CAD-FEA study on parameter sensitivity in the Grodzinski method for an axial load case

Mihail Aignatoaie*
Department of Mechanical Engineering, Mechatronics and Robotics, Faculty of Mechanical Engineering, Technical University “Gheorghe Asachi” of Iasi, Blvd. Mangeron 59-63, IASI-700050, Romania

Received 1 July 2013; accepted 11 February 2014

In mechanical design it is necessary to reduce the stress concentration in areas of the structure with sudden geometrical changes. Topology and shape optimization is a modern tendency, but it implies some theoretical and practical difficulties. As an alternative, there is so called “simplified shape optimization”, making use of several shapes, available within modern CAD editors (FILLET, ELLIPSE, etc.) or copying the forms of the bionic structures. The final decision depends on the performance of a specific shape in reducing the stress concentration but also its ability to be implemented in a particular arbitrary case. The paper presents the Grodzinski method, that can be used in stress concentration areas with practically any aspect ratio. There is hardly any reference in the literature on the influence of the $n$ parameter (the number of division intervals in geometrical construction). A CAD-FEA study is performed on the influence of increasing $n$ parameter. The test case is a notched specimen, axially loaded. There are analyzed stress concentration coefficients $K_t$, $\sigma_{vomiss}$ and some geometrical characteristics of the stress concentration area. The results of the Grodzinski profiles are compared with the alternative options FILLET and ELLIPSE.

Keywords: Stress concentration, Grodzinski method, Simplified shape optimisation, CAD, FEA

In a global economy subjected to rapid changes any competitive company is striving to produce high quality products in a short time, at convenient prices. This implies significant improvements in creativity design and the CAD process. In industrial design the engineer must find the best shapes in stress concentration areas in order to reduce the maximum stress.

The topology and shape optimisation is a modern option for improving the shape of mechanical structures. Its practical use is not simple because of some theoretical aspects and also from practical point of view. The necessary specialized software packages or the ability of use them with proficiency are not always available. When applied, the necessary time for generating the optimised final CAD model could be significant.

As an alternative, the engineer must rapidly adapt geometrical shapes able to reduce the stress concentration phenomenon in any general practical case. Sometimes this option is known as “simplified shape optimisation”. The classical option is the FILLET shape. Present CAD editors can also generate elliptic or hyperbolic shapes.

Modern tendencies include the use of bionic shapes, optimised by nature. A problem in this case is to find the best option to generate the bionic shape. Mattheck introduced a very simple algorithm that copies the shape of the trees. The algorithm of Mattheck could be implemented only in a space with an imposed aspect ratio. Generally speaking the use of a specific method could be difficult or even impossible to be implemented in a specific design with arbitrary given aspect ratio.

A classical, probably not very well known algorithm, developed by Grodzinski last century, has the ability to generate a shape for a stress concentrator with practically any given aspect ratio. The algorithm uses a graphical construction using $n$ as a parameter of divisions of the segments defining the concentrator area. There are hardly any additional indications or studies in the literature on the influence of the $n$ parameter in practical cases. The paper is trying to promote this method in a study on the influence of the sensitivity of the $n$ parameter on the generated shape and on the stress distribution. It continues a previous CAD-FEA study of the author in the direction of improving the Grodzinski algorithm.

The CAD models were generated applying “manually” the algorithm of Grodzinski in TurboCAD.

---

*E-mail: maignato@mail.tuiaasi.ro
package (IMSI, Inc, USA). The case-study has considered an axial loading. The stress concentration phenomenon is studied with finite element analysis (FEA) by using Algor (Autodesk, Inc.).

The validity of the FEA model was evaluated with special designed tool from Algor software: “Precision of von Misses Stress”. The stress distribution in all the considered cases has been analysed. As reference models for all Grodzinski study-cases there were also included the classical FILLET and ELLIPSE shape options.

**Test Case Study**

A notched machine element (Fig. 1), from AISI1010 steel, is encastered to the left end and axially loaded to the right end. It presents a sudden change in cross-section from AA’ to BB’. The geometrical construction of the stress concentrator must be implemented in an area with a given aspect ratio AB:AC = 1:3 (Fig. 1).

The process of generating the Grodzinski profile is shown in Fig. 2. The segments BA and AC are divided into same number \( n \) of segments. Figure 2 presents the case \( n = 10 \). The division intervals are noted 1, 2, … , 10, 11 on both BA and AC segments. Construction lines unite points having the same notation on BA and AC. The start point B, Fig. 1, and a series of points defined by the intersection between each two consecutive construction lines and finally the end point C define a spline curve. This curve defines the Grodzinski profile of the concentrator. Figure 3 presents the geometrical constructions and the final profiles for the cases \( n = 20 \) and \( n = 30 \), both generated “manually” in the CAD editor.

For comparison, this study also included the FILLET case (R = AB = 50 mm) and the ELLIPSE case with aspect ratio 1:3 (minor axis = AB = 50mm, major axis = AC = 150 mm). A CAD study of these profiles is presented in Fig. 4a. At this scale all the
Grodzinski cases \((n = 10, 20, 30)\) actually overlapped and no difference could be noticed. The details of these profiles in the vicinity of P1, P2 and P3 points of Fig. 4a are represented in magnified form in Figs 4b, 4c and 4d, respectively.

**FEA Analysis**

Due to the longitudinal axial symmetry of the test case study (Fig. 1), the CAD model and the FEA model (Fig. 4a), have considered only half of the initial geometry. The model is encastered at the left end in EF zone. The area FG is included into the longitudinal symmetry plane. Therefore, the boundary conditions were “rollers”. The axial load acting at the right end was uniformly distributed and corresponds to a value of 1 MPa.

The mesh was generated automatically, using 3D elements, type “brick” with 4, 5, 6 and 8 nodes. In order to have comparable results for all the considered cases (FILLET \(R=50\) mm, Grodzinski: \(n=10, 20, 30\) and ELLIPSE aspect ratio 1:3), the automatic mesher had the same settings for the basic parameters. A comparison of the FEA parameters for all the studied cases is presented in Table 1.

Algor software has a specific procedure to evaluate the precision of a FEA model. This method calculates the stepped changes in results from one element to the next. In an ideal model, the stress is expected to change smoothly between adjacent elements. In the process of meshing the model, there are always some changes in results from one element to the next and the results are not continuous. In a structural analysis the elements with shared nodes predict stresses independently at the node, and therefore the independent stress calculations provide a "precision" estimate for the model based on the von Misses stress. The precision of von Misses stress at a given node is calculated, with the Eq. (1).

<table>
<thead>
<tr>
<th>Test-Case FEA Model</th>
<th>No. of Elements (Bricks)</th>
<th>No. of Nodes</th>
<th>DOFs (Degrees of freedom)</th>
</tr>
</thead>
<tbody>
<tr>
<td>FILLET, Radius (R=50)</td>
<td>23269</td>
<td>40109</td>
<td>119321</td>
</tr>
<tr>
<td>Grodzinski, (n = 10)</td>
<td>27099</td>
<td>41922</td>
<td>124760</td>
</tr>
<tr>
<td>Grodzinski, (n = 20)</td>
<td>26131</td>
<td>41857</td>
<td>124565</td>
</tr>
<tr>
<td>Grodzinski, (n = 30)</td>
<td>25538</td>
<td>41679</td>
<td>124031</td>
</tr>
<tr>
<td>ELLIPSE, Aspect ratio 1:3</td>
<td>25975</td>
<td>42332</td>
<td>125990</td>
</tr>
</tbody>
</table>

Note: In all cases the meshing type was automatic with the same settings for basic parameters.
**Results and Discussion**

It has been investigated throughout the entire model the distribution of the stresses, for evaluating the performance of each and every model on the reduction of stresses concentration. Selective images of the distribution of \( \sigma_{\text{von Misses}} \) are presented in Figs 5, 7 and 9, and \( \sigma_x \) in Figs 6 and 8.

The effect of stress amplification can be described using the \( K_t \) coefficient, the ratio between the maximum stress at the notch divided by the nominal stress of the part, Eq. (2).

\[
K_t = \frac{\sigma_{\text{max}}}{\sigma_{\text{nom}}} \quad \text{... (2)}
\]

Fig. 5—Case \( R=50 \): von Misses stress distribution

Fig. 6—Grodzinski case \( n=10 \): \( \sigma_x \) stress distribution
Where, $\sigma_{\text{max}}$ = maximum value of the stress determined experimentally or calculated with computational methods; $\sigma_{\text{nom}}$ = the nominal value of the stress calculated with simple formulas known from the strength of materials.

In present case $\sigma_{\text{max}}$ could be defined by the maximum value of $\sigma_x$, for each and every studied model. $\sigma_{\text{nom}}$ is actually $\text{SigX} = 1$ MPa (Fig. 4a), the load uniformly applied in all the cases.

The CAD study was also oriented on other geometrical parameters of the considered models:

(i) the “length of profile (LP)”: it represents the length of the spline curve defined by point B- specific studied profile (FILLET, Grodzinski case $n=10, 20, 30$ or ELLIPSE aspect ratio 1:3)- point C, Fig. 4a;

(ii) the “Volume of material (VM)”: it corresponds to the volume of the model defined by the area: point A- point B- specific studied profile - point C – point A on the entire thickness = 1 mm of the models.

The ideal model should have a minimum stress concentration coefficient $K_1$ and $\sigma_{\text{von Misses}}$. A minimum VM indirectly means a minimum weight. An ideal shape of the stress concentrator should also assume a better behaviour to fatigue. Theoretically\textsuperscript{11,12} a larger area in the stress concentration zone could increase the probability of a fatigue failure. That is why a minimum value for the LP and the VM are more favourable for a better behaviour of a specific model to fatigue.

Table 2 presents the most important results from the CAD and FEA study for each and every considered case. Some of the presented FEA results could be extended to study cases different from those described in Fig. 4a by use of similitude\textsuperscript{13}.
Conclusions

The study presented the influence of the $n$ parameter (cases $n = 10$, 20 and 30) in Grodzinski method on the stress concentration (determining $K_t$ and $\sigma_{\text{von Misses}}$) and on geometrical parameters (the length of the profile and the volume of material in the concentrator area) directly influencing the behaviour on fatigue. The Grodzinski profiles were comparing with the FILLET case and the ELLIPSE case. The following conclusions can be drawn:

(i) The classical FILLET option has just the advantage of the smallest volume of material, which is more than 55% smaller than the values calculated in all the other cases. The other parameters, by comparison with the rest of the studied cases, are definitely not convenient: the length of the profile is more than 4% bigger and the stress concentration coefficient is with at least 38% bigger. The maximum von Misses stress is more than 42% bigger than the rest of the study-cases. FILLET case would be recommended only when the object of reducing the weight is more important than reducing the stress concentration.

(ii) The ELLIPSE case provides the shortest length of the profile and the biggest volume of material. The stress concentration coefficient $K_t$ and the maximum value of the von Misses stresses are significantly smaller than the FILLET option, however slightly higher than the Grodzinski cases.

(iii) The Grodzinski profiles have smallest values for the stress concentration coefficient and von Misses stresses. The length of the profile is slightly bigger than the ELLIPSE case but clearly smaller the FILLET option. The increase of the $n$ parameter in the Grodzinski method, for the studied cases produces a non very important decrease of the length of the profile and an insignificant increase of the volume of material. It has a favourable, small influence on reducing both $K_t$ and $\sigma_{\text{von Misses}}$.

All the Grodzinski profiles were manually constructed. The $n = 30$ case was actually a “limit of pain” in applying the method manually. The future alternative would be the construction of the Grodzinski profiles by use of special designed script applications.
in the CAD editor or by use of the Python language. An interesting tendency is the use of modern approaches\textsuperscript{14,15} in order to integrate the geometrical model with the FEA package. The optimisation process of the profiles shape could be automated by use of the programming language Python.

Although optimal shape of the stress concentrators improves stress distribution, usually it means very complex geometric shapes. Such machine elements could only be produced by use of modern CNC machine-tools. The Grodzinski method implies some difficulties in generating the CAD model or in manufacture. This procedure offers however an adaptability in practical arbitrary conditions and make possible to reduce the stress concentration better than other known options.

References

8. Mattheck C, Concealed shape laws of the nature - Optimum forms without computer (in German), (Forschungszentrum Karlsruhe GmbH, Karlsruhe, Germany) 2006, 1-79.