

# ROBUST DESIGN PROCESSES WITH CAD BASED FINITE ELEMENT MODELS

**Albert Albers<sup>1</sup>, Hans-Georg Enkler<sup>1</sup>, Thomas Maier<sup>1</sup> and Helge Weiler<sup>1</sup>**

<sup>1</sup>IPEK – Institute of Product Development Karlsruhe  
University of Karlsruhe, Germany

## ABSTRACT

Real components are subject to tolerances. These tolerances have to be taken into account when virtual product development methods like Finite Element Method (FEM) are applied, i.e. variation of components parameters must be included in simulations and optimizations. Nowadays, parametric CAD models are available at an early stage in the product development process. To allow the utilization of these CAD models to analyze the influence of variations, an automated process from the geometric model to the evaluation of the analyses results has to be established. In general, modern CAD software and optimization tools meet all requirements for such automation.

This article introduces an integrated and automated process that considers the described influences. The process has been realized with two common CAD environments. One process is based on CATIA V5. The geometric model is meshed by means of CATIA V5's built in Advanced Meshing Tools. For exporting the mesh and applying boundary conditions an external application is used. Subsequently, analyses were executed with MSC.Nastran. The Pro/Engineer based process includes modelling via PTC Pro/Engineer while Pro/Mechanica is used for the generation of the mesh. A Java application realizes control and reporting. MSC.Nastran is also used for analyses. The use of FE solvers integrated in CATIA V5 and Pro/Engineer wasn't considered – it was preferred to use established FEM solvers within the process. Potentials, limits and efforts, e.g. complexity in modelling, coming along with these methods will be discussed.

The described work is part of research project QUERON, financed by the German foundation *Stiftung Industrieforschung*.

*Keywords: Optimization, CAD, FEM, parameterized models*

## 1 INTRODUCTION

For a consequent shortening of the product development process (PDP) the application of optimization processes and the consideration of immanent tolerances in early phases of the PDP is mandatory. To achieve this, parametric geometry information are required within corresponding CAE processes. It is therefore obvious to use parametric finite element models (FE models) for the model generation in FE pre-processors. However, parametric modelling in common FE pre-processors is only supported rudimentarily. Therefore, parametric modelling within FE pre-processors can be very time consuming and is only feasible for a certain kind of parameters. Fields of application of parametric modelling are far reaching. Typical fields of application are mechanical engineering [2, 3], acoustics [4] or manufacturing simulation [5]. But the topic exceeds conventional mechanical engineering issues and can be used for medicine studies, for example [7].

Since the use of parametric design of CAD models has been established in common CAD environments [1], it is obvious to also use these existing parameters for CAE processes. This can be achieved by integration of CAE functionalities in CAD environments. As shown in [1], this integration has already begun. Especially CAD tools have been enhanced by analysis functionalities in the last few years. It is now possible to conduct parameter studies directly in CAD environments [1]. However, at the present time, one is mostly restricted to use internal FE solvers. Furthermore, the functionalities that are supplied in parameter studies and optimizations can not compete with possibilities of dedicated software packages as for example Optimus from Noesis.

In order to use the parametric geometry information from CAD models in combination with established CAE software, it is necessary to set up a continuous automated process starting from the CAD model to the result evaluation of the analysis. Modern CAD and optimization software offer the basis for such an automated process, but must be linked with each other via suitable interface software. Such a process has been realized at the Institute of Product Development Karlsruhe (IPEK). The process has been set up by means of the CAD environments CATIA V5 and Pro/Engineer, the solver MSC.Nastran and the optimization software Optimus supplied by Noesis. In the following, possibilities and restrictions of these approaches will be discussed.

The described work is part of the research project QUERON supported by the German foundation *Stiftung Industrieforschung*. QUERON deals with the integration of Robust Design Methods in existing development processes.

## 2 CURRENT UTILIZATION OF CAE PARAMETERIZATION

The parametric generation of components in 3D CAD tools has been established in many industry sectors [1]. However, a continuous utilization of these models has not yet been achieved. Two of the most common CAD environments are CATIA V5R16 and PTC Pro/Engineer Wildfire 3.0. Both offer possibilities to generate 3D volume meshes on the basis of CAD models and to define boundary conditions and loads for a FE analysis. They also supply integrated solvers to conduct FE analyses.

However, the utilization of the widely-spread solver MSC.Nastran was preferred for the described processes. In the case of CATIA, the generated analysis models can not be simply exported for the calculation with established solvers. It is only possible to export node and element definitions. This can be realized by means of additional workbenches such as Abaqus for CATIA, MSC.SimDesigner or LMS.VirtualLab. This of course means additional license expenses. Therefore, this was not considered for the referred project. For Pro/Engineer, there are no such additional FEM supporting workbenches. There is however the option to export MSC.Nastran and Ansys models directly from the CAD environment. These models do not need any further processing before calculation.

## 3 EXAMPLE OF APPLICATION

For the generation of both CAE processes, a planet carrier is used as a demonstrator. This demonstrator was developed by *PULSGETRIEBE Dipl.-Ing. W. & J. Puls GmbH & Co.* in Karlsruhe, Germany. *PULSGETRIEBE*, IPEK and other partners participate at the research project QUERON. The chosen component is supported by a bearing-arrangement as shown in Figure 1, right side. Load torques are induced through the contact surfaces at the bars and the hub. So far, six edge lengths and diameters as well as two radii have been varied.

The corresponding FE models consist of about 250000 elements. This leads to about 380000 nodes when using quadratic element approach and about 50000 nodes when linear elements are utilized.

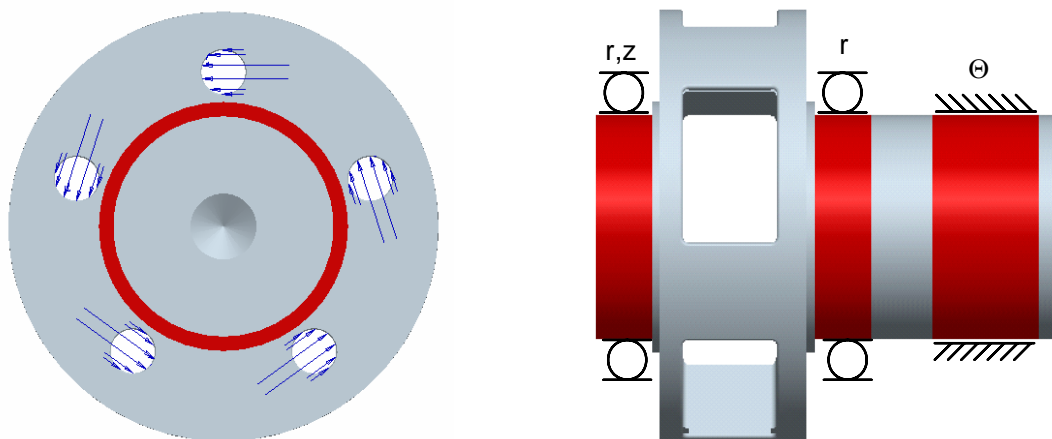


Figure 1. Loads and boundary conditions

#### 4 DESCRIPTION OF THE PROCEDURES

In this chapter, the procedures for joining CATIA – MSC.Nastran and Pro/Engineer – MSC.Nastran to automated process will be pointed out. It is assumed that a parameterized model has been set up within CATIA and Pro/Engineer.

##### 4.1 Procedure in CATIA

After the generation of the parameterized geometric models, an extraction of all of the solid’s surfaces is necessary. This is done within CATIA by means of the Generative Shape Design (GSD) module. These additional surfaces are important for the later export of the node and element information and for the application of the relevant boundary conditions. Depending on the boundary conditions and the needed mesh fineness, it may be necessary to split or combine the extracted surfaces. In the next phase, the mesh is built up manually – as usual before starting an analysis within CATIA. First of all, the surfaces extracted before are meshed. Secondly, a volume mesh is generated from the surface mesh, i.e. the volume mesh does not base on the geometrical model, but is directly based on the surface mesh. Thereby, the preliminary work for the optimization is completed.

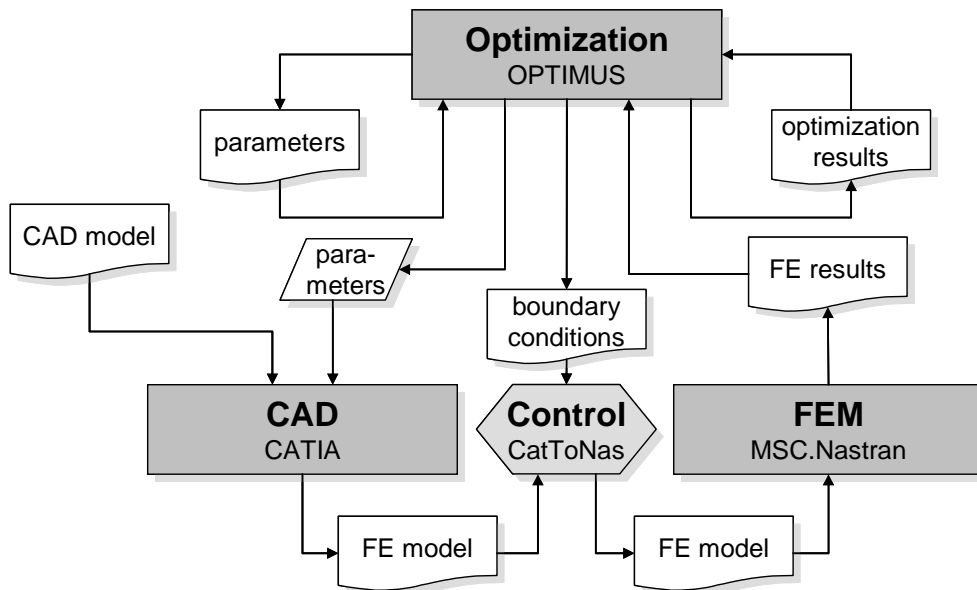


Figure 2. Flow chart of the procedure with CATIA and MSC.Nastran

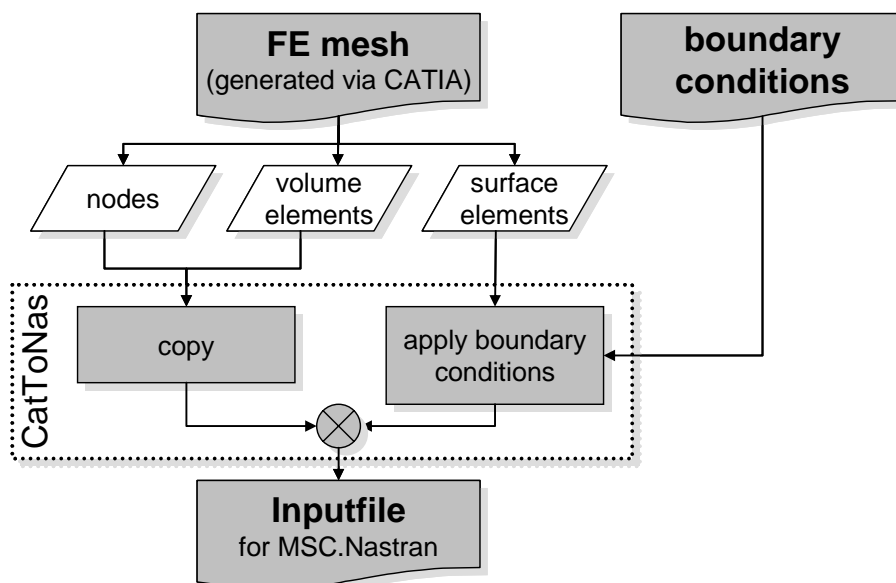


Figure 3. Procedure for the generation of an MSC.Nastran Inputfile from the CATIA V5 FE mesh and externally defined boundary conditions

Unlike a normal analysis, both surface and volume elements are exported along with the corresponding nodes. The volume mesh is sufficient for the actual analysis. The additional surface elements can be identified by an external application and utilized to apply boundary conditions and loads. Thus, the application enables the generation of an input deck ready for FE analysis. The input deck is passed on to MSC.Nastran and an analysis is performed. Analysis' results are evaluated and a new parameter set is generated. After passing this new parameter set to CATIA, CATIA automatically updates the solid model and the associated surfaces. Then, the surface meshed and the volume meshes are regenerated and exported. Afterwards, the external application again adds boundary conditions and the input deck is provided for evaluation with MSC.Nastran. Figure 2 shows the flow chart for joining CATIA and MSC.Nastran. The procedure for the generation of an MSC.Nastran input deck including both FE mesh and boundary conditions is shown in Figure 3.

#### 4.2 Procedure in Pro/Engineer

The first step is defining the representation of the component within Pro/Engineer and the integrated Pro/Mechanica. Besides common CAD-geometry, boundary conditions, loads and properties of the model have to be defined. The actual mesh has to be defined by element sizes, edge divisions, fixed points, and curves. The Java interface Jlink is used for the automation of Pro/Engineer. Any suitable external application can be utilized for the control of the Java application. The Java application controlling the process starts up Pro/Engineer, passes over the new set of parameters, regenerates the model and initiates the mesh generation. Afterwards, it commences an analysis and performs an evaluation of the results obtained by the analysis. After finishing the analysis by means of MSC.Nastran, the Java application evaluates the analysis results and prepares them for the parameter optimizer. Figure 4 gives an overview of the loop.

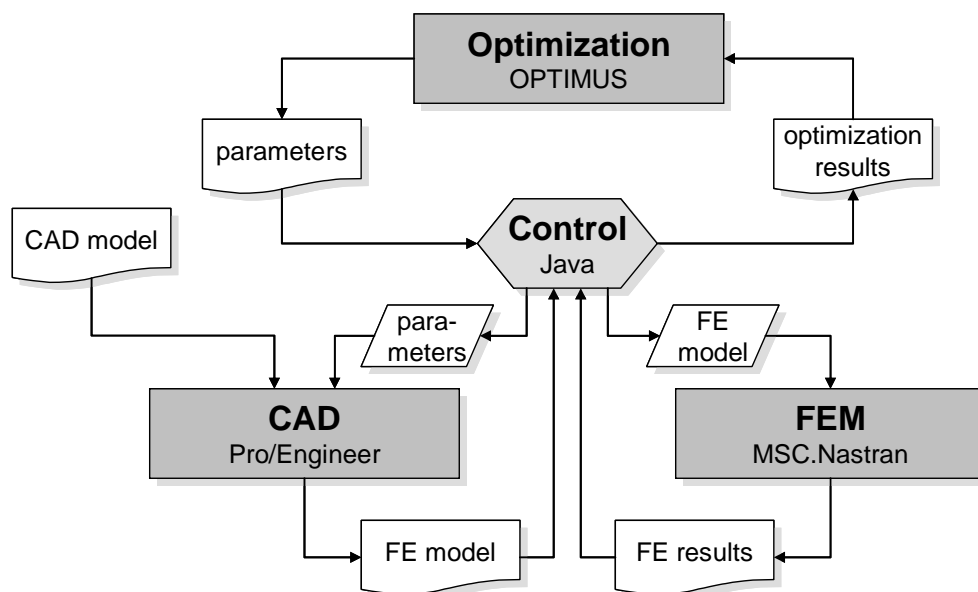


Figure 4. Flow chart of the procedure with Pro/Engineer and MSC.Nastran

## 5 COMPARISON OF BOTH METHODS

### 5.1 Modelling expenditure

It is relatively simple to define FE models out of existing CAD models in both applications. However, an appropriate parameterization from the constructional and manufacturing point of view is not necessarily suitable for an optimization or robust design analysis. It is usually required to adapt the parameterization and to consult with the design engineer in advance. This additional expenditure should already be considered in an early phase of the whole process. In some cases it is also necessary to generate additional geometry in order to define the exact definition of the loads and boundary conditions.

## 5.2 Possibilities to control the automatically generated FE mesh

In both processes, the FE mesh is regenerated during each calculation run. Thus, a control of the mesh is only possible via mesh parameters and not via manual intervention as in classical CAE processes. There are only restricted possibilities to control the mesh within Pro/Engineer. It is sometimes necessary to use additional geometry in order to generate comparable and good meshes in critical places. For an example to define additional curves as fix curves in radii in order to guarantee comparable meshes.

In contrast CATIA offers extended possibilities to control the mesh. There are different algorithms for meshing surfaces available. Furthermore, CATIA offers easy possibilities to analyse the mesh quality. However, the automation within the Advanced Meshing Toolbox is not failsafe. The influence of the whole parameter range on the generated mesh should be checked manually and the meshing parameters adapted if necessary

Both meshing tools do not show the same performance as established and specialized FE-preprocessors like MSC.Patran or Altair Hypermesh, but are efficient enough for our tasks.

## 5.3 Expenditure of time

After passing the parameter sets to the CAD software and updating the geometry, the FE mesh has to be generated. This generation of the FE model with about 250,000 linear elements requires about 90 seconds at a current workstation when using Pro/Engineer. It takes about 120 seconds when using CATIA. The remesh takes a little longer, if quadratic element approaches are used. The time needed for the start of the application should not be disregarded. In the case of smaller models, this can lead to a decisive extension of the total process. Thus, a parallel operation of a CAD application and optimization software in which the CAD application awaits new jobs is subject of current considerations. Furthermore, the time factor when analyzing text-based result data should not be underestimated. This lasts up to two minutes. Together with a linear FE calculation, a calculation run adds up to four to five minutes. When using quadratic element approaches, it takes about 35 minutes to finish the FE calculation whereby the remaining time expenses lose its significance.

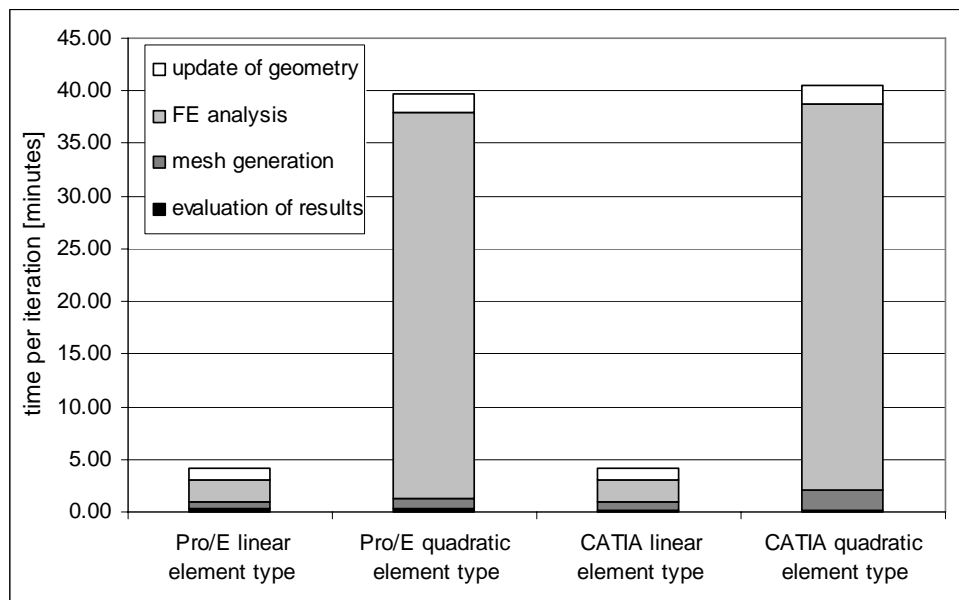


Figure 5. Comparison of the expenditure of time per iteration when using Pro/Engineer and CATIA

## 5.4 Evaluation of the analyses' results

A great disadvantage in evaluating Pro/Engineer models is the lack of a function to define group or set definition. Therefore, the model has to be divided into several properties in order to consider several component areas for evaluation. In CATIA this turns out to be comparatively simple. The component is divided automatically in several properties during the export. This can be achieved by manipulating

the surfaces that are derived from the solid volumes and/or by dividing the model into several volumes.

### 5.5 Expenditure in automation / adaptation

Pro/Engineer with Jlink offers a powerful interface for automation which allows accessing most of the objects and methods of a session. This however, only applies to the CAD part of the application. Within the integrated Pro/Mechanica environment, the automations can only be realized with macros that repeat manually performed user actions, which nevertheless can be performed in batch-mode without a user-interface. This restricts the generation of a standardized control and results in an additional programming expenditure when using a new model.

This applies similarly to the CATIA application. In the common CAD-workbenches most of the objects and methods can be accessed by means of VisualBasic or the CATIA script language. However, you will not have this option within the Advanced Meshing Tools. That's the reason why the above described procedure had been chosen – an existing mesh is automatically updated when the geometry has been changed.

## 6 ROBUSTNESS ANALYSIS

The presented workflow offers all necessary features for optimizing the considered parts and/or realizes robustness and reliability evaluations. In our case, both workflows have been used for evaluating the robustness of the relevant von-Mises-stresses against variations of the geometry of the part. In a first step, the sensitivity of the stresses has been evaluated in a structured DOE. This led to the identification of the six most important measures (shown in figure 6).

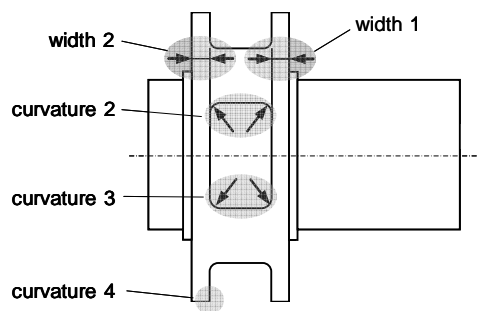


Figure 6. Six most relevant parameters

Based on a second DOE (Latin Hypercube), a response surface has been modelled, describing the dependencies between the stresses and the identified measures. A Least-Squares approach has been used. The identification and weighting of the different terms was done by the ORSM-Method (Optimal Response Surface Model) of OPTIMUS. After verifying the surface, a multi-criteria-optimization of the six identified design parameters was performed with the objective functions “von-Mises-stress” and “variance (sigma) of von-Mises-stress”. Figure 7 shows the resulting Pareto-front.

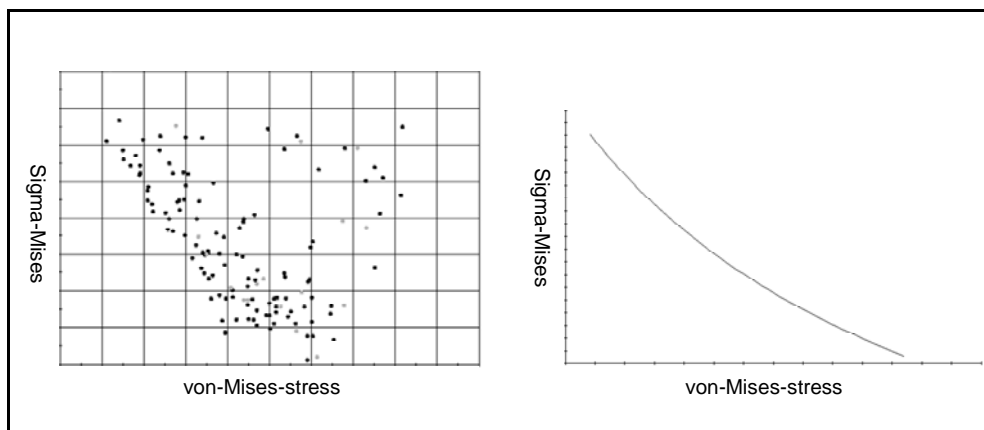


Figure 7. Pareto-front of the objectives stress and variance of the stress

The use of OPTIMUS allowed the integrated execution of all tasks needed, starting from the sensitivity analysis, the definition of the response surface and the calculation of the variance of the stresses up to the Pareto optimization.

## 7 CONCLUSIONS

An automated generation of FE meshes and FE analyses of components can be realized by the functional range of the current CAD software packages and additional scripts that are comparatively easy to handle. The utilization of third party software like Ansys Workspace, Abaqus for CATIA, MSC.SimDesigner or LMS VirtualLab is not necessary to achieve automated CAE processes. By using established solvers like MSC.Nastran and automation of the CAE processes, it is also possible to use the CAD software packages mentioned above in simulation departments. When joining CAD and FEM, there are further advantages: The application of established solvers enables a direct comparison of analyses' results with classically generated FE models. The applied boundary conditions are revisable and controllable. The processes presented offer all features for optimization and/or robustness and reliability evaluations of a component. Restrictions in automation are rather natural and are applicable to alternative processes such as mesh-morphing.

The processes have been developed within the research project QUERON and applied to one single component. For analyzing assemblies an enhanced process has to be developed.

## REFERENCES

- [1] Albers, A.; Binz, H.; Höhn, B.; Blacha, M. Häußler, P.; Jaros, M.; Minx, J. Simulation und Berechnung. In: *Innovationspotentiale in der Produktentwicklung*, Krause, Franke, Gausemeier, Hanser Verlag, 2006
- [2] Pedrero J. I.; Zanzi-C. Stress analysis of a face gear drive with conical pinion. In *VDI-Berichte, Vol. 1904.1*, 2005, pp 107-123 (VDI-Verlag, Düsseldorf, Germany)
- [3] Bratsberg K.; Xiros E.; Haugen J. O. Design and production of the high head Francis turbine for Svartisen power plant. In *Modern Technology in Hydraulic Energy Production, Proceedings of the 15th IAHR Symp., Int. Ass. for Hydraulic Res., Vol 2*, 1990, pp Q3.1-10 (Belgrade, Yugoslavia)
- [4] Steffens C.; Nussmann N.; Pilath C. Virtual product development techniques applied to powertrain acoustics. In *14. Aachener Kolloquium: Fahrzeug- und Motorentechnik, Vol. 1*, Aachen, Germany, October 2005, pp 277-291
- [5] Zaeh M. F.; Oertli T. Parametric finite element modelling of machine tool structures. In *Production Engineering, Research and Development, Vol. 12*, 2005, pp 129-132 (Wissenschaftliche Gesellschaft für Produktionstechnik (WGP), Berlin, Germany)
- [6] Rigal J. F.; Pupaza C.; Bedrin C. Model for simulation of vibrations during boring operations of complex surfaces. In *CIRP Annals*, Vol. 47, 1998, pp 51-54
- [7] Tao K.; Qu A.-L.; Wang D.-M.; Wang C.-T.. A Novel Method for Creating a 2-D Parameterized Finite Element Model of the Human Femur with Matching Artificial Hip Joint Based on X-Ray Image. In *Proceedings of the 2005 IEEE Engineering in Medicine and Biology 27th Annual Conference*, Shanghai, China, September 2005, pp 805-807
- [8] Krause, F.-L.; Franke; Gausemeier. *Innovationspotentiale in der Produktentwicklung*, Hanser Verlag, 2006

Contact: o. Prof. Dr.-Ing. Dr. h. c. Albert Albers  
University of Karlsruhe (TH)  
Institute of Product Development  
Kaiserstraße 12  
76131 Karlsruhe  
Germany  
Phone: +49 (721) 608 2371  
Fax: +49 (721) 608 6051  
e-mail: [albers@ipek.uni-karlsruhe.de](mailto:albers@ipek.uni-karlsruhe.de)  
URL: [www.ipek.uni-karlsruhe.de](http://www.ipek.uni-karlsruhe.de)

Contact: Dipl.-Ing. Hans-Georg Enkler  
University of Karlsruhe (TH)  
Institute of Product Development  
Kaiserstraße 12  
76131 Karlsruhe  
Germany  
Phone: +49 (721) 608 6495  
Fax: +49 (721) 608 6051  
e-mail: [enkler@ipek.uni-karlsruhe.de](mailto:enkler@ipek.uni-karlsruhe.de)  
URL: [www.ipek.uni-karlsruhe.de](http://www.ipek.uni-karlsruhe.de)

Contact: Dipl.-Ing. Thomas Maier  
University of Karlsruhe (TH)  
Institute of Product Development  
Kaiserstraße 12  
76131 Karlsruhe  
Germany  
Phone: +49 (721) 608 6957  
Fax: +49 (721) 608 6051  
e-mail: [maier@ipek.uni-karlsruhe.de](mailto:maier@ipek.uni-karlsruhe.de)  
URL: [www.ipek.uni-karlsruhe.de](http://www.ipek.uni-karlsruhe.de)

Contact: Dipl.-Ing. Helge Weiler  
University of Karlsruhe (TH)  
Institute of Product Development  
Kaiserstraße 12  
76131 Karlsruhe  
Germany  
Phone: +49 (721) 608 7033  
Fax: +49 (721) 608 6051  
e-mail: [weiler@ipek.uni-karlsruhe.de](mailto:weiler@ipek.uni-karlsruhe.de)  
URL: [www.ipek.uni-karlsruhe.de](http://www.ipek.uni-karlsruhe.de)