce those forces. While this option is very convenient, it is only supported for the normal linear analysis, but not for Nastran Sol 200. Therefore, the manual inertia relief option has been used for this paper.
Another difficulty is the load introduction. The aim is to run a shape optimisation of the connecting rod, so it is very important for the local stresses in the optimisation area to be accurate.

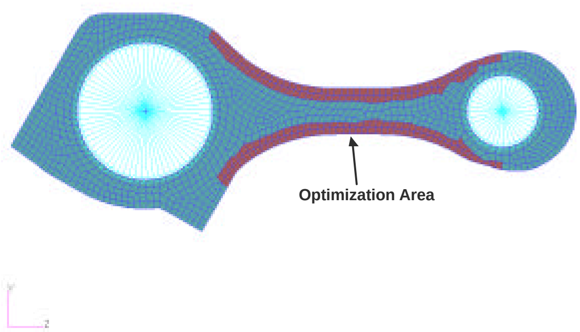


Figure 4: Shape optimisation area

Using the RBE2 element from the modal analysis will not result in a realistic stress distribution, since it will transmit the loads via tension and compression. On the other hand, a non-linear analysis including an accurate contact representation will result in much longer simulation times and is not supported for a Sol 200 optimisation.
A compromise, which does not give accurate contact stresses but a far improved load path and overall stress distribution, especially in the optimisation area, is the usage of Multipoint Constraints (MPCs), where those under tension are iteratively "opened". This could be done by MSC.Nastran's Linear Gap formulation, or, as it has been done here, by

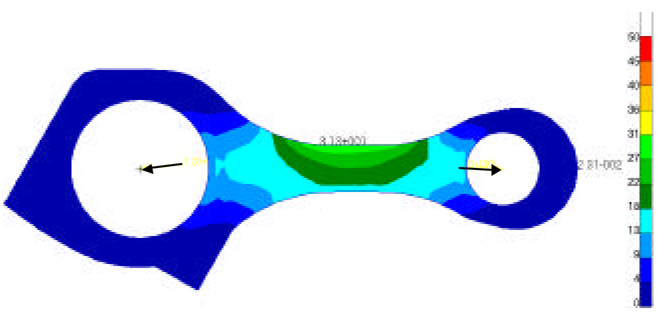


Figure 5: Rod under tension and acceleration, RBE2

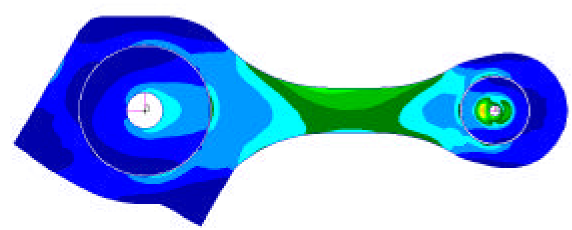


Figure 6: Rod under tension and acceleration, all MPCs closed.

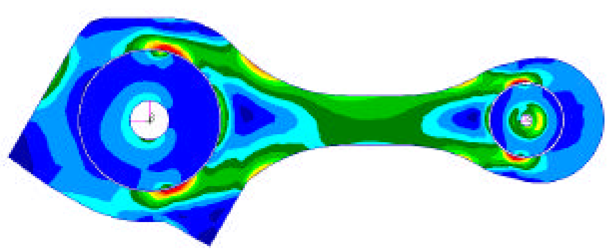


Figure 7: Rod under tension and acceleration, MPCs under tension opened.

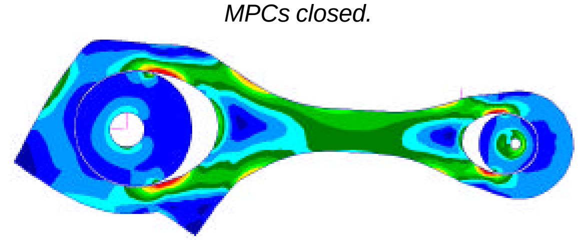


Figure 8: Rod under tension and acceleration, MPCs under tension opened, showing displacement (scaled).

an external routine, which deletes the MPCs under tension after each "contact iteration". Usually this can be done within 2-3 iterations. (In the example shown, in the first iteration 286 node-pairs have been released, the 26 and finally 8 out of 728 initally fixed node-pairs.) For this approach, the bolt has to be modelled and the MPCs have to be set up in the gap. Since all translational DOFs of the opposing nodes in the gaps are firmly connected, the "bolt" has been set up to have 1/10 of Young's Modulus of the rod and zero Poisson's Ratio. This results in a "cushion" effect and a smooth load introduction. The figures on this page show, how the stress distribution changes when those MPCs which are under tension are opened iteratively. It is obvious, that the stresses in the contact zone are

not realistic with this way of modelling, but the load path shows the expected behaviour in the optimisation area.
If the initial stress distribution had been used for optimisation, the optimiser might have removed material in regions where in reality the highest stresses occurred.

Optimization Setup

Introduction

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

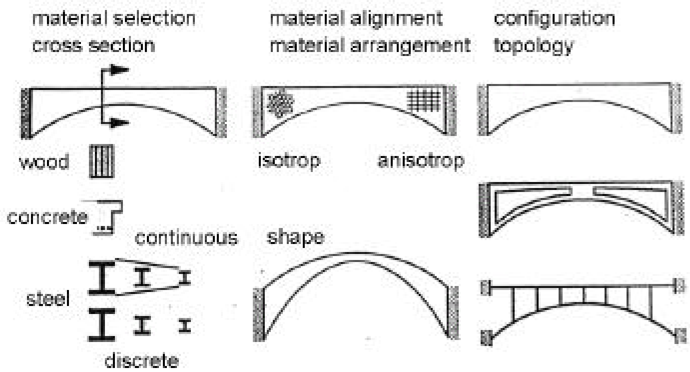


Figure 9: Fields of structural optimization [Kim-90]

Generally, the terms “sizing optimisation”, “shape optimisation” and “topology optimisation” are used for classification.
Besides the optimisation of elements properties and materials (like cross-sections of beams, sheet thicknesses, fibre orientations and more), MSC.Nastran Sol200 is able to optimise the shape of FEM models using so called shape basis vectors. These vectors define a relationship between the design variables of the optimisation and the shape change of the FEM model (see figure 10).

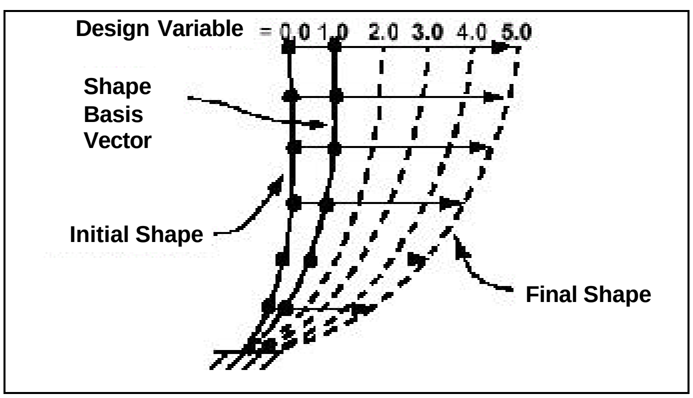


Figure 10: Shape Basis Vectors [Van-01]

They have to be defined before the optimisation can be started. The user is free to determine the method for setting them up. Common ways are geometrical defined deformations, eigenmodes, results of other, e.g. optimality criteria based optimisations, or artificial” load cases, which are usually not mechanically related to the real load cases. For the example dealt with here, the latter has been chosen.

Optimisation Parameters

Design Variables:

In order to change the shape of the rod in the optimisation area already illustrated in figure 4, pressure loads for surface deformation have been chosen.

The poisson's ratio for this auxiliary analysis has been set to zero so that all the affected nodes make a movement only in the yz plane. As mentioned before, there is no physical meaning behind these load cases, they are only used to generate shape basis vectors for the planned optimisation.

On the whole, 29 of such load cases have been set up, 14 on the top side, 15 on the bottom side (see figure 12). Each loadcase is slightly overlapping, so that a smooth surface can be formed by the superposition of the shape basis vectors. The mesh is locally adjusted by the movement of the inner nodes caused by the deformation.

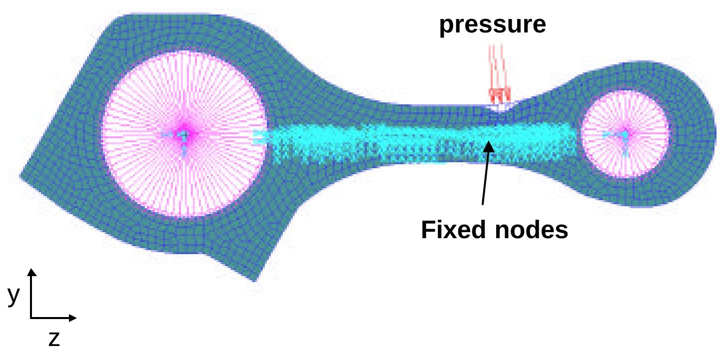


Figure 11: Load case to generate a shape basis vector

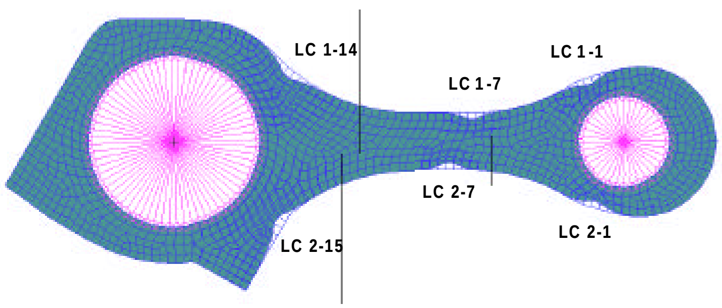


Figure 12: Examples of the 25 shape basis vectors

Objective Function:

In order to optimise the mechanical system's performance and to reduce the imbalance caused by the connecting rod, the minimisation of the rod mass has been chosen as objective function.

Constraints:

To ensure save operation of the rod without failure, a constraint has been set on its stresses. In MSC.Nastran Sol 200, the element Von Mises stresses can be limited. A value of 25 N/mm² has been chosen, which is below the maximum occurring stress in the design area of ca. 38 N/mm². The constraint is limited to the design area, which is not necessary since also stresses outside the design area could be reduced by the shape basis vectors. It has been done here to neglect the high stresses on the inner bearing diameter.

Additional so called "side constraints" limit the design variables directly. This means, that the maximum "shrinking" is limited to 3mm, the maximum growth is limited to 40mm. One reason for this limitation is to keep a reasonable rod design, the other is to control the occurring mesh distortion.

Software Setup and Dataflow

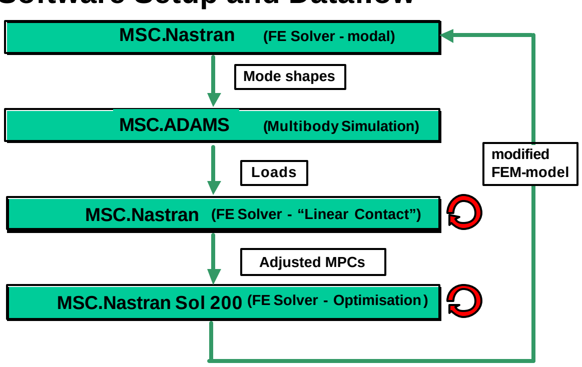


Figure 13: Dataflow of the automated optimisation approach

The software used for the control of the optimisation process and the data exchange is written in PERL. This has the following advantages for this application:

- since PERL is compiled just in time, it is very easy and lees time consuming, when the code needs to be adjusted or extended. No special linking and compiling is needed.
- PERL is extremely powerful for the fast modification of large ASCII files, like FEM data.
- It is platform independent and freely available for all important platforms.

There is no GUI for the setup of the approach.

The Adams and MSC.Nastran models have to be set up as before. Only the parts of the FEM-Model, which are changed during the optimisation have to be moved to an include-file. These include-files are then accordingly exchanged during the optimisation process. There are three necessary include-files:

- <Filename>_loads.bdf: Contains the latest loads of the Adams simulation.
- <Filename>_optdata.bdf: Contains the updated FEM entries which have been changed by the design variables.
- <Filename>_desvars.bdf: Contains the current state of the design variables.

In an individual configuration file amongst some other data the following can be defined:

- Simulation script and load output times of the multibody system simulation.
- Number of maximum internal Sol 200 optimisation loops
- Number of maximum complete optimisation loops

The necessity of complete optimisation loops/load updates

The update of loads out of Adams is necessary under the following two conditions:

- Large accelerations together with large changes of mass or mass distribution.
- Changes of the mechanical properties of the components which leads to different system behaviour.

For the first point, it doesn't seem to be obvious, why a new multibody system simulation is necessary for a load update. The generated Adams acceleration statements should be able to reflect the changes of the components mass properties. It is right, that the e.g. reduced mass will produce less inertia forces caused by the accelerations. The problem is, how the above mentioned equilibrium of forces is achieved.

All the proposed methods of [McC-01] to ensure this equilibrium are based on the assumption either that the inertia forces and the support forces are compensating each other. Therefore, additional supports will not change the stress distribution but only produce the minor forces for the exact equilibrium. Or the inertia relief will generate the accelerations necessary for the support forces on the interface nodes. There is no way, to generate the support forces, necessary to compensate the occurring inertia forces for the scenario shown here. This could only be done if the directions of the support forces could be predicted. Then, support-entries could be used in the FEM model.

Even worse, the inertia relief method will apply higher accelerations to the component, if its mass is reduced to fullfill the equilibrium with the support forces (initially calculated as reaction to the inertia of the larger mass, $F=m*a=const.$). This can result in larger stresses, which in reality would not be the case!

Simulation Results

Multibody System Simulation

For the introduced MBS model, the simulation has resulted in the strain energy graph as seen in *figure 14*. FEM calculations with exported FEM-loads of the simulation have shown that the four peaks of the strain energy give typical occurring stress distributions and the highest observed local stresses in the design area.

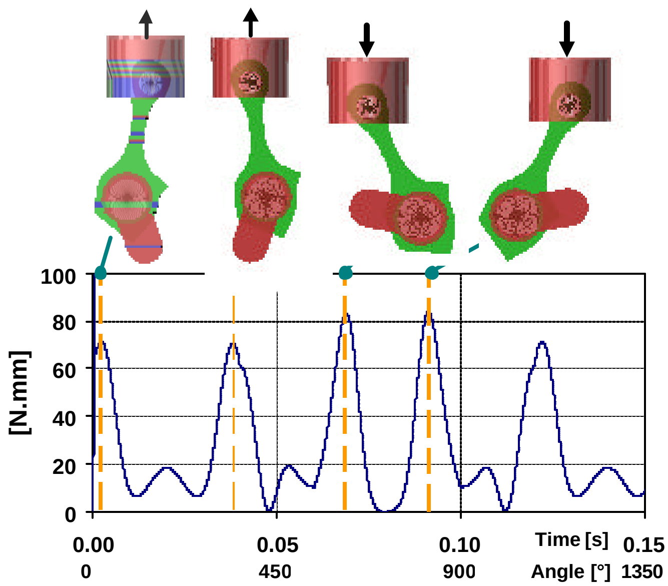


Figure 14: Strain energy over time of the flexible connecting rod during the Adams simulation

Large strain energies are a direct measure for large deformations of the flexible body, but do not guarantee the points of time with the maximum local stresses in the design area. Therefore it is recommended to investigate local stress for a lot more points of time to investigate the load path and the local stresses of all critical situations (e.g. a crank angle of 90°).

This manual procedure is one drawback of the proposed method. An automatic procedure for the determination of the largest local stresses would be beneficial.

FEM-Simulation

In *figure 15*, the results of the FEM simulation with the chosen Adams load cases can be seen. The MPCs under tension have already been released.

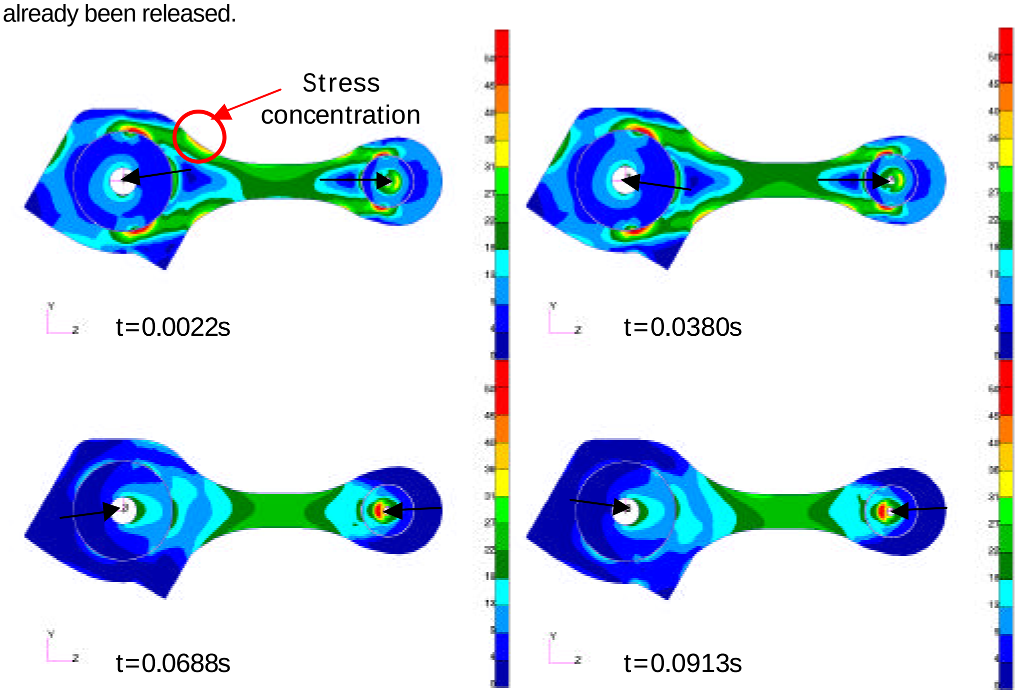


Figure 15: All four loadcases, as exported from Adams, FEM-analysis with "linear contact" representation.

The strain energies of the times of load export should be compared with the strain energies obtained by the FEM analysis, to ensure that a representative approximation of the body's flexibility is achieved by the chosen mode shapes. Without the linear contact, the differences were less then 1%. Due to the reduced stiffness of the FEM-model with contact, the elastic energy within the component rises up to two times. It is therefore questionable, if the FEM-model of the modal analysis without the contact is a reasonable representation of the flexibility of the component within the MBS simulation. As long as the deflections are small and their accuracy is not of importance, this is still a very good means for the computation of the loads of the rod. Those are, for the given scenario, hardly dependent of the deflections.

Optimisation

The shape optimisation has been carried out using the Modified Method of Feasable Directions (MMFD) which is the default algorithm for Sol 200. The maximum number of iterations has been limited to 30. In addition, the maximum number of constraints to observe has been set to 150 while no other default parameters have been changed.

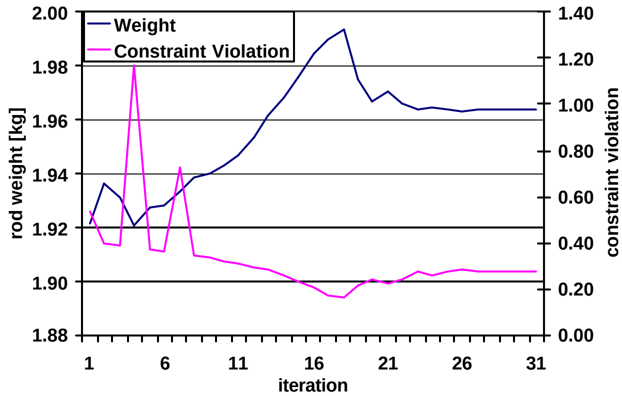


Figure 16: Objective function and constraint violation during the optimisation

This has resulted in an optimisation with 30 iterations, stopped by the maximum number of iterations. The progress of the objective function and the normalized constraint violation can be seen in *figure 16*. A constraint violation of 0 would indicate no constraint violation, while the final value shows that the model still violates the constraint by about 27%. The elements where these violations occur can be seen in *figure 18*.

It is unclear why the optimiser has not been able to find a feasible design, since the chart of the design variables in *figure 19* shows that none of the design variables has reached a side constraint. With the SQP algorithm, this problem of constraint violation has increased to even 37%. However, the optimisation a reached a stress reduction of 26% while gaining only 2% more weight!

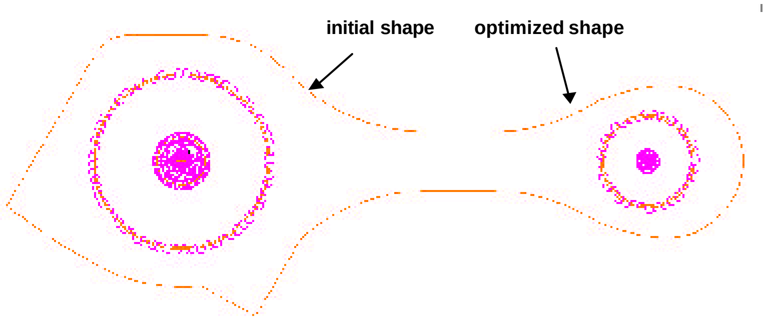


Figure 17: Optimized shape of the connecting rod

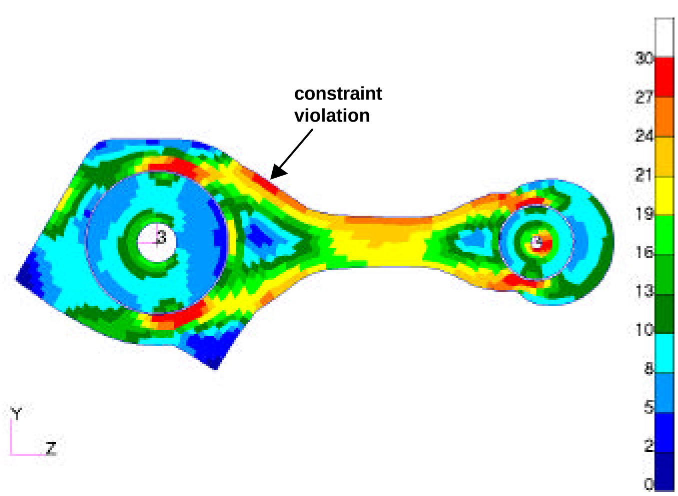


Figure 18: Constraint violations of Von Mises stresses at t=0.0022s for the final shape

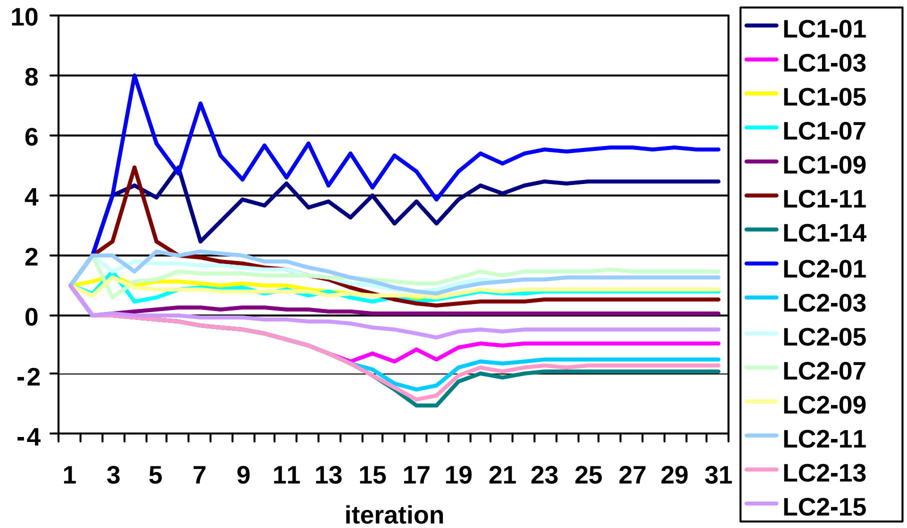


Figure 19: Examples of the design variable histories for the optimisation

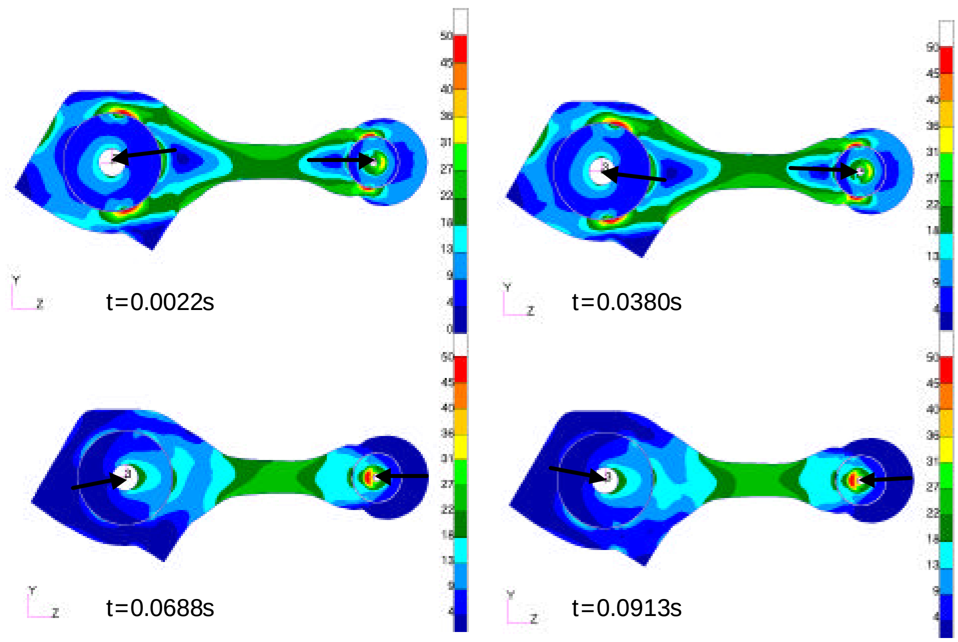


Figure 20: FEM results of the optimised connecting rod

The shape results as shown in *figure 20* look as expected. The shape of the optimisation area has been adjusted to the load path of the chosen load cases. This is best seen for the two take-in load cases, with the rod under tension. The stress concentrations in the optimisation area of the initial design have been removed (compare with *figure 15*).

Looking at the applied translational acceleration which is the initial acceleration plus the correction by the inertia relief in *figure 21*, the relationship to the model mass can be observed: if the model mass increases, the acceleration drops and vice versa. During the whole optimisation, the correction of the translational accelerations never exceed 4%. More critical are changes in the rotational accelerations, since these are a signal, that mass has been moved away from the centre of rotation and may cause larger inertia forces even if the overall mass stays constant. But the rotational acceleration corrections stay in the same order of magnitude over the whole optimisation process. Under these circumstances, the load update for this set-up is considered to be unnecessary.

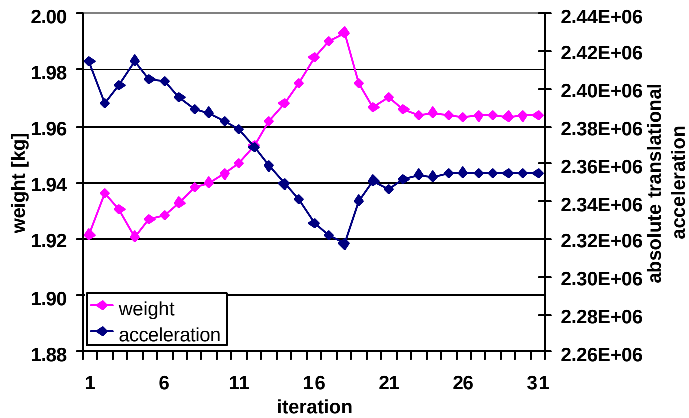


Figure 21: Comparison of model weight and acceleration

2nd Loop

The optimised connecting rod has then been reimported into the Adams model, and the same simulation has been run again.

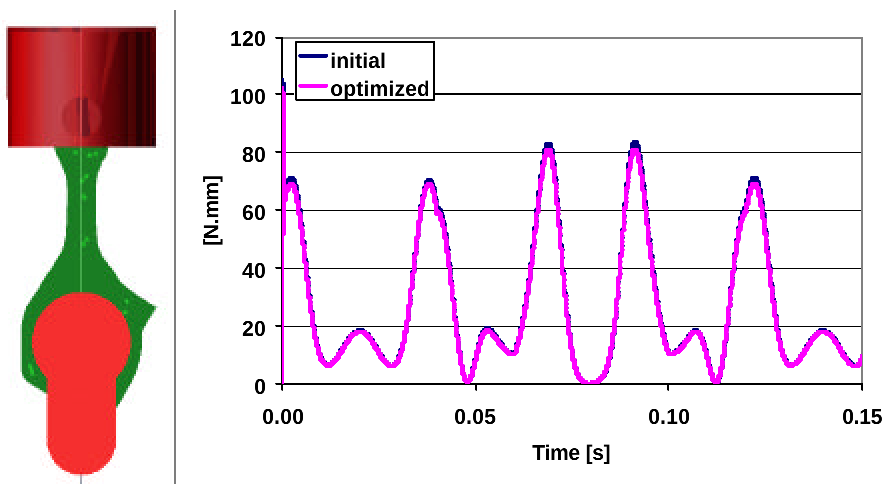


Figure 22: Strain energy history of the optimised rod during MBS simulation

Comparing the resulting strain energy history with the previous one as shown in *figure 22*, it can be computed that the new strain energy it about 1.5% lower. The forces of the exported MSC.Nastran load cases have changed less then 3%, so therefore, the overall optimisation process stops here, no further run of the Sol 200 optimisation software is necessary since no further improvement can be achieved under these boundary conditions.

Conclusions

Multibody system simulation is an excellent means for a quick and accurate generation of component loads for optimisation. It also allows an easy and fast verification, so see whether the component-based optimisation also improves the performance of the complete system.

Whether the update of the loads during the optimisation is worth the effort depends on the situation.

- If the system is highly dynamic and the loads are extremely dependent of the mass distribution. This situation could demand frequent load updates.
- If the optimisation heavily influences the mass distribution, which is more likely for topology optimisation than for shape optimisation.
- If the system is extremely sensitive towards flexibility changes of the component. This could lead to different loads or to different system behaviour for e.g. controlled systems.

The shape optimisation with MSC.Nastran Sol200 has the advantage, that the design variables, constraints and objective functions can be analytically defined. The price to pay for this flexibility are the high preprocessing needs, e.g. for the generation of shape basis vectors, and a large number of parameters which have to be set adequately. The needed number of iterations is often much higher than for MSC.Construct, which also is not limited to linear analyses.
In cases, where the lifetime of a component is the main focus, the coupled optimisation bears even more benefits. For existing load histories the advantages of optimisation based on fatigue analyses has been shown have been shown in previous works of the Institute of Machine Design [Ilz-00],[Ilz-01]. How a Adams MBS simulation can be used to easily generate and update those load histories for such optimisations will be shown in a new paper of the Institute to be published, soon.

Acknowledgements

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

References

[Cra-68] **Craig, R. R.; Bampton, M. C. C,** *Coupling of Substructures for Dynamic Analyses,* AIAA Journal Vol. 6, No. 7, 1968, pp. 1313 ff.

[Häu-01] **Häußler, P.; Emmrich, D.; et al**, *Automated Topology Optimization of Flexible Components in Hybrid Finite Element Multibody Systems using ADAMS/Flex and MSC.Construct,* MDI 2001 European Users Conference, November 2001, Berchtesgaden, Germany.

[Ilz-00] **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.

[Ilz-01] **Ilzhöfer, B.; Müller, O.; Häußler, P.; Albers, A.; Allinger, P.,** *Shape Optimisation Based On Liftime Prediction Measures,* ICED 2001 International Conference on Engineering Design Glasgow, August 21-23, 2001

[Kim-90] **Kimmich, S.,** *Strukturoptimierung und Sensibilitätsanalyse mit finiten Elementen*, PhD Thesis. Universität Stuttgart, 1990

[McC-01] **McConville, J.B.**, *A Survey of FEA-Based Stress Recovery Methods in ADAMS - Aircraft Model Case Study*, North American MDI User Conference 2001, June 19-20, 2001, Novi, Michigan

[Mül-99] **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.

[Ótt] **Óttarsson, G.,** *Modal Flexibility Implementation in ADAMS/Flex*

[Ótt-98] **Óttarsson, G.; Moore, G.; Minen, D.**, *MDI/ADAMS-MSC/NASTRAN Integration Using Component Mode Synthesis*, Americas User's Conference, MSC.Software, 1998

[Van-01] Vanderplaats, G: Design Optimization Training Course, 2001

Contact
Dipl.-Ing. Dieter Emmrich MSc BEng
Institute of Machine Design
University of Karlsruhe
Kaiserstraße 12
76131 Karlsruhe
Germany

Email: cae@mkl.uni-karlsruhe.de
Internet: http://www.mkl.uni-karlsruhe.de/