4th International Conference "Optimization of the Robots and Manipulators"
OPTIROB 2009- CONSTANTA, 28-31 May 2009
Organized by University "POLITEHNICA" of Bucharest
Faculty IMST, Department MSP, ROMANIA

APPLICATION OF CAM-FEM TECHNIQUES IN THE ESTABLISHMENT OF MILLING CONDITIONS OF THE PARTS WITH THIN WALLS SURFACES

Ionuț Ghionea[1], Adrian Ghionea[2], Ioan Tănase[3]
University Politehnica of Bucharest
Spl. Independenței nr. 313, Faculty of IMST
E-mail: ionut76@hotmail.com, ghionea@imst.msp.pub.ro, tanase@imst.msp.pub.ro

Abstract
The aim of the presented analysis is the diversity the parts' surfaces shapes, their materials and most of all, the working simulation possibilities using specialized software. Thus, in the paper are shown some results of the CATIA application in milling simulations of parts with thin wall surfaces using end mill tools. The considered surfaces have linear and curved directories – concave/convex and low stiffness or a high ratio for width/thickness. These situations are met in various large number of parts for moulds, rotor blades, aerospatial assemblies etc. The simulations offer values of the elastic deformations and stresses, these results being very helpful in the establishment of the working regime parameters in real conditions, in the adaptation of the technological processes to CNC machine-tools.

Keywords: CAM milling – CNC manufacturing - FEM analysis - stress - simulation.

1. Introduction
The use of modern CAD-CAM techniques in the parts manufacturing processes has a large application in the mechanical, automotive, aerospatial industry domains. The specialty literature [1], [3] also highlights remarkable results of FEM applications in order to determine the elastic deformations values and the stresses inducted in parts by the contact with the tools during the working processes. The influences of the technological system (machine-tool, tools, fixture devices) determine dimensional variations and irregularities of the processed surfaces, dynamical behaviour of the machine-tool in the cutting process and also the premature wear of the cutting edges [4]. The results of the surfaces generation by simulation, with the aid of CAM techniques (tool path, division pattern of the stock left for machining, different working strategies, etc.) offer a large number of data: processing times, surface accuracy (roughness size), behavior of the machine-tool in working conditions (power and cutting torque) etc. both for roughing and finishing operations. For the optimization of the technological process there are considered and applied various criteria, software environments, tables of data.

2. The CAM simulation
Preparing the piece for processing on a machine-tool with numerical control involves the generation of command information, all data is then stored in a preset order within a storage device. Programs can be generated directly on the machine, the operator writes the necessary instructions using the available interface or by using a CAD-CAM program and a virtual model of the piece [2], [3]. Defining the piece in a CAD environment is used as the entry date to generate the program with one of the complex existing programming languages. Thus, the simulation is justified to optimize the process because CAM programs elaborate the NC machine code. In the case of manual developing and writing, the allocated time is disproportional to the time of processing. The command for machine is provided

[1] - Lecturer Eng., Machine Manufacturing Technology Department
[2] - Prof. PhD. Eng., Machine and Manufacturing Systems Department
[3] - Prof. PhD. Eng., Machine and Manufacturing Systems Department

by the numerical control equipment (NEC), designed for the following types of machine tools: lathes, milling, grinding, hobbing machines. The machine-tools, by their kinematics structure, execute simple movements (rotation, translation) in relation to the numerically controlled axes. The axes of coordinates are assigned to couples of translation or rotation of the machine. Each processing program for the piece, compatible with the machine, consists of a sequence of sentences written in a logical sequence based on a specific syntax [2]. On the modern CNC equipments the user may program, in addition to coordinates, other geometrical information regarding the compensation of length and tool diameters. Thus, it is possible to do various corrections on the dimensions, clearances and vibrations appeared in the fixture devices of the piece on the machine-tool etc. In general, the corrections are introduced to the machine console by a specific address, and a group of digits for the correction.

For the purposes of this paper, it is considered a part having its 3D model made in CATIA Part Design module and presented in figure 1. The overall dimensions of the stock part are 100×100×32 mm.

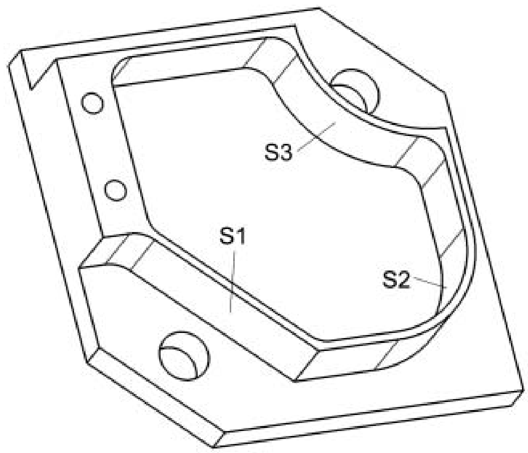

Fig. 1 – The part's 3D model

From the analysis of the part's surfaces it is identified the cavity delimited by a closed contour whose walls have thickness of 2 mm and 20 mm height. To express (by comparison) the material influence over the part's stiffness in manufacturing conditions, there are considered two materials: high-alloyed steel and aluminium alloy, having the yield strength of 2.5×10^8 N/m^2 and 9.5×10^7 N/m^2.

To manufacture the part it is used a 3-axis CNC vertical milling machine-tool having the following main characteristics: spindle speed: 20000 rpm with infinite variable speed range (direct drive spindle), movement on axes X = 1020 mm, Y = 550 mm, Z = 560 mm, the power of the principal electric engine: P = 15 kW (continuous rating), machining feedrate max: 8000 mm/min and rapid feedrate: 30000 mm/min. The machine-tool has a CNC Sinumerik controller.

The tools used in manufacturing simulation are chosen from a company catalogue [5]. Also, the toolholders are in correspondence with the spindle nose and the holding system of the machine-tool.

Below are presented (as figures and parameters) the main steps of the CAM simulation in the case of aluminium alloy working, with 75 HB.

a. *Face mill* (Fig. 2), one pass, D_c = 50 mm - tool diameter, h_m = 0.08 mm – average chip thickness, v_c = 960 m/min - cutting speed, n_c = 5600 rpm - spindle speed, v_f = 3400 mm/min - feed speed, P_c = 4.6 kW - cutting power for removal of chips, M_c = 7.7 Nm - cutting torque, Q = 204 cm^3/min - metal removal rate, f_z = 0.1 mm – feed per cutting edge, a_p = 2 mm - cutting depth, z_c = 5 – number of teeth, t_m = 9 s – machining time, t_t = 11 s – total time.

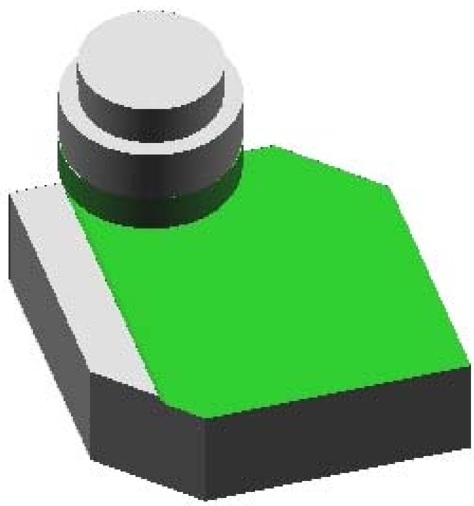

Fig. 2 - Face mill

b. *External roughing profile contouring mill* (Fig. 3), one pass, D_c = 25 mm, h_m = 0.04 mm, v_c = 1000 m/min, n_c = 12000 rpm, v_f = 2600 mm/min, P_c = 13 kW, M_c = 11 Nm, Q = 520 cm^3/min, f_z = 0.04 mm, a_p = 20 mm, z_c = 5, t_m = 19 s, t_t = 20 s.

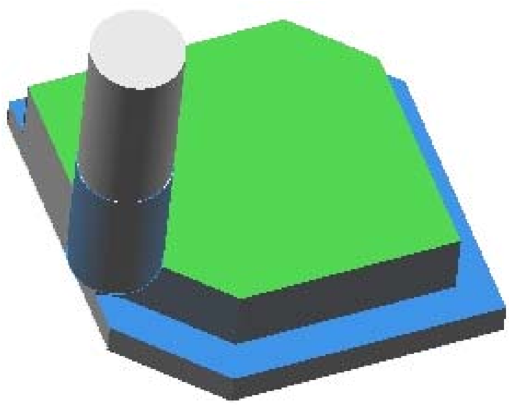

Fig. 3 - Profile contouring mill, rough surface

c. *External finishing profile contouring mill* (Fig. 4), two passes, D_c = 16 mm, h_m = 0.02 mm, v_c = 1000 m/min, n_c = 14000 rpm, v_f = 2400 mm/min, P_c = 11 kW, M_c = 5.3 Nm, Q = 340 cm^3/min, f_z = 0.02 mm, a_p = 20 mm, z_c = 6, t_m = 50 s, t_t = 52 s.

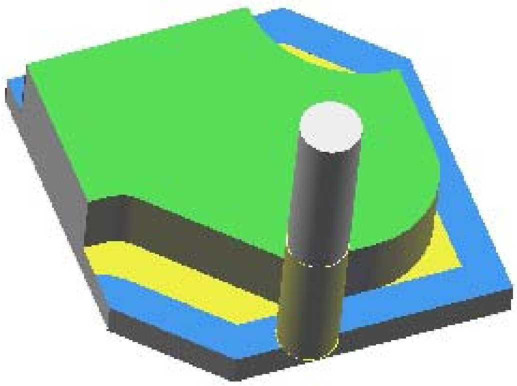

Fig. 4 - Profile contouring mill, finish surface

d. *Internal roughing profile pocketing mill* (Fig. 5), one pass, D_c = 16 mm, h_m = 0.02 mm, v_c = 1000 m/min, n_c = 14500 rpm, v_f = 1300 mm/min, P_c = 13 kW, M_c = 6.4 Nm, Q = 380 cm^3/min, f_z = 0.02 mm, a_p = 20 mm, z_c = 4, t_m = 24 s, t_t = 27 s.

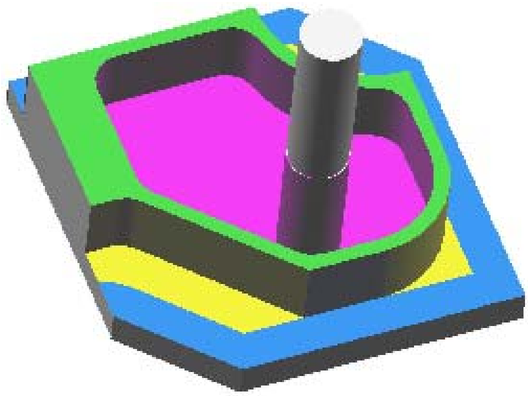

Fig. 5 - Profile pocketing mill, rough surface

e. *Internal finishing profile contouring mill* (Fig. 6), two passes, D_c = 12 mm, h_m = 0.03 mm, v_c = 735 m/min, k_{c1} = 700 N/mm^2 – specific cutting force, m_c = 0.25 – exponent, n_c = 19500 rpm, v_f = 8500 mm/min, P_c = 4.4 kW, f_z = 0.11 mm, a_p = 20 mm, a_e = 1 mm – working engagement, z_c = 4, t_m = 27 s, t_t = 31 s.

Fig. 6 - Profile contouring mill, finish surface

The tool path for this finishing mill process is shown in detail in figure 7.

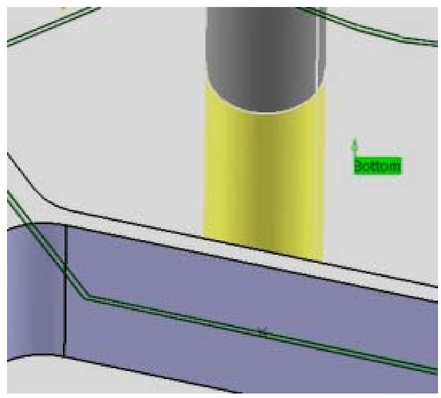

Fig. 7 - Detail with the tool path for the profile contouring mill, finish surface

f. *Drill Ø12 two holes* (Fig. 8), D_c = 12 mm, v_c = 200 m/min, n_c = 5300 rpm, v_f = 2100 mm/min, P_c = 3.5 kW, M_c = 6.4 Nm, f_z = 0.02 mm, a_p = 6 mm, z_c = 2, t_m = 5 s, t_t = 7 s.

Fig. 8 - Drilling two holes Ø12

g. *Drill Ø5 two holes* (Fig. 9), D_c = 5 mm, v_c = 140 m/min, n_c = 8900 rpm, v_f = 1800 mm/min, P_c = 0.7 kW, M_c =0.6 Nm, f_z = 0.02 mm, a_p = 2.5 mm, z_c = 2, t_m = 16 s, t_t = 18 s.

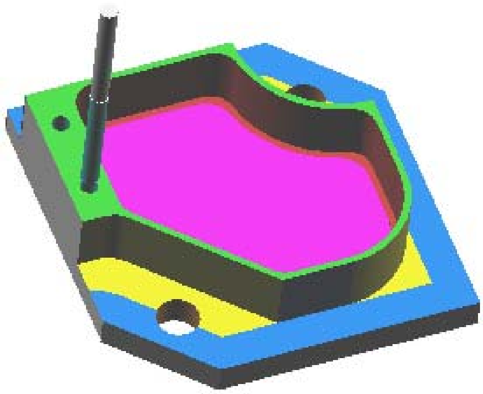

Fig. 9 - Drilling two holes Ø5

In the case the part would be processed of high-alloyed steel, with 200 HB, all the data above will be modified. Thus, for the operation e.*Internal finishing profile contouring mill* (Fig. 6) the parameters become: two passes, D_c = 12 mm, h_m = 0.03 mm, v_c = 235 m/min, k_{c1} = 1950 N/mm^2, m_c = 0.25, n_c = 6235 rpm, v_f = 2750 mm/min, P_c = 4.3 kW, f_z = 0.11 mm, a_p = 20 mm, a_e = 1 mm, z_c = 4, t_m = 30 s, t_t = 34 s.
To determine the cutting power for this operation, it is used the next equation:

$$P_c = \frac{a_p \cdot a_e \cdot v_f \cdot k_c}{60 \cdot 10^6 \cdot \eta}, \text{ [kW]} \qquad (1)$$

where: $k_c = k_{c1} \cdot h_m^{-mc}$ = 4685 N/mm^2 and η = 1.

```
N10 G0 G90 G40
N20 G40 M8
N30 G17
N40 G0 X0 Y0 Z25
N50 ;= TOOL CHANGE =
N60 ; DESC :
N70 ;===============
N80 T5 M06
N90 D5
N100 G0 G90 G40 G17
N110  F0 S0
N120 G64 SOFT
N130 S6250 M3
N140 G1 X18.5 Y-18.5 Z10 F8000 G94
N150 Z-22
N160 Y-51.479 F2700
N170 X48.521 Y-81.5
N180 X65
N190 G3 X81.5 Y-65 I0 J16.5
N200 G1 Y-47.55
N210 G2 X52.45 Y-18.5 I8.5 J37.55
N220 G1 X18.5
N230 Y-18 F500
N240 G3 X18 Y-18.5 I0 J-0.5
N250 G1 Y-51.479
N260 X18.146 Y-51.833
N270 X48.167 Y-81.854
N280 X48.521 Y-82
N290 X65
N300 G3 X82 Y-65 I0 J17
N310 G1 Y-47.55
N320 X81.909 Y-47.262
N330 X81.61 Y-47.062
N340 G2 X52.938 Y-18.39 I8.39 J37.062
N350 G1 X52.785 Y-18.129
N360 X52.45 Y-18
N370 X18.5
N380 Z10 F6000
N390 M5
N400 M5 M9
N410 M30
```

Fig. 10 - CNC code for the internal finishing profile contouring mill, high-alloyed steel case

After the CNC simulation of the considered operation, the resulted code is shown in figure 10.

3. The FEM simulation and analysis

The Finite Element Method (FEM) is one of the best existing approaches that accomplish a vast range of engineering calculus and simulation [1]. The method and FEM based programs becomes a main constituent of modern engineering computer-aided design. A FEM based analysis is now compulsory for high performance engineering tasks.
For many equipments and particularly for the machinery manufacturers, the resistance structure design is the most important component that has to be analyzed with FEM instruments; the structure comprises the whole mechanical system having a strictly loads absorption, ensure a specific meaning, together with the static and dynamic stability or to guarantee a nominal stiffness established by the designer in his specifications [1], [2].
For a specific product, the engineers should always consider constraints like: the number and intensity of the static loads as well as dynamic ones, the maximum strains values, different safety factors (for bucking, breaking or fatigue), the susceptibility at execution, mounting or errors appeared in operation, frequency characteristics, steady afterflow speed, product cycle of life, weight, material and moment of inertia, different loads stiffness, static and dynamic stability or simultaneous loads response [1].
The FEM analysis of a structure is in fact a numeric computation review. Hence, for a specific geometrical model, particular loads and constraints will result the required values of the deformation, stress, bearing reaction and natural frequency [1], [2].
In this FEM simulation it is considered a finishing end mill with the diameter D_c = 12 mm, number of cutting edges z_c = 4, pitch of the helical cutting edge P_{hc} = 37.4 mm, helix angle ω = 45^0. On the contact line between the mill and the part there are created four cutting spots placed on the height of the helix pitch. Two of the spots corresponding to the cutting depth (a_p = 20 mm) are shown in figure 11.

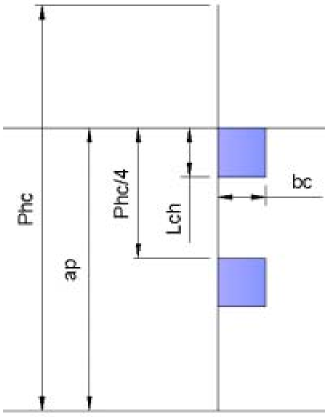

Fig. 11 – Positioning of the cutting spots

In order to establish these spots where the tangential medium cutting force is applied it was necessary to determine the contact length of each helical edge with the part. The dimensions of these spots and also the contact angle φ between the part and the cutting edge are calculated with the following equations:

$$b_c = \sqrt{\frac{D_c^2}{4} - \left(\frac{D_c}{2} - a_e\right)^2} = 3.32\text{ , [mm]} \quad (2)$$

$$L_{ch} = \frac{\sqrt{D_c \cdot a_e}}{tg\varpi} = 3.46\text{ , [mm]} \quad (3)$$

$$\varphi = 2 \cdot \sqrt{\frac{a_e}{D_c}} = 0.577\text{ , [rad]} \quad (4)$$

The medium cutting force acting on a contact spot for the contact angle φ is determined by the relation:

$$F_{m\varphi} = L_{ch} \cdot h_m \cdot k_c\text{ , [N]} \quad (5)$$

With the previously determined values for cutting power (aluminium alloy and high-alloyed steel) it is calculated the tangential medium cutting force:

$$F_{tm} = \frac{60000 \cdot P_c}{v_c}\text{ , [N]} \quad (6)$$

and has the values: 360 N, respectively, 1098 N.

The medium radial cutting force (which is practically applied on the two spots) is F_{rm} = (0.3 … 1) · F_{tm}, considered $F_{rm} = 0.45 \cdot F_{tm}$.

As follows, there are presented some results of FEM simulations and analysis in the cases of the part being processed of aluminium alloy and high-alloyed steel. Thus, the medium radial cutting force is different for each case (F_{rm} = 164 N – aluminium alloy and F_{rm} = 536 N – high alloyed steel).

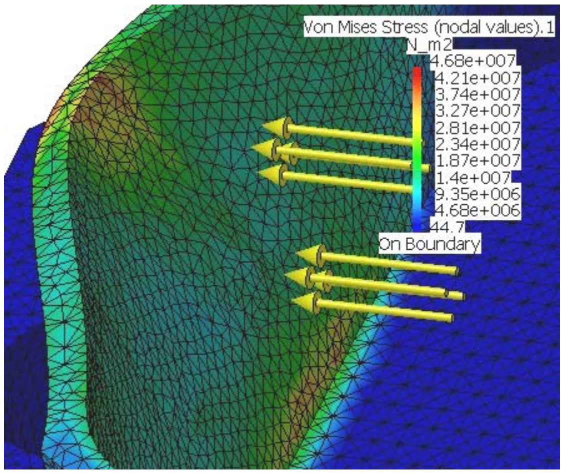

Fig. 12 – Example of Von Mises Stresses for the flat milled surface S1

Figure 12 shows the Von Mises Stresses calculated by the FEM analysis in the case of aluminium alloy and flat surface. For a force of 164 N applied on the two spots (Fig. 11) the results are: max. stress = 4.68×10^7 N/m^2 and max. elastic deformation = 0.062 mm (Fig. 13) with an error of 48.8 %.

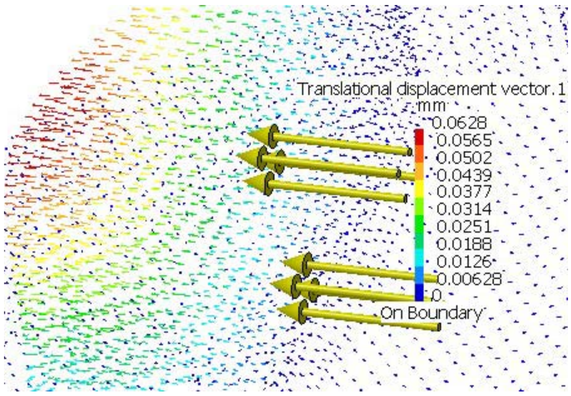

Fig. 13 – Example of deformation for the flat milled surface S1

That error is too high, so a new analysis is done after a refinement of the part structure. Although CATIA software defines a default net of nodes in the process named digitization, it is highly recommended for experienced users to edit this net and establish the dimension of the finite element, the maximum deviation of the virtual digitized model from the real model and the type of the finite element.

After the refinement, the error is 32.7 % and the values become: max. stress = 6.22×10^7 N/m^2 and max. elastic deformation = 0.083 mm.

In the high-alloyed steel case (flat surface S1, F_{rm} = 536 N), for an error of 48.8 % the max. stress = 1.53×10^8 N/m^2 and max. elastic deformation = 0.07 mm. Also, for an error of 31 % the values are: max. stress = 2×10^8 N/m^2, max. elastic deformation = 0.09 mm.

In the FEM practice, an error of 20% - 35% is acceptable and it's very close to the real case. Anyway, in both situations for the flat surface S1, the max. stresses are lower than the materials' yield strengths, so the surface is deformed only in the elastic domain.

On the convex surface S2 it is applied the same force for each considered material. In the aluminium alloy case, for an error of 38.3 % the max. stress = 3.47×10^7 N/m^2 and the max. elastic deformation = 0.014 mm.

After the refinement, the error is 25.5 % and the values become: max. stress = 5.35×10^7 N/m^2 and max. elastic deformation = 0.0175 mm.

Figure 14 shows an example of the Von Mises Stresses determination in the high-alloyed steel case. Thus, for an error of 38.13 % the max. stress = 1.12×10^8 N/m^2 and the max. elastic deformation = 0.016 mm (Fig. 15).

After the refinement is applied, the error is 25.3 % and the values are: max. stress = 1.34×10^8 N/m^2 and max. elastic deformation = 0.02 mm.

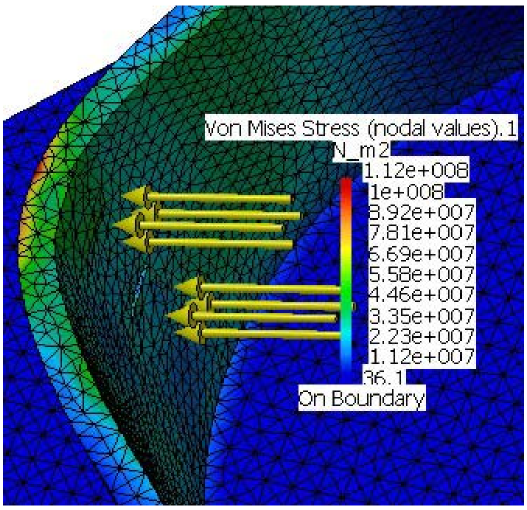

Fig. 14 – Example of Von Mises Stresses for the convex milled surface S2

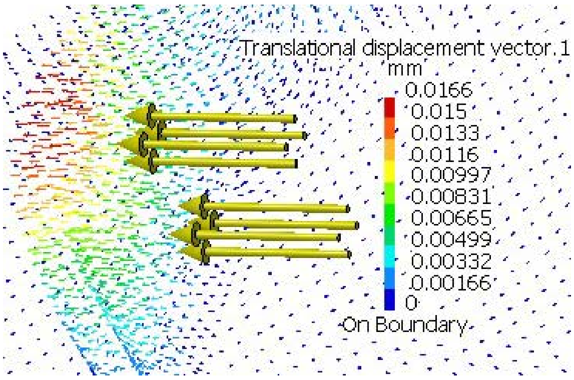

Fig. 15 – Example of deformation for the convex milled surface S2

The last analysis simulations consider that the radial cutting force is applied on the concave surface S3. In the aluminium alloy case, for an error of 34.4 % the max. stress = 3.39×10^7 N/m^2 (Fig. 16) and the max. elastic deformation = 0.0214 mm (Fig. 17).

After the refinement, the error is 22.3 % and the values become: max. stress = 4.28×10^7 N/m^2 and max. elastic deformation = 0.0245 mm.

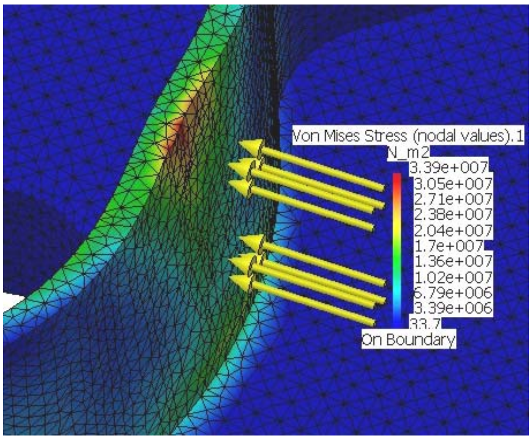

Fig. 16 – Example of Von Mises Stresses for the concave milled surface S3

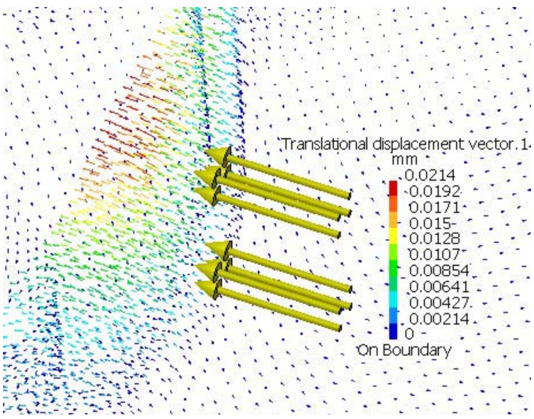

Fig. 17 – Example of deformation for the concave milled surface S3

In the high-alloyed steel case, for an error of 35.1 % the max. stress = 1.11×10^8 N/m^2 and the max. elastic deformation = 0.024 mm. After the refinement is applied, the error is 22.2 % and the values are: max. stress = 1.47×10^8 N/m^2 and max. elastic deformation = 0.027 mm.

For an increased precision and higher improved model for each simulation, the user may continue an iterative analysis and to refine the model to decrease the error. Such errors like those obtained in the previous simulations seem to be high, but they indicate in fact all the differences between the proposed virtual model and the real structure.

The user is interested in the outputs of the maximum stress that must not exceed the admissible material yield strengths.

4.Conclusions

The determination of the cutting forces with which the cutting edges action on the processed surface is made using the values of the cutting power, of the chip thickness and of the specific cutting force.

The radial cutting force is distributed on the contact spots between the cutting edges and the part. These spots have calculated dimensions corresponding to the cutting parameters. The simulations' results show that the stresses inducted in the part's thin walls not exceed the elastic domain. The maximum elastic deformations of the processed surfaces are greater in the case of the high-alloyed steel than in the case of the aluminium alloy.

It can be distinguished the dependence of the stresses and of the deformations with the shape of the processed surface. The elastic deformations are greater in the case of the flat surfaces than in the case of the convex or concave surfaces. Also, these deformations corresponding to the considered working regime (max. 0.09 mm) seem to be unacceptable, influencing the dimensions of the processed surfaces. If the aim is a higher precision, the parameters of the working regime should be decreased and the process simulation resumed.

5.References

1- Constantinescu, N. I., Sorohan, Şt., Pastramă, Şt., "The practice of finite element modeling and analysis". Editura PRINTECH, Bucureşti, ISBN 978-973-718-511-2, 2006.
2- Ghionea, I., "CATIA v5. Aplicații în inginerie mecanică". Editura BREN, Bucureşti, ISBN 978-973-648-843-6, 2009.
3- Kief, B. H., "NC/CNC Handbuch". Carl Hanser Verlag Wien, ISBN 3-446-21756-8, 2001/02.
4- Trent, E., Wrigt, P., "Metal cutting". Butterworth Hinemann, Oxford, Boston, 2000.
5- ***, "Cutting tools from Sandvik Coromant. Main catalogue. Turning. Milling. Drilling. Boring. Toolholding", C-2900: 6 ENG/01. Sweden, www.coromant.sandvik.com, 2008.