U.P.B. Sci. Bull., Series D, Vol. 72, Iss. 4, 2010 ISSN 1454-2358

OPTIMIZATION DESIGN FOR THE STRUCTURE OF AN RRR TYPE INDUSTRIAL ROBOT

Adrian GHIORGHE[1]

Articolul de față prezintă o metodologie de determinare a valorilor optime pentru variabilele de proiectare considerând criteriul reducerii de material din structura robotului industrial prin aplicarea un algoritm de optimizare structurală şi topologică. Această analiză bazată pe metoda elementelor finite constă în realizarea modelului parametrizat considerând mărimile geometrice ca variabile de proiectare pentru care se impun condiții restrictive în vederea construirii funcției obiectiv. Prin rularea recursivă a analizei MEF, variind valorile variabilelor de proiectare se evaluează variabilele de proiectare în vederea determinării unei configurații optimizate din punct de vedere al funcției obiectiv.

The current paper shows a methodology to determine the optimum values for the design parameters considering the criteria of reducing the material used to build the structure of industrial robot, using an structural optimization and topology algorithm. This analysis is based on the finite element method(FEM) and consists in completion of the design model using the dimensional data as parametri design variables to whom restriction conditions have been applied in order to achieve the object function. A recurrent FEM analysis using different parameters for the design variables was applied in order to assess an optimum composition of the object function.

Keywords: optimization, FEM analysis, topology, robot, structure

1. Introduction

The techniques of analysis and simulation of mechanical systems using the **finite element method** allows researchers and mechanical engineers to build mathematical models and to analyze the static and dynamic behavior of the structural elements directly on the computer, and optimization calculations, simulations, studies of similarity, etc.

Approximate numerical solutions, obtained through modeling for the proposed problems, have the following key advantages:

- it can be applied to bodies and real phenomena, regardless of their degree of complexity;
- are converge to solutions of proposed problems (results may be obtained with the desired accuracy;

[1] Eng., GENERAL TURBO – CNC Division, / e-mail:adrian.ghiorghe@gmail.com

- you can view the pictures, charts, graphs - intuitive and more diverse than in the case of exact solutions;
- allows to obtain a solution in a reasonable time;
- are economically advantageous.

It is a work environment that integrates design and analysis, in order to achieve optimized products, shortening manufacturing cycle.

It has the capability to create optimized versions of the products, reducing the need to manufacture a prototype to validate the results.

The challenge that robot manufacturing industry must face today is the speed of response to market demands. The key is to produce efficiently, at a low price and high quality in record time. Specialized companies provide information technology solutions that helps companies in various areas to reduce time from manufacturing to sales while reducing the costs dramatically.

Specialists allow customers to eliminate real prototypes and laboratory tests in the design phase, while they check and improve their processes and products.

Optimized structural design for the structures of the industrial robots have to meet certain criteria regarding dimensional design and shape, material consumption and adapt this to the functional requirements. For an optimized design of the robot structure the engineers normally consider all the aspects of industrial applications where the structure will be integrated. Specific requirements are related to the resistance of the elements, not to oversize the structure but also to guarantee minimum criteria of stability and security in operation and to fit the material and its shape with the above mentioned criteria. It is required to correlate all these with the kinematic model of the joints and from this basis to establish the loads and to build a dynamic model to determine the behaviour from this point of view.

The actual progress in the field of topological optimization – [1], [2], [3], [4], [5] is determined by the software progress in optimization on the basis of FEM, using the new approach that reaches a high accuracy of the results obtained from a system of equations, whose size depends on the complexity of the model examined and whose solution require an efficient numerical method to calculate.

The research indicates also as a major possibility to improve the characteristics of the structures, the use of new materials, while the modern methods for optimized design of a robot structure involves the use of CAD programs and finite element analysis (CAD-FEM). Different material databases are available for assessment within the virtual model. This kind of verification of the components and complete structure of robots becomes compulsory requisite in modern design as well as in optimization program.

On the other side, the theoretical models – [6], [7], [8] were built from many years as well as the less sophisticated FEM software based approaches – [9]. Currently, new software allows to develop methods of reporting issues into a computer model structure from which other information can be obtained also. The most important are static and dynamic analysis. The current paper describes a methodology to improve the researches in the above mentioned field.

2. Shape optimization techniques

Structural elements and assembled structures of the robot modelled with computer based technologies have as a first step an approximation of the physical structure. From several interactions based on experimental research compared with the virtual model, an accurate virtual structure can be obtained.

The theoretical and experimental research revealed that the main design variables are optimized structures of new materials to replace traditional materials characterized by poor physical and mechanical properties used for recursive modelling structure and optimization of the structure using algorithms and programs. To improve the static and dynamic behaviour of the industrial robot structure the following requirements must be accomplished:

- Minimum weight structure;
- Maximum static stiffness of structural elements;
- Minimum deformation – maximum precision at the end-effector;
- Highest possible resonance frequency out of the frequency range in the working environment;
- Minimum moments of inertia, with direct outcome in end-effector dynamics.

The stages of the FEM analysis – Fig. 1 for a structure of articulated robot includes the complete definition of the elements to mesh as well as all the the physical-mechanical properties of the material elements of the structure. Then, the conditions of restrictions imposed on them, due to actuators for parts by: placing restrictions on all nodes that are presumed to have contact between the leader and the led, the calculation of stiffness matrix for the structure, the placement static loads as forces and moments on the three axes, the nodes that define the links between the parts.

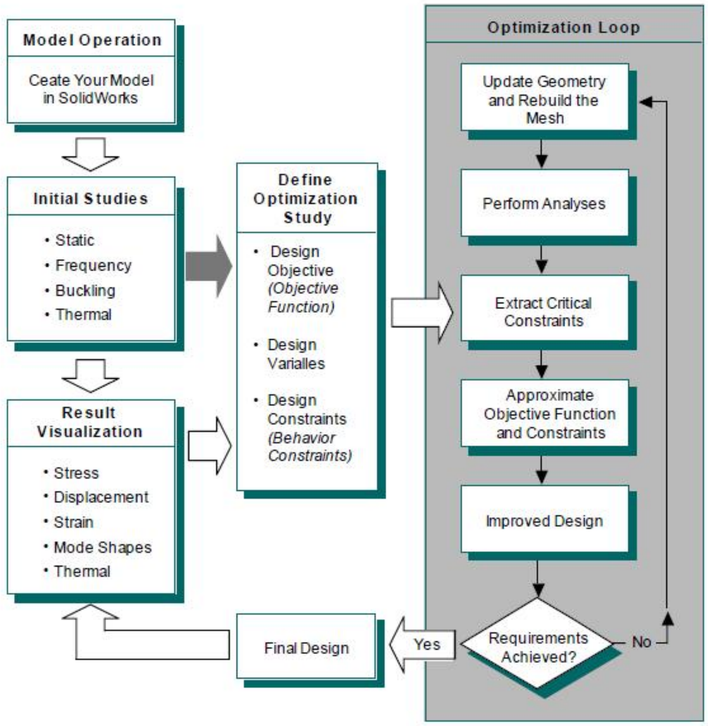

Fig. 1. Structural optimization chart

The static analysis determines the load and stress that arise within the structure, along the axes and an equivalent stress. The location where the major stress occurs gives important indication about the structure behaviour. This analysis determines the structure behaviour and main loads (torsion or bending) leading to conclusions relating the improvements to be considered (strengthening to be introduced).

The modal analysis by determining the natural frequency and own vibration eigenmodes, allows simulation of sub-assemblies for all the modes of vibration and general behaviour. This analysis gives information about the structure due to forces of gravity, isolated from the rest of the structure.

The structural optimization based on the previously mentioned static and modal analysis, that indicates when an upgrading such as rib or diaphragm is introduced, improves the structure behaviour. Here are shown: deformation, higher amplitudes of the own vibration eigenmodes, dynamic analysis, static loading, distribution of forces in multiple nodes, etc. The calculation of FEM is based on the classic equation – (1):

$$\{P\}^{(e)} = [k^{(e)}]\{\delta\}^{(e)} + \{F_{\varepsilon 0}\}^{(e)} + \{F_F\}^{(e)} \tag{1}$$

Several variants of robotic arms are proposed by placing ribs or diaphragms to decrease the weight by increasing structural stiffness of the elements regarding the experimental model of the robot.

Basically, the strain obtained experimentally is recorded and compared with the one given by static analysis. Then, the cause of the differences between experimental and theoretical model are examined and proposals for reducing this and improve the structure are mode. By reducing the robot structure weight while static rigidity is increasing, eigenfrequency values are improved.

Shape optimization, is obtained by optimizing opposite dimensional structure involving the variation of the limits, so the range of the structural analysis is changing. The structural shape optimization requires a considerable computing time, recurrent operations and user experience to solve a local problem that can improve certain aspects of structural behaviour.

Optimization of topological elements of an articulated structure of industrial robots is a fundamental consideration in order to optimize the structure components. Topological optimization can be applied to a single case of load or multiple loads, simultaneously applied[10]. For a number of different load cases, the weight function is defined (2), where w_i is the weight structure for the case of structural load energy compliant.

$$F(U_c^1, U_c^2, ..., U_c^i) = \sum_{i=1}^{k} w_i U_c^i \tag{2}$$

Topology optimization using FEM led to the removal of unwanted elements structure, but the final forms depend on the initial density of mesh used for finite element analysis.

The goal is to find the best use of the material for a structure subjected to the action of one or more forces. From this perspective topological optimization for structure design elements allows to achieve a maximum stiffness.

3. Structure modelling

The first step of FEM is to build of a CAD model for the complete structure of the robot Fanuc LR Mate 100, considering the restrictions in the joints using 3D solid modelling software – SolidWorks 2008. In order to perform a static analysis as well as the modal analysis by determining its natural frequency and vibration modes, a FEM based software has been used – Ansys 10.0 – Fig. 2 and Fig. 3.

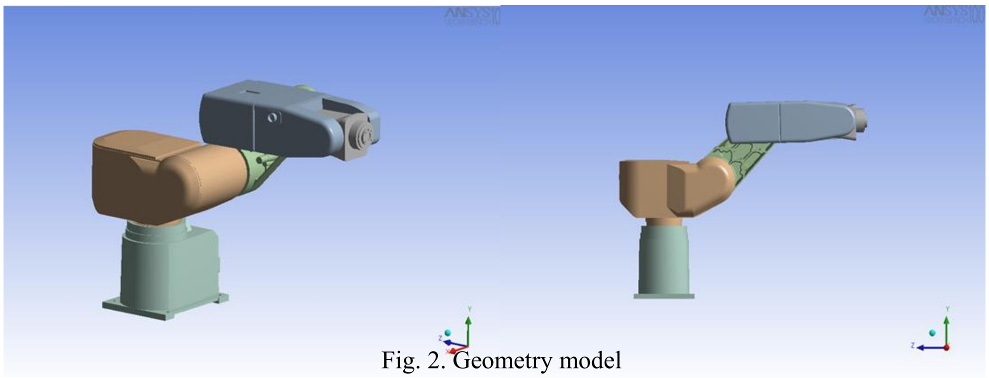

Fig. 2. Geometry model

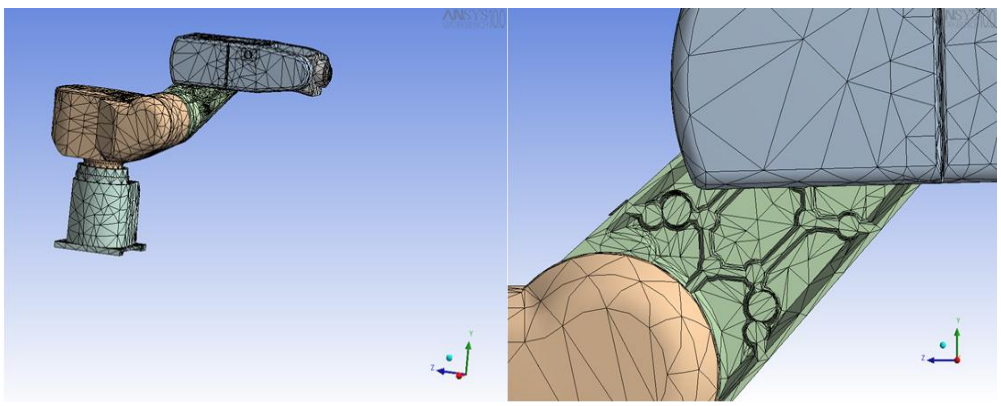

Fig. 3. The finite element mesh

For the FEM analysis of the robot structure, we considered as input data the following parameters: distributed loads Ox, Oy, Oz of 50kg ~ 490N gripper load and a temperature distribution in the working area of 50 ° C.

The structure was tested to a static analysis in order to obtain the total deformation and directions of principal stress and equivalent (von-Mises).

3. Results and interpretation

The results achieved in terms of maximum deformation on each axis, are presented in Tables below 1, 2 and 3.

Table 1

Total and axial deformations

Deformation	Total deformation	X axis deformation	Y axis deformation	Z axis deformation
Minimum (mm)	0	-0.068	-0.009	-0.323
Maximum (mm)	0.374	0.110	0.265	0.054

Table 2

Main deformation on loading conditions

Principal stress (10^8 Pa)	Minimum Principal stress	Middle Principal stress	Maximum Principal stress	Equivalent stress
Minimum (mm)	-9.894	-4.054	-3.413	0.691
Maximum (mm)	0.167	0.769	2.468	6.223

The final optimization step depends on the criteria to be improved, which sets the parameters for analysis. Analysis of parameters like deformation, stiffness or frequency must be considered in order to optimize the robot model.

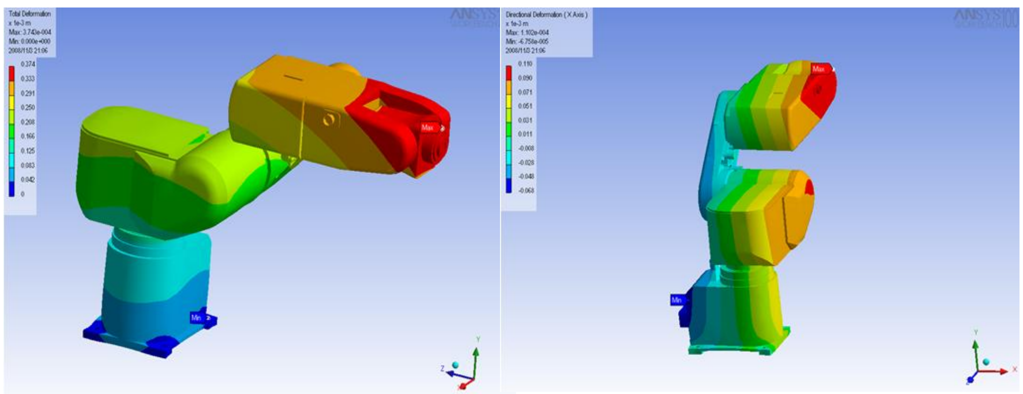

Fig. 4. Total deformation of the structure on Ox axis

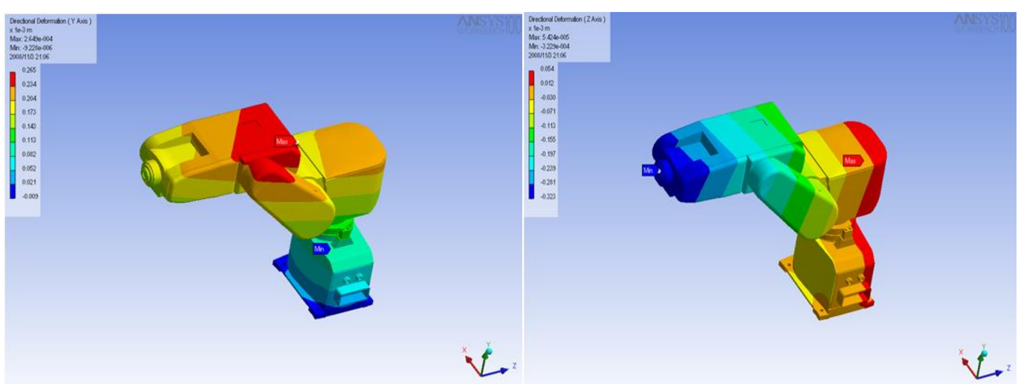

Fig. 5. Total deformation of the structure on Oy axis and Oz axis

The total deformation of the structure and deformation Ox axis is shown in Fig. 4 as well as the deformation corresponding on Oy axis and Oz axis– Fig. 5.

The main results from the static analysis includes: the maximum principal stress – Fig. 6, middle principal stress – Fig. 7, and the equivalent (von Mises) stress – Fig. 8.

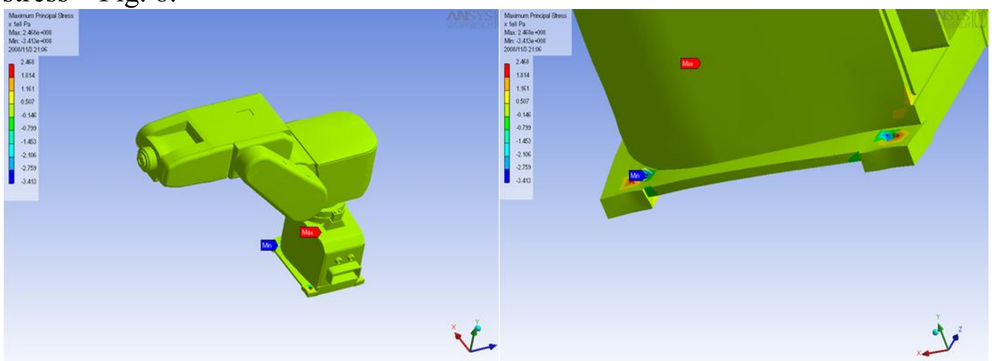

Fig. 6. Maximum principal stress results

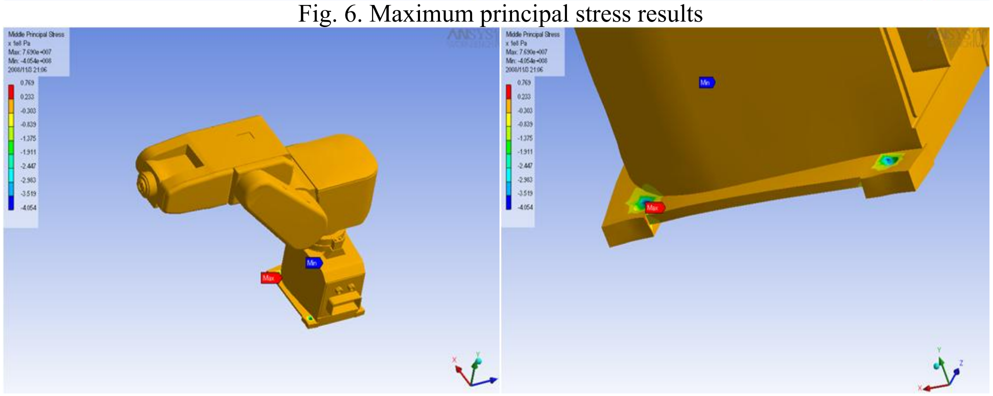

Fig. 7. Middle principal stress results

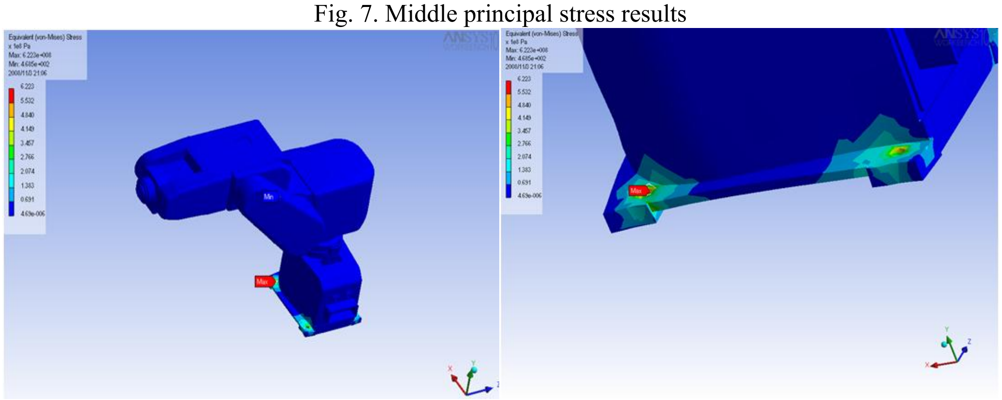

Fig. 8. Equivalent principal stress – von Mises

The second set of tests performed after static analysis is the modal analysis.
This analysis generated 4 frequency modes, as shown in Table 3 and the Figure 9.

Table 3

Frequency modes in range

Frequency Modes	1st	2nd	3rd	4th
Structure frequency (Hz)	51.57	58.95	153.24	187.65

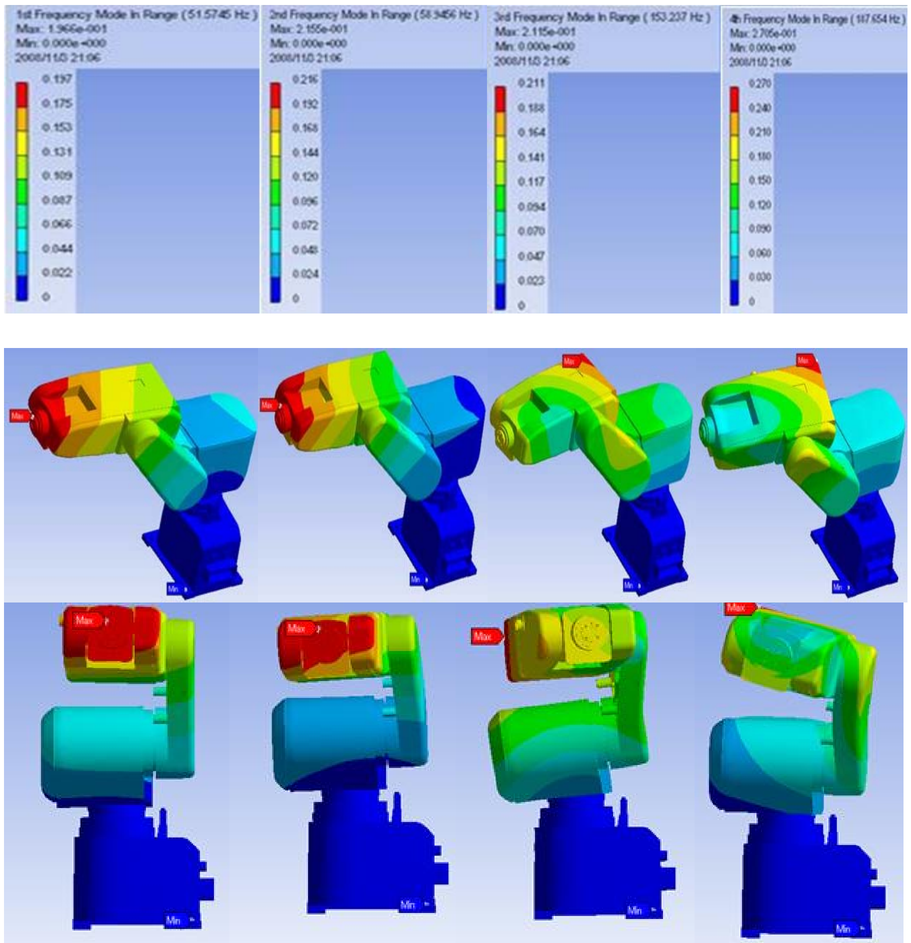

Fig. 9. Frequency modes in range

Using this analysis, in order to improve the characteristics, a part of the robotic arm structure was improved.

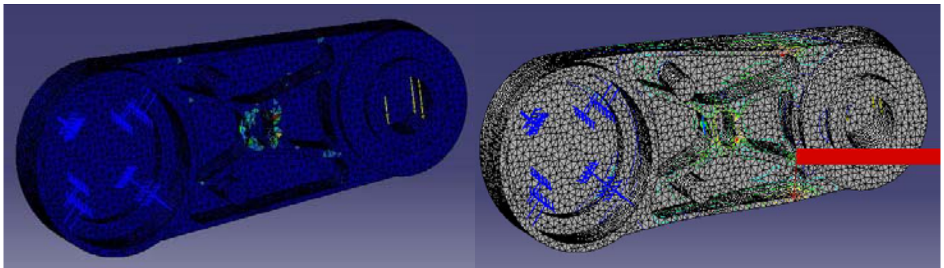

Fig. 10. Arm part structure isolation

The characteristics of this part are: material – aluminium alloy 7079 with Young's modulus $7.2*10^{10}$ N/m^2, Poisson's coefficient of 0.33, density of 2700 Kg/m^3, the thermal expansion coefficient of $2.5*10^{-5}$ and the allowable strength of $2.7*10^{10}$ N/m^2.

In order to optimize a structure element using static analysis, the robot arm has been isolated and a load of 490N has been applied on the main directions – Ox, Oy, Oz – Fig. 10. The meshed part has a mass of 3.55879 kg. with a solid standard mesh type having 63,152 elements, 100,181 nodes and feature size of 5.4833 mm. with a tolerance of 0.27416 mm.

A comparison between several shapes of the structure has been done. The first analysis comprises a possible shape – Fig. 10. For this, the results are mentioned in Table 4 – the equivalent von-Mises stress and Table 5 – the deformations along the three axis.

Table 4

Equivalent von Mises stress – first analysis

Stress	Minimum stress (N/m^2)	Position on axis (mm, mm, mm)	Maximum stress (N/m^2)	Position on axis (mm, mm, mm)
X axis stress	0.703244	(-10.6, -67.9, 0)	$3.97268*10^6$	(-42.5, 76.3, -64.1)
Y axis stress	5.49144	(43.3, -51.2, 0)	$1.62343*10^7$	(-1.9, 71.9, -24.1)
Z axis stress	6.27877	(43.3, -51.2, 0)	$1.63821*10^7$	(-1.9, 71.9, -24.1)

Table 5

Deformation over the three axis – first analysis

Deformation	Minimum stress (N/m^2)	Position on axis (mm, mm, mm)	Maximum stress (N/m^2)	Position on axis (mm, mm, mm)
X axis deformation	0 mm	(50.1, 6.1, -64.5)	0.0111476	(-2.7, 303.9, -24.1)
Y axis deformation	0 mm	(50.1, 6.1, -64.5)	0.0585494	(2.7, 303.9, -24.1)
Z axis deformation	0 mm	(50.1, 6.1, -64.5)	0.0601608	(2.7, 303.9, -24.1)

Figure 11 displays results of von-Mises stress, and the deformation, corresponding to the arm model calculation. The lowest value of the strees is at the bottom of the part while the maximum is at the top of it, and the dialog box contains explicit values. Light blue and dark blue colours indicates low stress, while the yellow to red indicates high tension.

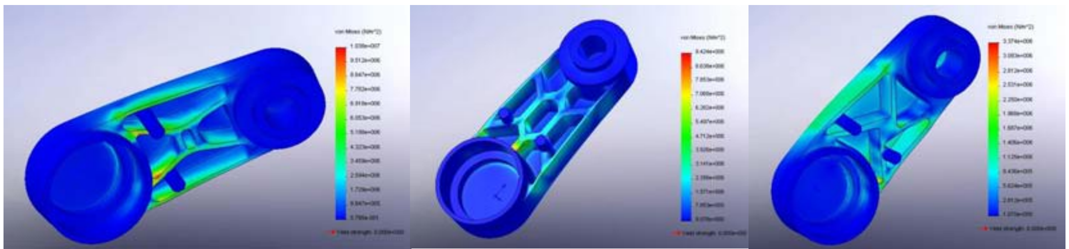

Fig. 11. Equivalent stress von-Mises

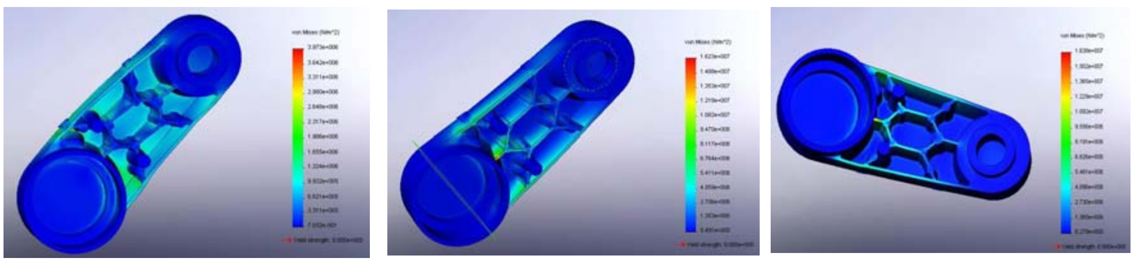

Fig. 12. Optimization of the part using von-Mises stress analysis

Figure 12 shows the second analysis for the other variant while the results are summarized in the Table 6 – the equivalent von-Mises stress and Table 7 – the deformations along the three axes.

Table 6

Equivalent von Mises stress – second analysis

Deformation	Minimum stress (N/m^2)	Position on axis (mm, mm, mm)	Maximum stress (N/m^2)	Position on axis (mm, mm, mm)
X axis stress	16.5776	(44.7, -50.2, 76)	$1.71093*10^7$	(58.9, 60.5, 49.1)
Y axis stress	1.73899	(-18.5, -66.4, 73)	$1.59826*10^6$	(58.9, 60.5, 49.1)
Z axis stress	1.07031	(-2.8, -68.9, 76)	$3.3744*10^6$	(58.9, 60.5, 49.1)

Table 7

Deformation over the three axis – second analysis

Deformation	Minimum stress (N/m^2)	Position on axis (mm, mm, mm)	Maximum stress (N/m^2)	Position on axis (mm, mm, mm)
X axis deformation	0 mm	(62.7, 0, 29.01)	0.0633982	(3.04, 304, 65.5)
Y axis deformation	0 mm	(62.7, 0, 29.01)	0.00483607	(3.0, 304, 65.5)
Z axis deformation	0 mm	(62.7, 0, 29.01)	0.010182	(-2.8, 303.9, 65.5)

4. Trends regarding the advanced design of industrial robots. FEM research.

Modern methods of design elements of the structure of robots require in certain phases the use of finite element analysis for obtaining safe projects in terms of structural strength and durability.

Using FEM analysis in the design process brings the following benefits: reduced design costs, reduced production costs, saving material, recognizing weaknesses, improves the quality of project.

Mechanical structure manufacturing of industrial robots has focused on two fundamental research directions:

• research, design and development of new materials to replace traditional materials characterized by poor physical and mechanical properties;

• modeling and optimization of structural elements shape by using algorithms and programs.

Lightweight materials such as aluminum alloys and composites in case of high speed mechanisms, have clearly demonstrated that the dynamic responses of flexible transmissions are governed by stiffness/weight reports and damping capacity of the components.

This is not a pure replacement for conventional materials, but an optimization is allowed that considers typical properties and technical possibilities of realization of composite materials.

Geometrical shape representation by using density-form function, provides an efficient calculation method for topology optimization. Optimization process tends then to solutions that are fully dense, material being removed from areas not required.

To optimize the shape of structural elements, programs were primarily developed for analysis and computer simulations. These are mathematical optimization programs or shape optimization with objective functions required.

A multi-criteria optimization has not yet been acomplished to respond simultaneously to many requirements.

Classical approaches to determine the rigidity of structural elements does not consider their tensor of inertia, which is difficult to calculate analytically. Therefore all calculations can be considered for designing and provides significant errors. A significant immediate solution is to use a software package for modeling, analysis and optimization through FEM, giving full information for any type of structure.

It is clear that a compromise must be sought between stiffness and damping characteristics of the material. Stiffness belongs to geometric modeling and damping is strongly dependent on intrinsic properties of the material. The influence of geometric modeling and the analyzed material on the damping and stiffness, must not be neglected.

5. Conclusions

The current research aimed to reduce the weight of the structure, minimizing its structural deformations on directions Ox, Oy, Oz in order to improve the stiffness, on the basis of calculus of the loading forces applied in a static study. The suggested variants may be chosen from better shape, weight, stiffness, static and dynamic behaviour.

Therefore the paper provides a methodology of analysis on the criteria of stiffness and mass while the static analysis determines a solution for a part of the arm that has been detailed. An optimization of a part of the structure from these points of view was performed and we have established the first analysis leading the best results.

Research methodology was developed to validate the theoretical results obtained by numerical analysis, finite element method (both those related to static analysis), the frequency modes and the optimization.

We have to point out that the design elements of the structure should carefully avoid discontinuities, variations in thickness, sudden changes of direction, small diameter holes and especially sharp angles in order to avoid concentration of stresses.

An improvement of the static behaviour of the elements of the structure led to finding a constructive solution from the point of view of optimum weight, shape and static stiffness with changes imposed by successive loads and choosing the best option. Design objective is to find the form that provides maximum stiffness for a given mass. Its structural shape optimization requires a considerable computing time, many trials, is very dependent on user experience rather than solve a local problem and can only improve certain aspects of structural behaviour.

Topological optimization is a technique to determine the spatial pattern of its structure or to establish optimum generalized form of continuous or composite structures. Topological optimization to obtain optimum generalized form of a structure is one of the priority areas of implementing the new techniques. The goal is to find the best use of the material for a structure subjected to the action of a force or more forces distribution.

From this perspective the aim of the topological optimization for structure design elements is to achieve a maximum rigidity and to optimize the topology with the mass control criteria and interference between the different composite materials.

REFERENCES

[1] *J. Sauter, B. Lauber, P. Häußler, D. Vieker* -"Structural Optimization – Integration and Gaps in Workflows of Numerical Simulation Processes",NAFEMS Seminar: "Integration of Numerical Simulation into the Development Process", November 17 - 18, 2003 Wiesbaden, Germany

[2] *R. Meske, F. Mulfinger, O. Warmuth,* NAFEMS Seminar "Modellieren von Baugruppen und Verbindungen für FE Berechnungen", "Topology and Shape Optimization of Components and Systems with Contact Boundary", Conditions/24.-25.April2002,Wiesbaden

[3] *J. Sauter, H. Fricke, Z. Güngör, G. Himmler, P. Hougardy, B. Lauber, O. Müller, W. Neithardt, R. Schirrmacher,* Internationaler "Integrierte Topologie- und Gestaltoptimierung im virtuellen Produktentstehungsprozess", Einbindung in die iViP Architektur und industrielle AnwendungVDI-Kongress/14.-15. September 2000/Würzburg

[4] *N. Bakhtiary, P. Allinger, M. Friedrich, F. Mulfinger, J. Sauter, O. Müller, M. Puchinger,* "A new Approach for Sizing, Shape and Topology Optimization", SAE International Congress and Exposition 1996, 26.-29. February 1996, Detroit/Michigan, USA.

[5] *O. Müller, A. Albers, J. Sauter, P. Allinger,* "Topology Optimization of Large Real World Structures", NAFEMS World Congress 1999, p.26.-28, 1999, Newport (Rhode Island), USA

[6] *Al. Dorin, R. Chirițoiu,*"Analysis of the articulated stuctures of industrial robots using finite element method", 1998, TCMM nr. 33, Ed. Tehnică, Bucureşti, ISBN 973-31-1238-0, pag.283-288;

[7] *Al. Dorin, R. Chirițoiu* "Optimizarea topologică prin metoda elementelor finite a structurilor articulate ale roboților industriali", (Topological optimization of structures by finite element method articulated industrial robots), Construcția de maşini, 2000 (52), nr,9, Romania

[8] *Al. Dorin, N. Predincea, R. Chirițoiu* "Soluții constructive pentru brațele de roboți", (Constructive solutions for robotic arms), Conferința științifică TEHNOMUS, Ediția a-IX-a, 30-31 mai 1997, Universitatea "Ştefan cel Mare" din SUCEAVA, Romania

[9] *G. Munteanu, A. Ghiorghe,* (2009), Kinematic chain and structure behavior analysis for 2nd joint of RRR type robot in order to increase its positioning accuracy, Annals of DAAAM for 2009 & Proceedings of the 20th International DAAAM Symposium - Intelligent Manufacturing & Automation: Theory, Practice & Education, ISSN 1726-9679, ISBN 978-3-901509-70-4, Editor B. Katalinic, Published by DAAAM International, Vienna, Austria 2009

[10] *S. Ioniță,* Analyse de la structure et de la topologie des mécanismes compliants planaires, U.P.B. Scientific Bulletin., Series D, Vol. 71, Iss. 2, 2009, pp. 126-132.