HOOK CRANE SHAPE DESIGN IMPROVEMENT FOR REDUCING THE MAXIMUM STRESS

DAN N. DUMITRIU , MARIUS IONESCU, CRISTIAN RUGINA

Abstract. The stress analysis of a hook crane submitted to some vertical static force is a simple FEM problem, approached here using SolidWorks Simulation software. Our purpose was to simulate different possibilities to slightly modify the shape of the hook crane, without adding too much material, in order to reduce the maximum stress induced in the hook. The shape design concerned here both the shape of the (transversal) cross-section and the shape of the hook longitudinal profile. The stress distribution is strongly depending on the shape of the hook crane, but it is almost independent on the hook material, since the bending mechanical stress, which is dominant in this combined bending and elongation case study, is not depending on the elastic modulus of the hook material. Obviously, the material choice is crucial from the point of view of the yield strength, usually hardened steels with increased yield strength are preferred. In what concerns the shape design improvement from the point of view of the cross-section (of equal area), our study confirmed that the trapezoidal section proposed generally in the literature corresponds to a reduced maximum induced stress. As a new result, a T-shape hook crane cross-section seems to prove better results that the trapezoidal section. To avoid stress concentrators, a combination between the trapezoidal and T-shape cross-sections has been also considered. The idea is to avoid as much as possible the discontinuity points on the outer contour of hook cross-section, since these discontinuity points are usually mechanical stress concentrators. A shape design improvement has been performed also in what concerns the longitudinal section/profile of the hook crane.

Key words: Hook crane, stress distribution, shape design improvement, smallest maximum stress, transversal section, longitudinal profile.

1. INTRODUCTION

The hook crane is a classical end-effector tool/device, used for lifting heavy loads by means of a lifting crane, in various industrial activities such as construction sites, shipyards, ports. Being used to carry tons of loads, the hook is thus a component subjected to strong mechanical stress. In this context, any shape design improvement which targets the reduction of the maximum stress induced in the hook material, becomes very important.

Institute of Solid Mechanics of the Romanian Academy, Bucharest

Ro. J. Techn. Sci. – Appl. Mechanics, Vol. 68, N° 1, P. 19–39, Bucharest, 2023

DOI: 10.59277/RJTS-AM.2023.1.02

To improve the shape and maximum induced stress of a hook crane, a Taguchi optimization approach was considered by Bhatkar *et al*. [1] in relation with an ANSYS software FEM study. They have considered three steel-based materials, quite similar in terms of Young's modulus and Poisson ratio, and with quite close tensile strengths. A trapezoidal section of the hook crane was considered, being close to the idea of a section of equal strength. The best suitable material for the manufacturing of the hook crane was simulated to be AISI-4340 alloy steel. The simulation was performed using a statical force of 150 kN, and considering a width of the trapezoidal section of about 180 mm, for this case study resulting by ANSYS simulation a maximum induced stress of 143.31 MPa, corresponding to the maximum/induced deformation of 1.5452 mm. The error compared with analytical results for this case study was found to be 5.21%.

A similar FEM static structural analysis using ANSYS software was performed by Divakari *et al*. [2], with the same purpose of "design and optimization" of a hook crane. Their conclusion was that "the material which is having less deformation will be more stable" [2]. Still from the material choice point of view, Zade [3] performed a hook crane FEM study considering three types of materials: a structural steel, wrought iron and an alluminium alloy. The results concluded that the smallest maximum stress is obtained for the hook crane made of structural steel and the one made of wrought iron.

A review of several research studies "*analysing crane hook of different cross sections and different materials*" has been recently performed by Singh et al. [4]. For example, Uddanwadiker [5] performed a crane hook stress analysis, validating the FEM ANSYS results by experimental photo-elasticity. The obtained stress distribution was also in good agreement with the analytical results, i.e., curved beam formulas available for the classical crane hook shapes [6, 7]. Some important suggestions to reduce failure of hooks are concluded by Uddanwadiker [5]: "crane hooks produced from forging are much stronger than that produced by casting"; "grain refinement process such as normalizing is advisable after forging"; processes such as welding or removal of metal from the hook should be avoided, since they clearly increase stress. In what concerns design/shape optimization, it is recommended to increase the thickness of the inner curvature of the hook [5], so the same trapezoid section is recommended as in literature, obviously with the large base of the trapezoid located on the inner side of the hook.

Similar crane hook FEM studies have been performed by several authors:

– Bhagyaraj et al. [8] performed a SolidWorks Simulation FEM analysis for a designed crane hook made up of Alloy 1.2367 (X38CrMoV5-3), which shows very good yield and tensile strength properties, both estimated at 2120 MPa. As a conclusion, the size of hooks made of such high-strength alloys can be reduced due to their increased yield strength properties.

– Lakshmana Moorthy and Prakash [9] performed crane hook ANSYS FEM simulations for two hardened-tempered alloy steels AISI 6150 (Chromium-Vanadium alloy steel, yield strength 1160 MPa) and AISI 4140 (Chrome alloy, yield strength 990 MPa), with the following material hardening contributions:

"Chromium improves corrosion resistance and hardenability; Manganese gives work hardening property; Vanadium improves wear resistance properties; Molybdenum helps in deep and thorough hardening, increases hot hardness and creep strength; Silicon improves oxidation resistance and hardness" [9]. The authors studied the shape design of the crane hook, concluding that the trapezoidal cross-section of the crane hook provides increased/improved stress to weight ratio, compared with circular or rectangular cross-sections.

Similar comparisons between trapezoidal, triangular, rectangular and circular cross-sections of a crane hook were simulated by Bergaley and Purohit [10], Gopichand *et al*. [11], Mehendale and Wankhade [14], etc, all concluding that the smallest maximum stress on the inner width of the crane hook corresponds to the trapezoidal cross-section. In order to further reduce the stress concentration, some curvature should be considered at the corner of the trapezoidal cross-section [10, 12, 13].

In fact, Gopichand et al. [11] applied a Taguchi optimization method for the following design parameters: the radius of curvature of the hook, its cross-section and the selected material.

Still in order to minimize the induced maximum stress, several studies [10, 12, 13] have tried to optimize the widths of the two bases of the trapezoidal cross-section (inner and outer width) of the hook crane. For example, in [13] the value of the large base B of the trapezoid, which is located at the inner side of the hook, is varied from 110 mm to 130 mm, whereas the value of the small base b of the trapezoid, located at the outer side of the hook, is varied from 45 mm to 55 mm. The purpose is obviously to find the best combination of B and b, corresponding to a simultaneous reduction of the maximum induced stress and of the crane hook mass, more precisely a trade-off must be found between maximum stress minimization and weight minimization. The displacement and strain distributions in the crane hook are also considered.

Methods other than FEM can obviously be used to simulate the crane hook elastic mechanical behaviour, for example Qin et al. [15] used the boundary interpolated reproducing kernel particle method (BIRKPM), deduced by combining the interpolated reproducing kernel particle (IRKP) method with the boundary integral equation (BIE) method which aims to solve elastic mechanics plane stress. From our point of view, the shape of the crane hook used in this paper [15] is very interesting, where the first quarter circle of the crane hook is flattened, being more a straight line rather than a quarter of circle. This hook shape will be considered as variant also in this paper, *in our search to find a hook shape minimizing the maximum induced stress for the same overall hook mass (of all considered candidate shapes)*.

Another shape modification idea to be tested in this paper is inspired by Leopardi and Strozzi [16], who assessed that a "paradoxical behaviour occurs in a curved beam subjected to bending, by laterally removing material from section zones close to the neutral axis. The bending stress diminution is often of the order of a few %, whereas the mass diminution may reach 10%" [16]. In the spirit of this idea, we are testing in this paper the T-shape cross-section of the crane hook, as alternative to trapezoidal or other cross-sections.

In what concerns the crane hook materials considered in the above-mentioned FEM studies, it is quite common to use alloy steel AISI 4340, as in [12], or alloy steel AISI 4140, as used in [13].

2. PROBLEM FORMULATION AND CASE STUDY

A classical hook crane is considered here, being fixed in its upper part as illustrated in Fig. 1a. A vertical force of 150 kN is statically applied, more precisely this force is uniformly distributed over a small application surface, as in a practical case.

Figure 1b presents the longitudinal profile of the hook crane considered in the first part of the shape design improvement study, where several transversal/cross sections are considered for the same longitudinal profile. The different cross-sections considered have the same area, so that to have the same overall mass of the hook crane. So, the study trying to find which is the cross-section which corresponds to the smallest maximum induced von Mises stress, is performed for hook cranes having almost the same mass, i.e., for cross-sections having the same area.

The considered material for the hook crane in this study is normalized AISI 4340 Ni-Cr-Mo steel, with the following mechanical properties: elastic modulus E = 205 000 N/mm^2 (MPa); Poisson's ration ν = 0.32; yield strength = 710 MPa. The stress distribution is strongly depending on the shape of the hook crane, but it is almost independent on the hook material, since the bending mechanical stress, which is dominant in this combined bending and elongation case study, is not depending on the elastic modulus of the hook material. Obviously, the material is important from the point of view of the yield strength, so that the hook crane to withstand higher mechanical stress. So, hardened steels with increased yield strength are used.

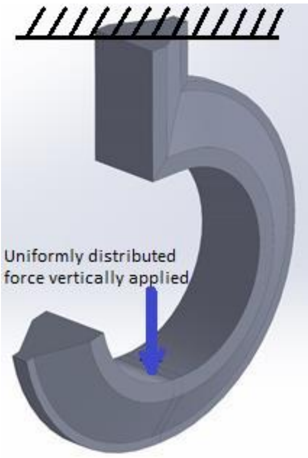


Fig. 1a – Hook crane in 3D representation.

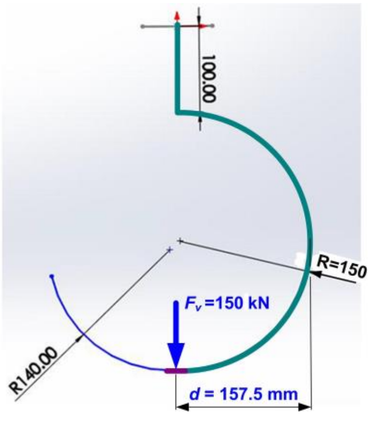


Fig. 1b – Longitudinal profile of the hook crane considered.

All FEM study results in this paper are obtained using SolidWorks Simulation software, more precisely the static FEM simulation module. Figure 2 shows the adaptive mesh used in this study, more precisely a curvature-based mesh with fine density was preferred, involving 611 267 total nodes and 421 174 total elements. For this fine mesh density provided by SolidWorks Simulation software, the simulations illustrated in Fig. 3 were performed in 33 seconds.

In comparison, when a medium mesh density provided by SolidWorks Simulation was used, involving 444 411 total nodes and 307 444 total elements, the computing time decreased slightly to 28 second. The maximum von Mises stress is obtained the same for the case of fine mesh density versus the case of medium mesh density, so the mesh convergence is verified.

The results below are performed using the adaptive mesh with the fine mesh density shown in Fig. 2, since the computation time is affordable.

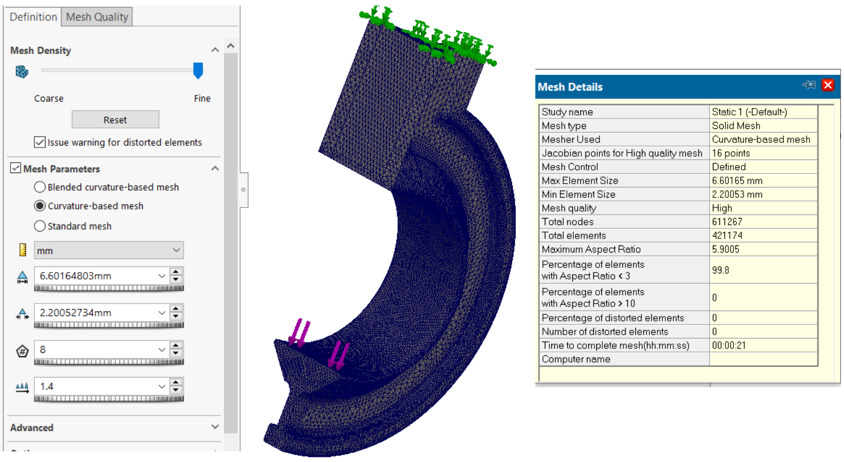
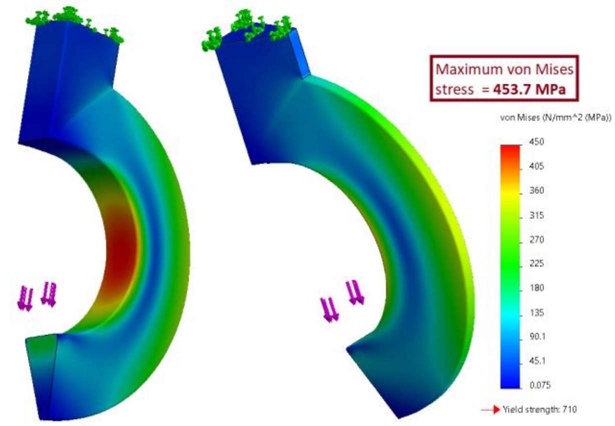
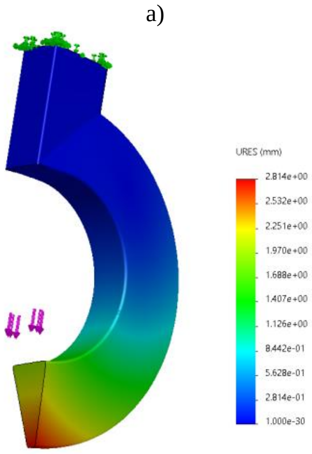


Fig. 2 – FEM SolidWorks Simulation mesh details, using a fine mesh density, as provided by SolidWorks Simulation.

Figure 3 shows the von Mises stress distribution for this case study where a vertical force $F_v = 150\,\text{kN}$ is statically applied on a crane hook of trapezoidal cross-section, with the following dimensions: parallel sides of $b_1 = 65\,\text{mm}$ and $b_2 = 15\,\text{mm}$, with $h = 100\,\text{mm}$ height. Two different angle views are presented, showing that a maximum von Mises stress of $\sigma_{\text{tot,max}} = 453.7\,\text{MPa}$ is obtained on the inner surface, in the median part of this circular shape hook. The total/resultant stress σ_{tot} is obtained by adding the bending stress σ_b corresponding to the hook bending and the direct/tensile stress σd corresponding to the hook tensile behaviour. The maximum displacement is 2.81 mm, obtained obviously in the bottom part of the hook.



a)



b)

Fig. 3 – FEM SolidWorks Simulation results obtained for a hook crane with trapezoidal cross-section, submitted to a vertical force of 150 kN, statically applied: a) distribution of the von Mises stress; b) displacements distribution.

In Fig. 3 and in the following FEM SolidWorks Simulation figures, the crane hook is cut in cross-section in the area where the vertical force is applied, so that to better visualize the stresses in this particular section, where increased stresses may arise due mainly to the direct/compression stress. Especially for hooks with T-shape cross-section or other shapes with edges and corners in the middle part of the cross-section, these edges and corners represent stress concentrators and the stresses will be increased in these areas, so we will pay special attention to these situations. "*The best is the enemy of the good*" (aphorism), so we will have to be careful that, when searching for a better cross-section shape than the trapezoidal one, the reduction of the maximum stress occurring on the hook inner surface for the new shape might unfortunately be associated with increased stresses in other areas, such as the one where the vertical force is applied.

3. ANALYTICAL SOLUTION AVAILABLE FOR HOOK STRESS ANALYSIS

An analytical solution for hook stress analysis is available in the literature [5, 6, 7, 10, 11, 14]. In the following, a brief detail of this simple analytical solution is presented for the case of a hook with trapezoidal cross-section. Figure 4 shows a part of the longitudinal (curvature) section and the trapezoidal cross-section of the considered hook, with the following dimensional notations and values (for this case study): $b_1 = b_{\text{int}} = 65\ \text{mm}$ and $b_2 = b_{\text{out}} = 15\ \text{mm}$ are the parallel sides of the trapezoidal section, the larger side $b_1 = b_{\text{int}}$ being located at the inner part of the hook; $h = 100\ \text{mm}$ is the height of the trapezoidal section; it results a trapezoidal section area $A = \dfrac{b_1 + b_2}{2} h = 4000\ \text{mm}^2$; $r_1 = r_{\text{int}} = 100\ \text{mm}$ is the radius at the inner surface/side of the hook and $r_2 = r_{\text{out}} = 200\ \text{mm}$ is the radius of at the outer hook surface.

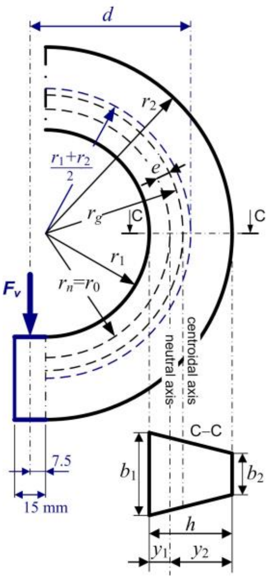


Fig. 4 – Partial longitudinal (curvature) section and trapezoidal cross-section, reproduced from [7], of the hook considered in Fig. 3.

For a homogenous hook, the radius of curvature r_g at centroidal axis and the radius of curvature at neutral axis $r_n = r_0$ are computed as follows [5, 7, 14]:

$$r_g = r_1 + \frac{h(b_1 + 2b_2)}{3(b_1 + b_2)} = 139.58\ \text{mm (in our case study)}; \tag{1}$$

$$r_0 = \frac{\left(\dfrac{b_1 + b_2}{2}\right)h}{\left(\dfrac{b_1 r_2 - b_2 r_1}{h}\right)\log\left(\dfrac{r_2}{r_1}\right) - \left(b_1 - b_2\right)} = 134.63\ \text{mm}\,. \tag{2}$$

(in our case study).

By definition, the neutral axis is the geometric place where the bending stress is null, while the centroidal axis is the geometric place of the centres of gravity of the hook cross-sections. The eccentricity e is the distance from the centroidal axis to the neutral axis [5, 7, 14]:

$$e = r_g - r_0 = 4.95\ \text{mm}. \tag{3}$$

For hook section C–C, the total/resultant stress σ_{tot} is composed from the direct/tensile stress σ_d and the bending stress σ_b :

$$\sigma_{\text{tot,C–C}} = \sigma_d + \sigma_b\,, \tag{4}$$

where

$$\sigma_d = \frac{F_v}{A} = 37.5\ \text{MPa}\,, \tag{5}$$

while the bending stress σ_b is given by [5, 7, 14]:

$$\sigma_b = -\frac{M}{Ae}\frac{y}{r_0 - y}\,. \tag{6}$$

In expression (6), the bending moment is given by $M = F_v d$, with d the perpendicular distance between F_v force axis and the vertical axis passing at middle distance between the inner and outer sides of the hook, while $y = r - r_0$ is the distance from the neutral axis to the point where the bending moment is computed. For the cross-section most distanced from the force axis, i.e., the hook section C–C, where the perpendicular distance $d = 157.5\ \text{mm}$ and the bending moment shows a maximum value $M_{\max} = 23\,625\,000\ \text{Nmm} = 23.625\ \text{kNm}$, the following bending stresses are computed at the inner surface and at the outer surface, as follows:

$$\begin{gathered} \text{– at inner surface: } \sigma_{b1} = -\frac{M}{Ae}\frac{r_1 - r_0}{r_1} = 412.55\ \text{MPa} > 0 \Rightarrow \\ \sigma_{\text{tot},1} = \sigma_d + \sigma_{b,1} = 450.05\ \text{MPa}; \end{gathered} \tag{7}$$

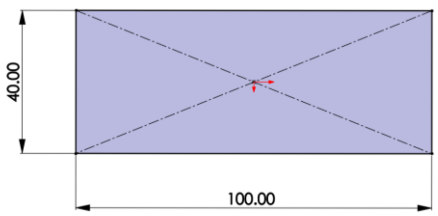
$$– \text{ at outer surface: } \sigma_{b2} = -\frac{M}{Ae}\frac{r_2 - r_0}{r_1} = –389.44 \text{ MPa} < 0 \Rightarrow$$
$$\sigma_{\text{tot},2} = \sigma_d + \sigma_{b,2} = –351.94 \text{ MPa}. \quad (8)$$

Let us compute the relative error between the value 450.05 MPa of the total stress obtained analytically in (7) and the value 453.7 MPa of the von Mises stress obtained using SolidWorks Simulation software FEM at the middle of the inner surface, where is located the maximum stress. The relative error is only 0.8 %, so extremely reduced, this verification confirms the validity of using SolidWorks Simulation, so the results obtained below are correct.

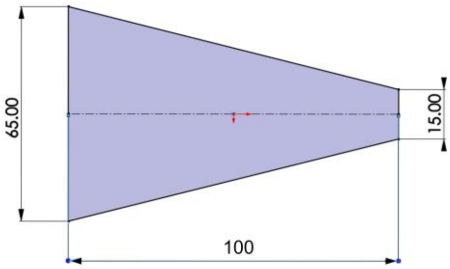
4. CROSS-SECTION SHAPE DESIGN IMPROVEMENT FOR REDUCING THE MAXIMUM MATERIAL STRESS

The design improvement proposed here concerns the testing of several shapes of the hook cross-section, in order to find the one that reduces the most the maximum von Mises stress, obtained at the inner surface/fibre of the crane hook. The different cross-section shapes must have the same area, so that the mass of the overall crane hook to be approximately the same. More precisely, the overall mass of the crane hook considered in this study is of approximately 2.8 kg.

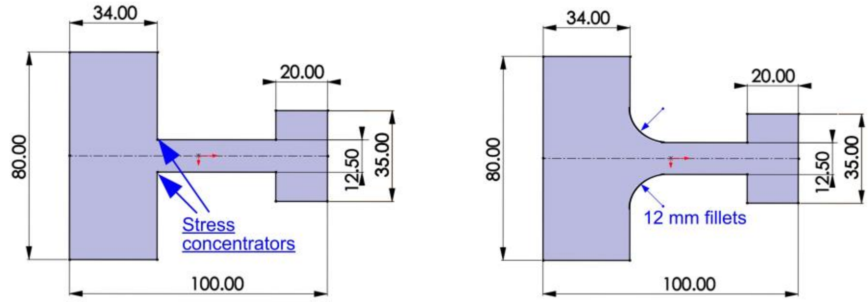
Figure 5 shows the different cross-sections (with similar area) considered in this study: rectangular, trapezoidal, T-shape, T-shape with 12 mm fillet in the stress concentrator area, combination between trapezoidal and T-shape (with 11.25 mm fillet, not really 12 mm due to the dimensions), combination between triangular and T-shape (with 12 mm fillet). Moreover, small rounding fillets with 3 mm radius have been generally applied to the outer edges. The influence of this outer edge small rounding fillets of 3 mm on the stress distribution is a minor one. In fact, only the fillet with 12 mm radius applied in the stress concentrator area (Figs. 5d, 5e and 5f) produces a real stress reduction effect, being applied in the stress concentrator area.



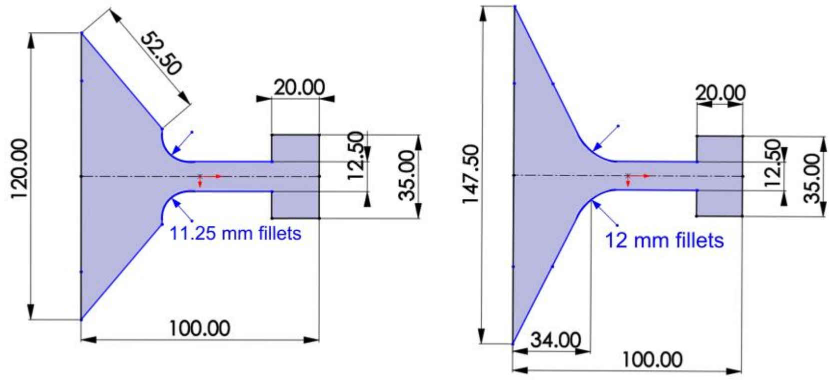
a)



b)



c) d)



e) f)

Fig. 5 – Transversal/cross sections considered in this study: a) rectangular; b) trapezoidal; c) T-shape; d) T-shape with 12 mm fillet in stress concentrator area; e) combination between trapezoidal and T-shape, filleted; f) combination between triangular and T-shape, filleted.

The evolution of the idea of what shape to use in order to reduce the maximum stress is well illustrated in Fig. 5: the rectangular shape is the simplest one, the trapezoidal shape is the one used predominantly in the literature [1, 3, 5–7, 9–14]. Figure 3a shows the von Mises stress distribution obtained by SolidWorks Simulation for the trapezoidal cross-section case, while the displacements of this simulation are presented in Fig. 3b, with a maximum value of 2.81 mm.

The idea of testing the T-shape cross section came from the fact that the T-shape shows a good bending behaviour, this idea/variant of shape being in agreement with Leopardi and Strozzi [16], which recommended "laterally removing material from section zones close to the neutral axis", in order to obtain a "bending stress diminution of the order of a few %". In fact, the T-shape form is a T-shape with a small base (see Fig. 5c), like a H-shape with one side much smaller than the other. This T-shape form of the cross-section showed increased stress in some small stress concentrator area, illustrated in Fig. 6. In order to reduce this stress concentrator effect, a fillet with 12 mm radius was applied in that area, as showed in Fig. 5d. The von Mises stress distribution obtained by SolidWorks Simulation for the T-shape cross-section with 12 mm fillet in stress concentrator area are presented in Fig. 7, the maximum stress being reduced from 477.5 MPa to 416.3 MPa due only to the 12 mm fillet.

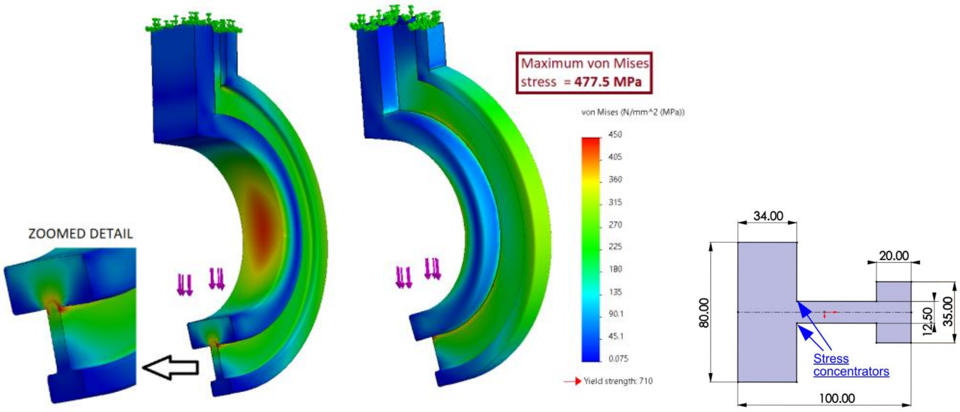


Fig. 6 – Distribution of the von Mises stress for the T-shape cross-section (two different views), illustrating the stress concentrator area (zoomed view). Maximum stress is 477.5 MPa.

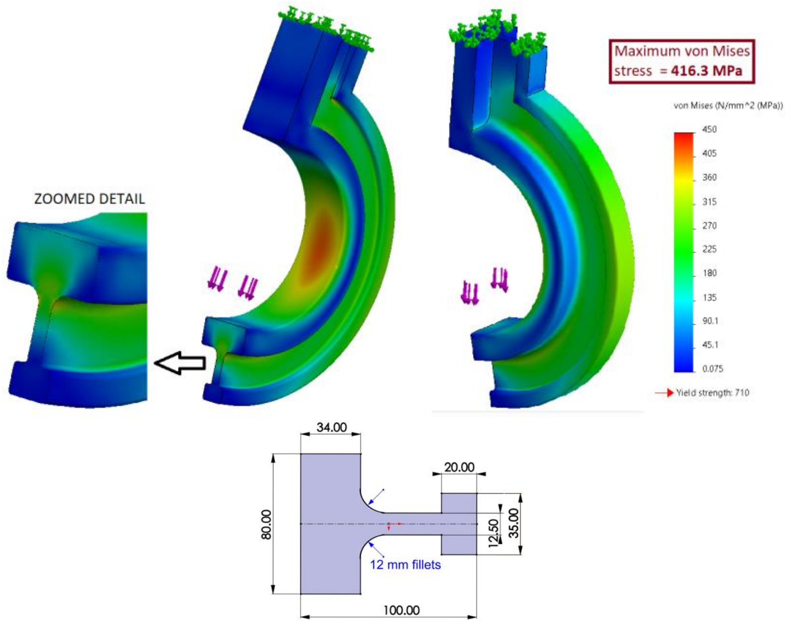


Fig. 7 – Distribution of the von Mises stress for the T-shape cross-section, with 12mm fillet applied in stress concentrator area. Maximum stress is 416.3 MPa.

The next idea was to combine the T-shape and the trapezoidal forms in a combined T-shape + trapezoidal section, keeping the fillet of 11.25 mm (almost 12 mm) radius in the stress concentrator area. As a parallel variant, a

T-shape + triangular section was also tested. Figure 8 shows the von Mises stress distribution results for the case of combined T-shape + trapezoidal section, with the 11.25 mm fillet, while Fig. 9 show the von Mises stress distribution results for the other case of combined T-shape + triangular section, with the 12 mm fillet. Two different angle views are presented, but also zoomed details of the cross-section in the bottom area where the vertical force F_v is applied, showing still some increased stresses in the stress concentrators area.

The evolution of the cross-section shapes tested in this paper, all illustrated in Figs. 5a–5f, has been presented above. Our main goal has been to reduce the maximum stress induced in the crane hook by a vertical force statically applied.

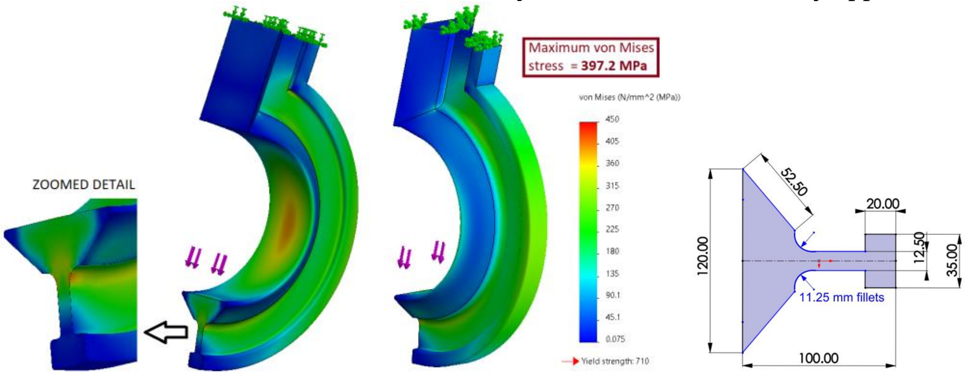


Fig. 8 – Distribution of the von Mises stress for the combined T-shape +trapezoidal cross-section case of this study, with 11.25 mm fillet applied in stress concentrator area. Maximum stress is 397.2 MPa.

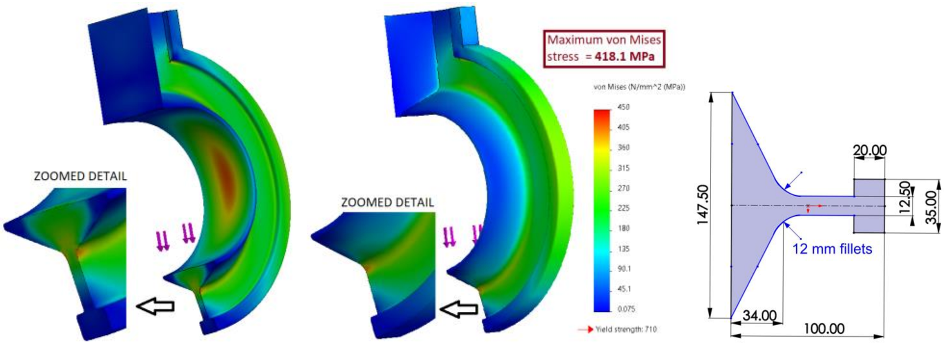


Fig. 9 – Distribution of the von Mises stress for the combined T-shape +triangular cross-section case of this study, with 12mm fillet applied in stress concentrator area. Maximum stress is 418.1 MPa.

Table 1 summarizes the values of the maximum von Mises stress $\sigma_{b_1,\mathrm{MAX}}$ for the different shapes detailed in Figs. 5a–5f, obtained using SolidWorks Simulation testing. The maximum displacements are presented as well, taking place in lower/bottom part of the hook, i.e., where the vertical force F_v is applied.

Table 1

Values of the maximum von Mises stress $\sigma_{b_1,\mathrm{MAX}}$ [MPa], for the different cross-section shapes considered in this study

Shape of the cross-section, with dimensions in mm	Maximum von Mises stress, in MPa	Maximum displacement, in mm
Rectangular section (Fig. 5a), 40x100	509.9	2.67
Trapezoidal section (Fig. 5b), b_1=60, b_1=20, h=100	455.7	2.73
Trapezoidal section (Fig. 5b), b_1=70, b_1=10, h=100	459.6	2.94
Trapezoidal section (Fig. 5b), b_1=65, b_1=15, h=100	**453.7**	**2.74**
T-shape section (dimensions in Fig. 5c)	477.5	2.86
T-shape section, filleted with $R = 12$ mm on the interior side (dimensions in Fig. 5d)	416.3	2.80
combined T-shape + trapezoidal section, filleted (Fig. 5e)	**397.2**	**2.58**
combined T-shape + triangular section, filleted (Fig. 5f)	418.1	2.65

The smallest maximum von Mises stress is obtained for the cross-section shape obtained by combining and mediating a T-shape section with a trapezoidal section. As already mentioned, Fig. 8 is the one showing the von Mises stress distribution obtained by SolidWorks Simulation for this case of combined T-shape + trapezoidal cross-section (from Fig. 5e). The application of the 11.25 mm fillet has really reduced the increased stress in the respective stress concentrator area, but one must further study the technological reliability of fabricating hooks with cross-sections based on T-shapes. The technological realization point of view is crucial, otherwise the improved solution remains only a theoretical/simulation one.

The maximum von Mises stress is 453.7 MPa for the trapezoidal cross-section shape, considering as reference (the mostly used in the literature). For the newly proposed T-shape + trapezoidal cross-section, the smallest maximum von Mises stress is obtained, i.e., 397.2 MPa. The reduction in the maximum von Mises stress is quite significant: $12.5\% = \frac{453.7 - 397.2}{453.7}$.

So, the newly proposed crane hook T-shape + trapezoidal cross-section shows a quite significant improvement for reducing the maximum material stress, in what concerns FEM theoretical simulation. Of course, technological issues must also be considered, this T-shape + trapezoidal section is more complicated than the simpler trapezoidal section, from the point of view hook casting or other technological manufacturing processes.

5. INFLUENCE OF HOOK LONGTUDINAL PROFILE ON THE BENDING STRESS. PARTICULAR CASE OF TRAPEZOIDAL CROSS-SECTION

In order to further reduce the maximum stress in the cross-section located at the middle of the longitudinal profile of the hook, the idea is to reduce the perpendicular distance d between F_v force axis and the vertical axis passing at middle distance between the inner and outer sides of the hook.

This perpendicular distance d has practically no influence of the direct stress component defined by (5). In what concerns the bending stress, it is defined analytically by (6), more precisely by (7) for the analytical bending stress at the inner surface where the maximum total stress occurs:

$$\sigma_{b,1} = -\frac{M}{Ae}\frac{r_1 - r_0}{r_1}. \tag{9}$$

Let us consider here only the particular case of trapezoidal cross-section, for which the analytical solution is presented in section 3. The different variables implied by the inner surface bending stress expression (7) depend on distance d as follows (we indicate only the variables that depend on d, for example cross-section area A does not depend on d, so it is not listed below):

• the bending moment $M = F_v\, d$ depends linearly on d;

• the radius at the inner side of the hook depends linearly on d, as follows: $r_1 = d - \frac{h}{2}$, where h is independent of d, depending only on the cross-section shape. Similarly, the outer radius is $r_2 = d + \frac{h}{2}$.

• the radius to the neutral axis r_0 is given by (2), so the dependence on d is as follows:

$$r_0 = \frac{\left(\frac{b_1 + b_2}{2}\right)h}{\left(\frac{b_1 r_2 - b_2 r_1}{h}\right)\log\left(\frac{r_2}{r_1}\right) - \left(b_1 - b_2\right)} =$$

$$= \frac{\left(\frac{b_1 + b_2}{2}\right)h}{\left[\frac{b_1\left(d + \frac{h}{2}\right) - b_2\left(d - \frac{h}{2}\right)}{h}\right]\log\left[\frac{\left(d + \frac{h}{2}\right)}{\left(d - \frac{h}{2}\right)}\right] - \left(b_1 - b_2\right)}; \tag{10}$$

• the eccentricity $e = r_g - r_0$, with r_g is given

$$r_g = r_1 + \frac{h(b_1 + 2b_2)}{3(b_1 + b_2)} = \left(d - \frac{h}{2}\right) + \frac{h(b_1 + 2b_2)}{3(b_1 + b_2)} \tag{11}$$

and r_0 given by (10).

Based on the expressions above of various variables as a function of distance d (used to compute the bending moment), the analytical bending stress $\sigma_{b,1}$ at the inner surface can be computed as a function of distance d based on (7), then the maximum total stress is computed based on (4). Table 2 presents the different values of the maximum total stress (computed analytically) as a function of perpendicular distance d, obviously with the maximum stress occurring at the middle of the inner side of the hook.

Table 2

Values of the analytical maximum total stress $\sigma_{\text{tot},1}$ for different perpendicular distances d

Distance d [mm]	Maximum total stress σ_{tot} [MPa]
67.5	425
72.5	383
77.5	364
82.5	357
87.5	**354**
92.5	356
97.5	359
107.5	369
117.5	383
127.5	398
137.5	415
147.5	432
157.5	450
167.7	468
177.5	487

Figure 10 illustrates this dependence of the analytical maximum total stress $\sigma_{\text{tot},1}$ computed using (4) as $\sigma_{\text{tot},1} = \sigma_d + \sigma_{b,1}$, occurring at the middle of the inner side of the hook, as a function of distance *d*, which is the distance used to compute the bending moment.

As showed in Table 2 and Fig. 10, where results are obtained by analytical calculation, the smallest maximum total stress $\sigma_{\text{tot}} = 354\,\text{MPa}$ is obtained for perpendicular distance $d = 87.5\,\text{mm} = 80 + 15/2$.

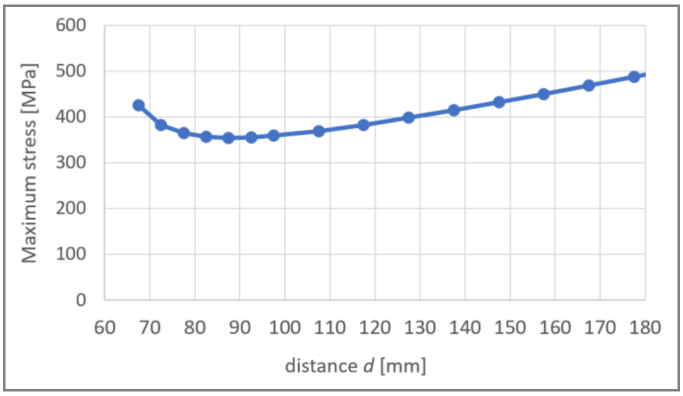
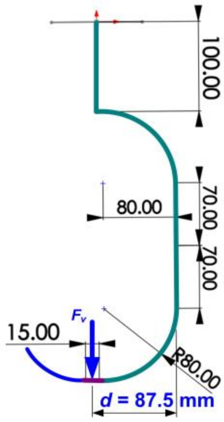


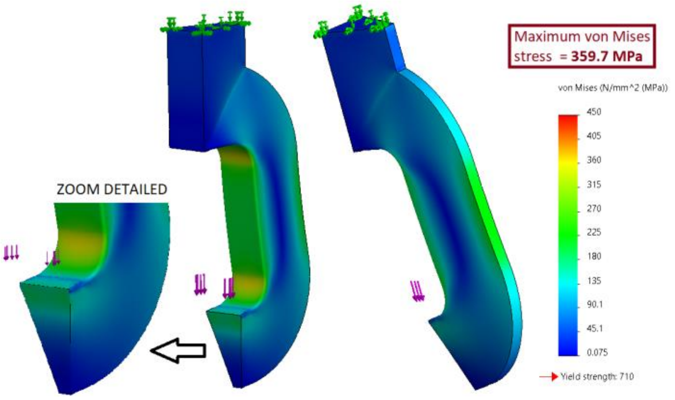
Fig. 10 – Dependence of the analytical maximum total stress σ_{tot}, as a function of distance *d*.

In practice, two longitudinal profiles have been tested by FEM simulation, both with the hook's distance $d = 87.5$ mm. The 1st variant of longitudinal profile considered here, with distance $d = 87.5$ mm, is shown in Fig. 11a. The two variants of longitudinal profile were tested for both trapezoidal cross-section and combined T-shape + trapezoidal cross-section (filleted with 12 mm radius in stress concentrator area).

Figure 11b presents the SolidWorks Simulation results concerning the von Mises stress induced in a crane hook with the longitudinal profile from Fig. 11a and with trapezoidal cross-section. The trapezoidal cross-section is used in this case study from Figs. 11 in order to compare the maximum von Mises stress of 359.7 MPa obtained by SolidWorks Simulation with the corresponding analytical value of 354 MPa. The difference is 1.6% (insignificant), so the correspondence between the theoretical calculation and the SolidWorks simulation is very accurate. A maximum displacement of 1.42 mm is obtained, in the bottom part of the hook.



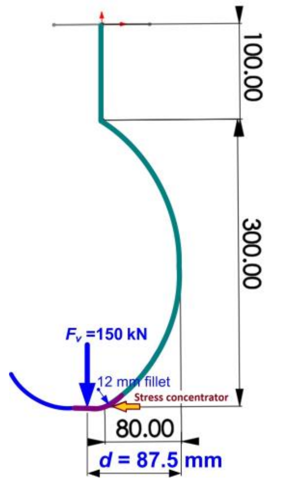
a)



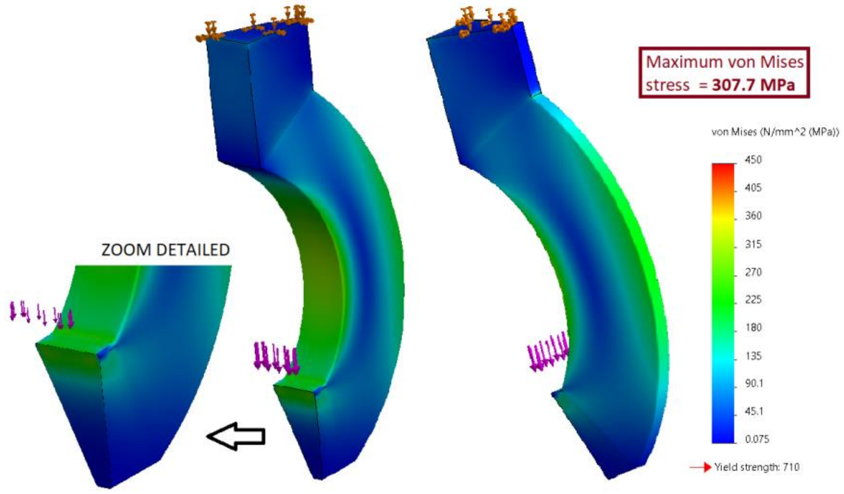
b)

Fig. 11 – a) 1st variant of less wide longitudinal profile of crane hook (distance d = 87.5 mm instead of 157.5 mm); b) distribution of the von Mises stress for this less wide longitudinal profile and for trapezoidal cross-section. Maximum stress is 359.7 MPa.

The 2nd variant of longitudinal profile with distance $d = 87.5$ mm is represented in Fig. 12a, with a stress concentrator occurring in the lower part of the hook. The solution for solving this stress concentrator is the same, i.e., applying a fillet with $R = 12$ mm in the stress concentrator zone, as shown in Fig. 12a. The distribution of the von Mises stress for this 2nd variant of less wide longitudinal profile and for trapezoidal cross-section is shown in Fig. 12b. The maximum stress obtained by SolidWorks Simulation is 307.7 MPa, in this case the difference/error with respect to the analytical solution is about 13%. The maximum displacement obtained in this case is 1.60 mm.



a)



b)

Fig. 12 – a) 2nd variant of less wide longitudinal profile of crane hook (with distance d = 87.5mm); b) distribution of the von Mises stress for this less wide longitudinal profile and for trapezoidal cross-section. Maximum stress is 307.7 MPa.

Table 3

Values of the maximum von Mises stress $\sigma_{b1,MAX}$ [MPa], for the considered longitudinal profiles with distance d = 87.5 mm

Shape of the longitudinal profile and of the (transversal) cross-section, with dimensions in mm	Maximum von Mises stress, in MPa	Maximum displacement, in mm
1st variant of longitudinal profile (Fig. 11a) with distance *d* = 87.5, with trapezoidal cross-section (Fig. 5b), b_1=65, b_1=15, *h*=100	**359.7**	**1.42**
2nd variant of longitudinal profile (Fig. 12a) with distance *d* = 87.5 and filleted in the stress concentrator zone, with trapezoidal cross-section (Fig. 5b), b_1=65, b_1=15, *h*=100	**307.7**	**1.60**
1st variant of longitudinal profile (Fig. 11a) with distance *d* = 87.5, with combined T-shape + trapezoidal section (Fig. 5e), filleted with $R = 12$ mm on the interior side	360.9	1.24
2nd variant of longitudinal profile (Fig. 12a) with distance *d* = 87.5, with combined T-shape + trapezoidal section (Fig. 5e), filleted with $R = 12$ mm on the interior side	325.7	1.23

Table 3 centralizes four simulation results for longitudinal profiles with distance $d = 87.5\,\text{mm}$:

- 1st variant (Fig. 11a) of longitudinal profile with trapezoidal cross-section;
- 2nd variant (Fig. 12a) of longitudinal profile with trapezoidal cross-section;

• respectively 1st variant (Fig. 11a) of longitudinal profile with combined T-shape + trapezoidal section, filleted with $R = 12\,\text{mm}$ on the interior side;

• finally 2nd variant (Fig. 12a) of longitudinal profile with combined T-shape + trapezoidal section, filleted with $R = 12\,\text{mm}$ on the interior side.

The FEM simulations for the two variants of less wide longitudinal profiles lead to the following conclusions: the trapezoidal cross-section seems here more reliable than the combined T-shape + trapezoidal cross section (filleted). For the 1st variant of longitudinal profile (Fig. 11a) with distance $d = 87.5\,\text{mm}$, the maximum stress is obtained 359.7 MPa (for trapezoidal cross-section) compared to 360.9 MPa (for T-shape + trapezoidal), so it is almost the same. But the trapezoidal cross-section is simpler and more reliable, without the problematic issue of undesired stress concentrators.

In what concerns the comparison between the 1st and the 2nd variants of longitudinal profile, the 2nd variant corresponds to a smaller maximum stress, i.e., 307.7 MPa compared to 359.7 MPa. But this 2nd variant is also associated with some stress concentrators in the lower part of the hook, so for the same reasons of simplicity and reliability the 1st variant of longitudinal profile (from Fig. 11a) might be preferred, even if the simulations performed show better results for the 2nd variant of longitudinal profile (from Fig. 12a) with trapezoidal cross-section.

So, this study leads to the conclusion that, from our point of view of FEM simulation results, the 2nd variant of longitudinal profile (Fig. 12a) with distance $d = 87.5\,\text{mm}$ and with trapezoidal cross-section (Fig. 5b), $b_1 = 65\,\text{mm}$, $b_1 = 15\,\text{mm}$, $h = 100\,\text{mm}$) is the recommended solution for decreasing the maximum stress induced in the crane hook. Compared with the longitudinal profile under the form of a partial circle with $R = 150\,\text{mm}$ (in this case, $d = 157.5\,\text{mm}$), the maximum von Mises stress is reduced from 453.7 MPa to 307.7 MPa. The reduction is important, approximately 32%. In what concerns the 1st variant of longitudinal profile (from Fig. 11a), the maximum von Mises stress is 359.7 MPa, so the reduction is 21% (compared with 453.7 MPa).

6. CONCLUSIONS

The approach in this paper tried to improve both the (transversal) cross-section and the longitudinal crane hook profile, so that to reduce the maximum stress induced in the material. In what concerns the transversal section, besides the trapezoidal cross-section widely used in the literature, a T-shape cross-section and, moreover, a combined T-shape + trapezoidal/triangular cross section have been tested. The T-shape + trapezoidal cross-section solution really reduces the maximum von Mises stress (12.5% reduction), but this solution is more complicated, showing the disadvantage of some stress concentrators (where it was necessary to apply the solution of rounding fillets with 12 mm radius), compared with the simple and reliable trapezoidal cross-section.

Another contribution of this paper concerns the longitudinal profile of crane hooks. An analytical calculation, confirmed by SolidWorks Simulation results, has shown that a less wide longitudinal profile, with the perpendicular distance *d* of 87.5 mm instead of 157.55 mm, corresponds to a reduction of the maximum von Mises stress induced of approximately 21% (for the 1st variant of longitudinal profile from Fig. 11a, or even 32% (for the 2nd variant of longitudinal profile from Fig. 12a).

To summarize, the two main contributions of this paper are: 1) proposing a combined T-shape + trapezoidal or triangular cross-section for crane hooks; 2) improvements of the hook longitudinal profile (FEM simulation and analytical contribution) so that to reduce the maximum induced stress in the material.

In conclusion, shape design improvement is possible both concerning the transversal section and the longitudinal profile of crane hooks. A significant reduction of the maximum stress induced in the material is possible: in this paper we have obtained such a reduction of the order of 10–20% (in average), for the case of an external force applied statically on a crane hook. Shape design improvement and optimization is thus possible for hooks and for other mechanical parts, always in close connection with the specificity of the concerned mechanical problem.

Technological issues must also be considered. Usually hook cranes are manufactured by forging, followed by normalizing heat treatment [5]. Further mechanical processing such as milling or welding are not recommended, because they increase stress concentration and new stress concentrators might occur. So, the crane hook forged as one complete object is recommended [7]. From this forging manufacturing point of view, the proposed T-shape + trapezoidal section is rather more complicated than the simpler trapezoidal transversal section. For the longitudinal study too, the proposed solution is also more complicated from the forging point of view, because fillets are necessary so that to reduce induced stress concentrators. But modern forging is capable of creating complex shapes, so only a further realization and testing of the proposed hook shapes can establish all technological issues.

Acknowledgements. The anonymous reviewers are acknowledged for improving the paper by their relevant requests. Our colleague Dr. Florin Isvoranu is acknowledged for the explanatory discussions concerning the technological processes involved in crane hooks manufacturing.

Received on February 15, 2022

REFERENCES

1. BHATKAR, O.P., JAVIR, A.V., ASHOK, S.A., SUMANT, S.S., EKNATH, U.S. SHIVRAM, K.K., *Finite Element Analysis of crane hook and optimization using Taguchi approach*, Journal of Advance Research in Mechanical & Civil Engineering, **2**, *3*, pp. 63–67, 2015.

2. DIVAKAR, G., RAMAKRISHNA, C., RAMANAMURTYNAIDU, S., TARANIKANTH, K., RAO, Y., *Design and optimization of crane hook*, Journal of Critical Reviews, **7**, *15*, pp. 2721–2727, 2020.
3. ZADE, M.R., *Finite Element Analysis and fatigue analysis of crane hook with different materials*, International Research Journal of Engineering and Technology, IRJET, **4**, pp. 1429–1436, 2017.
4. SINGH, P., JAIN, O., YADAV, P., NIKHIL, SHARMA, S.K., *Analysing crane hook of different cross sections and different materials*, In: Recent Advances in Mechanical Engineering – Select Proceedings of ITME 2019, eds. M. Muzammil, A. Chandra, P.K. Kankar, H. Kumar, , Springer, Singapore, pp. 575–5802021.
5. UDDANWADIKER, R., *Stress analysis of crane hook and validation by photo-elasticity*, Engineering, **3**, pp. 935–994, 2011.
6. TRIFKOVIĆ, S., RADIĆ, N., ŠEHOVAC, S., KOŠARAC, A., MILUTINOVIĆ, M., *Stress analysis of crane hook using FEM*, INFOTEH-JAHORINA, **10**, *C-2*, pp. 244–248, 2011.
7. KISHORE, K., GUJRE, V.S., CHOUDHARY, S., GUJRE, A.S., VISHWAKARMA, M., THIRUMURGAN, T., CHOUDHURY, M., ADHIKARY, M., KUMAR, A., *Failure analysis of a 24 T crane hook using multi-disciplinary approach*, Engineering Failure Analysis, **115**, p.104666, 2020.
8. BHAGYARAJ, G., SURYAPRAKASH, K., RAO, K.S., *Crane hook design and analysis*, International Research Journal of Engineering and Technology, IRJET, **4**, *9*, pp. 61–65, 2017.
9. LAKSHMANA MOORTHY, S.K., PRAKASH, B., *Design and analysis of crane hooks of different cross sections made of hardened-tempered alloy steel AISI 6150 and AISI 4140*, International Journal of Engineering Research & Technology, IJERT, **9**, *5*, pp. 435–439, 2020.
10. BERGALEY, A., PUROHIT, A., *Structural analysis of crane hook using finite element method*, International Journal of Science and Modern Engineering, IJISME, **1**, *10*, pp. 3–7, 2013.
11. GOPICHAND, A., LAKSHMI, R.V.S., MAHESHKRISHNA, B., *Optimization of design parameters for crane hook using Taguchi method*, International Journal of Innovative Research in Science, Engineering and Technology, IJIRSET, **2**, *12*, pp. 7780–7784, 2013.
12. SINGH, A., ROHILLA, V., *Optimization and fatigue analysis of a crane hook using finite element method*, International Journal of Recent advances in Mechanical Engineering (IJMECH), **4**, *4*, pp. 31–43, 2015.
13. SAHU, S., DEWANGAN, R., PATNAIK, M., YADAV, N., *Study of crane hook having trapezoidal section by finite element method & design of experiments*, International Journal of Modern Engineering Research, IJMER, **2**, *4*, pp. 2779–2781, 2012.
14. MEHENDALE, S.A., WANKHADE, S.R., *Design and analysis of EOT crane hook for various cross sections*, Int. J. of Current Engineering and Scientific Research (IJCESR), **3**, *12*, pp. 53–58, 2016.
15. QIN, Y.X., XIE, W.T., REN, H.P. LI, X., *Crane hook stress analysis upon boundary interpolated reproducing kernel particle method*, Engineering Analysis with Boundary Elements, **63**, pp. 74–81, 2016.
16. LEOPARDI, L., STROZZI, A., *Two extensions towards practical applications of a paradox in curved beams*, Procedia CIRP, 29th CIRP Design 2019, **84**, pp. 761–768, 2019.