The application of the Finite Element Method analysis in the process of designing the punching die for belt perforation

D Wojtkowiak, K Talaśka and A Fierek
Chair of Basics of Machine Design, Poznan University of Technology, Piotrowo 3 street, 61-138 Poznań, Poland
E-mail: dominik.wojtkowiak@put.poznan.pl

Abstract. The Finite Element Method (FEM) analysis has improved the process of designing modern machines, especially when the effective construction solution has to be found. In the paper authors have presented the application of this type of analysis based on the example of the construction of the punching die for belt perforation. For these machines the cooperation of the piercing punch and the die is crucial. Because of this, the analysis of the flexural rigidity of the head block and the punch plate provides useful information, which can be used to maintain proper alignment between the punch and the die during the perforation process. It may have a positive effect on the tool wear and the punching performance. Additionally, by using the topology optimization the mass of the punching die can be greatly reduced while maintaining enough rigidity. This effect is desired in automated machines for belt perforation, in which the punching die has to move with high velocities in order to create a designed pattern on the perforated belt with satisfying efficiency. In the presented research authors focused on determining the influence of the constructional parameters of the punching die on the deflection of the head-punch block. The obtained results and the presented methodology of performing the FEM analysis of such machines will help to improve their design process and find the effective construction solution.

1. Introduction
Metal punching process is widely discussed in the scientific literature [1–3] and it seems that for softer materials like polymer composite belts, punching dies can be designed based on the same constructional briefs. However, due to the extremely different mechanical properties of the processed materials, the construction of the punching die should be designed with various approaches and different aspects may have an influence on the perforation performance. In the literature, multiple examples of the influence of the mechanical properties of the processed material on the constructional features of the machines can be found [4–15], especially for natural polymers [4–6] and unclassical materials [7–15]. This suggests that testing the mechanical properties of the processed materials will provide useful information for the machine design process [12–21].

The main difference between the metal punching and polymer composite belt perforation is the recommended value of the punch-die clearance, which is 10 % for metals and 0.5–1 % for polymer composite belts [1, 13]. Maintaining that small value of clearance requires to take into consideration multiple technological and constructional restrictions. One of these is the minimization of the head-punch block deflection in which the punch is embedded directly or by using the punch chunk. In this paper the authors present the application of the Finite Element Method (FEM), which has a wide application in scientific research [12–14, 19, 22–26], to analyse the influence of the geometry of the
head-punch block on the strength and stiffness of the construction. Additionally, the methodology of using the obtained data to decrease the mass of the punching die is discussed.

The research was conducted on the basis of the punching die construction presented in figure 1. The punching die consists of two plates: the head-punch block and the base-die one. The relative movement of these two plates is provided by sets of guiding columns with guiding sleeves and ball bushings. Since the guides support the plate on one side only, the head-punch block is subjected to bending and its deflection is relatively big. The return motion of the plate is generated by a set of return springs, which are put between the plates. The force of the blank holder, which is used to hold the material during perforation, is generated by two springs on both sides of the piercing punch. The punch is mounted in the punch chunk, which is embedded in the socket in the head-punch block and it is held with a circular plate bolted to the head-punch block. In this research the main focus was put on the analysis of the construction of the head-punch block with FEM.

Figure 1. Construction of the punching die for polymer composite belts.

2. Methodology of the research

The analysis of the strength and rigidity of the head-punch blocks with various thickness was performed based on the computer simulation results obtained from ABAQUS. The presented methodology is based on the application of the Finite Element Method (FEM). The construction of the FEM model is presented in figure 2.

For the conducted research only static analyses were performed and the most adverse loading was assumed, in order to gain the peak values of the plate deflection and its stress. To simplify the model and decrease the computational time necessary to perform the simulation, the head-punch block was separated from the construction as a single instance. To map the real conditions to which the plate is subjected, the boundary condition BC1, which takes away all degrees of freedom, was applied to the inner cylindrical surfaces of the mounting holes of all three guiding sleeves. The loading consists of four forces which were applied as a pressure on the partitioned surfaces of the plate and their magnitudes were defined as a total force in Newtons. To establish its values, the results from previous research [13–14] and the constructional features of the prototype of the punching die were used.

The perforation force \( F_p \) is the force necessary to pierce the belt and create the hole. Its value depends on the mechanical properties of the belt [14, 19] and the geometry of the tool [12–14]. The properties of the elastomer materials and plastics of which the belts are made may vary significantly [14, 19]. This indicates that in order to analyse the most adverse loading, the value for the strongest and most rigid of the commonly perforated type of belts should be used. In the presented research, the peak perforation force for the TFL15S belt with polyamide core and the spherical bowl punch [14], which equals 2810 N was used. It was additionally increased 20 % to cover the excess caused by tool condition or process
parameters, and the final value 3372 N of perforation force was used. This load is transferred to the plate from the punch and its chunk, whose collar is embedded in the socket on the top surface of the head-punch block and blocked by the circular plate screwed with the head-punch block with two bolts washers and nuts. Since the chunk presses the circular plate and extends the screws, the force in the model was applied to the surface partition on the bottom surface of the instance, which covers the size of two washers M8 from the screw connections tightening the joint.

![Figure 2](image-url)

**Figure 2.** Construction of FEM model of the head-punch block of the punching die in ABAQUS: $F_p$ – perforation force, $F_{RS}$ – return springs force, $F_{BH}$ – blank holder force, $F_{pp}$ – pressure plate force.

The next two forces loading the plate are forces from blank holder springs $F_{BH}$ and from return springs $F_{RS}$. Both of these forces are applied to the bottom surface of the sockets on the bottom of the plate on which the springs are supported. To determine their values, it was assumed that the peak force will occur when the punch penetrates the whole belt. This indicates that return springs were compressed with displacement $c_0 = 5$ mm (thickness of the belt $t = 3.1$ mm and clearance between the punch and belt surface 1.9 mm displacement) from the initial deformation $x_0 = 5$ mm caused by the gravity load of the plate. Since springs are connected in parallel, the equivalent spring stiffness $k$ is the sum of each spring stiffness $k_i$. Based on the above-mentioned correlations, the stiffness of the return springs $k_{RS}$ can be calculated for various mass of the head-punch block from Eq. (1), where $g$ is the gravity acceleration $9.81$ m $s^{-2}$. Applying it to the Eq. (2), the magnitude of the force generated by the return springs can be determined. The force of the blank holder (Eq. (3)) depends of the thickness of the belt and the stiffness of its spring system, which is constant for all analysed cases and equals 143 N.

$$k_{RS}x_0 = mg ightarrow k_{RS} = \frac{mg}{x_0}$$  \hspace{1cm} (1)

$$F_{RS} = k_{RS}c_0 = \frac{mg}{x_0}c_0$$  \hspace{1cm} (2)

$$F_{BH} = k_{BH}t = 46.1 \frac{N}{mm} \cdot 3.1 \ mm \approx 143 \ N$$  \hspace{1cm} (3)

The last force is connected with the pneumatic drive system of the punching die and is represented as a pressure generated by a pressure plate fixed with pneumatic cylinders pistons on the top surface of
the head-punch block. Since the force will increase until the material is pierced, its peak value will be the sum of all the above-mentioned forces:

\[ F_{PP} = F_P + F_{RS} + F_{BH} \]  

(4)

Due to the complex geometry of the plate, the tetrahedral elements C3D10 were used to discretize the instance into the finite elements. The global size of each element was set as \( \frac{1}{4} \) of the plate thickness in order to maintain 4 elements on the thickness of the plate.

Tests were conducted for a thickness in a range of 15–40 mm, with an increment of 5 mm. It was assumed that the depth of the socket for a circular plate equals half of the plate thickness and the minimal thickness of the springs support sockets bottom is 5 mm.

3. Results of the FEM analyses of the head-punch block with various thickness

For each case two parameters were analysed: stress distribution in the head-punch block and the deflection of the plate in the direction of the punch axis (A). The examples of the simulation results for the block with thickness \( t_p = 30 \) mm are presented in figure 3 (stress distribution) and figure 4 (displacement).

Figure 3. Mises stress distribution \( \sigma_{mises} \) (Pa) on the head-punch block with thickness \( t_p = 30 \) mm.

Figure 4. Displacement \( f \) (m) in the direction of the punch axis (A) of the head-punch block with thickness \( t_p = 30 \) mm.

As can be observed, due to the symmetry of plate geometry and its loading, stress and displacement are evenly distributed on both sides of the plane consistent with the punch axis. Stress concentration occurs nearby the socket for the punch chunk and the threaded connections between the head-punch block and the circular plate. In the remaining part of the plate the stress level is much lower, however its distribution is non-uniform. Slight concentration is visible nearby the guiding column mounting holes and the socket for the circular plate, but the center of the plate seems to be idle. This indicates that the
geometry of the head-punch block may be modified and part of the material can be cut out without aggravation of the exploitation parameters of the punching die. For that cause the topology optimization was also performed and its results are presented in the following part of this paper.

Although information about the stress level in the analysed component of the machine and its deformation is useful in the design process, it does not provide the most crucial details about the cooperation between the punch and the die. As was proved in the authors’ previous research [13], the punch-die clearance $c$ for the belt perforation should be characterized by smaller values than in metal punching. Since the punch is fixed in the chunk, which is embedded rigidly in the head-punch block’s socket, the deflection of the plate will also impact the deflection of the punch. If the deformation exceeds $0.5c$, the contact between the cutting edge of the punch and the die will occur, which leads to tool wear or even its damage. It is worth mentioning that due to the few joints in the kinematic chain of the punching die, the deflection of the punch is just one of the components which will cause punch deflection. What will have much more influence on the punch is the guiding column deformation. For that reason we can assume that tool deflection caused individually by the head-punch deflection should not exceed $0.1c$.

To evaluate the punch deflection, it is necessary to find its correlation with the plate deflection $f_A$. To facilitate understanding of its derivation, the schematic is presented in figure 5. This schematic is valid only for small angles of deflection $\alpha < 5^\circ$ for which the Eq. (5) is valid. Based on this assumption, the correlation between both deflections can be presented as in Eq. (6):

$$\tan \alpha = \frac{f_A}{L} = \sin \alpha = \frac{f_{PN}}{H} \tag{5}$$

$$f_{PN} = \frac{H}{L} \cdot f_A \tag{6}$$

**Figure 5.** Schematic of correlation between the punch deflection $f_{PN}$ and the plate deflection $f_A$.

Based on the previous discussion and the FEM analyses performed for 6 different plate thicknesses the results were obtained and they are presented in table 1. It is obvious that by increasing the plate thickness its mass rises with approximately linear increments. It is the most advantageous for a punching die to use a plate as light as possible, in order to decrease the inertia of the dynamically moving part of the machine. Additionally, it would greatly reduce the power consumption of the drive system responsible for the linear movement of the punching die relatively to the belt. For that reason we should be looking for the thinnest plate whose stress and deflection do not exceed the limit values.
Table 1. Results of the FEM analyses of the head-punch block stress and deflection.

| Plate thickness $t_P$ (mm) | Mass of the plate $m_P$ (kg) | Max Mises stress $\sigma_{max}$ (MPa) | Plate deflection in the punch axis $f_A$ (mm) | Punch deflection $f_{PN}$ (mm) |
|---------------------------|-----------------------------|-------------------------------------|-----------------------------------------------|-------------------------------|
| 15                        | 13.259                      | 119.3                               | 0.226                                         | 0.0391                        |
| 20                        | 17.628                      | 65.9                                | 0.107                                         | 0.0185                        |
| 25                        | 22.016                      | 42.2                                | 0.06                                          | 0.0104                        |
| 30                        | 26.432                      | 32.7                                | 0.038                                         | 0.0065                        |
| 35                        | 30.906                      | 25.9                                | 0.026                                         | 0.0045                        |
| 40                        | 35.381                      | 21.5                                | 0.019                                         | 0.0033                        |

The correlation between the plate thickness and the peak stress is presented in figure 6a, while the influence of this parameter on the punch and plate deflection is shown in figure 6b. As can be observed, all correlations are negative and nonlinear ones. This indicates that at some point there is no necessity for further increase of the plate thickness because the influence tends to be negligible. If we analyse the peak stress which occurs in the machine part, it should be compared with the allowable value. In that case the plate is often made for low carbon steel S275 which has Yield point at $\sigma_{pl} = 275 \text{ MPa}$. If we assume static load and use the safety factor $x_e = 2$, the stress should not exceed $137.5 \text{ MPa}$. So technically, even thickness 15 should be enough using only strength criteria. However, the punching die works with multiple cycles per minute, which indicates that dynamic allowable stress should be considered instead. Since the loading in unilateral, the allowable value should be restricted to 60 % of the static value, which means it equals $82.5 \text{ MPa}$. For that reason the thickness 15 mm is too small and the plate should have at least 20 mm.

![Figure 6a](image1.png)

**Figure 6a.** The correlation between the thickness of the head-punch block $t_P$ and the peak Mises stress $\sigma_{max}$.

![Figure 6b](image2.png)

**Figure 6b.** The correlation between the thickness of the head-punch block $t_P$ and the deflection of the punch $f_{PN}$ and plate $f_A$.

In machine design very often the strength criteria is fulfilled, but the rigidity criteria is not. If we analyse the punch deflection, it should not exceed 0.1 cm as was previously stated. For belt perforation the clearance should be as small as possible. It was established in previous research [13] that for holes up to 10 mm diameter the clearance $c = 0.1 \text{ mm}$ meets both hole quality and technological requirements for part machining, however for some belts (e.g. polyurethane toothed belts reinforced with aramide fibers) lower values are desired. If we use the limit value of 0.01 mm the minimal thickness of the plate rises up to 30 mm. This value was used in the further analysis and the prototype of the automated belt punching die.
4. Topology optimization of the head-punch block

Topology optimization is a design tool which uses FEM to distribute a prescribed amount of the material over a given design domain in order to minimize a scalar objective function for a determined set of constraints. As was stated previously, the non-uniform stress distribution observed in the computer simulation results suggests that part of the material is idle and may be cut out to decrease the mass of the component. For that reason the optimization process in ABAQUS was performed to determine how the geometry should be modified. The construction of the FEM model is almost identical as the one presented in figure 2 and described in section 2. Only a few more parameters like design responses, objective functions, optimization constraints, geometrical restrictions and a maximum number of cycles have to be defined.

Design responses are the inputs to the optimization and they can be read from the output database file. They are associated with the region of the model, but consist of only a single scalar value, such as maximum stress in the region or the total volume of the instance. It may be also connected with a particular load case or step of the analysis [27–29]. In the presented research two design responses were specified as a single term associated with the whole model, which are strain energy and volume.

Objective functions define the goal of the optimization and are extracted from the design response, such as minimum stress or deflection. They can be formulated from multiple design responses. In that case the objective function is calculated by adding the values which need to be minimized and maximized, taking into consideration the weighting factors to highlight the most important correlations [27–29]. In the topology optimization of the head-punch block the minimization of the strain energy was chosen.

Optimization constraints restrict the value of the design response as either percentage value (50 % of the initial volume) or specified value (deflection under 0.5 mm). Additionally, the manufacturing constraints or geometric ones can be applied. This induces that for a complex topology optimization we can restrict that the machine part will be casted, which will require to maintain certain proportions of the dimensions or we can specify that certain dimensions cannot be modified, like the diameter of a bearing surface. Fulfilling these constraints has a priority over achieving the minimization or maximization of the objective function [27–29]. For the presented analysis the volume constraint was used and it was specified as a fraction value of the initial value less or equal 0.3. Additionally, the geometric restrictions were set as a frozen area on all surfaces to which the BC or load are applied and for the hole used to mount the screw, which tightens the return springs.

Figure 7. Results of the topology optimization – Mises stress distribution $\sigma_{\text{mises}}$ (Pa).

The model described above was implemented to ABAQUS and the results are presented in figure 7. Although the maximum cycles were set to 100, only 33 cycles were necessary to reach the goal of the
objective function. As can be observed, some of the material necessary to maintain the integrity of the part was also erased. This is why the presented results should be treated as guidelines, which can be used in the design process of the head-punch block with reduced mass by combining them with conventional machine design and technological guidelines. The results of the design process based on the topology optimization is presented in figure 8. The mass of the plate was reduced to 12.86 kg, which is the 48.7 % of the initial mass.

**Figure 8.** Drawing of the head-punch block with reduced mass.

After finishing the design process the final effect was once more tested to verify the correctness of the proposed construction. The results are presented in figures 9 and 10. The maximal Mises stress has increased to 36.4 MPa (11.3 % growth) and the plate deflection rose up to 0.062 mm (63.2 % growth), which gives 0.011 mm punch deflection. If we compare obtained results with the data from table 1, we have achieved a solution of mass lower than the thinnest plate analysed in the research \((t_p = 15 \text{ mm})\), but the deflection and the stress is similar to the plate with only 5 mm less thickness than the base one. Although punch deflection slightly exceeds the limit value 0.01 mm, the obtained deviation could be treated as acceptable one. Of course further analyses concerned with the distribution of the guiding columns and their deflection should be performed in order to gain the full set of data needed for a punching die design process.

If we analyse the stress map presented in figure 9b we can observe uniform distribution which proves that the usage of the material was greatly improved by using the topology optimization. In figure 9c we can see that the relatively thin wall structure in the neighbourhood of the guiding column is subjected to low stress, which decreases the chance of the material failure in that region.
5. Conclusions
The presented research shows that the application of the Finite Element Method in the design process of the punching die may be very helpful and leads to more effective construction solutions. The key to proper analysis is to create a FEM model which will precisely imitate the real conditions and at the same time be as simple as possible to reduce the computational time of the calculations. The topology optimization is a useful tool in the design process, but only if we combine it with the engineering knowledge of classic machine design and take into consideration the machining process. Otherwise we will obtain a solution with reduced weight, but the costs will grow drastically. The presented research also proved that increasing the dimensions of the machine part too much would not grant us proportional benefits and it is important to find the middle-ground solution. It also shows that designing the punching dies for belts, which are softer materials than metals and seem to be easier to perforate, causes much more trouble and is very interesting from the scientific point of view.

Figure 9. Results of the FEM analyses of the head-punch block with reduced mass – Mises stress distribution $\sigma_{\text{mises}}$ (Pa).

Figure 10. Displacement $f$ (m) in the direction of the punch axis (A) of the head-punch block with reduced mass.
6. References

[1] Soares J A, Gipela M L, Lajarin S and Marcondes P V P 2012 Study of the punch–die clearance influence on the sheared edge quality of thick sheets Int. J. Adv. Manuf. Technol. 65 451–457

[2] Singh U P, Streppel A H and Kals H J J 1992 Design study of a punching/blanking tool J. Mat. Process. Technol. 33 331–345

[3] Wojtkowiak D, Talaśka K, Malujda I and Domek G 2017 Vacuum conveyor belts perforation–methods, materials and problems Mechanik 12 1138–1142

[4] Berdychowski M, Talaśka K and Wilczyński D 2019 Evaluation of the possibility of using the Drucker-Prager-Cap model in simulations of the densification process of shredded natural materials MATEC Web of Conferences 254 02018

[5] Wilczyński D, Malujda I, Górecki J and Domek G 2019 Experimental research on the process of cutting transport belts MATEC Web of Conferences 254 05014

[6] Wilczyński D, Talaśka K, Malujda I and Jankowiak P 2018 Experimental research on biomass cutting process MATEC Web of Conferences 157 07016

[7] Górecki J, Malujda I and Wilczyński D 2019 The influence of geometrical parameters of the forming channel on the boundary value of the axial force in the agglomeration process of dry ice MATEC Web of Conferences 254 05001

[8] Górecki J, Malujda I, Wilczyński D and Wojtkowiak D 2019 Influence of the face surface shape of the piston on the limit value of compaction stress in the process of dry ice agglomeration MATEC Web of Conferences 254 06001

[9] Górecki J, Malujda I, Talaśka K, Wilczyński D and Wojtkowiak D 2018 Influence of geometrical parameters of divergent sleeve on the value of limit stress MATEC Web of Conferences 157 05006

[10] Walęsa K, Malujda I and Talaśka K 2018 Butt welding of round drive belts Acta Mech. et Auto. 12 115–126

[11] Walęsa K, Malujda I, Górecki J and Wilczyński D 2019 The temperature distribution during heating in hot plate welding process MATEC Web of Conferences 254 02033

[12] Wojtkowiak D and Talaśka K 2019 The influence of the piercing punch profile on the stress distribution on its cutting edge MATEC Web of Conferences 254 02001

[13] Wojtkowiak D and Talaśka K 2019 Determination of the effective geometrical features of the piercing punch for composite belts Int. J. Adv. Manuf. Technol. 104 315–332

[14] Wojtkowiak D, Talaśka K, Malujda I and Domek G 2018 Estimation of the perforation force for composite conveyor belts taking into consideration the shape of the piercing punch Int. J. Adv. Manuf. Technol. 98 2539–2561

[15] Wojtkowiak D, Talaśka K, Malujda I and Domek G 2018 Analysis of the influence of the cutting edge geometry on parameters of the perforation process for conveyor and transmission belts MATEC Web of Conferences 157 01022

[16] Kukla M, Talaśka K and Malujda I 2019 Damping in magnetorheological elastomers under compressive stress MATEC Web of Conferences 254 06002

[17] Kukla M, Górecki J, Maluda I, Talaśka K and Tarkowski P 2017 The determination of mechanical properties of magnetorheological elastomers (MREs) Procedia Engineering 177 324–330

[18] Ondrusova D, Bozekova S, Bunova L, Pajtusova M, Labaj I, Dubec A and Vrskova J 2018 Modification of alternative additives and their effect on the rubber properties MATEC Web of Conferences 157 07007

[19] Talaśka K and Wojtkowiak D 2018 Modelling mechanical properties of the multilayer composite materials with the polyamide core MATEC Web of Conferences 157 02052

[20] Walęsa K, Mysiukiewicz O, Pietrzak M, Górecki J and Wilczyński D 2019 Preliminary research of the thermomechanical properties of the round drive belts MATEC Web of Conferences 254 06007

[21] Wilczyński D, Malujda I, Talaśka K and Dlugi R 2017 The study of mechanical properties of natural polymers in the compacting process Procedia Engineering 177 411–418
[22] Domański T, Piekarska W, Kubiak M, Saternus Z and Goszczyńska-Króliszewska D 2018 Numerical simulations of progressive hardening by using ABAQUS FEA software MATEC Web of Conferences 157 02007
[22] Gawronska E 2019 A Sequential approach to numerical simulations of solidification with domain and time decomposition Applied Sciences-Basel 9 1972
[23] Dyja R, Gawronska E and Grossner A 2017 Numerical problems related to solving the navier-stokes equations in connection with the heat transfer with the use of FEM Procedia Engineering 177 78–85
[24] Murcinkova Z, Novak P, Kompis V and Zmindak M 2018 Homogenization of the finite-length fibre composite materials by boundary meshless type method Archive of Applied Mechanics 88 789–804
[25] Sapietova A, Novak P, Saga M, Sulka P and Sapieta M 2019 Dynamic and stress analysis of a locking mechanism in the ANSYS workbench software environment Advances in Science and Technology-Research Journal 13 23–28
[26] Saternus Z 2017 Computer methods for determination of deformations in welded closed profiles Procedia Engineering 177 188–195
[27] Zhao Y, Guo W, Duan S and Xing L 2017 A novel substructure-based topology optimization method for the design of wing structure Int. J. Simul. Multisci. Des. Optim. 8 A5
[28] Zhu J-H, Zhang W-H, Xia L 2016 Topology optimization in aircraft and aerospace structures design Arch. Computat. Methods Eng. 23 595–622
[29] Abaqus User’s Manual, Version 6.14, Dassault Systèmes Simulia Corp. Providence Rhode Island, USA 2013