ISSN 0208-7774 TRIBOLOGIA 6/2019 p. 5–16
DOI: 10.5604/01.3001.0013.7759

Henryk CZARNECKI*, **Krzysztof TUBIELEWICZ****, **Andrzej ZABORSKI*****,
Michał TAGOWSKI****, **Hubert MICHALCZUK*******

FEM SIMULATION OF MATERIAL STRAIN IN CORNER OF FORMING TAP DURING COLD THREAD SHAPING

SYMULACJA NUMERYCZNA WYTĘŻENIA MATERIAŁU NAROŻA GWINTOWNIKA WYGNIATAJĄCEGO W PROCESIE KSZTAŁTOWANIA GWINTU NA ZIMNO

Key words: FEM Simulation, forming tap, threads, cold forming.

Abstract The paper presents a computer simulation of the influence of the corner construction of a forming tap on the tool material strain. The obtained visual representation is presented of the course of thread groove formation and the distribution of stresses and deformations in the workpiece and tool. The obtained operational stress distributions allow the observation of places with maximum loads and thus exposed to accelerated wear. This helps to select the most advantageous outline for the tool adapted to the specific application in the technological process without the need for long-term operational tests.

Słowa kluczowe: symulacja numeryczna, gwintownik wygniatający, gwint, formowanie na zimno.

Streszczenie W opracowaniu przedstawiono symulację komputerową wpływu konstrukcji naroża gwintownika wygniatającego na wytężenie materiału narzędzia. Otrzymano wizualne przedstawienie przebiegu powstawania bruzdy gwintu i rozkład naprężeń i odkształceń w materiale obrabianym i narzędziu. Otrzymane rozkłady naprężeń eksploatacyjnych pozwalają na obserwację miejsc maksymalnie obciążonych a tym samym narażonych na przyspieszone zużycie. Pomaga to wybrać zarys najkorzystniejszy dla narzędzia przystosowany do konkretnego zastosowania w procesie technologicznym bez konieczności długotrwałych badań eksploatacyjnych.

INTRODUCTION

The use of new construction materials and increased demands on the reliability of machinery and equipment, as well as the desire to ensure higher quality products while increasing production efficiency and automation, are the direct causes of the search for and implementation of new threading technologies. In recent years, the use of cold forming of internal threads with the use of tools called forming taps, whose work can be roughly compared with the screwing of self-tapping screws **[L. 1, 2]**, has become more and more common. The possibility to change technology is due to the similarity of threading kinematics to thread cutting, the many common design features of the tools used, and the use of the same machine tools in the manufacturing process. The technology of plastic forming of internal threads can be applied to all types of materials provided that their elongation As > 8% and hardness does not exceed 28–30 HRC and especially for difficult-to-machine materials **[L. 3]**.

In order to reduce friction during forming a thread, the axial section of the taps has a polygon-like outline with rounded corners called ridges. The working part

* ORCID: 0000-0001-7135-3733. Magellana 6/17 Street, 51-505 Wrocław, Poland, e-mail: czarneck @itm.pcz.pl.
** ORCID: 0000-0002-7900-1131. Zajączka 15/4 Street, Częstochowa, Poland, e-mail: krzysztoftubielewicz@op.pl.
*** ORCID: 0000-0003-1738-5034. Czestochowa University of Technology, Institute of Mechanical Technologies, Armii Krajowej 21 Ave, 42-200 Częstochowa, Poland, e-mail: zaborski@itm.pcz.czest.pl.
**** ORCID: 0000-0001-9975-9372. Czestochowa University of Technology, Institute of Mechanical Technologies, Armii Krajowej 21 Ave, 42-200 Częstochowa, Poland, e-mail: michalt@itm.pcz.pl.
***** ORCID: 0000-0002-3132-5217. Piaskowa 22B Street, 42-242 Rędziny, MAG-RYŚ Nowoczesna Narzędziownia, Poland, e-mail: hubertmichalczuk@poczta.onet.pl.

consists of a forming part with a cone-shaped shape or the latest structures with a curvilinear shape. This part of the tool performs the main work of shaping. The calibration part has a small convergence in the shank direction and determines the dimensional and shape accuracy of the thread **(Fig. 1) [L. 3, 4]**.

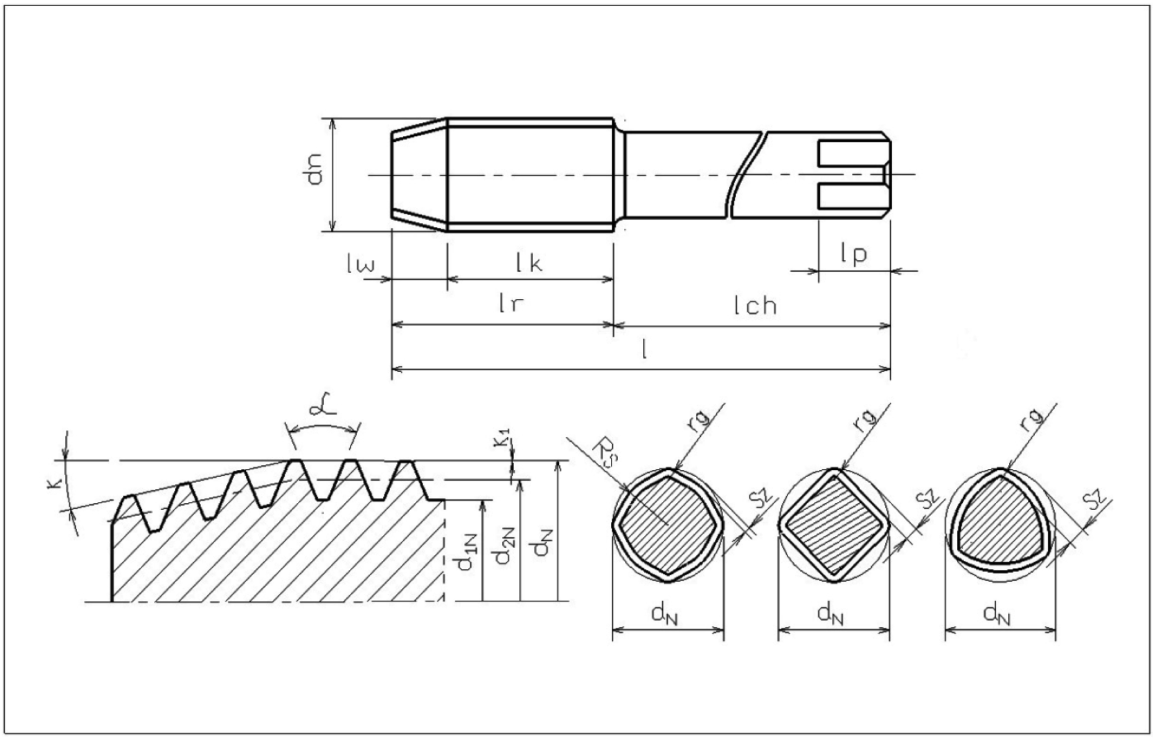

Fig. 1. Construction of forming tap: Sz – turning dimension, R_S – inner circle radius, r_g – ridge radius, α – thread angle, κ – chamfer angle, $κ_1$ – taper angle, (other markings in accordance with the standards, as for cutting threads) [L. 5]

Rys. 1. Konstrukcja gwintownika wygniatającego: Sz – wielkość zatoczenia, R_S – promień okręgu wewnętrznego, r_g – promień grani, α – kąt zarysu gwintu, κ – kąt przystawienia gwintownika, $κ_1$ – kąt zbieżności stożkowej gwintownika, (pozostałe oznaczenia zgodne z normami tak jak dla gwintowników skrawających) **[L. 5]**

By changing the radius of the corner (ridge) which makes the main material deformation, it is possible to influence the torque and the tool life within certain limits. As a result of the action of the ridges of the forming taps, the deformed material moves between the sides of the thread outline and fills in the free spaces **[L. 6]**. Depending on the diameter of the exit hole, a full or partial thread outline is obtained.

Such an appearance of the pressed thread tip is allowed by the amendment to the PN-84/M-82054/01 standard (Bolts, screws and nuts, Surface condition.) developed by the scientific team of the Institute of Technology and Production Automation of the Częstochowa University of Technology and published in PKNMiJ Bulletin No. 5 of May 1991, item 33 **[L. 7]**.

To optimize and evaluate the impact of changing the geometric parameters of a tool on the correctness of its work, it is necessary to make the trial taps and then to perform tests, which extends the entire modernization cycle and increases the costs of process preparation.

The rapid development of computer techniques creates new opportunities for modelling and simulation using numerical techniques to conduct experiments on real or virtual models describing a given process. More and more often, simulation modelling becomes a basis or a complementary method of research and evaluation of a real process and explanation of phenomena connected with it, and it creates the possibility of its forecasting **[L. 8]**.

Therefore, in the works of the scientific team at the Częstochowa University of Technology, computer techniques were used to solve an important problem concerning the influence of the geometric outline of the shape of a tool. Along with the use of proprietary software and applications (standard engineering packages) allowing the simulation of the process of cold forming of threads, the material strength of a taps corner was determined. It is assumed that strain is a measure of hazardous condition manifested by the appearance of local permanent deformations or cracks at any point in the component and that it depends solely on the components of the stress condition and the mechanical properties of the material. The weakest link principle is used to assess whole-body strength, i.e. the whole-body strength is determined by the point at which the greatest reduced stresses are present.

In order to simulate and thoroughly observe the phenomena accompanying the thread forming process and observe the elastic-plastic state of stresses and deformations, it was decided to conduct analyses using the Abaqus application. For this purpose, a model of a taps in the Catia system was developed, which was then loaded into the Abaqus system.

TAP MODELLING

When conducting a theoretical analysis or experimental research on a certain selected physical phenomenon, we try to reproduce it using a specific model. In the case of theoretical analysis, this model will be a mathematical model, and in the case of experimental research, a physical model.

Modelling of structures, tools, or technological processes **[L. 8]** can be done by analogy or by appropriate modelling. Proper modelling consists in building models with the preservation of features of geometric similarity in relation to the object being modelled. Proper modelling can be direct and indirect, during which the phenomena and states occurring in reality are studied. It is the most common modelling. Proper modelling of the object according to the similarity theory is very important for the possibility of transferring the results of model tests to the real object. Therefore, the Catia V5 program was used to create a three-dimensional model of the forming tap. It is a modern integrated CAD/CAM/CAE system used to support basically the entire cycle of activities related to the construction and manufacturing process of the product **[L. 9]**.

Work on the model of the forming tap began with the preparation of the grip part of the designed tool. Next, the forming part of a four-ridged tap with curvilinear shape in an axial cross-sectional was prepared. Both the forming and calibrating parts were designed so that their surfaces were the bottom of the furrow on which thread was later applied.

The shape of the cross-section significantly influences the process of forming and the life of the thread. Properly designed and parameterized cross-section contour greatly facilitates the introduction of changes. During the stage of cross-section design, it is possible to monitor many factors influencing thread formation (e.g., angle of contact with the material being processed, the thickness of the forming layer, whether forming takes place on the rounded surface, position of contact points, height of the corner part to the tangent), thanks to which it is possible to eliminate irregularities in the subsequent operation of the tool **[L. 10]**.

The last, and at the same time the most difficult, operation to develop the shape of the tool was to form the thread of the tap. This operation completed the process of preparing the model, the result of which is shown in **Fig. 2**. The drawing also shows the cross-sections (axial and normal) of the resulting tap.

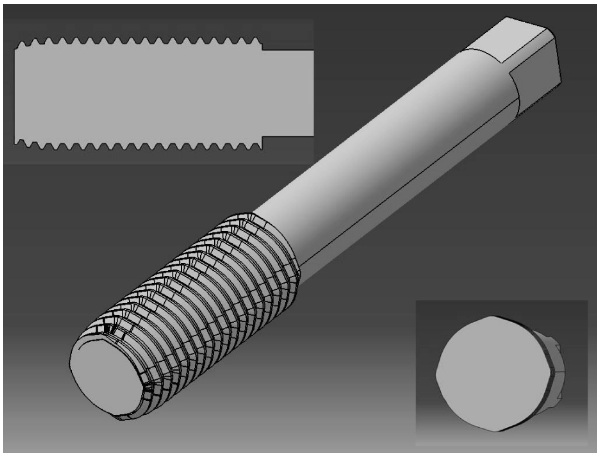

Fig. 2. Catia V5's forming tap model with visible axial and normal cross-section

Rys. 2. Model gwintownika wygniatającego opracowany w programie Catia V5 z widocznym przekrojem osiowym i normalnym

SIMULATION OF THE THREAD FORMING PROCESS IN THE ABAQUS

In order to obtain an analysis of the loads occurring at the tool ridge and in the hole, FEM simulations were performed during threading, using Abaqus, one of the world's most common software packages used to perform numerical simulations. It is used to solve complex engineering problems, in particular, to evaluate the strength of machine elements and structures, as well as to simulate highly complex linear and non-linear issues in solids mechanics and fluid mechanics. The application, using the finite element method, ensures proper assessment of emerging phenomena and is widely used in the aviation, automotive, shipbuilding, machine, metallurgy, and mining industries **[L. 11]**.

In this case, the main purpose of the simulation analyses is to determine the influence of the shape of the ridge on the loads occurring during the thread forming process and to observe the way the material flows. Therefore, in the first stage, it is necessary to prepare a numerical model.

To do so, a tap **(Fig. 2)** made in Catia V5 software was used from which three ridges were separated. This procedure was necessary, because the analysis for the whole tap would be very time-consuming in numerical calculations and difficult to perform in the conditions of the Institute due to the required computing power of computers. Such a simplified model of the tap and hole was imported into the Abaqus 6.9-3 system and scaled to 0.001 in order to adapt it to the system of SI units: length – meter, weight – kilogram, stress – Pascal, speed – meter per second **(Fig. 3)**.

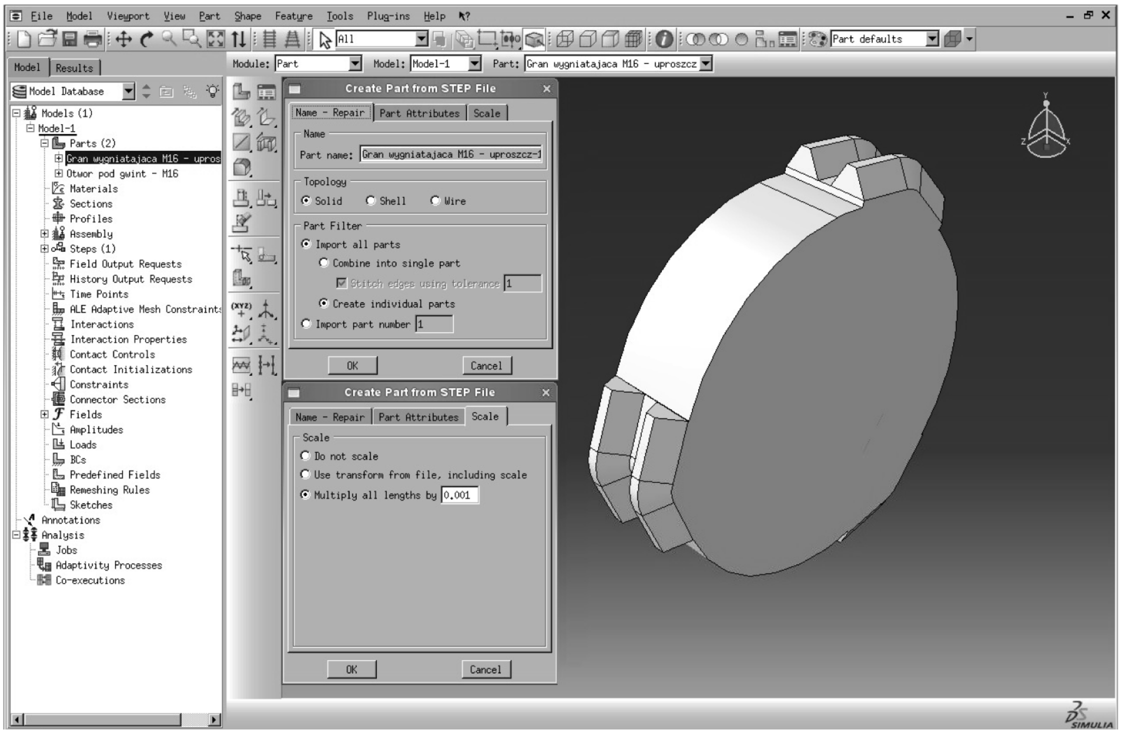

Fig. 3. View of the geometry of the forming tap imported into the Abaqus solver
Rys. 3. Widok geometrii gwintownika wygniatającego zaimportowanej do programu Abaqus

Next the material of the tap (high-speed steel HSS SW7M – 1.3343: compressive yield strength 3235 N/mm^2, E = 205GPa, G = 79 GPa density 8135 kg/m^3, Poisson`s ratio 0.28) and the hole (aluminium 2017A – 3.1325: yield strength 280 N/mm$^2$, E = 71,8 GPa, G = 27 GPa, density 2790 kg/m$^3$, Poisson`s ratio 0.33) were defined. During the tests, the decision was made to tap in aluminium in order to increase the chances of observing the phenomena accompanying the plastic flow of homogeneous material and its strengthening. In order to represent the actual conditions during the thread forming process for the material used, it was additionally described using points read from the material reinforcement curve (defining the plasticity of 2017A (3.1325) from the tensile curve). This operation made it possible in a more precise way to reproduce the elastic-plastic conditions occurring during the formation of a thread outline and to take into account the phenomenon of the reinforcement of the material resulting from the passage through the material of successive ridges. Parameters characterizing material properties were determined on the basis of standards.

Depending on the behaviour of the material after exceeding the yield point, plasticity may be ideal or with reinforcement. The yield point for a perfect plastic material is a constant value, so the reinforcement of the material may be omitted. In this case, the elastic-plastic material model with non-linear reinforcement was used to observe the deformation behaviour of the material **(Fig. 4)**. This model assumes that yielding stresses increase when the yield point is exceeded, resulting in a reinforcement of the material.

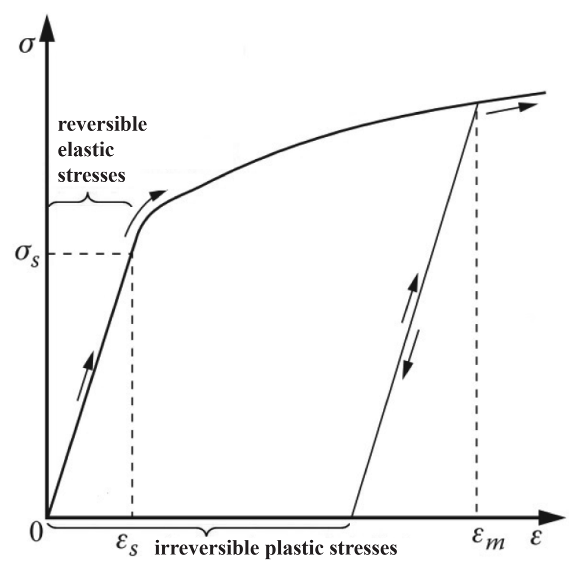

Fig. 4. Multi-linear model of the elastoplastic material [L. 12]
Rys. 4. Model materiału sprężysto-plastycznego ze wzmocnieniem nieliniowym **[L. 12]**

In the next step, the material sections were defined. This activity made it possible to assign different materials to particular elements of the numerical model and, by introducing appropriate geometrical cross-sections, to introduce different sizes of the finite elements depending on the importance of particular areas in the construction of the numerical task and the computational power possessed. Then, through the Assembly module, the model was assembled and the initial position of the tool in relation to the hole being machined was defined. We also introduced the reference point RP-1, to which the forming ridge was „attached" and for which boundary conditions will be set describing the kinematics of this element **(Fig. 5)**.

The tools included in the STEP module have been used to define the details of the numerical analysis that the system should perform in individual time steps and to determine which calculation values should be presented in the form of results. The whole calculation task can contain any number of steps, constituting a separate analysis. For calculations carried out in several steps, each step is based on the initial model resulting from the calculations in the preceding step.

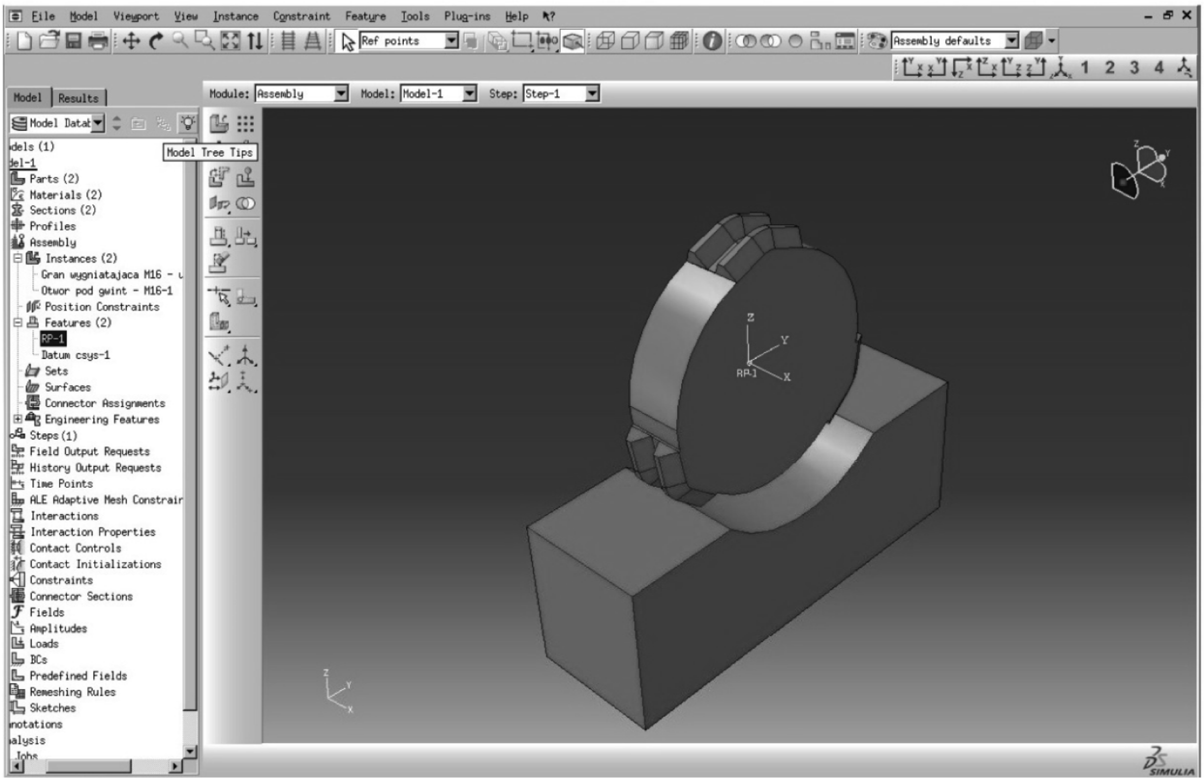

Fig. 5. View of the Assembly module – fixing the position of the elements in relation to each other
Rys. 5. Widok okna modułu Assembly – ustalenie położenia elementów względem siebie

Each model has a so-called initial step, in which the initial and boundary conditions of the model should be determined. In order to precisely present the phenomenon, additionally 4 new steps were introduced describing the calculation task: step 1 – start-up, step 2 – contact and transition of the first pair of ridges, step 3 – transition of the second pair of ridges, step 4 – transition of the third pair of ridges.

Contact properties are defined in the Interaction Properties tab. The surface to surface contact type was adopted and the appropriate contact pairs were indicated. In all simulations, a constant coefficient of friction $\mu = 0.1$ was determined which reflects the conditions occurring during the thread forming using ACP non-emulsifying oil as a lubricating and cooling agent **[L. 13]**. In cases where other coefficients of friction were used ($\mu = 0.4$–0.8), excessive deformations of some finite elements of the hole zone were observed, which, in real conditions, could mean the formation of overgrowths on the surfaces of the forming taps' ridges.

After defining contact properties, it was necessary to enter boundary conditions and initial conditions for the whole model. In this case, the conditions applied were that the threaded hole should be restrained and the tool should be „allowed to rotate" with given velocity. It was assumed that the tool rotated at an angular speed of 52.36 rad/s, which corresponds to a speed of 500 rpm (actual tapping speed of 10 m/min), at which a M16 thread in 2017A (3.1325) were made in practice. The angular displacement of the tap was realized on the basis of the assumed angular velocity and the time in which the rotation takes place, determined in successive steps (0.12 s). **Figure 6** shows the way of introducing the boundary conditions and the velocity of the tap.

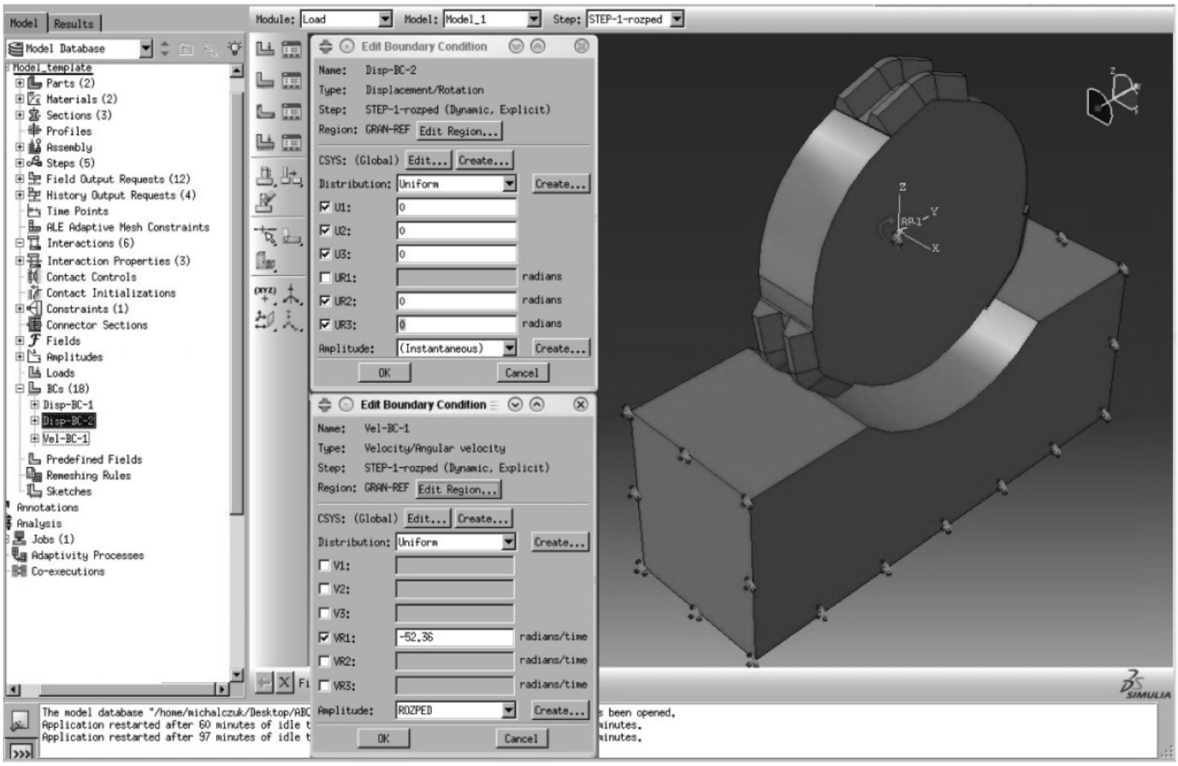

Fig. 6. Tool boundary conditions: restraint and rotation
Rys. 6. Widok utwierdzenia narzędzia oraz nadanie mu obrotu

In the next stage, a finite element mesh was imposed, whose properties **(Tab. 1)** are able be changed depending on the accuracy of calculations we want to obtain, as well as the available computing power **(Fig. 7)**.

Table 1. Finite element properties
Tabela 1. Przyjęte właściwości elementu skończonego

Finite element properties	**Tool ridge**	**Hole**
Shape	hexahedral	hexahedral
Type:	C3D8R	C3D8R
Library:	Explicit	Explicit
Element size [mm]:	0.114–0.791	0.076–0.657
No. of elements:	7442	27120

The preparation of an appropriate finite element mesh and the way it is superimposed affects the accuracy of the obtained results and limits the time needed to complete the calculation task. Therefore, in the case presented here, the mesh was made in such way that it is properly thicken in the places of the occurrence of

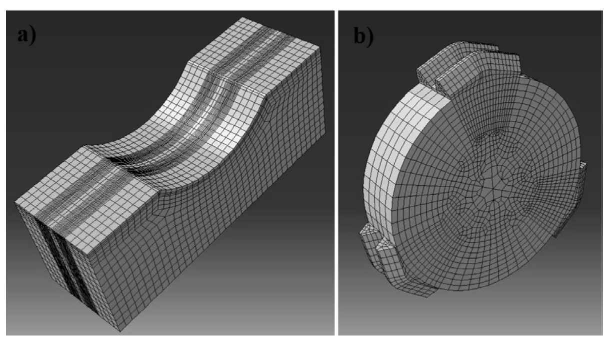

Fig. 7. Finite element grid view: a) on the material, b) on the tool
Rys. 7. Widok siatki elementów skończonych: a) na materiale, b) na narzędziu

deformation zones in order to obtain more precise results for these zones. However, in areas of lesser importance for this research, larger elements were introduced to simplify the calculations and reduce the need of memory capacity and CPU`s load, thus minimizing the duration of calculations. It was decided to thicken the mesh in the areas in the deformation zone at the hole, directly under the tool (0.076 mm), and in the tool's ridge zone (0.114 mm), where the basic work takes place during the thread forming process, and where the shape of the tool has a significant influence on the correct course of the forming process.

The last step that had to be taken in order to start the numerical analysis was to create a computational task specifying the following: the type of task, the area of allocated memory, parallel processing/multithreading, and precision of analysis. The calculations were conducted on eight cores with reserved 8GB of RAM for the whole task. Parallel processing type: threads.

SIMULATION RESULTS ANALYSIS

During the thread forming process, ridges penetrate into the hole on the depth 0.118 mm each, and they made the first three passages starting the thread forming process in the hole fragment and then the last three passages in the hole fragment with the pre-modelled thread groove. These procedures make it possible to observe the phenomena accompanying the process of thread shaping in the initial phase, when the basic deformation work is carried out, up to the final phase, together with the profile rolling operation when the final dimensions and shape of the thread are given. In this way, the adopted constructional solutions and the shape of the tap ridge were evaluated.

The work of the M16 tap designed for forming threads through holes was examined. During the study, the most important information was provided by stress distribution (Huber-Mises) and deformation maps in the deformation zone and in the tool material on the ridge at the point of its contact with the workpiece.

Figure 8 illustrates the stress distribution on the ridge and in the material deformed during the formation of the thread groove. On the basis of the obtained results, it can be stated that the highest stresses occur in the area of material deformation and on the ridge at the point of contact between the tool and the workpiece.

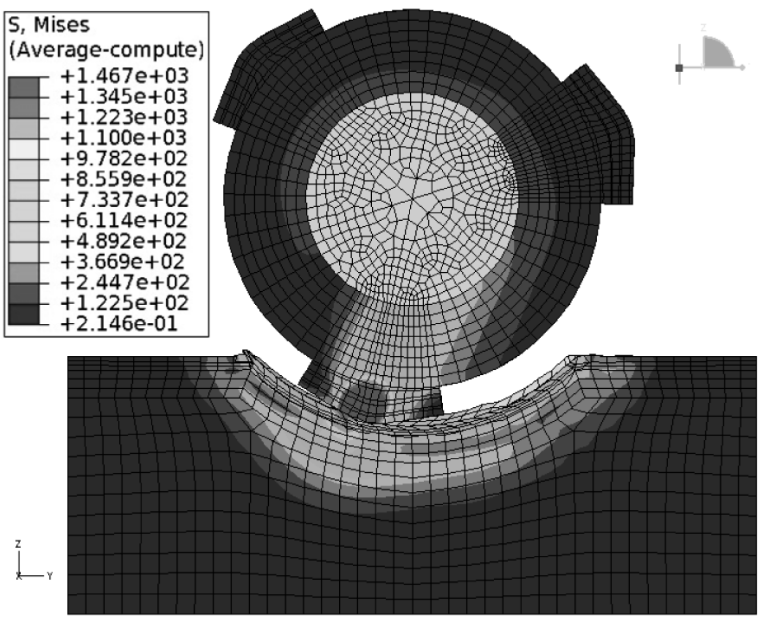

Fig. 8. Distribution of the reduced stress on the tap ridge and in the workpiece in the cross-section through the centre of the ridge during the thread forming

Rys. 8. Rozkład naprężeń zredukowanych na grani gwintownika i w materiale obrabianym w przekroju poprzecznym przez środek grani podczas wygniatania gwintu

Figure 9 presents a list of reduced stress maps for the next three transitions of the forming ridges through the hole, which are set in the same angular position for a clear results comparison.

The highest stresses were observed on the forming ridge at the point of contact with the material in which the basic deformation process takes place and in which the highest tool loads occurs. During the first crossing, stress value 1296 MPa was read, during the second crossing, it was 1467 MPa, and during the third one, the value was equal to 1607 MPa. The increasing value of stresses is caused by the position of the forming ridge which depth is increasing inside the material being processed. The increase in value is certainly influenced by the increasing contact surface of the tool with the material being processed and its hardening due to successive passes. The cross-sectional area shows an increase in stress on the ridge part of the tap and an increasing zone in the threaded hole that increases with each passing. This indicates an increase in the mechanical properties of the element subjected to this type of plastic deformation.

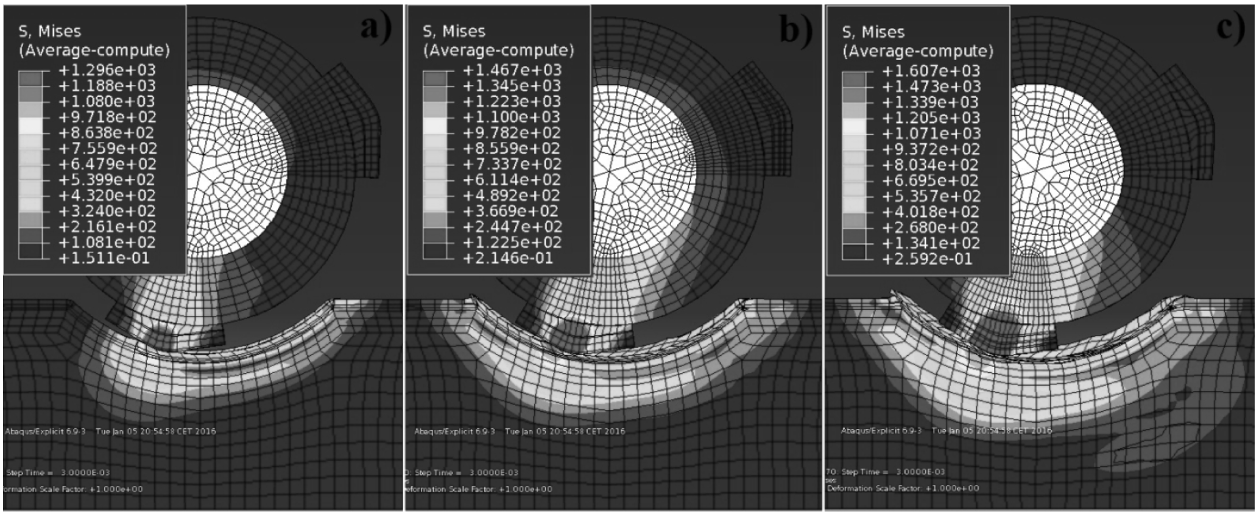

Fig. 9. Distribution of reduced stress in the cross-section in the following three transitions (a, b, c) of the tapping ridge through the hole

Rys. 9. Rozkład naprężeń zredukowanych w przekroju poprzecznym w kolejnych trzech przejściach (a, b, c) grani wygniatającej przez otwór

The view of reduced stresses in a separate fragment of the forming tool **(Fig. 10)** allows identification of the distribution of loads occurring on the thread forming ridges and the method of their formation depending on the outline of the corner shape. The loads are created symmetrically on the ridges, which may indicate that the thread forming process simulation has been modelled correctly and is proceeding without any disturbances.

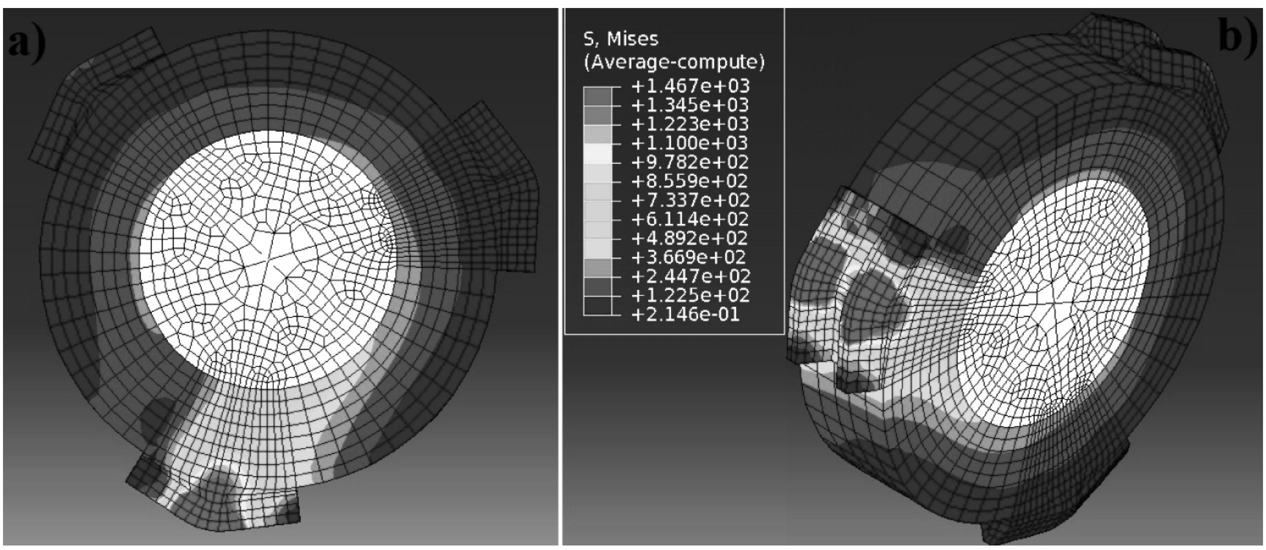

Fig. 10. Reduced stress map: a) on the ridge, b) isometric view

Rys. 10. Mapa naprężeń zredukowanych: a) na grani, b) widok izometryczny

To determine the most loaded point on the tap ridge, the colour map has been scaled up **(Fig. 11)** so that the maximum reduced stresses at the front of the ridge can be located in the area where the tool is in contact with the machined material. This confirms the fact that the shape of this tool element has a significant influence on the thread forming process. In the practice of the process implementation in this area of the tool one may observe the appearance of traces of wear caused by a significant load of the tool under operating conditions.

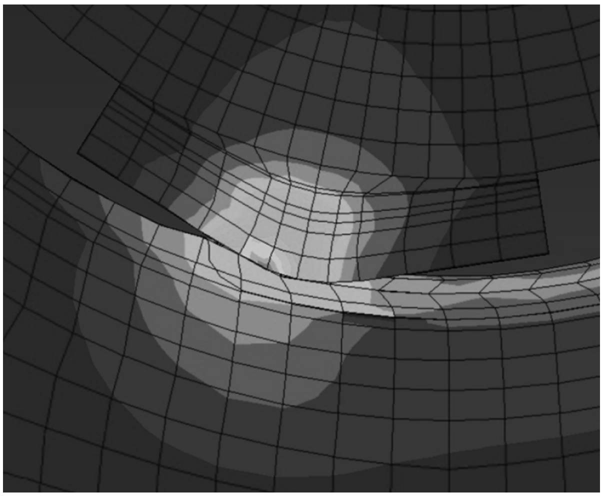

Fig. 11. Image of the location of reduced stress in the cross-section concentrated in the most heavily loaded zone of the tapping ridge

Rys. 11. Obraz lokalizacji naprężeń zredukowanych w przekroju poprzecznym skoncentrowanych w najbardziej obciążonej strefie grani wygniatającej

Before the ridge, a slight swaging occurs, which is caused by the intensive flow of material in front of the tool and may confirm the theory of the so-called wave formation, which indicates the proper definition of the material model adopted for the hole, used during the simulation of the phenomenon. The observation of deformations in the axial section **(Fig. 12)** also gives the possibility to locate the maximum stresses occurring on the thread profile of the tool. The highest stresses in the thread profile zone occur at both sharp corners of the profile apex and are located symmetrically in relation to its centre. They also coincide with the character and course of wear of the thread in this area. **Figure 12** also shows the thread forming process between the sides of the tool. The material is pressed into the free space between the tool windings, which results in a pre-forming of the thread outline. It can be seen from the distribution of the finite element mesh boundaries that the flowing material in the groove of the tap starts to form a characteristic indentation in the middle of the thread top from the very beginning of the process.

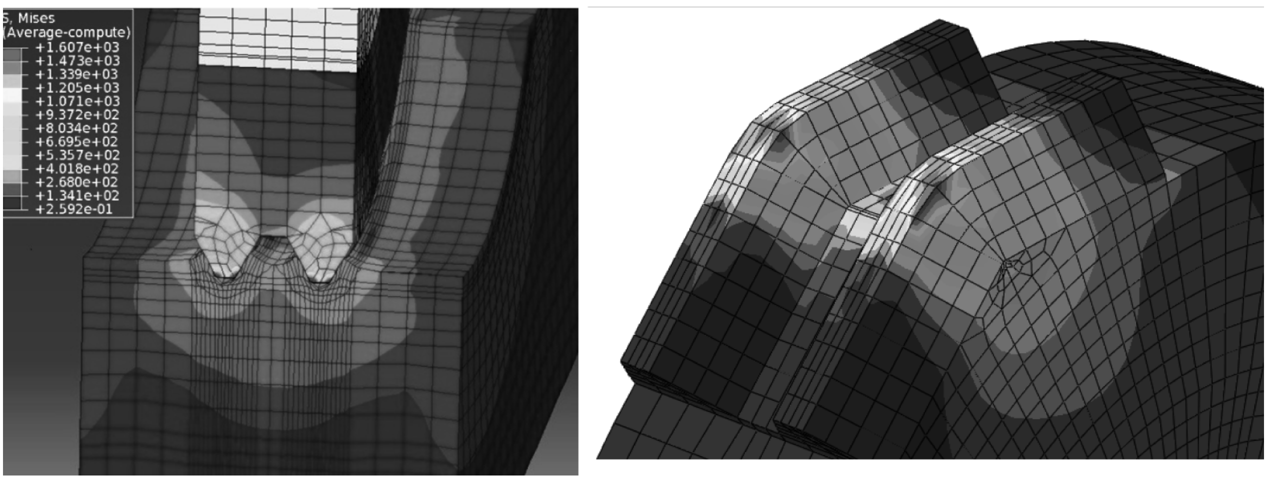

Fig. 12. Reduced stress image in axial cross-sections (left) and isometric view of the most heavily loaded points of the tap (right)

Rys. 12. Obraz naprężeń zredukowanych w przekroju osiowym (po lewej) i w widoku izometrycznym – widok najbardziej obciążonych miejsc gwintownika (po prawej)

As mentioned earlier, the highest reduced stresses in the tap occurs on the forming ridge and reach a value of approx. 1610 MPa. However, they are much lower than the limit of tensile strength R_m of the tool material made of high-speed steel SW7M (1.3343), which is 2180 MPa. Therefore, it is not possible to create any permanent deformations on the threaded sections of the tap.

The results of the simulation can be illustrated by graphs of maximum reduced stress and torque during thread forming **(Fig. 13)**. The horizontal axis describes the duration of the analysis of the thread forming process and the time intervals (0–0.04; 0.04–0.08; 0.08–0.012) of the subsequent transitions of the three ridges.

On the basis of the presented studies, it can be stated that the Abaqus solver allows the simulation of the process of cold forming of the internal thread. The applied solutions allow one to observe the process of thread forming by the ridge of the tap and to determine the values of reduced stresses on the tool and in the deformed material. Therefore, it was decided to determine the influence of the transverse shape of the taps on the course of the threading process.

For this purpose, the cross-sectional shapes of M16 taps with the parameters (dimensions) shown in **Fig. 14** were simulated.

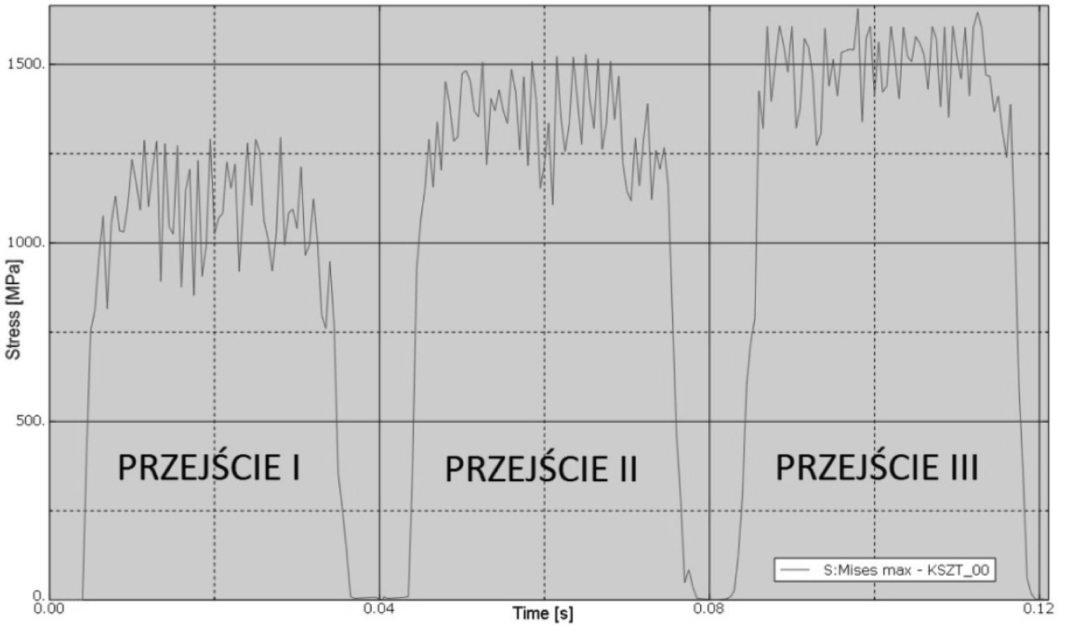

Fig. 13. Example of maximum reduced stress during thread forming
Rys. 13. Przykład przebiegu maksymalnych naprężeń zredukowanych występujących podczas wygniatania

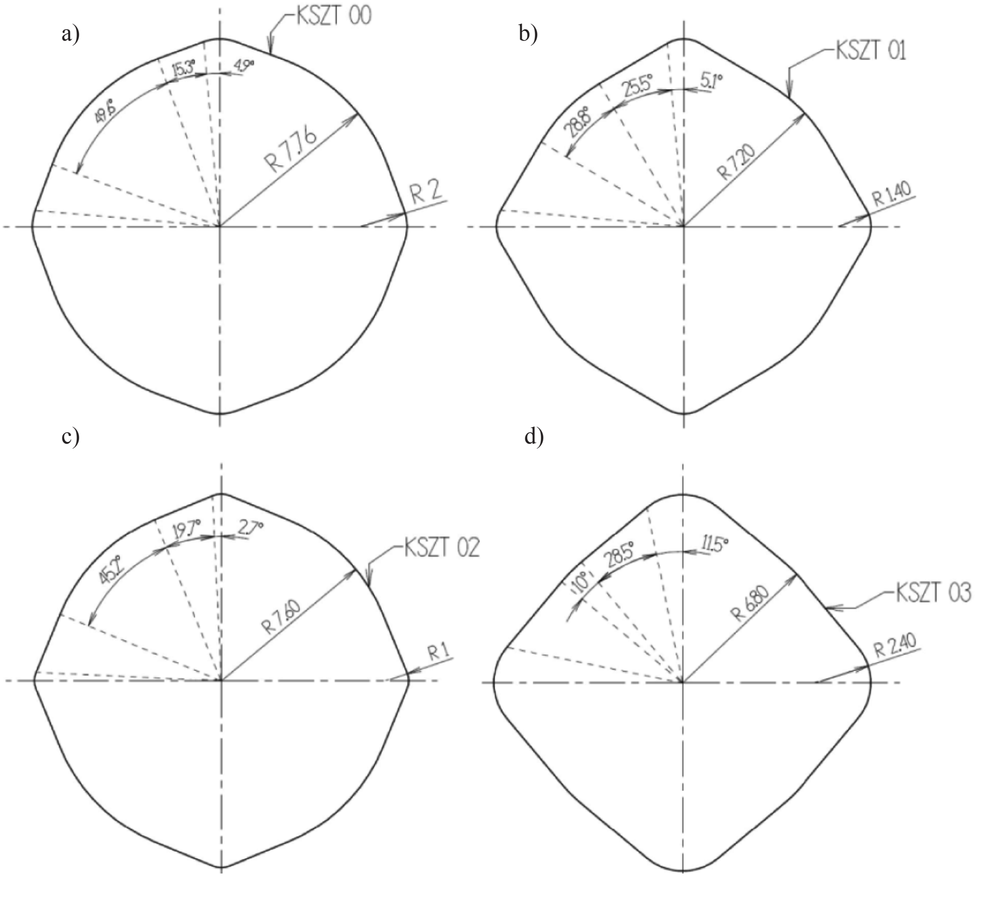

Fig. 14. Cross-sectional outlines of the tap, analysed during the simulation of the threading process
Rys. 14. Zarysy przekroju poprzecznego gwintownika poddane analizie podczas symulacji procesu wygniatania gwintu

In order to compare the load values for the examined cross-section profiles, the courses of maximum reduced stresses were then superimposed on each other **(Fig. 15)**.

The stress curves were approximated by the LOWESS method, which is the so-called „Robust Locally-Weighted Regression". It is a method of smoothing (averaging value waveforms) used for data on scatter charts in the x-y system. In this method, polynomial fit to the data using weighted least squares is assumed, and the value waveforms represent a clear general picture of dependencies between two variables **(Fig. 16) [L. 10]**.

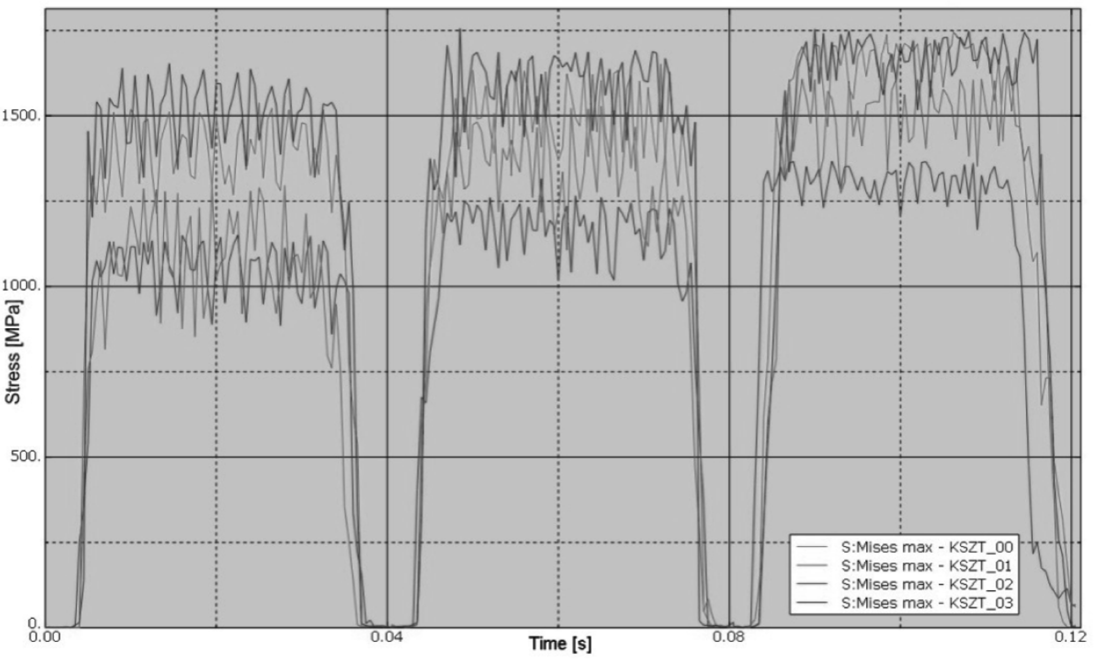

Fig. 15. Maximum reduced stresses for all numerically analysed cross-sectional profiles
Rys. 15. Maksymalne naprężenia zredukowane dla wszystkich analizowanych zarysów przekroju poprzecznego poddanych analizie numerycznej

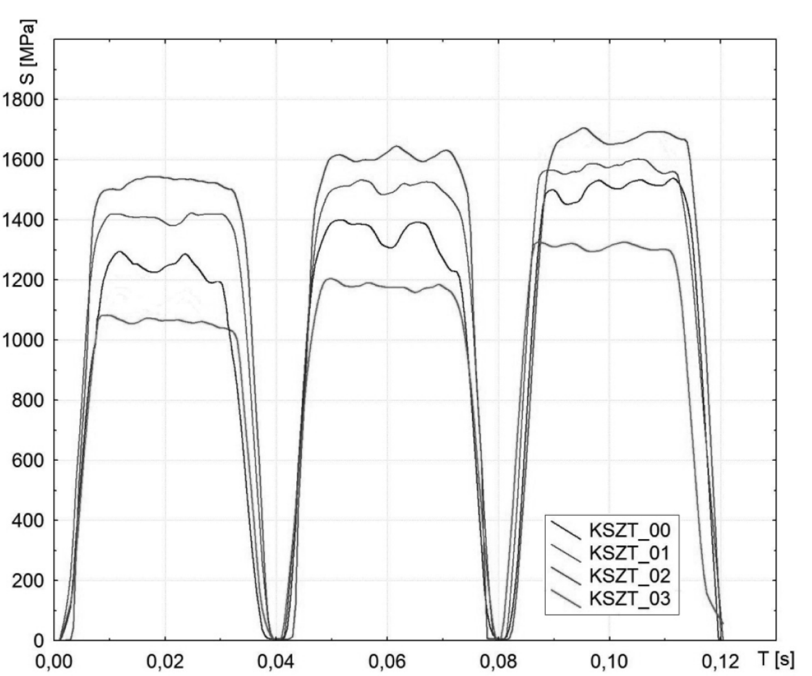

Fig. 16. Comparison of approximated maximum reduced stress for all cross-section profiles
Rys. 16. Porównanie aproksymowanych przebiegów maksymalnych naprężeń zredukowanych dla wszystkich analizowanych zarysów przekroju poprzecznego

The analysis of the influence of the corner shape of the tool on the values of stresses on the threaded sections showed that the modification of the structural parameters of the tool in the cross-section causes a significant difference in the value of stress, as evidenced by the presented results.

SUMMARY

Thanks to the possibility of performing various analyses and simulations of tap work, it is possible to eliminate errors in their shape or improve the preparation of implementation of this type of machining in production conditions. The Abaqus system is a desirable means of simulating the interaction between the selected shape of a taps and the workpiece. Thus, it can be helpful in designing a tool for specific applications. The finite element method allows for various types of strength simulations and computational analyses carried out during the design process on virtual models (close to real ones), which eliminates the need to perform tools (including cams, or a program for a numerical grinder), which require a lot of time and tests at research stations. It allows one to visualize the process of thread shaping, which is practically impossible in a laboratory experiment.

A simulation of strength loads aimed to modernising the shape of the tap has provided information that allows a designer to introduce new solutions in the shape and dimensions at the initial step of design process. Numerical analysis and the results obtained from experimental testing will help explain a number of phenomena occurring during the plastic deformation process and provide a wealth of data to assist in the design of this type of tool. The proposed simulation method allows one to determine the influence of a change in the shape of the tool corner on the area where the local stress concentration on the tap ridges and in the machined material occurs. This allows one to determine the changes in the moment of forming occurring in different phases of the thread forming process. Thanks to the simulation, it was possible to analyse the load on the tool's ridge and to determine places subjected to wear. Such information is important for the entire threading process and can be used as guidelines for the design of the manufacturing process of thread cold forming, which has also been confirmed by experimentation during industrial implementation.

Preparation of this type of simulation, however, requires a huge engineering experience, due to many aspects which should be taken into consideration. The whole task is quite time consuming, but only in the phase of its development.

REFERENCES

1. Czarnecki H., Michalczuk H.: Komputerowo wspomagana analiza zarysu poprzecznego gwintowników wygniatających, Mechanik 7/2014, CD, s. 117–126.
2. Czarnecki H., Michalczuk H.: Wspomaganie komputerowe projektowania gwintowników wygniatających i symulacja procesu wygniatanie gwintu, Współczesne problemy technologii obróbki przez nagniatanie, t. 4. Monografia pod redakcją Przybylskiego W., Politechnika Gdańska, Gdańsk 2014.
3. Łyczko K.: Technologia narzędzi i wygniatanie gwintów wewnętrznych, Wyd. PCz, Częstochowa 1999.
4. Łyczko K.: Gwintowniki wygniatające, WNT, Warszawa.
5. Michalczuk H., Czarnecki H.: Modelowanie i symulacja procesów wygniatania gwintów wewnętrznych na zimno. Mechanik nr 11/2014, CD, s. 13–24.
6. Czarnecki H., Michalczuk H., Tagowski M.: Changes in the properties of the surface layer of the internal thread made by cold forming, Tribologia 6/2016, s. 11–20.
7. PN-84/M-82054/0: Dotycząca kształtu wierzchołka wygniatanego gwintu.
8. Bąk R., Burczyński T.: Wytrzymałość materiałów z elementami ujęcia komputerowego, WNT, Warszawa 2001.
9. Michalczuk H., Czarnecki H.: Parametryzacja gwintowników wygniatających w systemie Catia V5, Mechanik nr 7/2015.
10. Michalczuk H.: Wpływ wybranych parametrów konstrukcyjnych gwintownika wygniatającego na kształtowanie gwintu, Praca doktorska 2018. Politechnika Częstochowska, Wydział Inżynierii Mechanicznej i Informatyki. Promotor prof. PCz. dr hab. inż. Czarnecki H.
11. Abaqus Analysis User's Manual Volume II Version 6.7, DS Simulia.
12. Zaborski A.P.: Teoria plastyczności i stanów granicznych, Politechnika Krakowska, Kraków 2010.
13. Czarnecki H.: A Program for Analysis of the Cross-Sectional Contour of Embossing Taps, Management and Control of Manufacturing Processes. Eds. Antoni Świć, Jerzy Lipski, Wyd. Lubelskie Towarzystwo Naukowe, Lublin 2011, s. 80–92.