*Design of Machines and Structures, Vol. 7, No. 1 (2017), pp. 5–14.*

# SAFETY INCREASE WITH MATERIAL REMOVAL

CSISZÁR L. RICHÁRD
Budapest University Of Technology and Economics,
Department of Machine and Product Design

**Abstract:** This study is about an optimization of an axial tensioned plate with a central single circular hole. The objective is to increase the safety with remove material from the plate. The study contains the method of the optimization, topology and shape optimization.

***Keywords:*** *optimization, topology optimization, shape optimization*

## 1. INTRODUCTION

The era of computer simulation the machine design process getting shorter. The simulation replaces the expensive and time-consuming physical tests. The structural optimization was only privileged by the researches but with commercial software the product designers can use these tools easily.

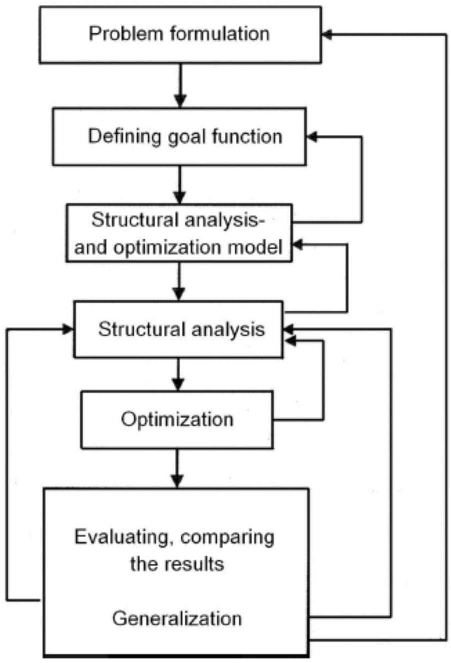


*Figure 1. Design process with optimization step [3]*

# 2. TASK PRE-PROCESS

## 2.1. Define the problem [2]

The task is to find a geometry which gives the highest safety, the geometry seen below.

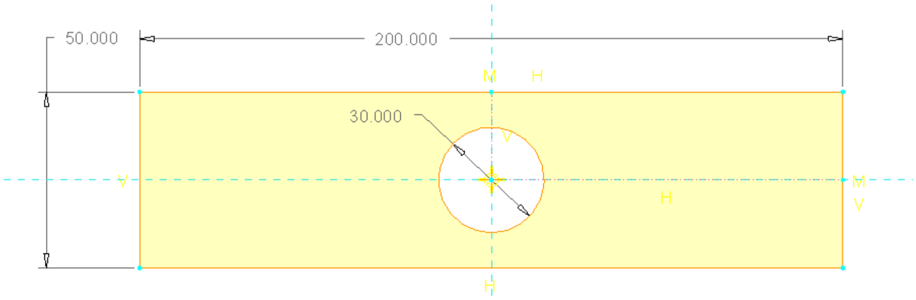


*Figure 2. Original geometry*

The material of the plate is S355 EN 10025:2004 structural steel. Around the hole for bolt must be a non-removable material 20 mm outside the hole. The tension force is 20 kN. The safety is equal to the maximum von Mises stress divided by the yield strength.

## 2.2. Data and information collecting

S355 EN 10025:2004 has the following properties:

*Table 1. Material properties*

| | |
|---|---|
| Yield strength | 355 MPa |
| Young modulus | 210 GPa |
| Poisson ratio | 0.3 |
| Density | 7850 kg/m$^3$ |

The pre calculations for the structure can see below. This is a stress concentration area example [1].

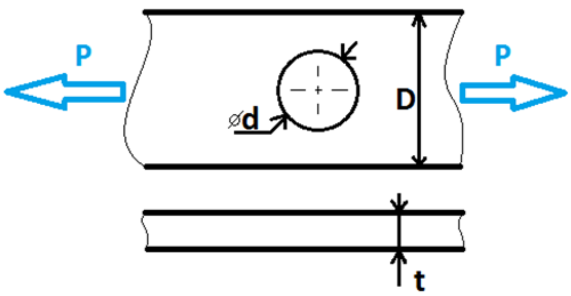


*Figure 3. Parameters of the calculation*

Structural stress calculation $$\sigma_{nom} = \frac{P}{[t \cdot (D-d)]} = 200\ MPa \quad (1)$$

Define $K_t$ factor $$0 \leq d/D \leq 1 \quad (2)$$

$$K_t = 3.000 - 3.140\left(\frac{d}{D}\right) + 3.667\left(\frac{d}{D}\right)^2 - 1.527\left(\frac{d}{D}\right)^3 = 2.11 \quad (3)$$

The maximum principal stress $$\sigma_{max} = K_t\sigma_{nom} = 422\ \text{MPa} \quad (4)$$

It means that the plate with a hole has the maximum tension stress of 422 MPa. It is higher than the yield strength so the safety is lower than acceptable.

### 2.3. Definition of variables

The design variables can be seen in *Figure 4*, which is the green area. The elements around the constraints and elements close to the hole must be a non-design space.

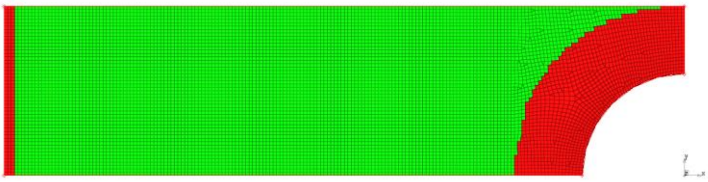

*Figure 4. Design variables in quarter model*

### 2.4. Objective of the optimization

The objective is the mass reduction, because the task is to remove material and reduce stress too. The algorithm search lighter structure, just the case of mass minimizing objective.

### 2.5. Constraints of the optimization

The constraint is the yield strength, so none of the elements can reach the 355 MPa von Mises stress**.**

## 3. OPTIMIZATION PROCESS

### 3.1. FE model

The first step is to build up the FE model of the structure. It is an obligatory task, because of the comparison with the analytical calculation. This is the input of the optimization, so it must be eligible. The analysis and the optimization is making by HyperWorks®. The geometry, the loads and the constraints are symmetrical, so symmetrical simplification can be used. The model is calculated in 2D plane stress so 2D elements can be used. The constraints can be seen on Figure 5. dX = 0 at the lower edge of the model and dY = 0 at the right side of the structure. The loads applied on the left side of the structure, the half of the structures has half of the loads and it is 10 kN.

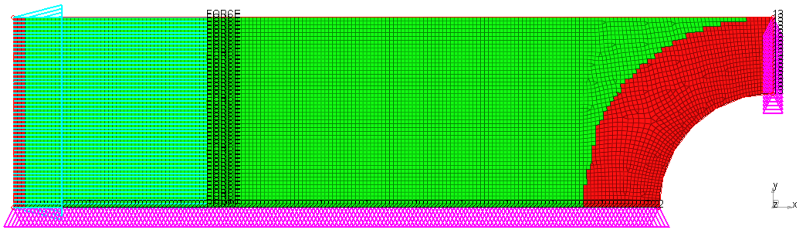

*Figure 5. FEA model*

The result of the analysis can be seen on *Figure 6*. The differences between the analytical *(Equation 1–4)* and computational results are 3% so it is acceptable.

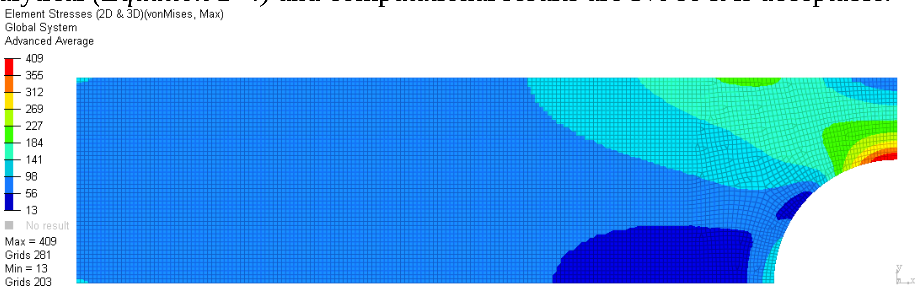


*Figure 6. Result of the FEA*

### 3.2. Parameters of the Optimization

The parameters in the first iteration are the task defined parameters. The variables are topology variables; that mean that the algorithm can change the relative modulus of the elements from almost zero to the real value of the rigidity. The total volume fraction response function has applied. The non-design space elements also have a response and it is the static stress response. The stress in the design space can be constrained with maximum von Mises stress as a parameter of the design variable, not as a response function. The non-design space is constrained as well, 355 MPa is the upper boundary. The objective is to minimize the volume fraction.

### 3.3. Results of the optimization

The process of optimization the algorithm reduces the volume and pay attention to the responses, stress responses in this case. These steps can be seen on the *Figure 7*. In the graph shows that in the second step the constraint is satisfied, so the stresses are lower than 355 MPa after the third iteration, the volume is dropping monotonous. The 19th iteration the constraint is penetrated so the next iteration the volume begins to rise but just for a few iterations. The convergence criteria reached after the 40 iteration so the iteration process stopped.

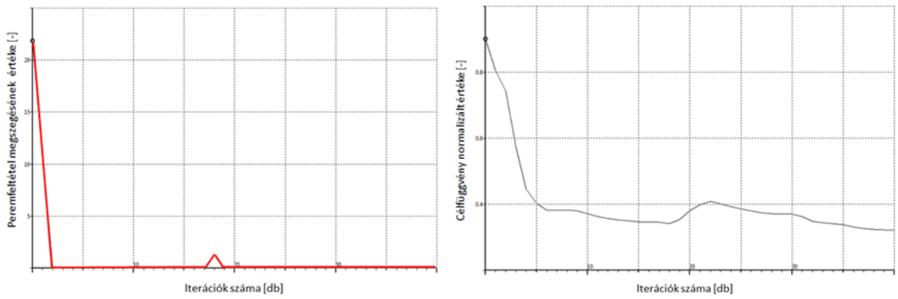


*Figure 7. Iteration steps*

The rough result can be seen on *Figure 8*. The explanation of the result is the following. The picture a shows the density plots from 0 to 1 value. The red area show the element with E0 young modulus so the original one. The blue areas have much lower modules and density values almost zero. The real values in the reality can be zero or E0 nothing between, so that is why the structure must be interpreted. The *b)* picture shows the stress plot of the rough result.

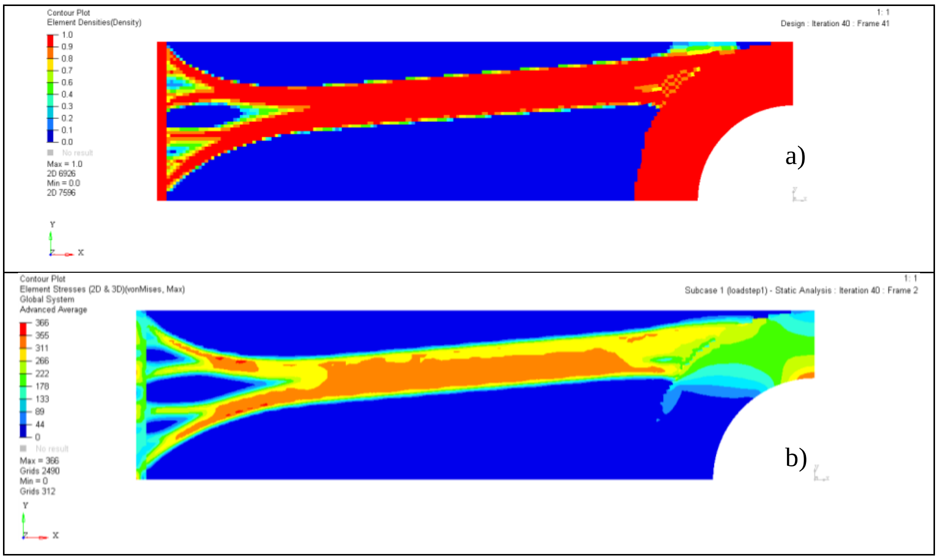


*Figure 8. Result of optimization*

## 4. INTERPRETATION AND VERIFICATION

### 4.1. Automatic smooth

OSSMOOTH® is a build in program, where the results can be interpreted automatic. The first step is to delete the elements which are lower, than the input parameter, here 0.5 is used. The next step is that the algorithm smoothed the surfaces of this rough area and then it could re-mesh the smoothed surface it is on the *Figure 9*. The most useful feature is that the constraints and the loads are reapplied by the program.

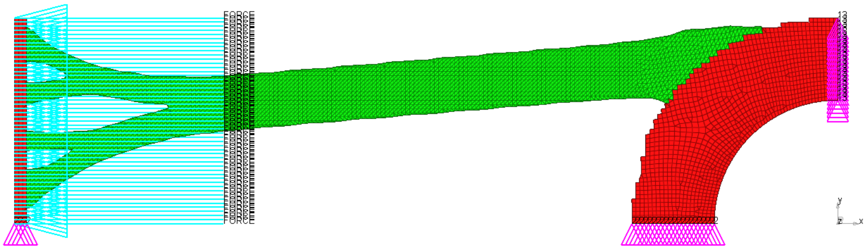

*Figure 9. Smoothed geometry*

### 4.2. Manual smooth

The interpreted geometry can be created in a CAD program. The process is easy with 2D structure. Take the picture of the rough optimization result, paste it into a plane and draw it. In 3D the rough surface of the result can be exported to *.stl file format. The 2D interpretation can be seen on *Figure 10.*

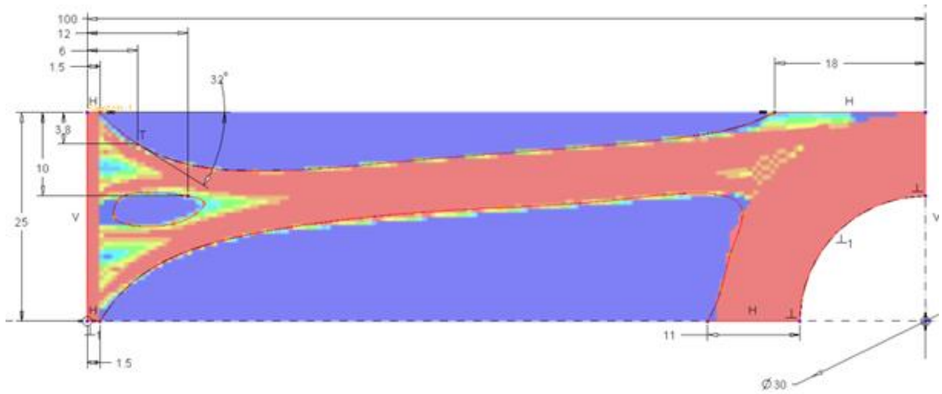

*Figure 10. Manual interpretation*

### 4.3. Usability of the result

The validation of the interpreted geometry is the next step. The *Figure 11 a)* part shows the non-interpreted result, this is feasible, but if the almost zero rigidity elements are disappeared the geometry is unfeasible. The *b)* shows the automatic interpreted geometry, *c)* shows the manual interpreted geometry, and none of them is feasible. The result needs a small amount of modification.

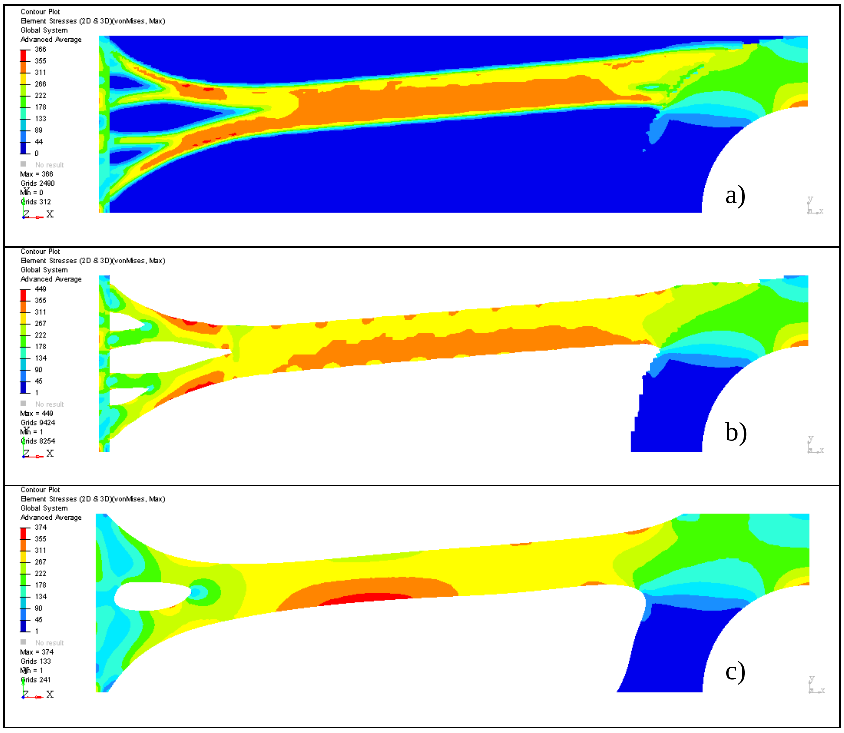


*Figure 11. Validation of the result*

### 4.4. Re-optimizing the result

The shape optimization is a tool for re-optimize the almost fine structure. The method can modify the nodes coordinates *Figure 12*, and with that local stress concentrations can be removed. The loads and constraints are the same, as in the FE model.

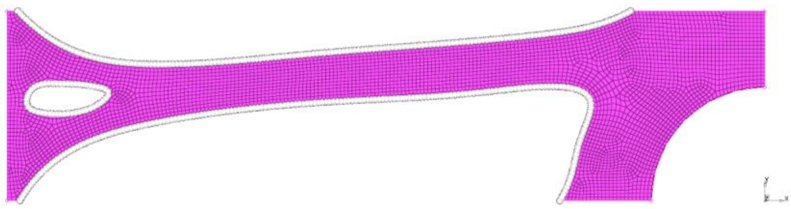

*Figure 12. Design variables of the shape optimization*

The task is to reduce stress on hot spots. The objective was the volume reduction but now it is reduced, so here the goal is to find the feasible geometry. The software can use minmax objective for stress response, it means that the previous maximum stress should be reduce for the next step. The shape optimization should be limited by the geometry because it can be larger than the original geometry. The node coordinates are limited Y+ directions in the upper side and Y– directions in the lower side. The results can be seen on *Figure 13*. The *a)* show the starting geometry; in the *b)* the final iteration stress plot can be seen. The *c)* shows the shape change magnitude in mm.

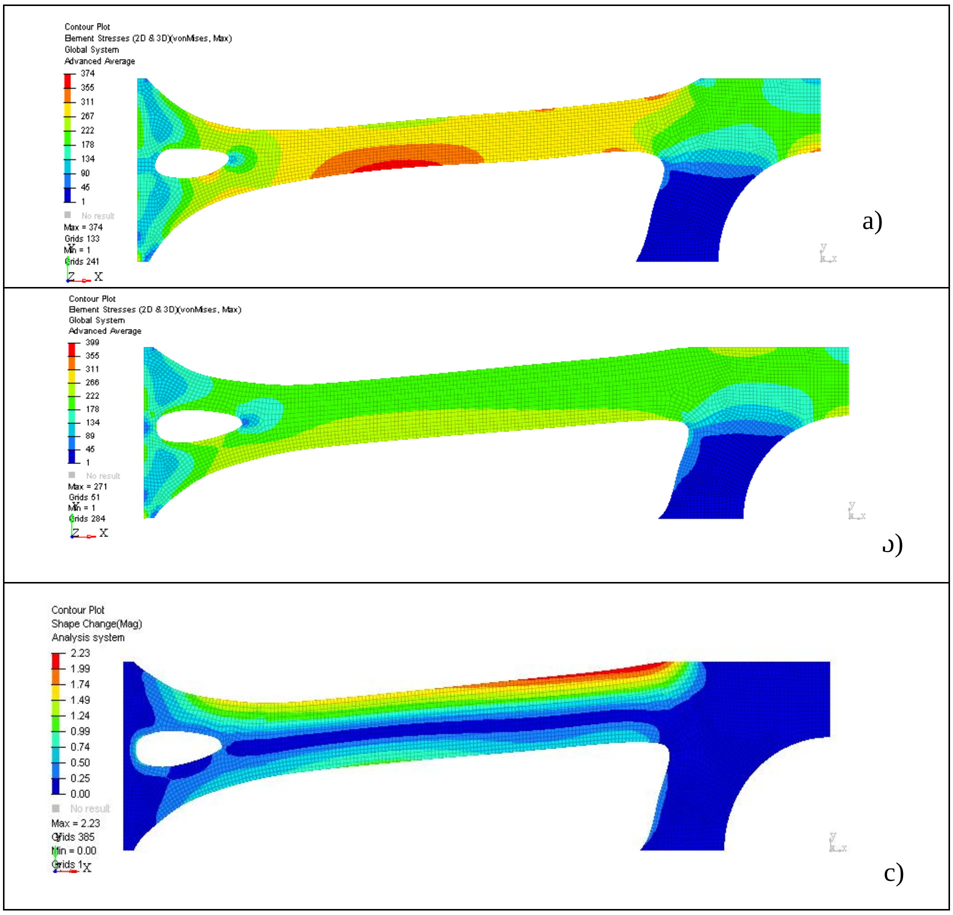


*Figure 13. Minmax stress objective*

If the case is to reach the minimum mass than the parameters of the optimization is different. The objective is the same as in the topology optimization, this is volume minimizing, the stresses constrained for the whole structure. The result of this optimization can be seen on *Figure 14 a)* shows the starting geometry, *b)* shows the final iteration, and the *c)* shows the change of the shape.

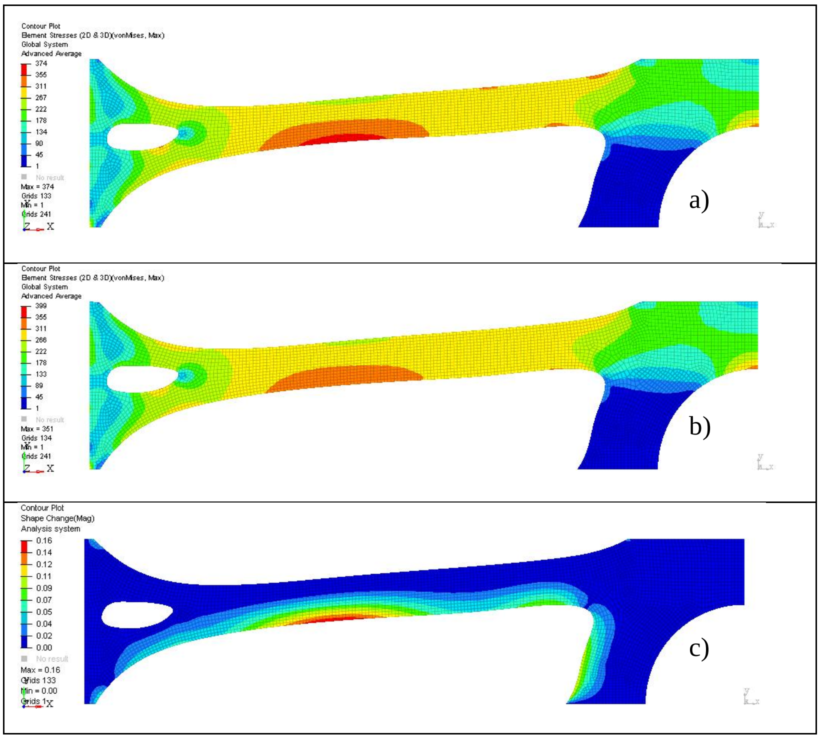


*Figure 14. Minimizing volume*

## 5. SUMMARY

The refine process the first task objective has to be interpreted, for define the sub task objectives. The final geometry can be difficult to manufacture, or can cause more problems for the non-constrained areas. The problems can be predicted with the detailed task intractable. The real difficulty is to formulate these constraints. The time that the optimization process needs the 50% is to refine the input data and not the calculations. The software knowledge is a base of the optimization, because it is can-not solve every problem. The final geometry can be seen on the *Figure 15* ones from the built-in ones. Further task is to develop the subassembly-kit and the tip-list.

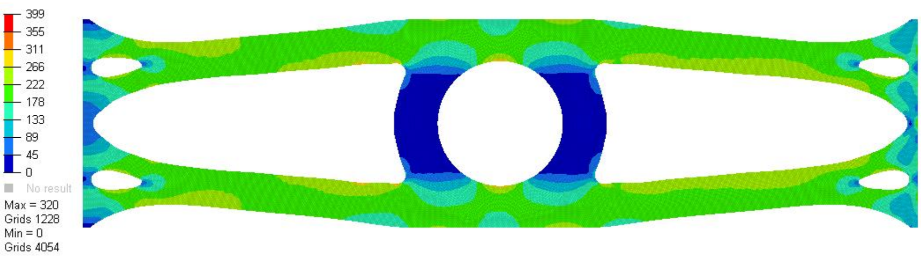


*Figure 15. Final geometry stress plot*

## 6. ACKNOWLEDGEMENT

The recent study and publication was realized within the Knorr-Bremse Scholarship Program supported by the Knorr-Bremse Rail Systems Budapest.

## 7. REFERENCES

[1] PILKEY, W. D. (2005): *Formulas for Stress, Strain, and Structural Matrices*. 2nd Edition, John Wiley & Sons.

[2] ARORA, J. (1989): *Introduction to Optimum Design*. McGraw-Hill.

[3] ERDŐSNÉ SÉLLEY Csilla–GYURECZ György–JANIK József–KÖRTÉLYESI Gábor: *Mérnöki optimalizáció*.