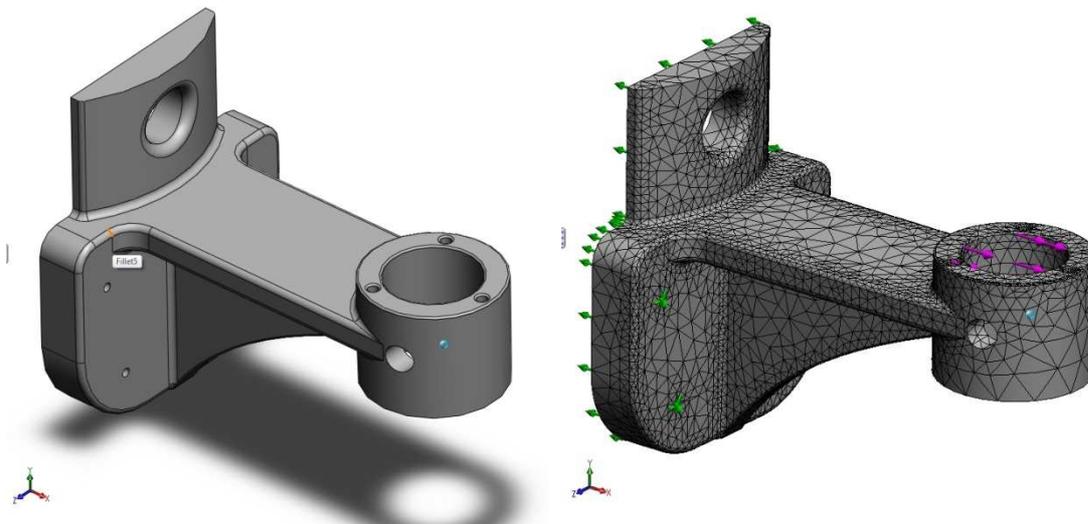


Engineering optimization example

The title of example:	Shape optimization of a 3D part using CAE system
Number of example:	OPT-BME-5
Level of example:	intermediate
Short description of the example:	In connection with the tenth chapter of the Engineering Optimization teaching material this example explains the steps of building shape optimization model in an integrated CAE system. The design variables are chosen from the parametric dimensions of the body, and for structural analysis the built in FE solver is used.

1. Definition of the problem:

Create a FE and optimization model for the given part. The left side of steel bracket is fixed, while the inside of the vertical hole is loaded in x direction with 10 kN. The shape of the lower rib should be optimized in order keep the Mises stresses under 100 MPa and the maximum deformation should not reach 0,12 mm. The goal of the optimization is weight minimization.



We assume that the structure geometry is available in the selected CAE system, and such a feature-tree, which meets the criteria laid down in the 10th chapter of the textbook.

In addition, we will need to set up the structural analysis problem, which in our example is going to be a linear static problem, but it can be whatever kind, which the numerical simulation module of the CAE system supports, and which results the optimization module reaches. As the sensitivity analysis section also pointed out, more attention should be paid to the preparation of an appropriate finite element mesh, in order to obtain sufficiently accurate results for structural analysis.

So we assume that the continuous parametric and the discretized geometry with the necessary data for structure analysis is available (boundary conditions, material properties).

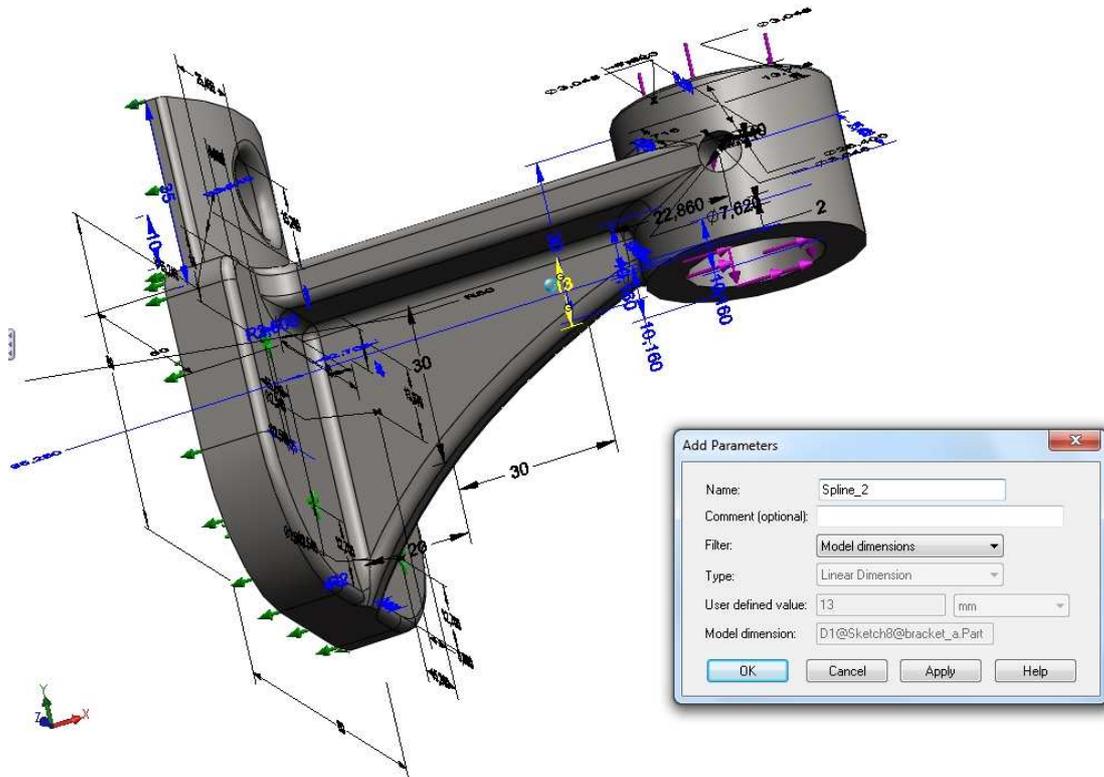
Remark: In this example we review that - regardless of software environment - what are the basic steps needed to solve a shape optimization problem by the use of 3D feature based CAE system, using numerical structure analysis and built-in optimization module. All the leading CAE systems have (SolidWorks, Pro/Engineer, Catia etc.) - though with slightly different

capabilities – a finite element simulation and a parametric shape optimization module, so this problem can be solved with all the three mentioned systems.

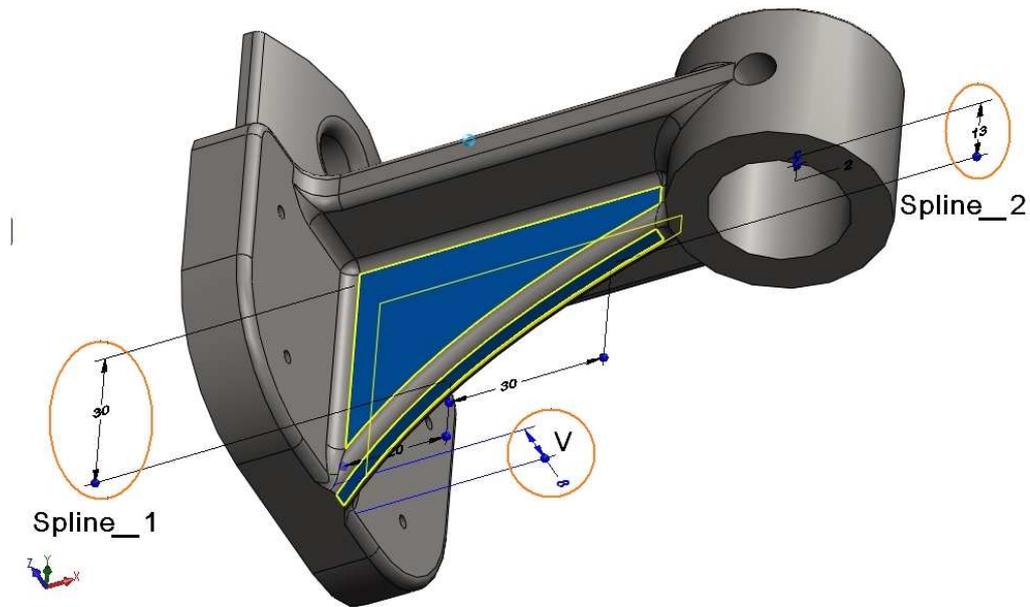
2. The steps of the solution:

1st step: Selection of design variables and setting its upper and lower limits

One of the most significant steps of solving shape optimization problem is the selection of design variables. The design variables must be independent data (usually the typical size of the parametric CAD geometry's contour). The specification manner of design variables happens by selecting the desired dimension on the CAE system's graphical interface:

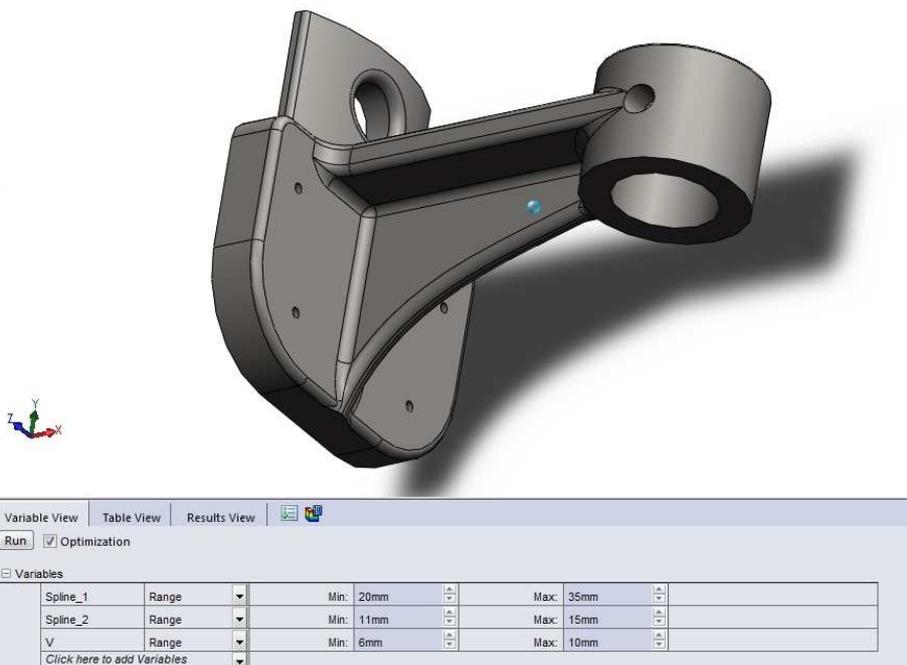


This procedure must be repeated for each design variable used. During the first optimization attempts we shouldn't use too many design variables because of the potential complexity of the task. If a problem occurs during the optimization loop, it is more difficult to determine the cause of the problem in case of large number of design variables. In this example we show the optimization of stiffening ribs on the bottom, with three design variables, from which two are the coordinates of the two different splines points that determine the contour of the rib (which are named spline_1 and spline 2), while the third variable is the thickness of the rib, designated by V:



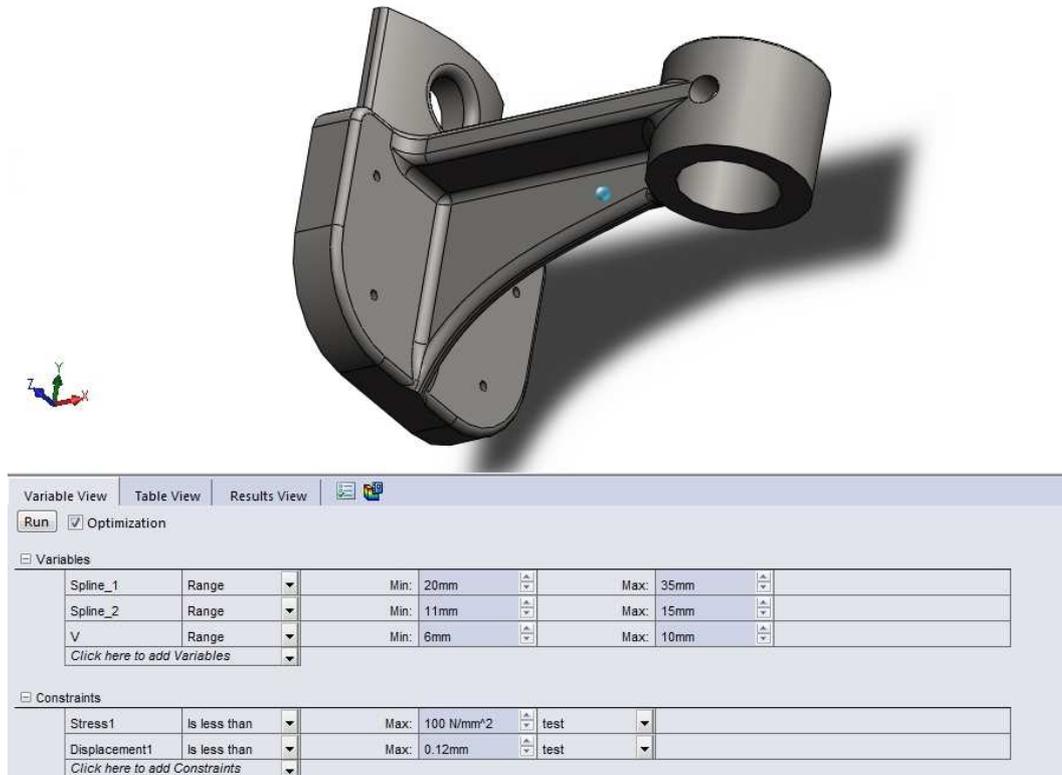
In case the design variables are chosen, the next step is to set the search range. For each design variable a lower and an upper limit has to be determined for the optimization, in which the current value of the design variable falls. We have summarized the chosen initial values and limits of the design variables in the following table, and the defined design variables with his limits in a CAE system can be seen in the following figure.

Design Variable Name	Initial Value [mm]	Lower Limit [mm]	Upper Limit [mm]
Spline_1	30	20	35
Spline_2	13	11	15
V	8	6	10



2nd step: Enter the optimization constraints

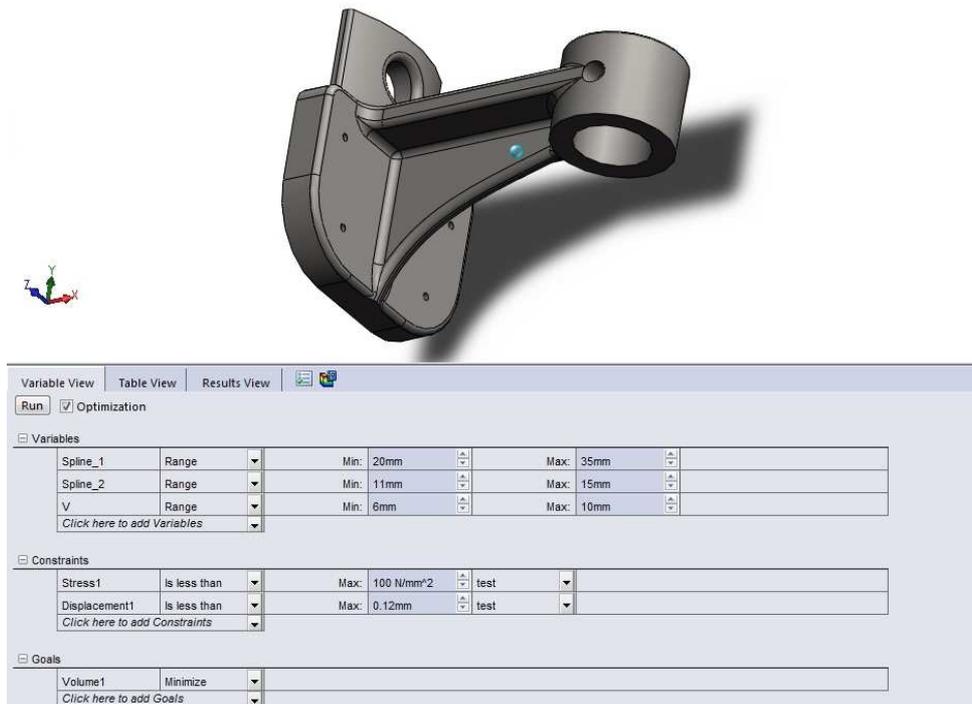
Generally the optimization conditions will serve to guarantee the structure functionality; therefore it's usually based on the structure analysis results. In this example, maximum Mises stress was constrained to 100MPa and the holder's maximum deformation was not to exceed 0,12 mm. The definition of the optimization constraints are shown in the following figure:



Generally in the real engineering praxis, a structure should carry more than one load cases, so we have to perform the optimization parallel with multiple load cases. For this type of the problems special attention should be paid for selecting the CAE system, because not all optimizing system is suitable for the optimization of multiple load cases.

3th step: Defining the goal function

Structural analysis is not necessary in the evaluation of some objective function, such as volume and mass. While in other cases structural analysis calculations are necessary for the design point evaluation, such as the maximum stress, maximum deformation, or the Eigen frequency. Similarly to specifying the optimization criteria, we have the opportunity to enter the scope of the objective function, which may be the whole structure, or part of it's as well. In the example the minimization of the volume is prescribed, this can be seen in the following figure:



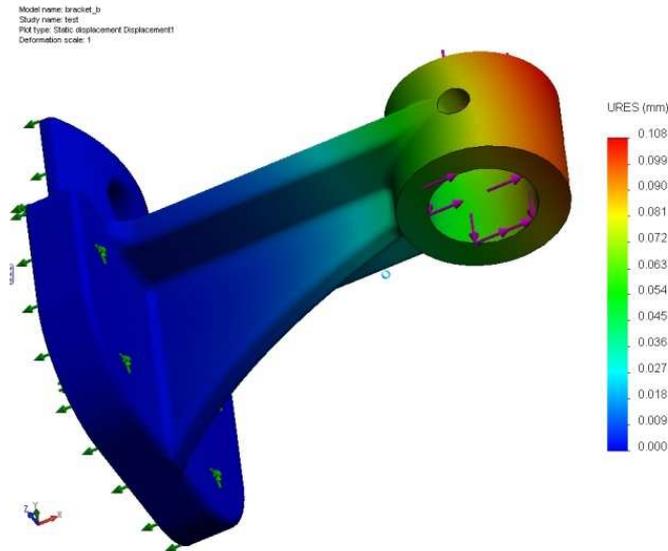
4th step: Setting up the Optimization parameters

After specifying the optimization problem, the optimization algorithm settings follow. However this is program specific, for example if several algorithms are built into the CAE system (such as Pro/Engineer and Catia) it's possible to select the optimization algorithm used from the list (there is only one algorithm in the Solidworks system, but two different levels of accuracy). In many cases it is possible to adjust the convergence parameters or stopping criteria as well.

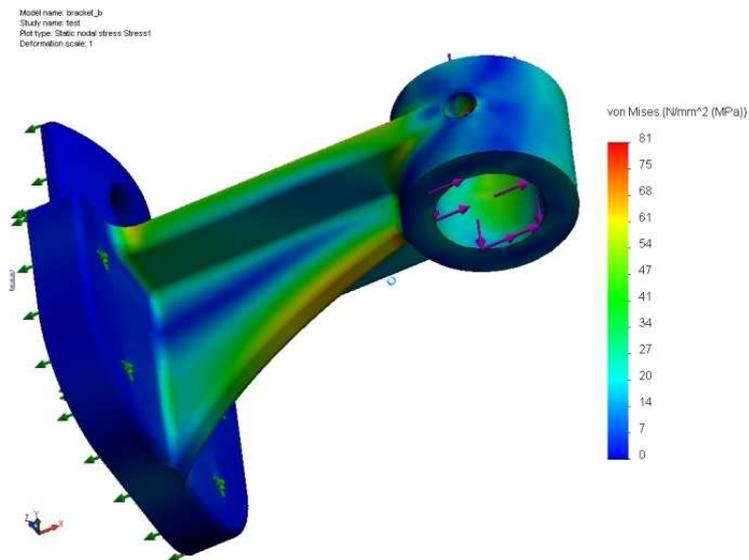
5th step: Run the optimization, evaluate the results

After a successful optimization cycle it should always be investigated, if the optimization results obtained certainly meet the optimization criteria, because there are such methods that interpolate the results reducing the time required for calculation and thus seek an approximate solution.

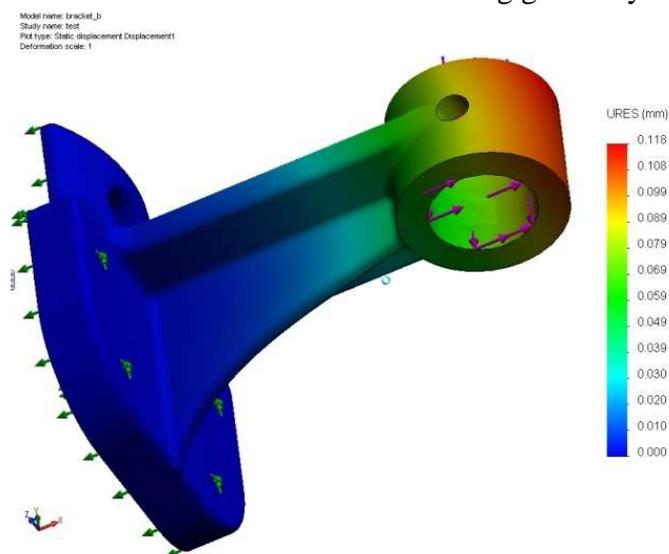
In our example it's obvious that the location of the stress maximum has changed, but both the initial and the optimized geometry meet all the optimization criteria:



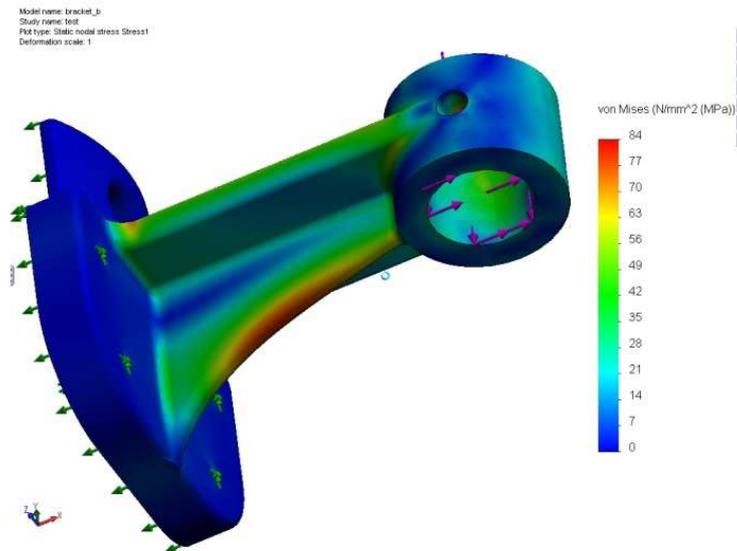
The displacement field of the starting geometry



Stress distribution in the starting geometry



The displacement field of the optimized geometry

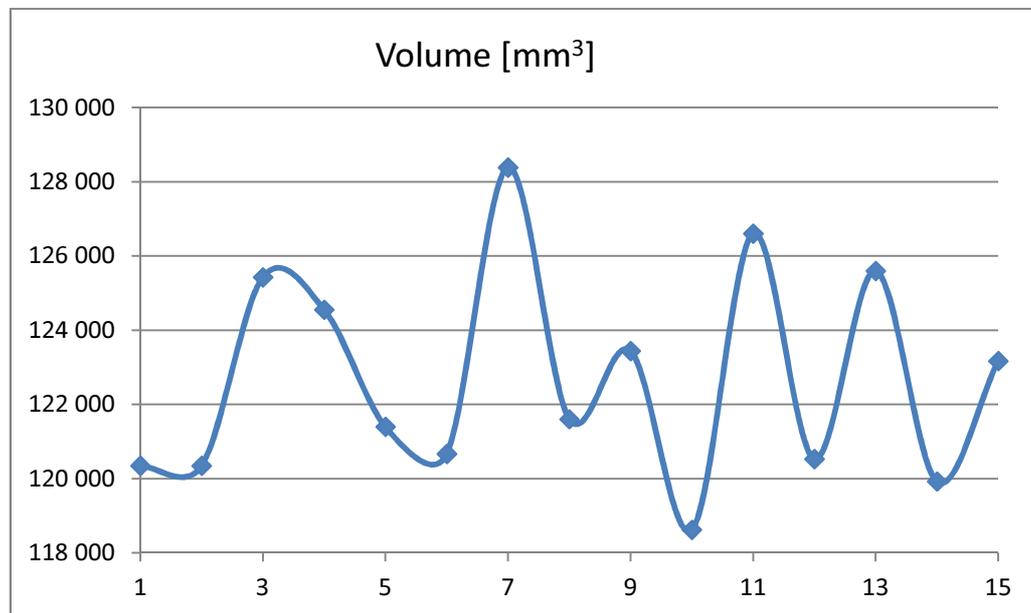
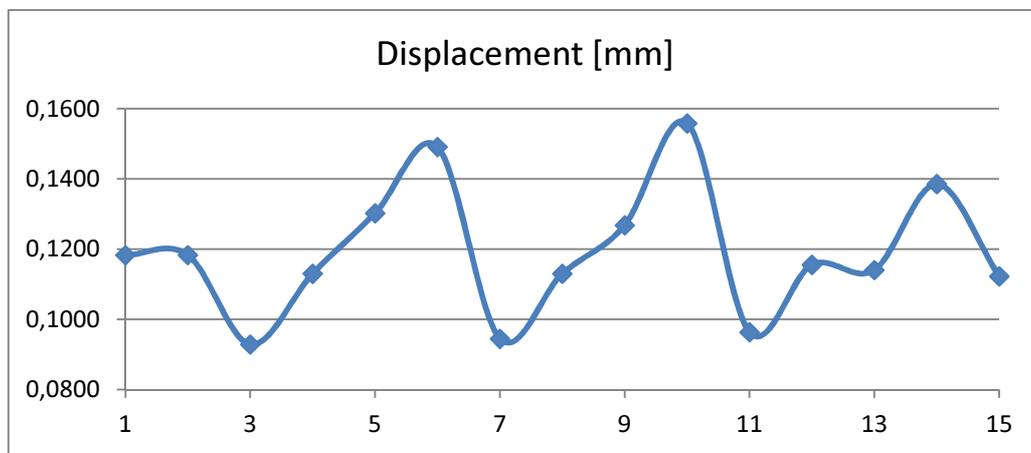
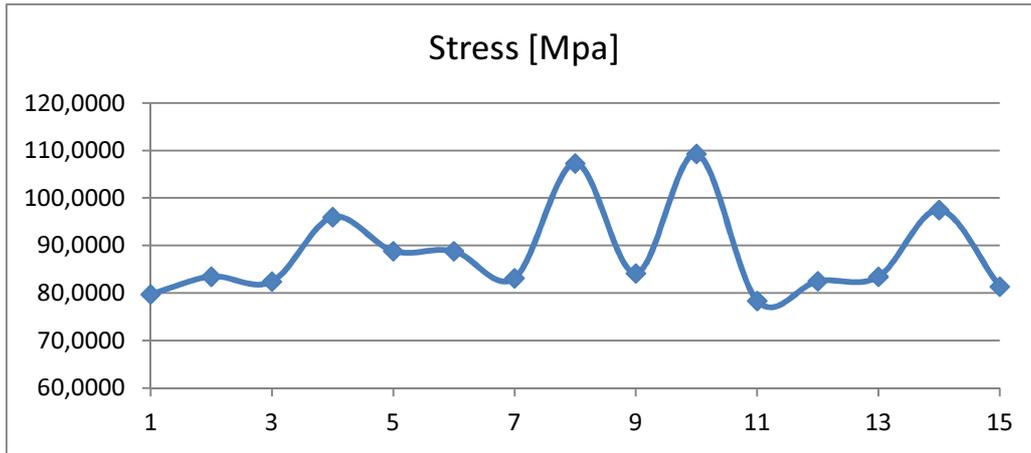


Stress distribution in the optimized geometry

The design variable values proposed by the optimization procedure are shown in the following table „the optimization result” column.

Design Variable Name	Initial Value [mm]	Lower limit [mm]	Upper limit [mm]	The optimization result [mm]	Optimized value [mm]
Spline_1	30	20	35	26,626	27
Spline_2	13	11	15	14,900	15
V	8	6	10	6,006	6

It is also worth examining the history of the optimization: for example the optimization constraints in what examined points are met the requirements and in where not (see in the next figure). Geometries tested during optimization are shown in the attached video. (geometrical changes.wmv).



6th step: Refine the optimization model (in regards of the search range and design variables)

Even in case of a successful optimization cycle, it is worth examining, that the optimization result is at the edge of the searching domain, and if so whether it is possible to change the upper or lower limit of the design variable. If the construction or production opportunities or the engineering considerations permit, the optimization can be continued in a modified (enlarged) searching space, in order to achieve a better design.

In our example shows, that the optimization result is away from the limits in the case of Spline_1 variable. Although in case of Spline_2 and V it's close, but doesn't reach the limits, so we didn't change the search range.

If the search range would be no longer changed, consideration should be given if introducing new design variables would be required in order to refine the design. This can result in a more precise definition of the recently changed component of the part. For example, we can insert another point into the spline curve describing the arch of the lower rib (which is currently changed by Spline_1 and Spline_2 design variables). By the change of the 3 point's coordinates there is a greater degree of freedom to more accurately determine the desired shape. This procedure is called a refinement of the optimization model.

7th step: Preparation of proper manufacturing design

Since in this example the optimization uses continuous design variables, therefore the resulting optimal design variable values should be adjusted due to manufacturability (see the last column in the optimization result table), and a structure analysis for the modified structure should be carried out. In this example, the result of the simulation shows, that the suggested optimized parts also meet the optimization criteria.