The 4th International Conference
″Computational Mechanics
and Virtual Engineering″
COMEC 2011
20-22 OCTOBER 2011, Brasov, Romania

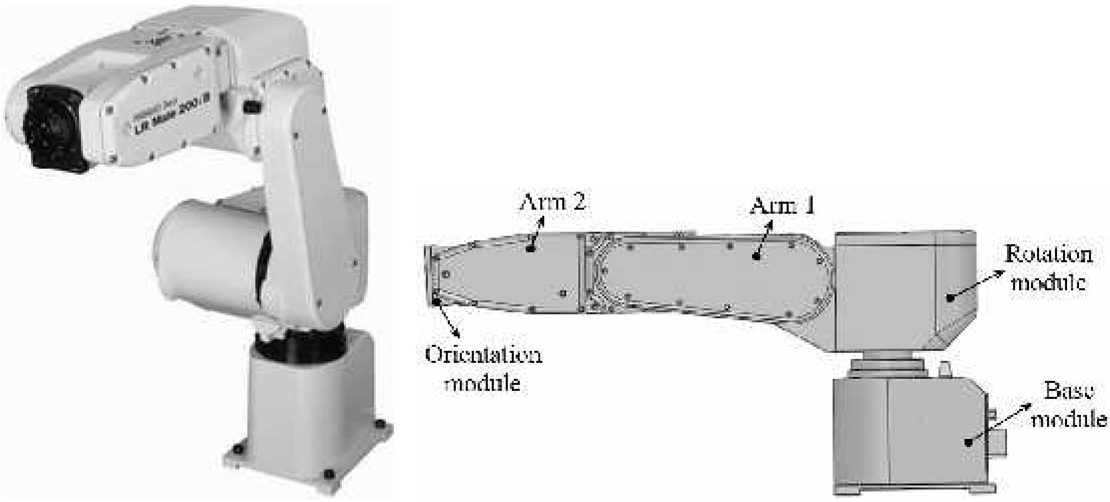
NUMERICAL STUDY OF DISPLACEMENTS AND STRESSES OF AN INDUSTRIAL SERIAL ROBOT

Mihaela Simion[1], Adrian Ioan Botean[2], Mircea Bejan[3]
[1] Technical University of Cluj-Napoca, Cluj-Napoca, ROMANIA, mihap_elly@yahoo.com
[2] Technical University of Cluj-Napoca, Cluj-Napoca, ROMANIA, Adrian.Ioan.Botean@rezi.utcluj.ro
[3] Technical University of Cluj-Napoca, ROMANIA, Mircea.Bejan@rezi.utcluj.ro

Abstract. *In this paper the authors present the study of displacements and stresses for the mechanical structure of an industrial serial robot using finite element method. This numerical study is done when the robot is solicited by its own weigh and by the maximum weigh which the robot can manipulate.*
Key words: *finite element, robot, solicited, displacements, stresses.*

1. INTRODUCTION

The purpose of this paper is to determine analytical the efforts and the state of deformation and stresses using Finite Element method for the industrial serial robot Fanuc LR Mate 100iB.
The industrial serial robot Fanuc LR Mate 100iB is constituted by five rotation modules, each module having one degree of freedom. The driving motors of the robot are electric and the maximum handling load is 5 kg.
In figure 1 a) is presented the real model of the robot and in figure 1 b) is shown the CAD model of the robot, realized using SolidWorks software, and also its components.



a) b)
Figure 1: The industrial serial robot Fanuc LR Mate 100iB: a) real model; b) CAD model

2. ANALITIC CALCULUS OF THE EFFORTS

To perform the analytic study of efforts has chosen a particularly working position of the robot, namely when the arms of the robot are extended to maximum (figure 1b).

Based on CAD modeling has built a simplified scheme (static determined framework) of robot's structure (figure 2), when has considered the following assumptions:

- The own weight of each module from robot's structure are represented by uniformly distributed loads $q_i, i = 1 \div 6$, respectively by concentrated force F_1;
- $l_i, i = 1 \div 7$ represent the distances between gravity center of driven motors of the robot's modulus;
- F_3 represent the maximum handling load ($F_3 = 50\ N$);
- The static determined framework is fixed to the robot's base (point 1 from figure 2).

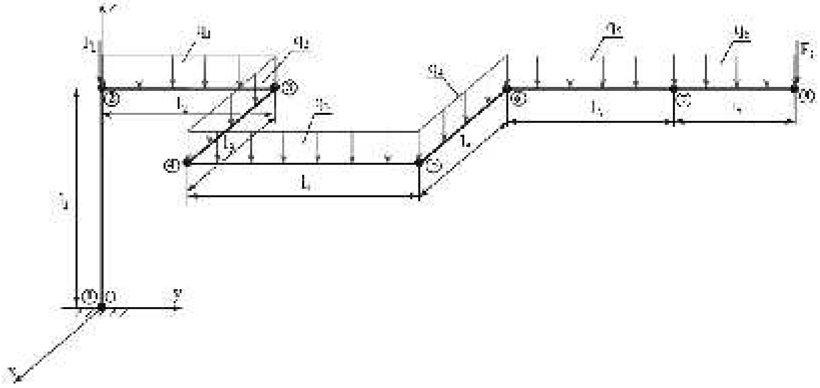


Figure 2: Simplified scheme of the structure of the robot

For writing the analytic relation of efforts on each interval of the simplified scheme, has used the usual relations from literature (these analytical relations are shown in table 1), and also has given the input dates:

$l_1 = 216\ mm$ $\quad q_1 = 0{,}68\ N/mm$ $\quad F_1 = 90\ N$

$l_2 = 145\ mm$ $\quad q_2 = q_4 = 0{,}74\ N/mm$ $\quad F_2 = 75\ N$

$l_3 = l_5 = 134\ mm$ $\quad q_5 = 0{,}99\ N/mm$ $\quad F_3 = 50\ N$

$l_4 = 250\ mm$ $\quad q_6 = 0{,}14\ N/mm$

$l_6 = 101\ mm$

$l_7 = 104\ mm$

In figure 3 has represented the effort diagrams (axial, shears, bending and torsion momentums), and also on these diagrams are given the analytic values obtained for the studied working position of the robot.

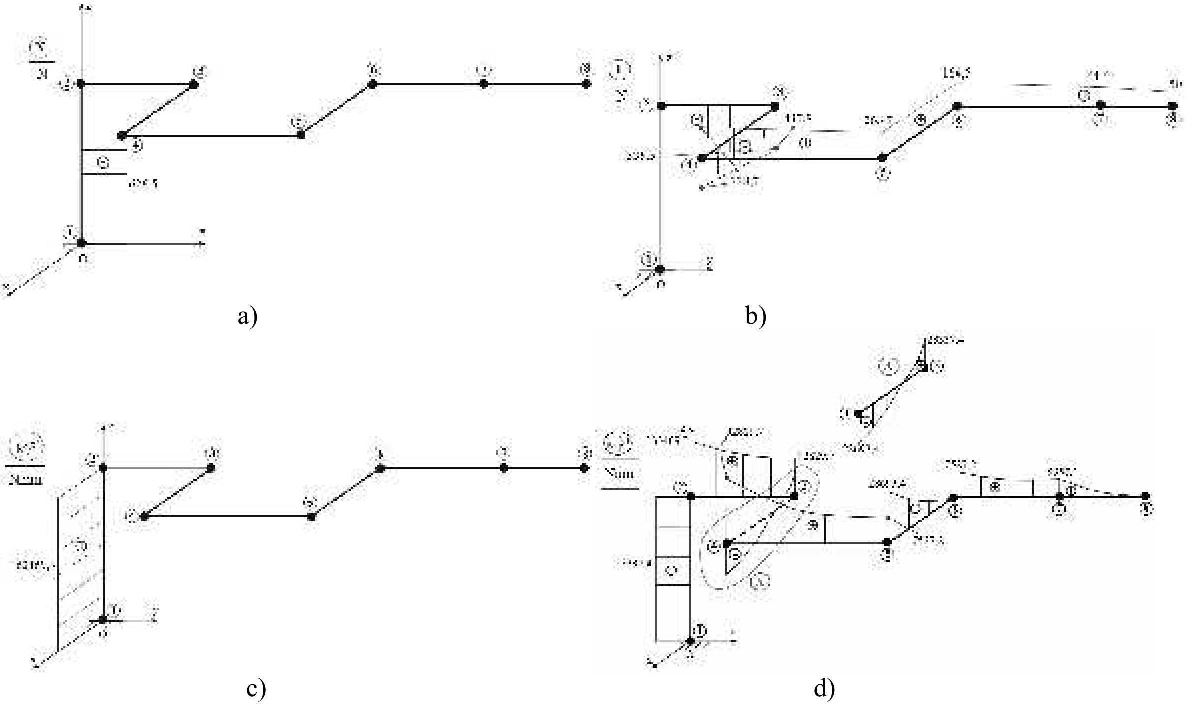
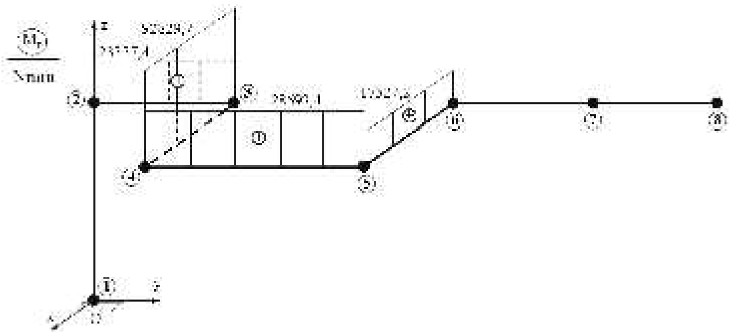


Figure 3: Effort diagrams for the particularly studied position: a), b) axial and shear efforts; c), d) bending moments after *x* and *y* axis



e)

Figure 3: Effort diagrams for the particularly studied position: e) torsion moments (continuation)

Table 1: Analytic equations of efforts

Nr. crt.	Reactions: V_1 N	Reactions: M_x Nmm	Reactions: M_y Nmm	Interval	N N	T N	$M_i^{(x)}$ Nmm	$M_i^{(y)}$ Nmm	M_t Nmm
1				1-2	$N_{1-2} = -V_1$	0	$M_{1-2}^{(x)} = M_x$	$M_{1-2}^{(y)} = M_y$	0
2	$\sum Y = 0 \Rightarrow V_1 - F_1 - q_1 l_1 - q_2 l_2 - q_3 l_3 - q_4 l_4 - q_5 l_5 - q_6 l_6 - F_3 = 0$	$\sum M_x = 0 \Rightarrow q_1 \frac{l_1^2}{2} + q_2 l_2 l_1 + q_3 l_3 \left(\frac{l_3}{2} + l_2\right) + q_4 l_4 (l_3 - l_2) + q_5 l_5 \left(\frac{l_5}{2} + l_3 + l_2\right) + q_6 l_6 \left(\frac{l_6}{2} - l_2 + l_4 + l_5\right) + F_3 (l_5 + l_6 + l_4 + l_3) = 0$ [illegible]	$\sum M_y = 0 \Rightarrow q_3 \frac{l_3^2}{2} + q_4 \frac{l_4^2}{2} - q_5 l_5 l_4 = 0$	2-3	0	$T_{2-3} = F_1 - V_1 + q_1 x$	0	$M_{2-3}^{(y)} = M_y - V_1 x + q_1 \frac{x^2}{2}$	$M_{t_{2-3}} = -M_y$
3				3-4	0	$T_{3-4} = F_1 - V_1 + q_1 l_1 x + q_2 x$	0	$M_{3-4}^{(y)} = M_y + F_1 x - V_1 x + q_1 l_1 + q_2 \frac{x^2}{2}$	$M_{t_{3-4}} = -F_3 (l_1 + l_3)$ [illegible] $q_5 l_5 \left(l_1 + l_5 \frac{l_5}{2}\right)$ $- q_4 l_4 \left(l_3 + \frac{l_4}{2}\right) - q_4 \frac{l_4^2}{2} - q_6 l_6 l_4$
4				5-4	0	$T_{5-4} = -F_3 - q_6 l_6 - q_5 l_5 - q_4 l_4 - q_3 x$	$M_{5-4}^{(x)} = F_3 (l_5 + l_6) +$ [illegible] $q_5 l_5 \left(l_6 + \frac{l_5}{2}\right)$ [illegible] $q_6 \frac{l_6^2}{2}$	[illegible]	0
5				6-5	0	$T_{6-5} = -F_3 + q_6 l_6 + q_5 x$	0	$M_{6-5}^{(y)} = -F_3 x + q_6 l_6 x + q_5 l_5 x + q_5 \frac{x^2}{2}$	$M_{t_{6-5}} = F_3 (l_5 + l_6) +$ [illegible] $q_5 l_5 \left(l_6 + \frac{l_5}{2}\right) + q_6 \frac{l_6^2}{2}$
6				7-6	0	$T_{7-6} = -F_3 + q_6 l_6 + q_7 x$	0	[illegible]	0
7				8-7	0	$T_{8-7} = -F_3 + q_8 x$	0	$M_{8-7}^{(y)} = -F_3 x + q_8 \frac{x^2}{2}$	0

2. STUDY OF DISPLACEMENTS AND STRESSES USING FEM

To determine the displacements and stresses on entire strength structure of the robot, has realized a statically numerical study with Finite Element Method (FEM), using Ansys 12.1 software (Static Structural).
The numerical study was realized based on CAD model of the industrial serial robot Fanuc LR Mate 100iB shown in figure 1b, and the material which it is made the strength structure is iron alloy, having the following mechanical characteristics:

- Density: $\rho = 7200\ kg/m^3$;
- Young's modulus: $E = 1{,}1 \cdot 10^5\ \ N/mm^2$;
- Poisson ratio: $\upsilon = 0{,}28$;
- Tensile ultimate strength: $\sigma_{r_i} = 280\ \ N/mm^2$.

To perform the numerical study the following simplified hypothesis and constrains was considered:

- were not taken in study gears and bearings, and also the screws;
- the covers and their corresponding modules of the robot was considered a single body;
- the robot was fixed in the feet from the robot's base;
- the robot was loaded with forces and/or moments corresponding to nodes 2, 3, 6, 7 and 8, representing the gravity center of driving motors plus the maximum handling force F_3 = 50 N;

- the mesh of the structure was done automatically, choosing the dimension of finite element 30 mm, resulting 170.241 elements and 288.561 nodes.

To realize the numerical study its starts from the base of the robot, adding one module at a time, finally obtaining the global distribution of displacements and stresses. This approach allows observing the behavior of the structure under loading, and also the evolution of distribution of displacements and stresses, from the base module to orientation module.

In figures 4 and 5 are presented the distribution of total displacements and von Misses stresses obtained with FEM for the particular working position taken into study, and for a better visualization are given details in the areas where the equivalent stresses are maximum.

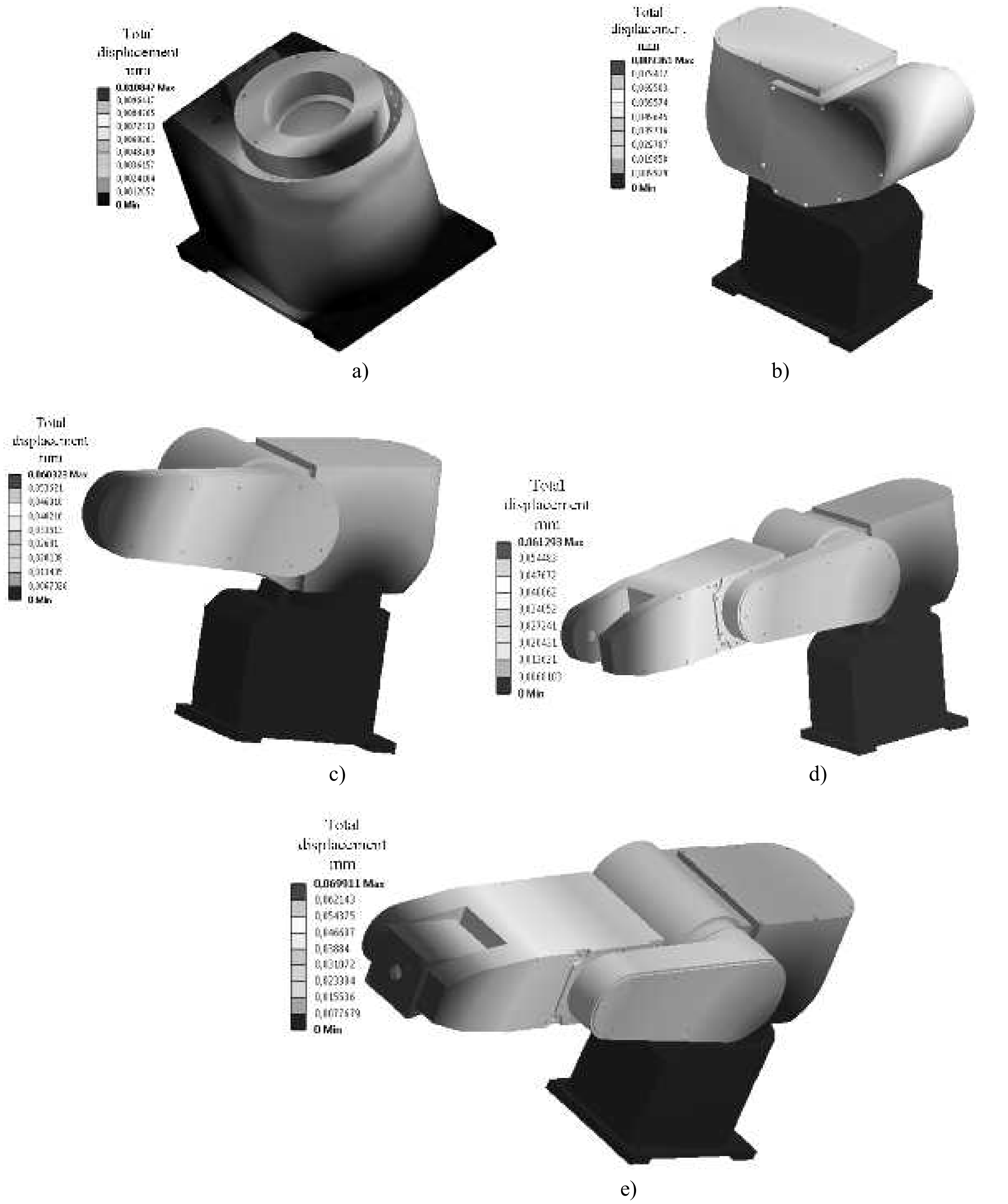


Figure 4: Distribution of displacements of the robot's strength structure with FEM

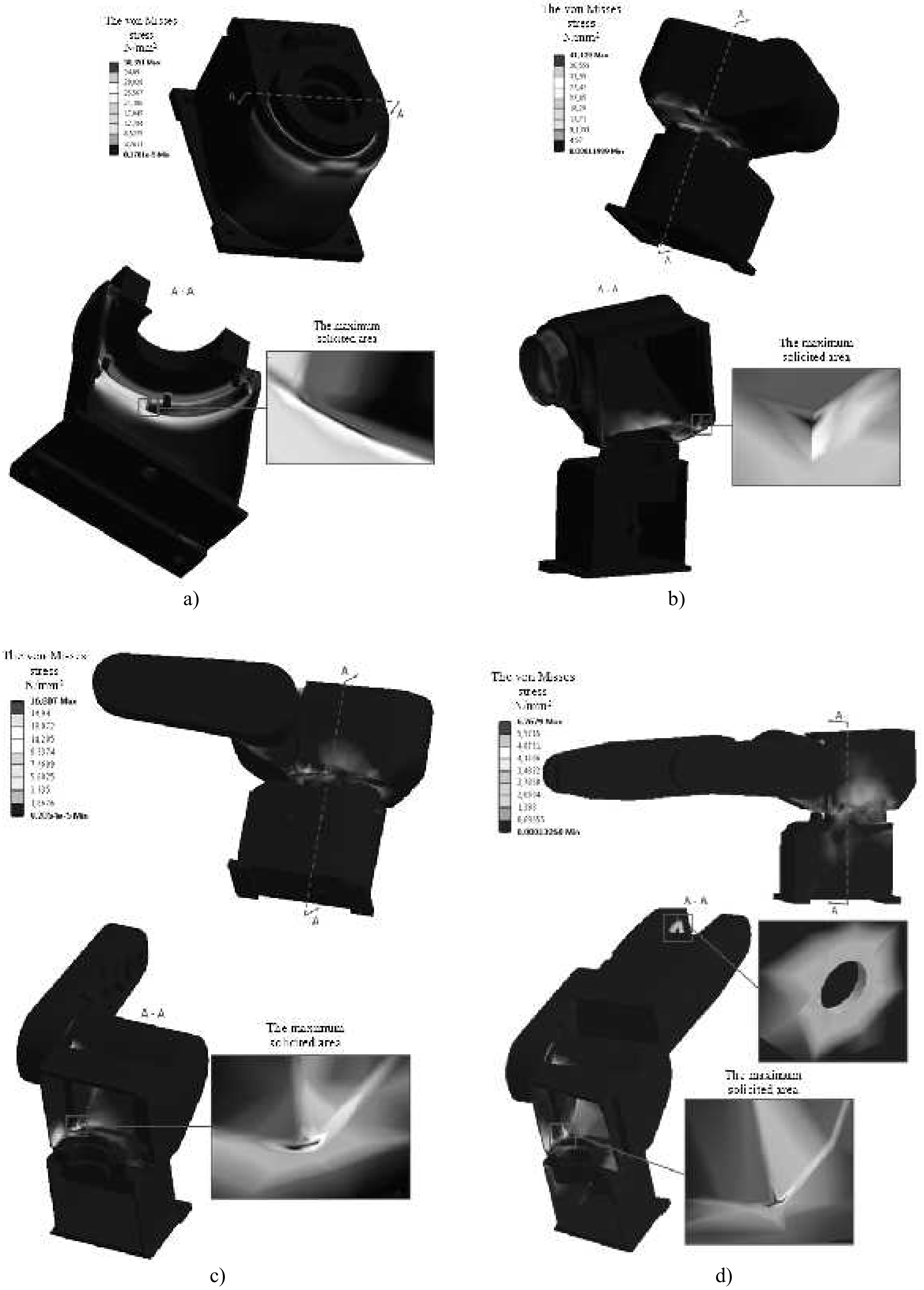
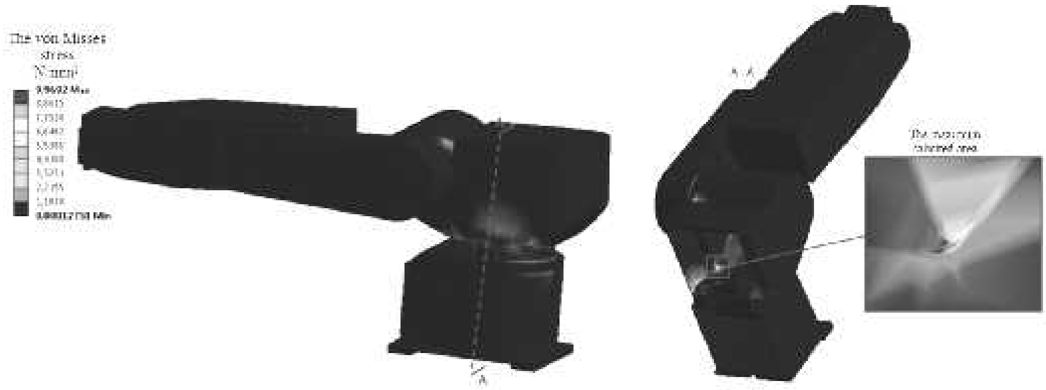


Figure 5: Distribution of stresses of the robot’s strength structure obtained with FEM



e)

Figure 5: Distribution of stresses of the robot's strength structure obtained with FEM (continuation)

4. CONCLUSION

In this paper has presented:

- an analytic study to determine the efforts and, then has build the diagrams of efforts for the strength structure of the industrial serial robot Fanuc LR Mate 100iB;
- the numerical study to determine the state of displacements and stresses of the robot's strength structure;
- for the numerical study has considered an particular working position of the robot.

Having in view the analytical and numerical study it can be concluded:

- observing the diagrams of efforts, the highest values has obtained for the bending moments after x axis (interval 1-2) and after y axis (interval 2-3), which corresponds to the base module, respectively to rotation module (figure 3c and d);
- from numerical study, the highest value of displacement has obtained in the case of rotation module (figure 4b), but the interested value is that at the free end of the strength structure, when the robot is loaded with the maximum handling force (figure 4 e);
- the robot being designed to work to height accuracy, the value of displacement obtained numerically at the clamping device area can affect the working accuracy of the robot, like in case of studied position;
- the maximum values of von Misses stresses has obtained to the rotation module (figure 5b), but the von Misses stress corresponding to entire structure (figure 5e) is quiet reduced, so it will not have a great influence on the strength structure of the robot. Equivalent von Misses stress values are influenced by the CAD model of the structure and construction of various elements such as beams connection etc.

Further numerical studies (static and dynamic studies) and experiments will be made in order to obtain more exactly data about the state of deformations and stresses for the Fanuc LR Mate 100iB robot.

ACKNOWLEDGMENT: This paper was supported by the project "Doctoral studies in engineering sciences for developing the knowledge based society - SIDOC" contract no. POSDRU/88/1.5/S/60078, project co-funded from European Social Fund through Sectorial Operational Program Human Resources 2007-2013.

BIBLIOGRAFIE

[1] Bejan, M., Rezistenţa materialelor, Vol. 1 şi 2, Ediţia a IV-a, Ed. AGIR, Bucureşti, Ed. MEGA, Cluj-Napoca, 2009.

[2] Simion, Mihaela, Tensiuni şi deformaţii în structura de rezistenţă a unui robot industrial serial, Raport de Cercetare nr. 2 în cadrul doctoratului, Cluj-Napoca, 2011.

[3] Simion, Mihaela, Botean, A.I., Bejan, M., Analityc study of efforts from the structure of an industrial serial robot 5R, Acta Tehnica Napocensis, Series: Applied Mathematics and Mechanics, vol. 54, Issue 1, ISSN 1221-5872, pag. 225-228, Cluj-Napoca, 2011.

[4] Simion, Mihaela, Botean, A.I., Bejan, M., Studiul analitic şi numeric al deplasărilor pentru o grindă articulată, Ştiinţă şi Inginerie, vol. 19, ISSN 2067 – 7138, pag. 657-666, Sebeş, 2011.

[5] tutoriale ANSYS 12.1

[6] *** www.fanucrobotics.com