Computer aided generation of construction model for construction similarity theory

M. Cielniak*, P. Gendarz

Institute of Engineering Processes Automation and Integrated Manufacturing Systems, Silesian University of Technology,
ul. Konarskiego 18a, 44-100 Gliwice, Poland
* Corresponding author: E-mail address: mateusz.cielniak@polsl.pl

Analysis and modelling

ABSTRACT

Purpose: The main aim of research was to analyze computer tools which can help to construction model generating process.

Design/methodology/approach: Computer aided methods were used to construction attributes selecting process: parameterization, relations defining, finite elements method mesh adaptation and optimization of the construction model. This stages were developed in advanced graphical program.

Findings: Computer tools were used to generate construction model. This tools allow to reduce construction – design stage time consumption and better material utilization. This allows to create series of types at lower costs and faster.

Research limitations/implications: The optimization of construction model with Finite Element Analysis for many variables is time consuming.

Practical implications: Presented method was applied to series of types construction model generation process.

Originality/value: Described analysis presents computer aided construction model generation process applied in construction similarity theory.

Keywords: Constructional design; Engineering design; Similarity theory

1. Introduction

Currently the market situation changes very dynamically. Product which was new yesterday, now is outdated. Competition on the market puts against manufacturer very difficult task. He must provide a product that meets customer requirements in terms of quality and price. Design time is reduced continuously.

Therefore, one of the stages of the construction – design process necessary to satisfy the above conditions is the optimization. It is very important to choose proper dimensions values because of relation between construction attributes, parameters and relative costs. To achieve that, the complex analysis must be done. It is very difficult and time consuming to do it by step by step so the computer powerful tools were developed.

2. Similarity theory

In the engineering many constructions are similar. In the catalogues there are elements with the same character design and various dimensions values. This values depends on the construction similarity conditions. The construction similarity theory says that basing on construction model it is possible to elaborate construction series of types, where states will be the same in whole series. Those states are described by system and they are defined by similarity conditions and unified characteristic attributes [2, 3, 14]. The system is coupling and transforming relations arrangement. Those relations can be described by mathematical functions which describe physical effect states, stereomechanical states or other simple states. The main aim is to maintain constant states in whole series, according to construction model.

There are many different similarities: geometric, time, force, electric, thermal, photometric [9, 11].

Based on that it is possible to distinguish the similarity between construction attributes and parameters $\varphi_l = f(\varphi_a)$.

In manufacturing the relation between construction attributes and relative costs $\varphi_k = f(\varphi_l)$ can be described. It is very helpful to cost estimation process on an early construction – design stage [1].

The construction series of types is based on construction model. Because of that, the construction model has to be practical verified and optimized in reference to criteria which follow from [4, 5, 7]:

- Technical Purposefulness Rights,
- Manufacture Potential Rights,
- Economic Rights.

There are computer tools which helps defining the construction model. Some of these are presented in this paper.

3. Variantional analysis

A special tool to create a construction model for the newly created ordered family structure (pre-ordering) was developed by Siemens PLM Software company in a graphics program I-DEAS. This method for variational analysis uses VGX parameterization (Variational Design Technology) and verification using the FEM [10, 13].

In the traditional FEM analysis, for the vector of needs $\overline{X}_i^u$, the geometric and material construction attributes $[y_{m,zn}^{e_j}, y_{(t)m,zn}^{e_j}, y_{0,nz}^{e_j}]$ has assigned fixed values. Because of the criteria: maximum stress, strain and minimum weight, the significant dimensions values of the element $y_{max,nz}^{e_j} \leq y_{0,nz}^{e_j} \leq y_{min,nz}^{e_j}$ are modified in the iterative way. Each iteration needs to repeat a very time-consuming FEM calculations, Fig. 1.

In the variant analysis the unknown values of the quantitative construction attributes, such as the outer diameter of the sleeve TU2, bore diameter TU5, take the limits $[y_{min,nz}^{e_j}, y_{max,nz}^{e_j}]$ for the vector of needs $\overline{X}_i^u$. Model transformation in the variant analysis

is shown in Figure 2. Variant Analysis by VGX parameterization enables user to create a consistent record of geometry, taking into account all the dimensions of the possible variables values space. This allows to carry a single FEM analysis which reduce optimization time of quantitative values for the construction model. The adopted values ranges must take into account the relationship between the parameters and dimensions and between dimensions of interacting elements.
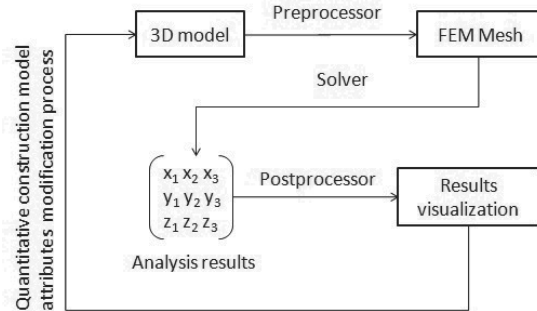

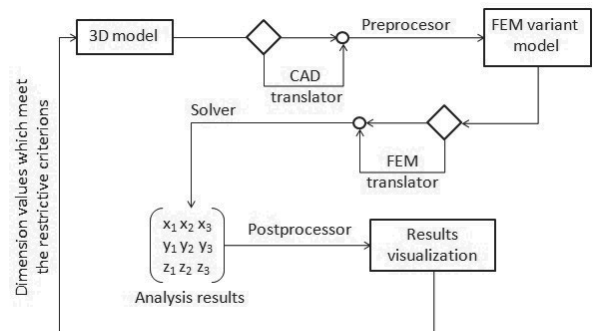Fig. 1. FEM calculation process diagram

Fig. 2. The traditional model of verification of the quantitative design features with FEM use and an variants analysis

This method allows the dimension values modification to meet the optimization criteria with iterative FEM runs use.

For the construction model, the following criteria optimization was defined:

- K1. minimum weight,
- K2. maximum stress $\sigma = 125$ [MPa],
- K3. deformation limit $\delta = 0.005$ [mm].
  There are the following stages of variants analysis:
- creation of an element model,
- FEA model creation (Baseline Analysis), Figs. 3, 4:
  - material construction attributes definition;
  - mesh generation process;
  - determination of loads and boundary conditions;
- dimensions sensitivity analysis (Sensitivity Analysis) (Fig. 5),
- Parametric Analysis for the selected dimensions of the dimensions limits (Fig. 6),
- dimensions optimization on the basis of criteria (Figs. 7, 8 and 9),
- model modification based on the result of optimization, Table 1.
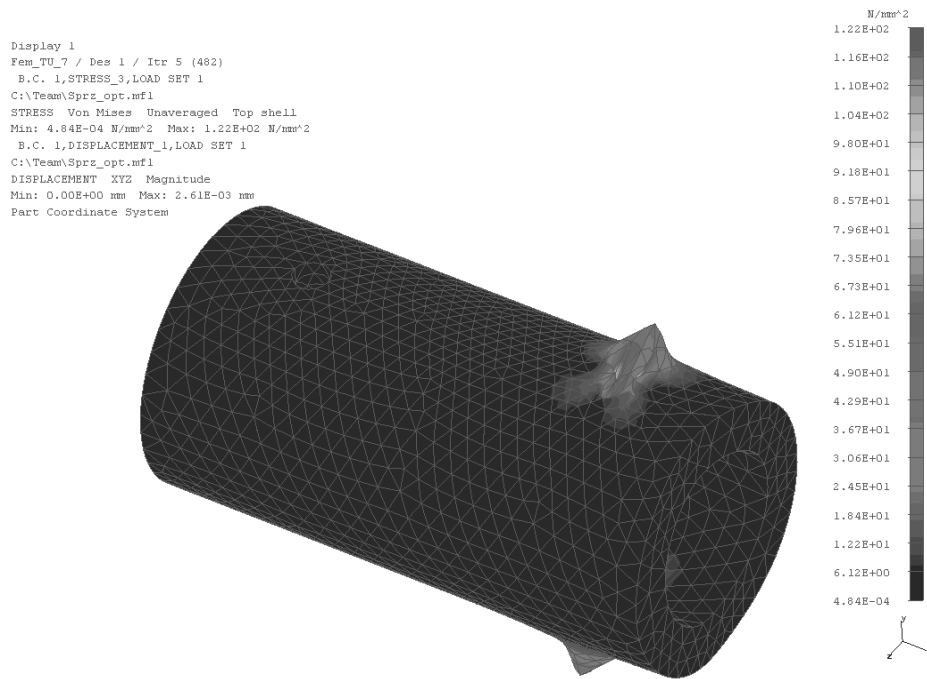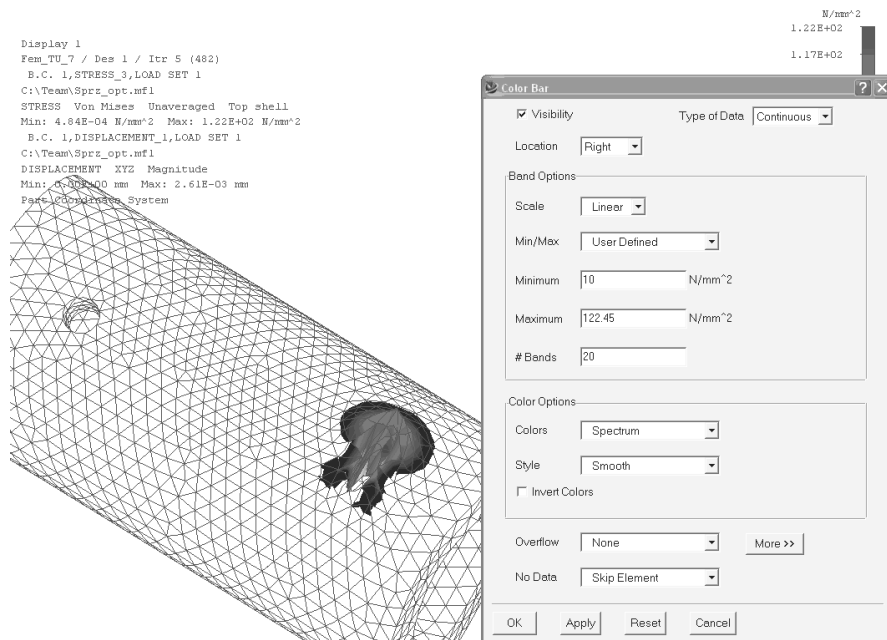
Fig. 3. Stress state

Fig. 4. Stress around hole

Table 1.
Result of the optimization of the dimensions

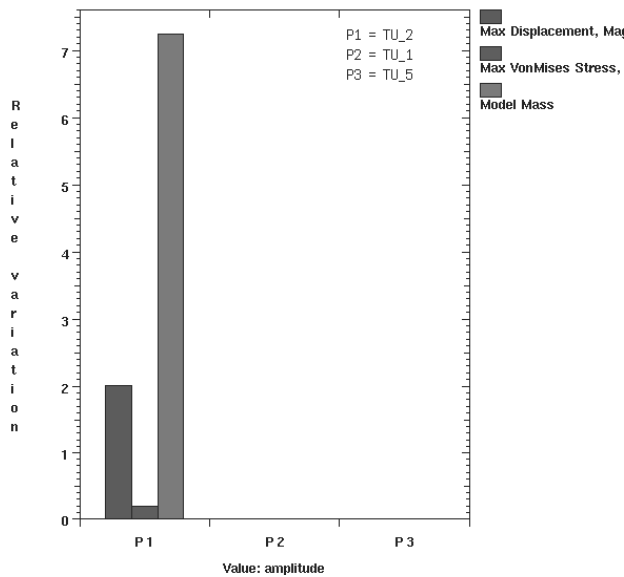| No	i	Mass		Stress		Strain		TU2		TU5		Iterations
		before	after	before	after	before	after	before	after	before	after	
		[kg]		[MPa]		[mm]		[mm]		[mm]		
1	0	1.510	1.240	80.8	92.80	0.00284	0.00319	60.0	56.92	10.00	8.57	3/5

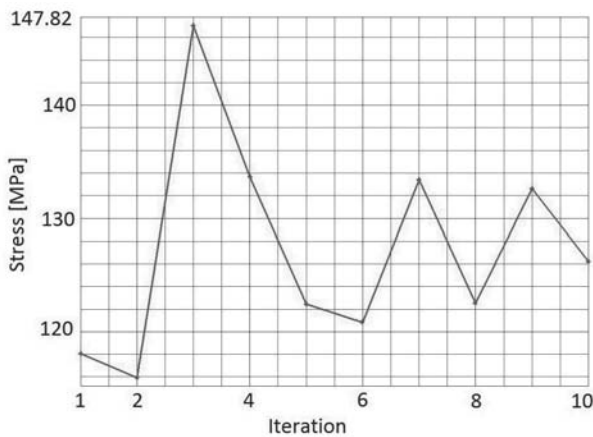Fig. 5. Sensitivity analysis for K1, K2, K3 criterion
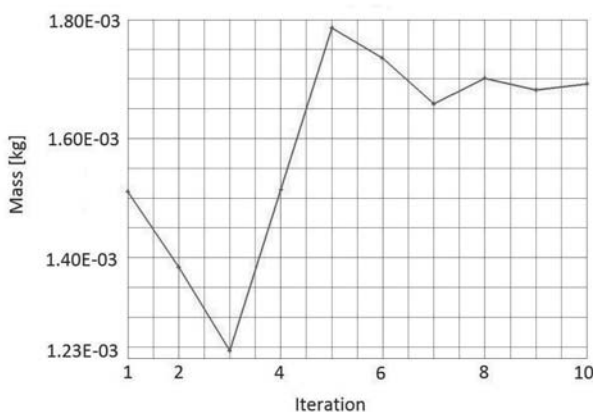

Fig. 6. Stress values in iterations
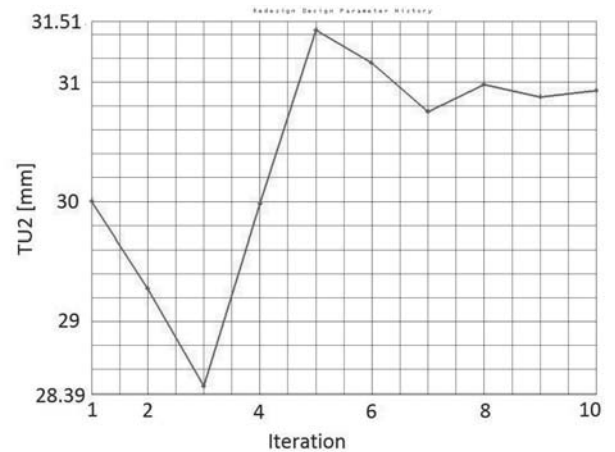

Fig. 7. Mass values in iterations

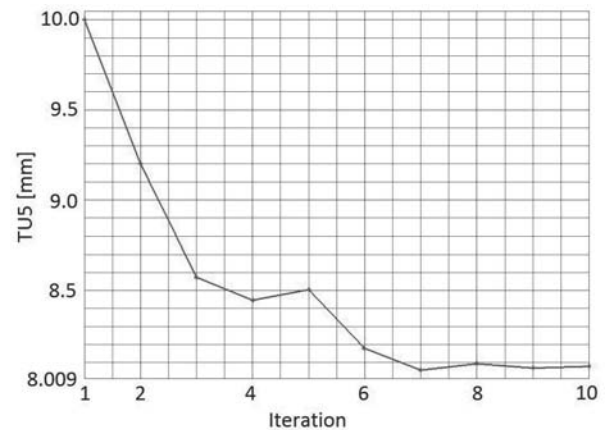Fig. 8. Outer diameter value TU2 in iterations

Fig. 9. Hole diameter values TU5 in iterations

FEM analysis results are shown on Figs. 3 and 4. Highest stresses are around holes and are equal to 122 MPa.

Outer diameter *TU2* and hole diameter *TU5* have highest impact to stresses. The inner diameter *TU1* cannot be modified because it has direct connection to clutch parameter *d*. This conclusion was confirmed by sensitivity analysis, Fig. 5.

Results presented in Table 1 indicates small changes in the value of the modified dimensions. This is due to the fact that the analyzed construction model meets the criteria for K1, K2 and K3, which provides the correct material usage and correct selection of quantitative design features. These features will not be modified, and will be equal to values before optimization.

4. Mesh adaptation

The quality of the verification results depends on the way of simulation preparation. It is important to properly model in advanced graphics program the boundary conditions and appropriate distribution of the finite element mesh.

NX program from Siemens PLM Software offers a special tool to automatically adapt the mesh to the stress state to ensure that the error caused by an accumulation of stresses in the nodes will be minimized. Based on the constraints and loads the stresses in the individual elements are estimated (Von – Misses hypothesis) [8, 15]. If the obtained result does not meet the defined conditions the elements grid is redefined.

Terms defined in the preparation of the analysis are:

- Stress Percentage Error - specifies the percentage ratio of the strain energy component to the total strain energy model. Its value lay in the range from 0 to 100 and is a real number. If that value is higher than the assumed the finite elements distribution is modified,
- Stress Error Deviation - absolute value of the difference described above; real positive number with no upper limit,
- If any parameter is equal to zero then it is not taken into account in the analysis. If both of the parameters are different from zero the analysis is completed after the fulfillment of a stricter criterion,
- Max iterations - the number ranging from 3 to 10, setting a maximum number of mesh modifications,
- Minimum Element Size - the minimum size of an item greater than or equal to zero.

After determining these parameters the simulation (Solution) must be specified from which boundary conditions will be used to carry out the adaptation.

The process of adjusting the finite elements mesh will be presented on the example of a pneumatic gripper element (Fig. 10).
The process begins when a mesh size of the element gives a result with 6.1% error. This accuracy is not satisfactory so the adaptation process was prepared for the assumed parameters:

- stress percentage error 3%,
- deviation of stress error equal to zero stress, and therefore ignored in the analysis,
- maximum of five iterations,
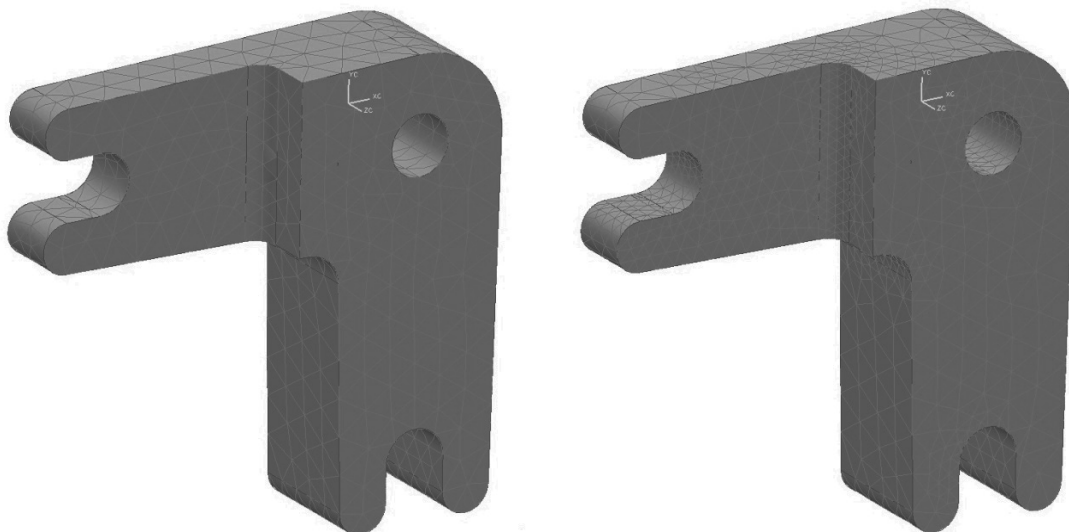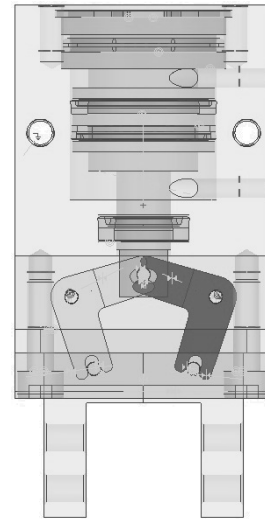- minimum size of the element – 0.

Fig. 10. Pneumatic gripper

Already in the first iteration, the error was reduced to 2.95% (Fig. 11).

Analyzing the new mesh can be seen that, in places where power was applied or constrained, mesh is more dense. This causes smaller stress error.

5. Optimization

Optimization is the criteria choice from the field of possible solutions [6]. At this stage, on the one hand there is a field of possible solutions arising during conceiving process, on the other system criteria.

Fig. 11. Mesh adaptation

The sources of criteria system creation are [6]:
- design – construction assumptions,
- product right of existence:
  - the technical reason for expediency;
  - the production capacity;
  - the economic right;
- principles of design science:
  - the principle of optimal load;
  - optimal materials;
  - optimal stability;
  - the optimum size of the relations involved.

Resulting from these criteria are: the criteria for stress, stiffness, frequency, dynamic, durability and reliability.

Optimization can be subjected to the system size, material, character design and topology.

Many machine elements and equipment has a very high degree of complexity. Due to that even the most experienced engineer is not always able to take all factors into account [12]. Modern software engineering comes to help him in such situations by offering tools that support the process of optimization.

One of the leading producers of software CAD / CAM / CAE is a Siemens PLM Software. The NX program includes a special tool to optimize the element dimensions values by strength verification using Finite Element Method. This tool is available in the *Advanced Simulation Module*.

The principle of this tool stems from the stress analysis file structure (Fig. 12). The verification process by finite element method starts with the CAD model. Based on it the idealized model is generated (the default _i appended to the name, e.g. model1_i.prt), which is a copy of the standard item. This allows user to simplify the model and prepare it properly by removing parts that do not affect the outcome (e.g., small diameter holes, undercutting, chamfering), enter the breakdown of walls, etc. without interfering in main CAD file. On the basis of so-prepared model the new file is created with the extension .fem, which has defined the finite element mesh and plastic characteristics. User can define multiple .fem files associated with one idealized model. The last is the file with .sim extension containing the boundary conditions and other items specific to the solver [12].

The optimization process is invoked from the simulation level. While it is running the dimensions values in the file containing the model are blocked and editing them is impossible. It is very important that the user can not interfere in the dimension values modified by the program.

The optimization process proceeds in an iterative way using an Altair HyperOpt algorithm developed by Altair Engineering company. This algorithm is responsible for appropriate variables values adjustments to achieve the target. When the process is started, each of the defined variable is modified by a certain amount and then a model is updated. Rebuilding the model requires re-definition of the finite element mesh. The modified item is verified using Nastran solver. The result is compared with the intended purpose. On the basis of the difference between the obtained result and stated objective the next adjustment of variables take place and the above steps are repeated in the next iteration. After reaching the convergence parameters or the maximum number of iterations the process is interrupted and the results are printed.

The target definition is the first step in preparing the analysis. User can choose between the aim characterizing the objects one, two, or three-dimensional mode. This could be stress, strain, displacement, weight or volume. Engineer must indicate when the process will minimize / maximize a given variable or the targeted, specific value must be achieved. Then he has to define the upper or lower limits, which cannot be exceeded. As in the target case that can be stress, strain, displacement, weight or volume. The next step is to identify variables that will be the subject of modification. The program allows access to all model parameters. Finally it is necessary to specify the conditions which end the analysis. The first is the maximum number of iterations. If other conditions are not met before the analysis is stopped after reaching the number specified here. Another, *Max Constraint Violation*, determines the percentage deviation of pre-specified limit; when achieved the process ends. *Relative Convergence* – the convergence defines the relative percentage difference between last two results and the *Absolute Convergence* – by analogy, their absolute values. If the difference between last two results is less than that, the program exits, because further modifications of variables no longer have a significant impact on the outcome. From *Perturbation Fraction* depends on how the variables may be modified in subsequent iterations.

NX program works with Excel spreadsheet. The optimization results are presented in a clear form of graphs and tables. In addition, each step of verification is available in the postprocessor window so user can trace the stress distribution in each iteration.

The method of optimization by NX will be presented on an element of pneumatic gripper (Fig. 10).

Optimization has been subjected to an element of a gripper arm. The objective is to minimize weight by reducing the radius on the end of the part, without exceeding the allowable stress values. Although variable is only one dimension in the process, it is difficult to immediately give this value, which allow to achieve the maximum effort of the material.
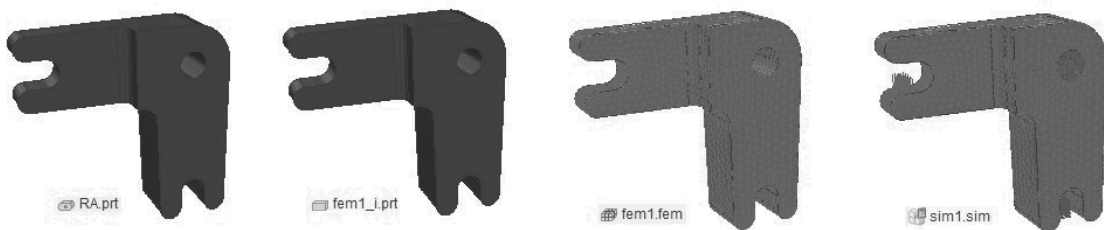

Fig. 12. Analysis file structure

The simulation preparation process is the same as a standard FEM verification. The locations of maximum stress is necessary to mesh density. In the case of such a process and prepared for a pre-established dimensions, the maximum stress was obtained at 120.8 MPa. The permissible stresses are at the level of 200 MPa, so the use of the material is not optimal. The following inputs to the optimization was defined: the objective – weight minimization, limits - the maximum stress equal to 200 MPa, the variables - the radius of the element. The maximum number of iterations – 8. Convergence parameters: the deviation limit - 2.5%, the relative convergence - 2.5, absolute convergence – 0.001, the coefficient perturbations 0.2.
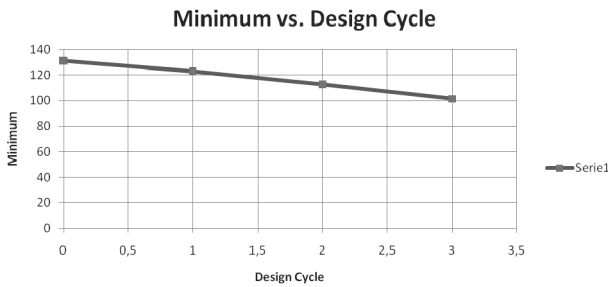
Minimum vs. Design Cycle

Fig. 13. Mass values

For these data it was possible to reduce the radius from 1.8 mm to 1.22 mm, which allowed to reduce the element weight from 131.3 g to 101.4 g – ca 23% (Fig. 13) at a stress of up to 199.6 MPa (Figs. 14 and 15).
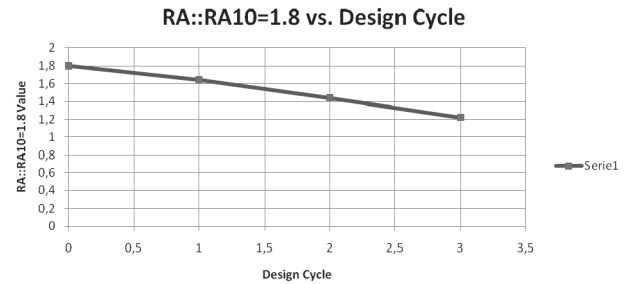
RA::RA10=1.8 vs. Design Cycle

Fig. 14. Variable values

6. Conclusions

The main limitation of the presented tools in the case of elements with a high degree of complexity is the calculation time consumption. When the process runs in iterative way the FEM calculations are performed repeatedly. In the case of a complicated model and too many variables, computation time may increase to an unacceptable level. It is important to simplify the model as much as possible and define the minimum number of variables.
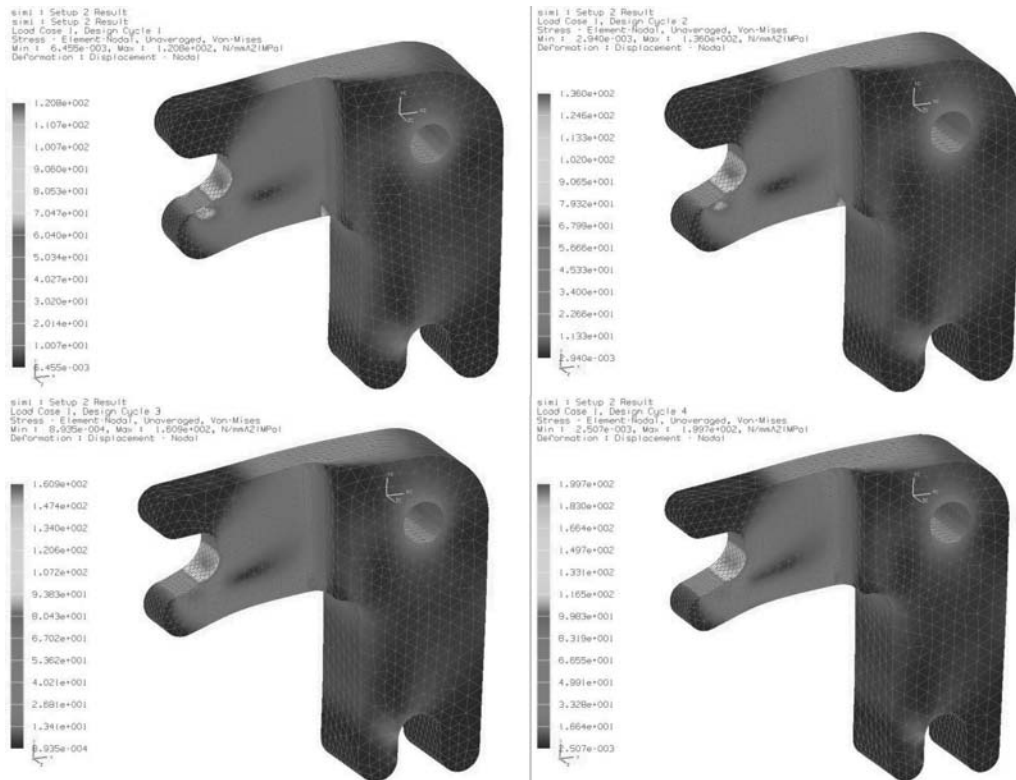

Fig. 15. Stress distribution in the iterations

The main advantage resulting from the application of the presented methods is primarily time savings compared with manual analysis of the impact parameter on the result. On several variables benefit of the tool will be even greater. For the automatic optimization of many variables may be the only solution. Properly executed process will result in better use of the material, which will shorten the manufacturing time and therefore its cost, which will contribute to reducing the losses of materials and energy. These savings will be directly reflected in the financial profit of the company.

Acknowledgements

References

[1] Z. Adamczyk, K. Kociolek, CAD/CAM technological environment creation as an interactive application on the Web, Journal of Materials Processing Technology 109/3 (2001) 222-228.

[2] J. Dietrych, Machines Construction Basis, WNT, Warsaw, 1995 (in Polish).

[3] J. Dietrych, System and construction, WNT, Warsaw, 1985 (in Polish).

[4] P. Gendarz, Calculation of dimensions for the construction series of types, Construction 50 H.10 (1998) 23-28 (in German).

[5] P. Gendarz, M. Cielniak, Models of construction attributes selection process in ordered construction families, Journal of Achievements in Materials and Manufacturing Engineering 43/1 (2010) 280-287.

[6] R. Rząsiński, P. Gendarz, Technological operators of series of types technology creating, Journal of Achievements in Materials and Manufacturing Engineering 41 (2010) 155-163.

[7] E. Gerhard, Series development. Contact & Study Band 105, Expert Verlag, Grafenaut, 1984 (in German).

[8] Z. Orłoś, Strength of materials, WNT, Warsaw, 1996, (in Polish).

[9] G. Pahl, W. Beitz: Construction science, WNT, Warsaw, 1984 (in Polish).

[10] R. Rzasinski, P. Gendarz, The creation of ordered technologies on the basic of ordered constructions of machines, Proceedings of the 3rd Scientific International Conference "Materials, Mechanical and Manufacturing Engineering" MMME'2005, Gliwice - Wisła, 2005, 203-207.

[11] K. Roth, Designing with design catalogues 2, Springer-Verlag, Berlin, 1994.

[12] S. Samuel, E. Weeks, B. Stevenson, Advanced Simulation Using Nastran NX5 / NX6, Design Visionaries, USA.

[13] W. Tarnowski, Basics of technical designing, WNT, Warsaw, 1997.

[14] S. Tkaczyk, M. Skucha, The application of the Balanced Scorecard Method in Organization Management,. Proceedings of the 9th Scientific International Conference "Achievements in Mechanical and Materials Engineering" AMME'2000, Gliwice - Sopot - Gdańsk, 2000, 549-552.

[15] J. Wróbel, Computer technique for mechanics, PWN, Warsaw, 1994.