

## INTEGRATING THE ABILITY FOR TOPOLOGY OPTIMIZATION IN A COMMERCIAL CAD-SYSTEM

Schmelcher, Johannes; Stetter, Ralf; Till, Markus

University of Applied Sciences Ravensburg-Weingarten, Germany

### Abstract

An promising approach in optimizing parts with respect to their weight and simultaneously preserve their functionality, is designing parts in a way, where material is only present in regions, which are important for the functionality, and all other regions which do not contribute to the stiffness remain void. This is called topology optimization and leads to a design which is optimal concerning the distribution of forces.

Since these benefits of topology optimization are readily known, research activities are very high in the past decades. Numerous methods have been developed and existing methods have been further advanced. Research activities can be divided into density-based methods, hard-kill methods, boundary variation methods and biologically inspired methods. One aspect which is not the focus of papers currently published is the integration of the actual optimization method into the product development process. Therefore this work focuses on the integration of topology optimization into product development processes by developing an easy deployable topology optimization tool which is completely integrated in the CAD system Creo Parametric 2.0.

**Keywords:** Computational design synthesis, Computer aided design (CAD), Design for X (DfX), Optimisation

### Contact:

Prof. Dr.-Ing. Ralf Stetter  
University of Applied Sciences Ravensburg-Weingarten  
Mechanical Engineering  
Germany  
ralf.stetter@hs-weingarten.de

Please cite this paper as:

Surnames, Initials: *Title of paper*. In: Proceedings of the 20th International Conference on Engineering Design (ICED15), Vol. nn: Title of Volume, Milan, Italy, 27.-30.07.2015

# 1 INTRODUCTION

Topology optimization has gathered a wide acceptance and application in the design of products and their components. Since products nowadays have to fulfil a variety of guidelines and laws (e.g. guideline 2005/32/EG (EuP), ecodesign regulation), a multitude of requirements have to be considered in the product development process. Therefore processes like VDI 2221 are not run through in a linear manner but with iterative loops, leading to the necessity of creating models and performing analyses multiple times. Regarding topology optimization, this means that extra work has to be done in order to create simulation models and transform the results in new design proposals. These drawbacks are widely accepted, since by using topology optimization in early stages of product development process significant savings in weight can be achieved (Krog et al. 2011).

Since the benefit of topology optimization for the final product properties are readily known, research activities are very high in the past decades. Numerous methods have been developed and existing methods have been further advanced. But one aspect, which is not the focus of papers currently published, is the quite important integration of the actual optimization method into the product development process.

Therefore this work focuses not on developing new methods of topology optimization or advancing existing ones. In fact, this paper focuses on the integration of topology optimization into product development processes by developing an easy deployable tool which is completely integrated in the CAD system Creo Parametric 2.0 and hence seamlessly embeds topology optimization into existing workflows.

## 2 THEORETICAL BACKGROUND

### 2.1 State of the art

Modern CAE programs offer various possibilities for supporting the developer in the development process with simulation tools for nearly every case of application. This enhancement of the virtual prediction of product performance and properties comes along with a reduction of iteration loops and hardware prototypes.

Especially in early stages of the product development process, where the main goal is not the detailed simulation of producibility but the evaluation of the geometry concerning loads and boundary conditions, fewer tools are available. Topology optimization will optimize the design itself, so an extended concept of integrating the original and resulting geometry is required. In order to give an overview over the available software tools, a short survey of these programs based on their functional characteristics is given. According to the information available in their respective datasheets the programs can be classified into three different groups.

The first group can be formed of the tools, which are directly integrated in the current CAD systems and therefore offer the advantage of the intuitive usability due to the familiar graphical interface. As representatives of this group NX 8.5 or Creo Parametric 2.0 could be mentioned. They offer the possibility of specific variation of dimensionings in order to find an enhanced objective function due to a better shape of the part. Mostly the objective functions concern stress peaks or natural frequencies. The calculated shape of the part by using these optimization processes resembles the one in the original state, since only dimensions have been altered. This implies that only local, not global optima of the objective function can be achieved and therefore this group of tools is not as powerful as proper simulation tools.

Tools which usability is similar to those of proper CAD systems but which are independent programs can be merged into the second group. For example Altair HyperShape/CATIA or CATOPO should be mentioned which connect directly to CAD systems and thus enable undisturbed workflows and efficient optimization processes. Those tools do not only vary dimensions of parts but compute a design proposal which is optimal regarding the loads and boundary conditions. So they are capable of doing real topology optimization but with the drawback that the optimized part has to be retransferred to the CAD system via interfaces.

The last group subsumes the current simulation tools, which are exclusively used for simulation purposes and thus very powerful. Members of this group could be, for instance Abaqus 6.13 or ANSYS 14.5, and they all offer possibilities for extensive topology optimization processes. The enormous advantage of these tools is the range of functions allowing the user to exert great influence

in the configuration of the optimization model and thus the quality of the optimized part. This leads necessarily to a more complex usability which is why these tools are not employed in parallel to the current engineering process.

This short survey of CAE programs capable of performing numerical optimizations shows that no solution is available which enables the user to optimize his/her part in parallel to the design process and to fulfil the parametrical basis of CAD systems.

## 2.2 Optimization disciplines

According to Bendsøe and Sigmund (2003) the field of numerical optimization disciplines can be divided into three different categories (compare Figure 1).

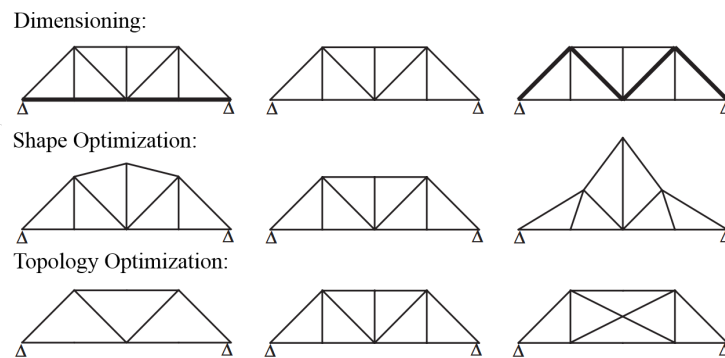


Figure 1. Different optimization disciplines

### Dimensioning:

The optimization process by means of variation of dimensions e.g. the thickness of metal sheets or the cross sectional areas of beams or rods typically take places for truss-like or structure like components (Harzheim 2008). The goal of this way of optimization is to minimize the compliancy or to maximize the stiffness with regard to distinctive boundary conditions. These boundary conditions could be the maximum allowable total volume or the total weight of all components or the upper or lower limit of cross sectional area (Christensen and Klarbring 2009).

### Shape Optimization:

The principle of this method is to optimize a component due to modifications performed on the shape of the part, whereas the actual topology remains unchanged (Schumacher 2013). The purpose of this optimization process is to change the component shape in order to reduce stress peaks and therefore improve the endurance strength respectively the durability (Harzheim 2008). Compared to the dimensioning, shape optimization is considerably more intricate since the variables in the optimization process do not only represent thicknesses or cross sectional areas but complex shaped boundary curves. One possible approach is to perform the optimization process on the basis of the FE mesh by means of specific modification to the boundary nodes (Harzheim 2008). Another approach uses mathematical functions to describe the boundary curves or surfaces, which are directly controllable by the use of variables.

### Topology Optimization:

While the shape optimization changes only boundary areas and not the actual topology of components, topology optimization modifies the whole design of the component. This implies that also boundary areas are changed and thus topology optimization includes shape optimization. The intention of topology optimization is to determine the structure of a component, which possesses either minimal compliancy with maximum stiffness or an acceptable compliancy with minimal total weight (Bendsøe and Sigmund 2003). This optimization problem can be solved by employing an approach developed by Bendsøe and Kikuchi (1988), which determines the optimal distribution of material in an available space. This available space of the component has to be discretized by means of a FE mesh whereby a microstructure is assigned to each finite element. The theoretical microstructure can be seen as a porous material containing holes in a specific percentage. To influence the material properties of the microstructure, the proportion of solid and void areas and the shape of void areas can be changed. By adjusting the anisotropic material properties of every single finite element, the purpose of the optimization process is, to form continuous areas where the elements are either solid or void (Schumacher 2013). If anisotropic material properties are not required or even undesired, the attempt

to describe the finite elements as microstructures can be dropped. In this case the characterization takes place solely using young's modulus, which is proportional to the density of each finite element. This correlation is nonlinear and ensures that the densities take only the values "0" (void) and "1" (solid) and not intermediate values. Thus the quantity of variables decreases to only one variable per finite element, resulting in an efficient way of computing the optimal structure (Rozvany 2001). Due to the isotropic material performance and the nonlinear correlation between the density and the young's modulus, this approach is called *Solid Isotropic Material with Penalization (SIMP)*.

### 2.3 Theory of the SIMP-Method

The most important information for the optimization process is the desired volume of the optimized structure or the volume fraction of the optimized structure compared to the original structure. If a finite element gets filled with material another element has to get void in order to satisfy the volume constraint. Additionally information about the load and boundary conditions are needed. As objective function, which is used to determine whether the structure is stiff or too flexible, the strain energy can be used. If the strain energy is low, the compliancy of the structure is low and thus the stiffness is high and vice versa. Equation (1) describes the strain energy of the system

$$G(\rho) = \mathbf{u}^T * \mathbf{K} * \mathbf{u} \quad (1)$$

where  $\mathbf{u}$  is the deformation vector and  $\mathbf{K}$  the stiffness matrix of the system. A volume constraint has to be introduced to avoid, that the optimization process seeks the trivial but mechanically uninteresting solution of the complete filling of the available space. The mentioned correlation between the densities  $\rho$  of the finite elements and their young's modulus  $E$  can be described using an equation where the potency of the densities with a penalization factor  $p$  is multiplied with the young's modulus  $E_0$ . This results in the behaviour of the young's modulus that intermediate densities are uneconomically concerning the gain of stiffness compared to the additional weight. Therefore the densities will tend to the limits of the allowed densities, when the penalization factor is chosen sufficiently high.

$$E = \rho^p * E_0 \quad (2)$$

Since the densities of the finite elements are not allowed to get greater than "1", a constraint for the upper limit of the densities has to be introduced. Also a constraint for the lower limit of the densities has to be formulated to prevent the stiffness matrix getting singular due to densities less or equal "0". So the whole optimization problem can be described as follows.

$$\begin{aligned} \min \{ & G(\rho) = \mathbf{u}^T * \mathbf{K} * \mathbf{u} \} \\ \text{s.t. } & \mathbf{K}(\rho) * \mathbf{u} = \mathbf{f} \\ & r^M = \sum_{\text{Elements}} \rho_e * V_e - M_{\max} \leq 0 \\ & r_i^{\rho+} = \rho_i - 1 \leq 0 \\ & r_i^{\rho-} = \rho_{\min} - \rho_i \leq 0 \end{aligned} \quad (3)$$

The deformation vector multiplied with the stiffness matrix has to form the external forces acting upon the system, whereas the quantities  $r^M$ ,  $r_i^{\rho+}$  and  $r_i^{\rho-}$  denote the material constraint and the upper respectively lower limit of the densities. Here  $\rho_e$  and  $V_e$  are the density and the volume of a single finite element while  $M_{\max}$  stands for the maximum allowed material of the optimized structure.

The mathematical procedure of solving this rather complex optimization problem is the optimality criteria method. Due to the similarity to development processes of engineers, where changes are evaluated and conclusions are drawn with regard to the better fulfilment of requirements, the optimality criteria method suits well for the SIMP-method (Spillers and MacBain 2009). The requirements for the optimality criteria method are the mathematical description of the optimality criteria and an algorithm depicting the style of modifications to the structure in order to achieve the optimality criteria (Hafka and Gürdal 1992). The optimality criteria can be developed using a Lagrange Multiplier, where the objective function and the constraints can be merged into an auxiliary function whose first partial derivatives vanishes at the extreme point. Thus the following relationship dependent of the Lagrange Multiplier  $\lambda$ , has to be applied.

$$B_i^k = \frac{p * \rho^{p-1} * E_0 * \mathbf{u}_i * \mathbf{K}_{i,0} * \mathbf{u}_i^T}{\lambda * V_i} = 1 \quad (4)$$

Using this condition for optimality, Bendsøe and Sigmund (2003) developed a heuristic update scheme, which adjusts the density from one iteration step to the other independently from the densities of the other finite elements.

$$\rho_i^{k+1} = \begin{cases} \max\{(1 - \xi) * \rho_i^k, \rho_{min}\} & \text{if } \rho_i^k * (B_i^k)^\eta \leq \max\{(1 - \xi) * \rho_i^k, \rho_{min}\} \\ \min\{(1 + \xi) * \rho_i^k, 1\} & \text{if } \rho_i^k * (B_i^k)^\eta \geq \min\{(1 + \xi) * \rho_i^k, 1\} \\ \rho_i^k * (B_i^k)^\eta & \text{else} \end{cases} \quad (5)$$

Here  $\xi$  denotes the maximum change of the density from one iteration step to the other and  $\eta$  acts as a numerical damping coefficient. As previously mentioned this update scheme is purely heuristic and only by means of numerical experiments the functionality has been proven. Only Ananiev (2005) made attempts to analyze the update scheme and he could verify the mathematical correctness.

The structure, which can be computed using the update scheme, depends strongly upon the FE mesh. This results in different optimal structures, when different element sizes are used for discretization. To cope with this issue, the partial derivatives are modified by a weighted average of the partial derivatives in a distinctive radius around a certain finite element. A side benefit is the avoidance of checkerboard-like structures leading to non-realistic structures (Bendsøe and Sigmund 2003).

### 3 DEVELOPMENT OF THE 2D-OPTIMIZATION-MODULE

Due to the lack of tools allowing the engineer to perform topology optimization in the design process, the development of this optimization module focuses on a tight integration in the CAD system Creo Parametric 2.0. Therefore as many features provided by the CAD system as possible are used. The 2D-Optimization-Module is dedicated for optimizing flat components e.g. sheet metal parts. Since these components are often manufactured using laser-cutting or waterjet-cutting technologies it must be ensured that the resulting geometry is suitable for these manufacturing processes. To guarantee this, the three-dimensional component is reduced to a two-dimensional plane in which the optimization process takes place. Therefore forces perpendicular to this plane are ignored in the optimization process. Furthermore, the three-dimensional optimization of sheet metals with a small thickness compared to the length and width is uneconomic since the increasing costs for manufacturing do not result in significant savings in weight.

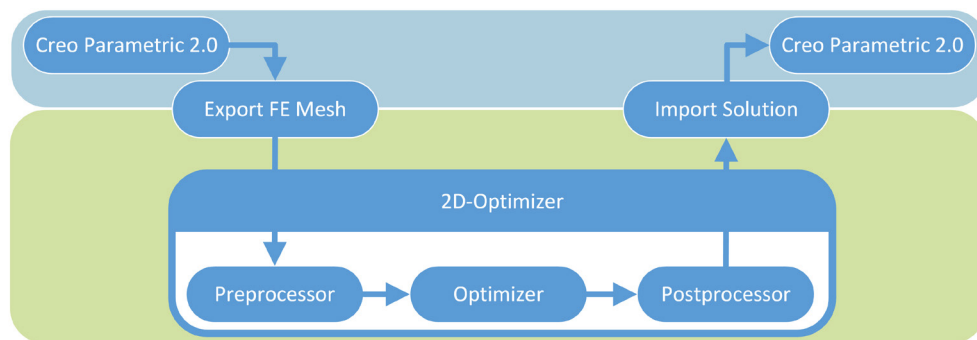


Figure 2. Procedure of the 2D-Optimization-Module

The close integration starts with the modification of the ribbon (set of toolbars), which is expanded by a button to switch from the normal modelling environment of components to the extended FE-mode. Here the user has to define the loads and boundary conditions according to the stresses and strains in reality. After he/she has started the actual optimization process by clicking another button added to the ribbon, the optimization module checks the model to prevent failures due to missing loads or boundary conditions. Additionally mesh controls have to be added automatically to the simulation model since Creo Parametric 2.0 uses the P-method for computing strains and therefore the FE mesh is very coarse. However the FE mesh has to be adequately fine in order to get to appropriate solutions in the optimization process. The FE mesh of the fine discretized component and the applied loads and boundary conditions are automatically exported in an ANSYS format text file. Due to the focus of the optimization module on improving sheet metal parts or components which are flat, the three-dimensional FE mesh has to be transformed in a two-dimensional FE mesh. Therefore not linear tetrahedron but constant strain triangles are used for assembling the global stiffness matrix. After the

completion of the optimization process the FE mesh consists of elements, whose densities are nearly or equal to “1” and element with densities near to “0”. In order to ensure an adequate automatic creation of features in Creo Parametric 2.0, regions where the densities of finite elements are less than a specific value are combined. These regions are described by one curve which surrounds all elements of the specific region and is used to create extrusion features.

Upon completion of the automatic creation of extrusion features, the CAD model contains the optimized structure according to the SIMP-method and concurrently preserves the parametrical basis of CAD systems (compare Figure 2).

### 3.1 Exporting the FE mesh

The first step, which influences the resulting geometry most, is that the user has to define the loads and boundary conditions. These assumptions have to be as realistic as possible otherwise the design proposal will not be realistic and the component will fail under real life conditions.

Once the user has defined the loads and boundary conditions according to reality and started the optimization-module, the first of the three program parts is executed, which realizes the interface between Creo Parametric 2.0 and the 2D-Optimizer. To guarantee that the optimization process does not abort due to missing loads or boundary conditions, the first task is to check whether the FE model is defined properly or not. For this purpose a HTML document is created automatically, which summarizes the FE model properties. It contains information concerning mesh controls, loads and boundary conditions, and can therefore be parsed in order to find particular keywords indicating the presence of the corresponding FE model feature. If at least one of the required features cannot be found, the further program flow is cancelled and the user gets indicated that certain features are missing (Figure 3).

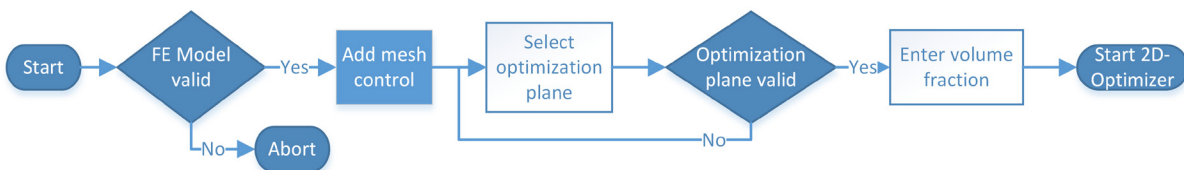


Figure 3. Program sequence of Export FE mesh (required user inputs are depicted unfilled)

After the validation of the FE model has succeeded, a mesh control has to be added in order to restrict the element size dependent on the component’s volume. First the TOOLKIT function *ProSolidMassPropertyGet* is used for automatically determining the volume and subsequently computing the required element size due to a favoured number of elements of the FE mesh. Since the shape of tetrahedrons in a FE mesh can be distorted massively, the computation of the required element size is only an estimation on the basis of a perfect tetrahedron. The value of the edge length of a perfect tetrahedron is assigned to a mesh control ensuring that no element of the FE mesh possesses an edge length greater than this value. The intended value of elements in the FE mesh is 100,000 because of the suitable size of the text file and the relatively low computation time. The FE mesh with the loads and boundary conditions can be exported in an ANSYS-format text file using the TOOLKIT function *ProFemmeshExport*.

Due to the restriction of only optimizing flat components, the whole optimization process can be reduced to two dimensions since forces in normal direction to the plane can be assumed constant. The optimization plane cannot be determined automatically, so the user is prompted to select one. An appropriate plane requires that the loads and boundary conditions are adjoining to the optimization plane and the load vector lies in the plane. Additionally it has to be considered that only surfaces of the component can be chosen, which are parallel to one of the three coordinate planes. The type of the selected surface can be determined by the function *ProSurfaceTypeGet* and only the type *PRO\_SRF\_PLANE* is accepted. Once an acceptable surface has been chosen, it has to be checked, whether the surface is parallel to a coordinate plane or not. Therefore the normal vector of the surface is identified and only if one component is equal to one, the plane is parallel to a coordinate plane and accepted. If the selections made by the user are not valid, he is prompted to select another surface. Finally the user is requested to enter the desired volume fraction of the optimized component compared to the original one. The values permitted to be entered are 0.01 to 0.99 and Creo Parametric 2.0 verifies if these limitations are satisfied, otherwise the user has to repeat his input.



Once all information required for the actual optimization process are available, the 2D-Optimizer is started via the command-line. Additional information are transferred to the starting program which includes the storage location of the text file, the direction of the optimization plane and their distance to the point of origin and finally the volume fraction.

### 3.2 2D-Optimizer

Once the 2D-Optimizer was started by the program part Export FE mesh, the information passed via command-line argument are itemized and transferred to corresponding variables. Then the preprocessor of the 2D-Optimizer is started.

#### 3.2.1 Preprocessor

The pre-processor transforms the FE mesh stored in an ANSYS-format text file, from a three-dimensional tetrahedron mesh into a two-dimensional triangle mesh. For this purpose it is necessary to read the text file, filter the containing information and store them in variables. Due to the structured setup of the text file all required information can be read in line by line. By means of a specific string preceding each line, the information can be distinguished by nodes, elements, forces or boundary conditions. The program sequence is shown in Figure 4.

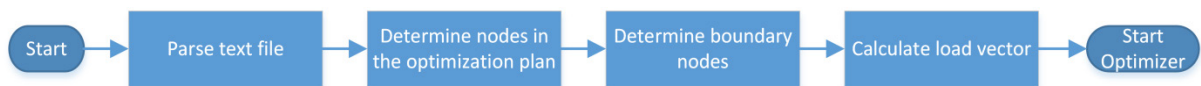


Figure 4. Program sequence of the preprocessor

Since the FE mesh has to be transformed into a mesh made of constant strain triangles, the tetrahedrons have to be identified where one of the four triangles lies in the optimization plane. This can be done by checking the three nodes of each triangle whether the nodes coincide with the optimization plane or not. If this verification is positive the information are stored and later used for constructing the element stiffness matrix of the constant strain triangle.

For calculating the load vector the boundary nodes have to be determined since loads are only allowed to act on the edge of the optimization plane. Due to the special build-up of the triangular mesh, every triangle can be reviewed independently from the others. If one triangle is in touch with other triangles on all three sides, the element is not a boundary element but an inner one. So if a triangle has no contact to another triangle on one side it is a boundary element and the information concerning the nodes are saved. After sorting the boundary nodes so that geometrically adjacent nodes are also stored adjoining in the variable the prerequisites are fulfilled for computing the load vectors. Each geometrically coherent area of loads acting upon boundary nodes are summarized to a separate load vector so that as many load vectors are created as coherent areas of loads are existing.

#### 3.2.2 Optimizer

The actual optimization process takes places within a conditional loop. The execution of the loop depends on the maximum density change of a finite element for the last iteration step. If this change is less than 0.01, it is assumed that the iterative solution has converged and a structure has developed consisting only of void and solid finite elements.

The process begins with the assembling of the global stiffness matrix using constant strain triangles (Figure 5).

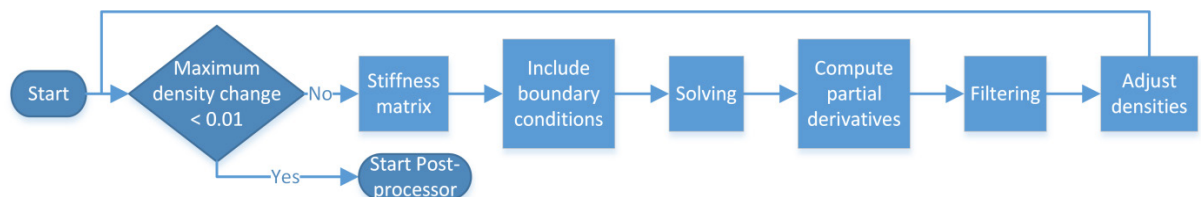


Figure 5. Program sequence of the optimizer

This element is capable of modelling a plane stress state like the one produced by reducing the three-dimensional problem to two dimensions. Indeed the linear displacement assumption leads to constant stresses and strains within the element, but therefore the computational cost are lower. This must not

be underrated since several hundreds of iteration loops are necessary with at least one FE analysis even for simple components. After the boundary conditions have been added to the global stiffness matrix, the actual process of solving the system of linear equations can be carried out. For this purpose an iterative equation solver is employed which operates on the principle of conjugated gradients. Following this classical FE analysis, the actual optimization process starts with the computation of the partial derivatives of equation (1) with respect to the densities of each finite element. These derivatives are filtered in order to avoid mesh-dependent solutions and to prevent the formation of checkerboard-like structures. Finally the densities of the finite elements have to be adjusted according to the update scheme presented in equation (5). If only minor density changes occur, the process has converged and the postprocessor is being called.

### 3.2.3 Postprocessor

After the iteration process has completed, the component consists of finite elements whose densities are “0” and “1”, but there are transient areas with values in the range of 0 to 1. The next task is to extract all elements which tend to have densities near zero. Due to the transient area the formation of continuous areas with a fixed density-limit is inappropriate. Therefore, the user can control the density-limit. The initial value of the density-limit is 0.3 since this has proven to be a convenient value leading to suitable structures. For computing the boundary curves, a similar approach to the determination of boundary nodes can be used. Therefore the density of the adjoining element is considered. With this approach it is possible to identify the nodes depicting the boundary between elements whose density is greater than the limit and elements whose density is less than the density-limit. The resulting volume matches not exactly the desired volume fraction but differs slightly (Figure 6).

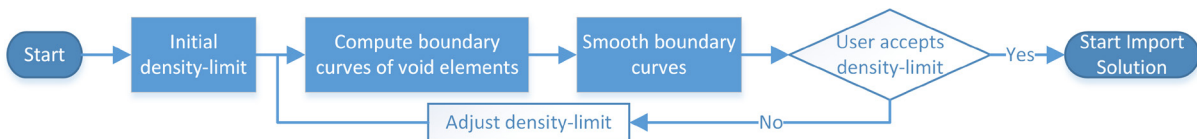


Figure 6. Program sequence of the postprocessor (required user inputs are depicted unfilled)

To ensure that the boundary curves are rather smoothly, a laplacian filter is used, which modifies the coordinates of a node through the weighted average of the two adjacent ones. Since the single use of the filter produces not sufficiently smooth curves, the filter is applied three times. If the user is satisfied with the shape of the boundary curves, he accepts the density-limit and the backtransformation process of these curves to Creo Parametric 2.0 is started.

### 3.3 Import Solution

Once the actual topology optimization process has finished and the points which are defining the splines are stored in a text file, the modification of the CAD model can start. These modifications can be done using extrusion features, whose central elements are 2D sketches on a flat surface or coordinate plane. The sketches are only allowed to contain closed curves which are then moved by a specific value perpendicular to the sketch plane and therefore forming a prism. With these solid bodies material can be added or removed from the original component.

For transferring the splines, computed in the 2D-Optimizer for each closed curve, a new extrusion feature will be created on the surface selected in the program part Export FE mesh. On this surface a spline is created which consists of the points in the text file and the resulting sketch is moved by the component thickness. The resulting solid body is used for removing the enclosed material of the original component. Actually, the creation of features happens interactively by the user, but since the process of importing the splines shall be fully automated, the relevant data has to be generated by the TOOLKIT application in form of a feature tree.

Once the creation of the feature tree concerning the orientation and the placement of the sketching plane is completed, the spline points can be added to the sketch. Since a sketch is not defined in the global coordinate system, in which the user models the component, but in a local one related to the sketching plane, the points have to be transformed. For this transformation process a horizontal and a vertical reference are necessary which are perpendicular to the sketching plane. Therefore two



coordinate planes according to the standard orientation of the component are selected automatically. Using these two references Creo Parametric 2.0 is able to compute a transformation matrix which transfers the points from the global to the local coordinate system. All transformed points are stored in an array and can be transferred to the sketch. To guarantee that the splines describe the component outline as good as possible, the closed curve is not realized with only one spline but with several ones. If the included angle formed by three points and the included angle of adjacent points indicate that an unsteady transition occurs, a new spline is introduced. This can happen if the spline passes over from the inside of the component to the boundary area or if the spline coincides with the boundary area and edges occur. If all points have been added to the sketch, it must be dimensioned. For this purpose TOOLKIT offers the function *ProSectionAutodim* which creates automatically the required dimensions and the appropriate values. These dimensions have to be evaluated concerning their effect on the geometry with the command *ProSectionSolve* and finally the whole sketch can be regenerated using the function *ProSectionRegenerate*.

If the text file contains still other spline points, the procedure is repeated until all computed areas, which do not contribute to the stiffness of the component, are transferred into features of a CAD model.

#### 4 VALIDATION

For validation purposes the structures created with the program presented in this paper were compared with structures optimized with the proven powerful simulation tool Abaqus 6.13. The design domain which has to be optimized is a cantilever beam with a single force at the centre of the free end. After completion of the optimization process, the resulting structures are depicted in Figure 7. The similarities are unmistakable. In order to compare the resulting structures in detail, a FE analysis was carried out. The maximum stresses are 0,92 MPa for the structure optimized with Abaqus 6.13 respectively 1,36 MPa for the developed program. Comparing the compliances results in  $1,013 \cdot 10^{-3}$  mm for Abaqus 6.13 and  $7,880 \cdot 10^{-4}$  mm for the 2D-optimization-module. Additionally it is remarkable, that the desired volume fraction of 50% was not reached by both programs. Abaqus 6.13 optimized the structure to a volume fraction of 59,25% and the developed program to 53,92%.

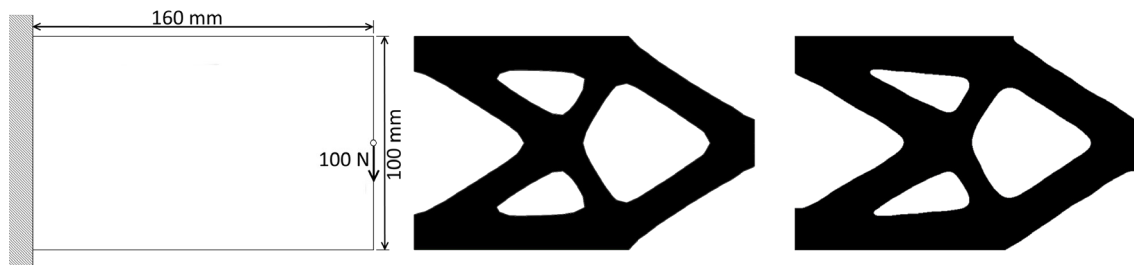


Figure 7. Design domain (left) and the optimized structures with Abaqus 6.13 (middle) and the developed program (right)

#### 5 APPLICATION EXAMPLE

To demonstrate the performance and the applicability of the 2D-Optimization-Modul in a practical application, the topology optimization process of a real component is presented. The component is the central, supporting part of an innovative tiltable e-bike which is under development at the Hochschule Ravensburg-Weingarten. A prototype for ergonomic verification consists of two metal sheets and connects the handlebars, the bottom bracket, the seat and the rear end of the vehicle. To achieve the greatest possible potential for lightweight design, the two identical metal sheets are supposed to be optimized in order to obtain the maximum stiffness with minimized weight. The loads, which are to be considered in the optimization process, are the weight of the driver, the pedal power and the vertical wheel force of the front wheel. Due to the early stage of development the load conditions are based on assumptions which are assumed to be realistic and can be seen in Figure 8. The absolute value of the forces is not relevant since only the ratio of the forces among themselves influences the optimization process. For completion of the simulation model, a fixed support is added to the rear vertical component surface. The objective of the optimization process is to minimize the weight of the component to 50%.

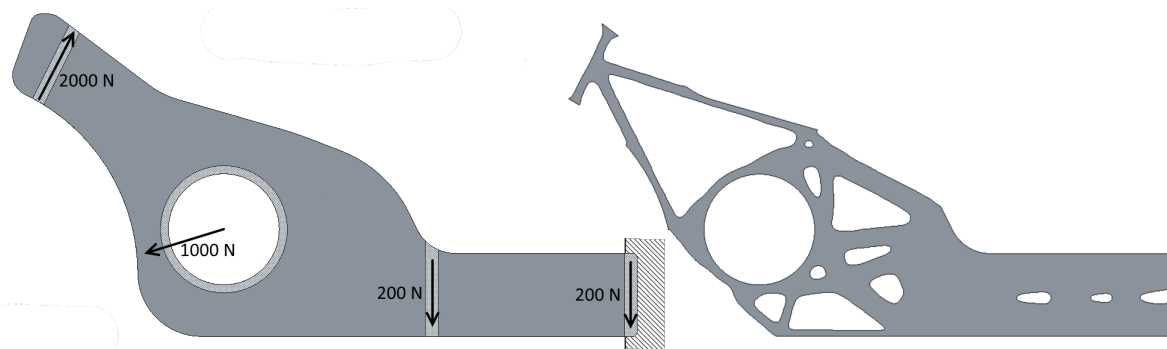


Figure 8. Applied loads to the component and the resulting CAD model

Once the optimization process and the transformation of the void areas into the CAD model has finished, the component has the appearance which can be seen in Figure 8. The weight decreased from 6.75 kg to 3.96 kg whereas the maximum displacement increased from 24.6 mm to 29.9 mm. Thus, the weight is reduced by 41.3% and the displacement grows by 21.5%.

## 6 CONCLUSION

This paper presented an approach to integrate topology optimization into existing product development processes without the need of new software. The developed optimization module integrates seamlessly into an existing CAD system and therefore can gather wide acceptance due to the known workflows. Currently only thin components can be optimized using this optimization module but the extension to three-dimensional components can easily be obtained. The transformation of void areas into CAD features needs further research effort especially in 3D. Another aspect which has to be considered then is the manufacturability of the design proposal. Frequently design proposals generated from three-dimensional topology optimization are not feasible with current manufacturing processes apart from 3D-printing. Therefore manufacturing constraints have to be introduced in order to generate appropriate design proposals (Michailidis 2014). Another important aspect is the control of the optimization process by the occurring stresses. This eliminates the need to specify a fixed amount of material and can lead to ready-to-use components. An approach similar to Lin and Sheu (2009) could be integrated to improve the presented work.

By means of a practical application, the functionality of the integrated optimization process could be shown. The resulting component shows more than proportional stiffness compared to the reduction of its weight. More practical applications will be investigated in the future steps of the project.

## REFERENCES

- Bendsøe, M., Kikuchi, N. (1988) Generating optimal topologie in structural design using a homogenization method. *Computer Methods in Applied Mechanics and Engineering*, Vol. 71, No. 2, pp. 197-224.
- Bendsøe, M., Sigmund, O. (2003) *Topology optimization – Theory, methods and applications*. Berlin: Springer.
- Christensen, P., Klarbring, A. (2009) *An introduction to structural optimization*. Dordrecht: Springer.
- Harzheim, L. (2008) *Strukturoptimierung – Grundlagen und Anwendungen*. Frankfurt/Main: Harri Deutsch.
- Krog, L., Tucker, A., Rollema, G. (2011) *Application of Topology, Sizing and Shape Optimization Methods to Optimal Design of Aircraft Components*. Airbus UK Ltd, Altair Engineering Ltd.
- Lin, C., Sheu, F. (2009) Adaptive volume constraint algorithm for stress limit-based topology optimization. *Computer-Aided Design*, Vol. 41, No. 9, pp. 685–694.
- Michailidis, G. (2014) *Manufacturing Constraints and Multi-Phase Shape and Topology Optimization via a Level-Set Method*. Palaiseau: Ecole Polytechnique X.
- Rozvany, G. (2001) Aims, scope, methods, history and unified-terminology of computer-aided topology optimization in structural mechanics. *Structural and Multidisciplinary Optimization*, Vol. 21, No. 2, pp. 90-108.
- Schumacher, A. (2013) *Optimierung mechanischer Strukturen – Grundlagen und industrielle Anwendungen*. Berlin: Vieweg + Teubner.
- Spillers, W., MacBain, K. (2009) *Structural Optimization*. Berlin: Springer.