A NEW APPROACH FOR THE PREPROCESSING OF TOSCA OPTIMIZATION TASKS IN ANSA

[1]Lauber, Boris*, [2]Korbetis, George, [2]Marmaras, Dinos,
[1]FE-DESIGN GmbH, Germany, [2]BETA CAE Systems S.A., Greece

KEYWORDS – Non-parametric structural optimization, shape optimization, bead optimization, topology optimization, TOSCA, ANSA

ABSTRACT - Non-parametric optimization methods are an efficient way to find an optimal design on a very early phase of the design process. Topology optimization may be used to find new design proposals based upon a maximum design space. Shape optimization is reducing local stress peaks or increasing components lifetimes by modifying the components surfaces only. Bead optimization increases the stiffness of sheet metal structures by introducing stiffeners. These different types of structural optimization are available in the non-parametric optimization software TOSCA. A completely new approach of executing preprocessing for non-parametric optimization tasks will be presented in the ANSA environment. The intuitive optimization environment provides an easy-to use optimization preprocessor that supports the complete design process. The use of the application will be demonstrated on existing examples of structural optimization.

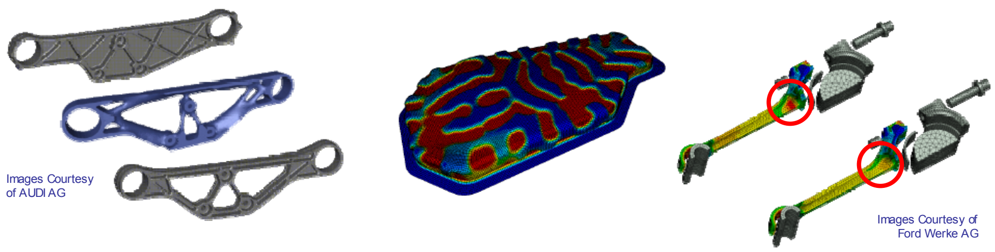
TECHNICAL PAPER -

1. INTRODUCTION

The development of lightweight designs is a quite challenging task. Not only the target has to be fulfilled but also all relevant constraints and restrictions have to be considered. The engineer is faced with a large number of requirements during the design phase of a new component. Different optimization methodologies exist, which are helping the designer and analyst to design right the first time. Design proposals may be found that are already optimal considering different requirements. But not only the optimization algorithms must be powerful and able to solve the optimization problem. The complete Optimization procedure must be available in an easy to use Optimization environment.

2. THE OPTIMIZATION SYSTEM TOSCA

A non-parametric optimization approach as used in the program TOSCA allows one to use standard industrial CAE-solvers (e.g. ABAQUS, ANSYS, MSC.Nastran, MSC.Marc, NX Nastran, or Permas) for solving large-scale optimization problems in the fields of topology, shape and bead optimization. Using existing CAE solvers for solving optimization problems is essential in the industry. Additionally to the finite element input deck, an additional file containing the optimization task definition has to be defined. All objectives, constraints and existing manufacturing restrictions have to be defined here.



Transverse Link optimized with **TOSCA.topology** | Oil Pan optimized with **TOSCA.bead** | Connection Rod optimized with **TOSCA.shape**

Fig. 1: Optimization methods available in TOSCA

TOPOLOGY OPTIMIZATION

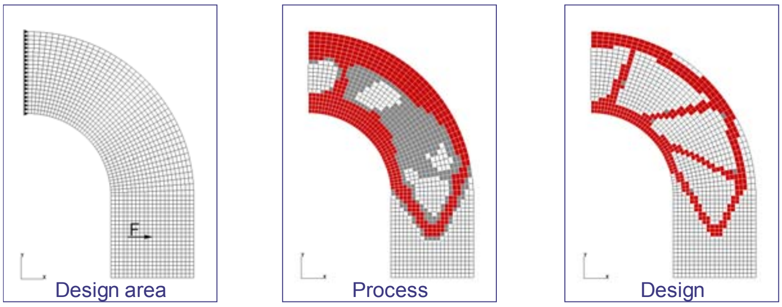


Fig. 2: Topology optimization

Topology optimization is used to find a design proposal in a very early design phase. The maximum allowed design space has to be available as finite element input deck. All relevant loadcases have to be applied on the design space model. A typical objective for topology optimization is the minimization of compliance corresponding to maximize the stiffness using a mass constraint for a given amount of material. Also the dynamic behaviour of components may be a target of the optimization by specifying objectives or constraints on the eigenfrequencies. The optimization of the mechanical behaviour of the components is very often combined with restrictions from the manufacturing. For casting parts, undercuts must be avoided in draw direction in order to be able to manufacture the part by casting. For some manufacturing types, restrictions on the minimum or maximum wall thickness have to be applied.

SHAPE OPTIMIZATION

Shape optimization is used in a later phase of the design process. The components topology remains unchanged in shape optimization. Only the components surface is modified in order to reduce local stresses or to increase the components lifetime. In TOSCA, a non-parametric optimization approach is implemented. No parameterization of the part is necessary. Each node on the surface of the component is a possible design node. A powerful mesh-smoothing algorithm is adapting the mesh after the design node movement in order to keep a good quality mesh for the analysis.

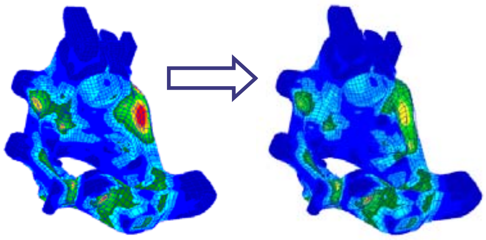


Fig. 3: Shape Optimization

BEAD OPTIMIZATION

Bead optimization is used to find the optimal bead layout for sheet metal parts. For complex loading conditions, a good bead design is hard to find. The bead optimization module helps to find a good location and orientation of bead stiffeners in a sheet metal structure. In bead optimization, the dynamic behaviour of the components is the most important target.

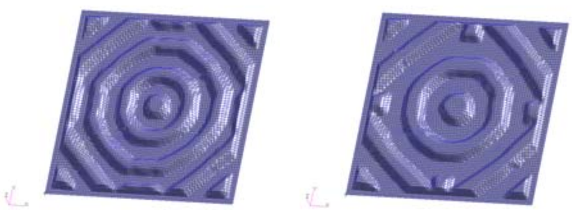


Fig. 4: Bead optimization

3. INTERACTIVE PREPROCESSING OF TOSCA TASKS IN ANSA

The definition of an optimization task contains the definition of the optimization target by defining the objective function and constraints, and the definition of existing restrictions concerning the allowed modification of the design variables. These definitions have to be made additionally to the setup of the finite element model for the analysis.
The new environment for the setup of Optimization Tasks in ANSA is designed that the user is guided by a task manager that defines the step-by-step procedure.

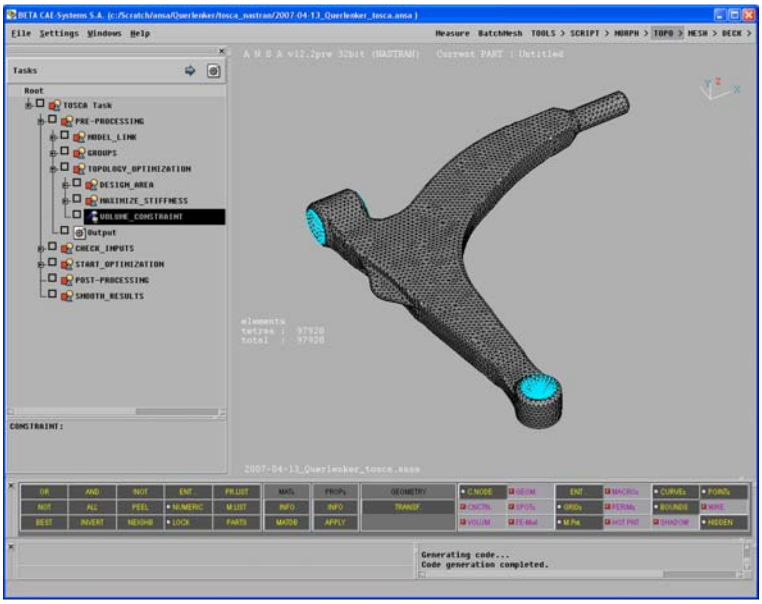


Fig. 5 : TOSCA Environment in ANSA

PRE PROCESSING

The complete preprocessing of the optimization task for topology, shape or bead optimization is executed. First, the link to the finite element model, which is used for optimization, is generated. The corresponding finite element model is loaded and visualized in the working area.
The user may now specify groups, which should be used for the optimization definition. Typical groups are design nodes, design elements, or groups for which restrictions have to be defined. If the groups are already defined in the finite element deck, they are imported into the database and may directly be used for the setup of the optimization task.

Dependent on the selected type of optimization, the design variables are defined and all necessary manufacturing constraints may be added as design variable constraints.
The definition of the objective function and constraints is added to finish the setup of the optimization task.

CHECK INPUTS

All dependencies are automatically checked while output the model. The user can additionally add additional checks e.g. to check complex restriction definitions for shape optimization. The checks are executed after the update of the preprocessing task.

START OPTIMIZATION

The next step is to execute the complete optimization. This may be done interactively but the more convenient way is to output the necessary input files and to start the optimization on a computation server. The optimization is an iterative procedure, where the finite element solver and the TOSCA optimization module is launched in each design cycle. The modification of the input file is made by the TOSCA application dependent on the selected type of optimization.

POST-PROCESSING

After the optimization is executed, the results are prepared for post-processing. Additionally to the finite element results, specific optimization results are provided. For topology optimization, the material distribution is output on the mesh of the design space. For shape and bead optimization, the optimization movement is output for post-processing.

SMOOTH RESULTS

The topology optimization results in very rough design proposals, where elements from the initial design space model have a very low density and are hidden for post-processing. A result based smoothing of these results allows generating a smooth surface distribution of the topology optimization results. These smoothed surfaces may be used for further processing of the results in the design process.
Additionally to the smoothed surfaces, cutting splines may be requested in order to be able to transfer the optimization results to CAD systems. The user selects a coordinate system axis and the cutting splines are calculated normal to the selected axis. An arbitrary number of cuts may be requested, that is output as Iges Splines to be imported into different CAD System for generation of new CAD data.

4. OPTIMIZATION EXAMPLE: WISHBONE

The example to be optimized is a wishbone. The optimization target is to reduce the weight of the component, and not to exceed the maximum stresses in the design space. The proposed workflow is to start with a topology optimization in order to find a minimum compliance design for a certain target volume. The next step is the redesign and validation of the optimized component considering the von Mises stress distribution. To reduce the Maximum stresses in the design, a shape optimization is executed at the end to minimize local stress peaks in the design.

GENERATION OF DESIGN SPACE MODEL

The first step in topology optimization is to determine the maximum available design space. A finite element model of the design space is generated and all loads and boundary conditions have to be applied to the design space model. In our example, a predecessor design existed. The cavities where filled with material in order to have a larger design space for the optimizer.

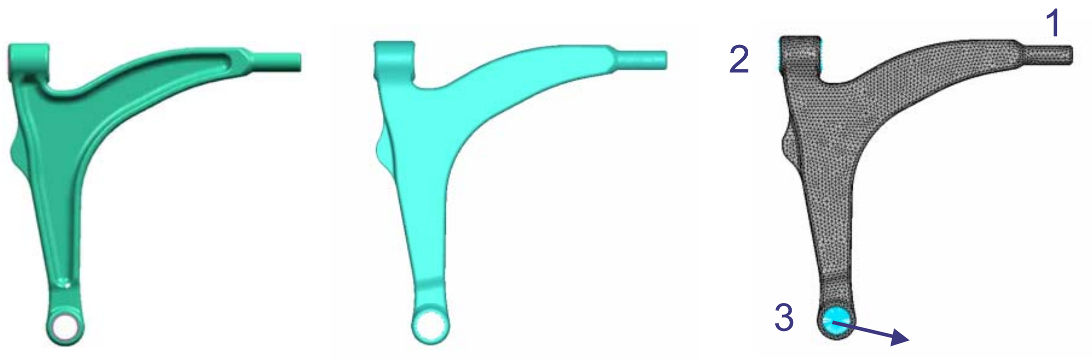


Fig. 6: Initial Design, Design Space Geometry and Finite Element Mesh for Topology Optimization

The design space model has to be prepared for the finite element analysis. All relevant loadcases for the layout of the component have to be applied. For this example, the following loading conditions are applied for the 3 loading points:

Point 1: Fixed in direction 2,3,5,6
Point 2: Fixed in direction 1,2,3,5,6
Point 3: Braking load

TOPOLOGY OPTIMIZATION

Additionally to the finite element deck, the definitions for the optimization have to be made. This information may be setup in the TOSCA.environment.

The first step is the definition the design area. The density of each element in the design area is a design variable that changes during the iterative modification procedure. For the wishbone example, additional design variable constraints have to be defined, that restrict the modification of the design variables. The areas of the 3 loading points and the area of the bracket are frozen for the optimization procedure. Additionally, a demolding constraint is activated with a parting plane lying in the symmetry plane of the structure. The wishbone is produced by a forging procedure so no undercuts are allowed for the design proposal.

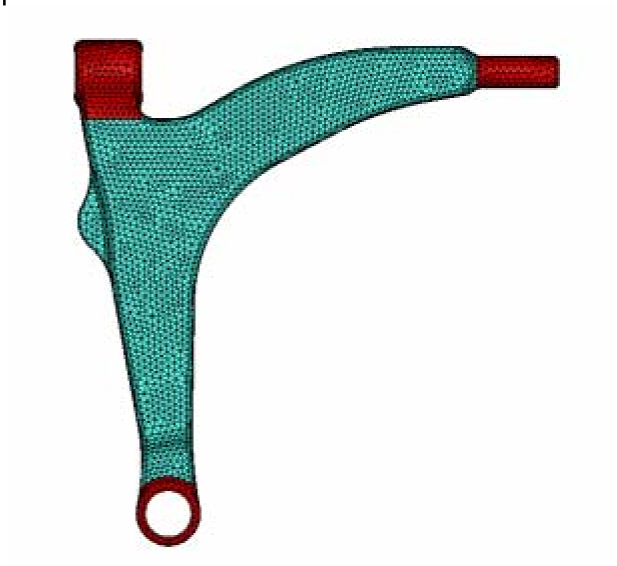


Fig. 7: Frozen Elements (red) for Topology Optimization

The objective is to find a minimum compliance design with a constraint on the volume. Topology optimization is started and the iterative optimization procedure is launched.

POST-PROCESSING OF TOPOLOGY OPTIMIZATION RESULTS IN μETA

Topology optimization results of TOSCA may be post-processed in different ways. During the optimization loop, the finite element solver is executed in each design cycle with a modified input deck having the actual property values for the elements. So for each design cycle, a native finite element results file exists if the user selected to keep all result files. These files may be used for a standard post-processing. Additionally, TOSCA exports the additional topology results. The material properties are exported to a VTF file that may be visualized with the TOSCA.view module that is used for a quick post-processing of the results. These specific results for topology optimization may also be exported to result files that can directly be imported by μETA. In μETA, the material distribution may be visualized as contour plot, iso-surfaces and also animations may be generated.

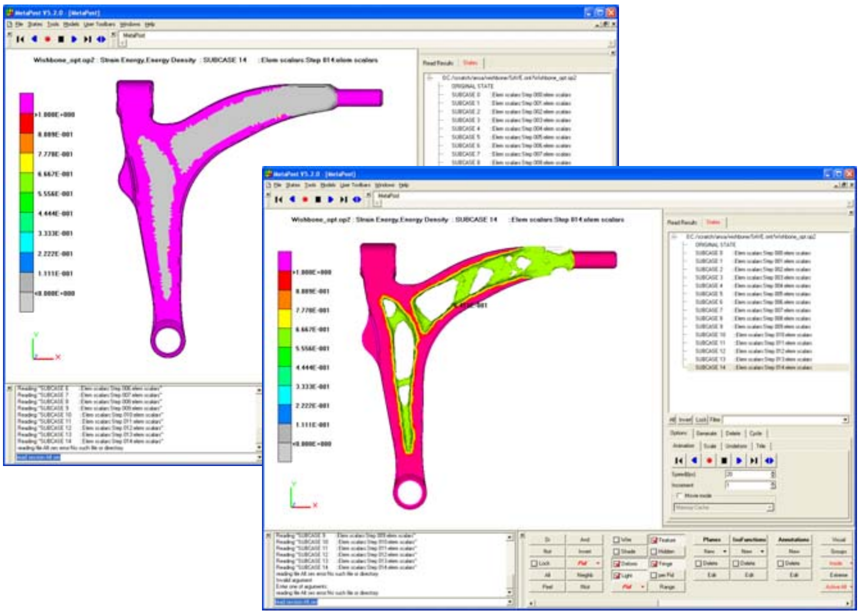


Fig. 8: Contour Plot of Material distribution and Iso-surface in μETA

SMOOTHING TOPOLOGY RESULTS

The smoothing procedure generates a new surface mesh from the material distribution of the results from topology optimization. All void elements are now removed from the structure and a new component for transfer to CAD or for the generation of a new finite element deck is generated.

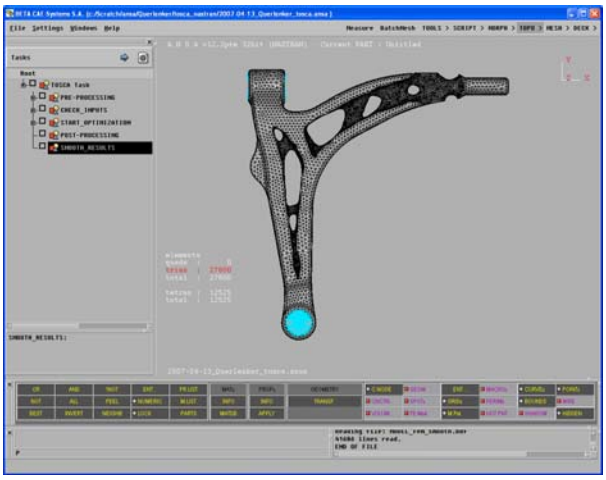


Fig. 9: Smoothed design of the wishbone imported in ANSA

VALIDATION OF TOPOLOGY OPTIMIZATION RESULTS

After the smoothed design is imported into ANSA, the surface mesh quality is checked and if necessary a reconstruct is performed. The so-generated shell mesh is used for the generation of a new solid tetra mesh that may now be used for a validation run. All boundary conditions from the initial design space model are applied and a new analysis is executed.

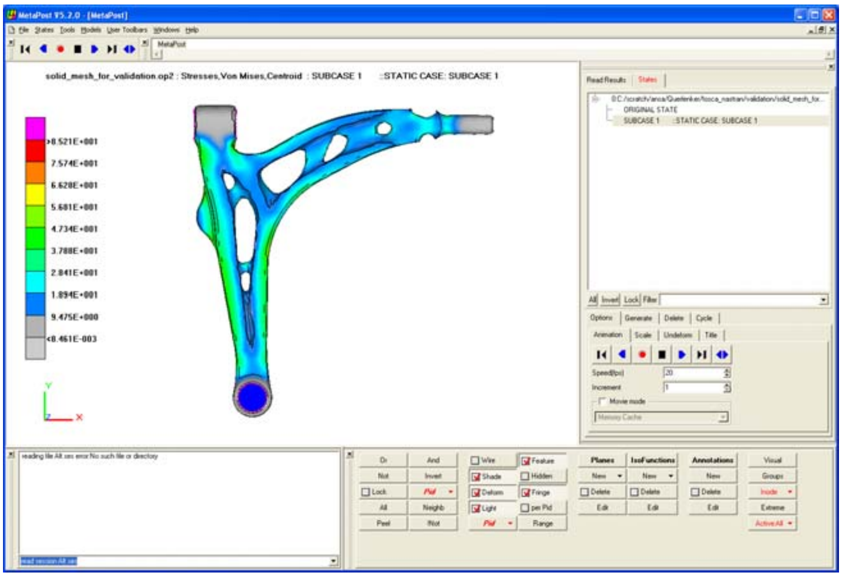


Fig. 10: Post-processing of validation run in μETA

GENERATION OF GEOMETRY

Additionally to the generation of a surface mesh, geometry information may be generated by using TOSCA.smooth. For the wishbone, 80 cuts in x and y direction as well as 10 cuts in z-direction are calculated. For each cut through the model, a cutting spline on the surface of the smoothed component is output as parametric or interpolated cubic spline in IGES format.

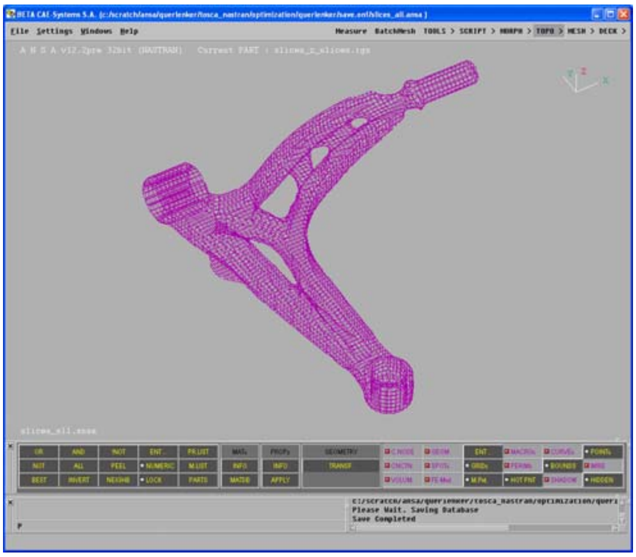


Fig. 10: Cutting splines for CAD transfer

New geometry information is generated from the results of topology optimization. During the generation of the new geometry, all detailed requirements for manufacturing are taken into account and the design proposal from topology optimization is used as design draft.

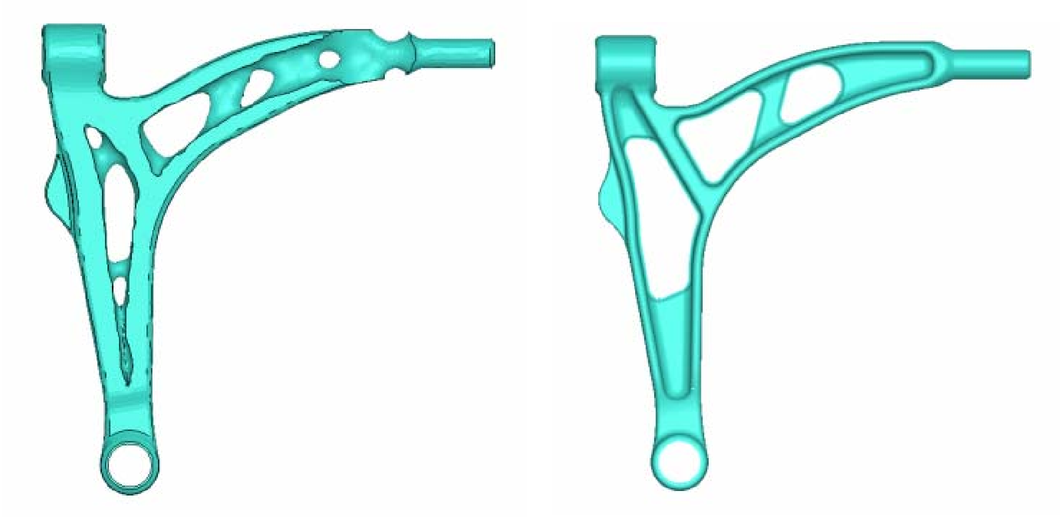


Fig. 11: Generation of new geometry from design proposal

5. CONCLUSION

Non-parametric structural optimization is an efficient tool for the improvement of components. Additionally to the optimization algorithms, the handling for the setup of the optimization task plays an important role. The setup of the optimization task should be made in an intuitive way. The new TOSCA environment is a tool for the guided setup of the topology, shape or bead optimization task in TOSCA. But not only the preprocessing is supported – the complete optimization process starting with the setup, including the start and post-processing of the optimization and the smoothing and CAD transfer is included. This environment helps the user to setup and control the optimization tasks in an efficient way.
The TOSCA optimization environment in ANSA supports the definition of topology, shape and bead optimization tasks. The complete optimization workflow may be setup including the smoothing and generation of a new solid finite element mesh from topology optimization to perform a validation analysis. In case of areas of critical stresses after topology optimization, a shape optimization task may directly be defined in order to optimize the components shape.

REFERENCES

(1) M.P.Bendsoe, O.Sigmund (2003), Topology optimization Theory, Methods and Applications, Springer

(2) Maute, K. (1998) 'Topologie- und Formoptimierung von dünnwandigen Tragwerken', University of Stuttgart., Institute of Structural Mechanics, Ph. D. Theses.

(3) Bendsoe M. P., Kikuchi N: Generating Optimal Topologies in Structural Design Using a Homogenization Method, Computer Methods in Applied Mechanics and Engineering, 71_197:224, 1988

(4) Meske, R., Sauter, J., Güngör, Z. (2001) 'Recent improvements in topology and shape optimization and the integration into the virtual product developments process', NAFEMS World Congress, Como (Italy).