Proceedings in Manufacturing Systems
Proceedings in Manufacturing Systems, Volume 9, Issue 2, 2014, 87–92 ISSN 2067-9238

COMPUTER AIDED ENGINEERING OF INDUSTRIAL ROBOTS

Cristina PUPĂZĂ[1,*], **George CONSTANTIN**[2], **Ștefan NEGRILĂ**[3]

[1)] Assoc. Prof., PhD, Machines and Manufacturing Systems Department, University “Politehnica” of Bucharest, Romania
[2)] Prof., PhD, Machines and Manufacturing Systems Department, University “Politehnica” of Bucharest, Romania
[3)] MSC-Eng., Assystem Romania S.R.L., Bucharest, Romania

Abstract: *The paper presents a comprehensive CAE perspective on industrial robots. It is a synthesis of a recent research and provides an overview on the most efficient types of modeling, simulation and optimization techniques that can be accessed for industrial robots. The paper also contains a state of art review in the domain. The CAE study was performed on a simplified model of an industrial robot. Results of the static, modal analysis and an extended kinematic study are presented and coupled with FEM optimization procedures. All the results are analyzed in respect with the influence of the static and dynamic behavior on the positioning accuracy of the robot. The model preparation stages are detailed, the simulation procedures are presented and the results are explained. Using a limited number of simulations, the CAE optimization allowed the exploration of an extended design space, taking into account a large number of variants and identifying the best design through-out a ranking and sorting scheme. Because the simulations and the optimization procedures supposed a reduced, but still important number of FEM solutions, a special attention has been paid to the model preparation stages. Further research is in progress on a more detailed model and information regarding the stiffness of the joints will be also considered.*

Key words: *robot, FEM, static, dynamic, kinematics, simulation, optimization, coupled procedures.*

1. INTRODUCTION

Industrial robots are a special category of mechanical structures, which are mostly open (serial) structures, small exceptions being the Gantry type and parallel type robots. They are used more and more in industry and their performances become more important when the movements take place with high speeds, with large accelerations, or when the transient periods of braking and acceleration are significant. Also, there is an important influence of the mass concentration on the structural elements such as arms, knowing that the driving motors, together with all the elements of the driving kinematic chains (gears, belt pulleys, belts, bearings, etc.) are placed on the robot arms. These are mass concentrators that lead to high inertial forces in the case of high speed movements. Accuracy is a particularly important feature when the industrial robots find their place in precision applications. In this case the movements must be done through the appropriate trajectory and the position controlled. A prior off-line simulation evaluates the precision performances and makes the appropriate corrections.

In the design process of the industrial robots several stages are taken into account [1, 2]:

- Specification of the robot (payload, workspace, etc.);
- Structural CAD Design, based on component library;
- Kinematic chains design;
- Choosing and checking the drive system;
- Modeling-simulation with blocks (Block Digital Simulation) [3];
- Finite element analysis (FEM) of the components and of the assembly (static and dynamic);
- Modeling and simulation of the robot as multibody system (Multi Body Simulation – MBS);
- Creation of virtual prototypes and their analysis;
- Integration of the control in virtual prototype simulation (Coupled Control);
- Virtual prototype validation;
- Changes of the real prototype.

Typically, in the process of the industrial robots design there is the necessity of the dynamic properties evaluation. It is necessary to obtain information regarding the accuracy, movement ranges, workspace, the stiffness of the robot and the behavior under dynamic loads in general.

Simulation and analysis of these systems is realized considering the flexibility of the robot arm (elastic ones), most commonly using FEM [4]. However, if the system is of high complexity, having a large number of degrees of freedom, and in the case of large displacements, the FEM analysis becomes very laborious in the preparatory phase of the model (preprocessing) and requires calculations that are time consuming. In addition, the large number of parts and joints bring new difficulties for the analyst of these complex systems regarding the solution convergence.

---

[*] Corresponding author: University *Politehnica* of Bucharest, 313 Splaiul Independentei Avenue, Bucharest 6, Romania
Tel.: +40214029174;
E-mail addresses: *cristinapupazal@yahoo.co.uk* (C. Pupaza), *george,constantin@icmas.eu* (G. Constantin); *negrila_stefan@yahoo.com* (Ș. Negrilă).

Table 1

Combined modeling and simulation of the robots

Platform / Model	**RBS**	**FEM**	**DBS**
RBS		* ANSYS + Rigid body module	*** MATLAB + Simulink & SimMechanics
FEM	* ADAMS + Flexible-body module		*** MATLAB + model order reduction
DBS	** ADAMS + Coupled control	** ANSYS + Coupled control	

RBS – Rigid Body Simulation;
FEM – Finite Element Method;
DBS – Digital Block Simulation;

* Mixed structural – FEM + RBS
** Mechatronic model
(structure + integrated control)
*** Mechatronic model
(control + equivalent structure)

Another part of the modeling and simulation process is the one that takes into account the control system of the robot motion (kinematic and dynamic). It is less common in the simulation, and can lead to new difficulties. Thus, the necessity for modeling and simulation of the robot from the dynamic point of view, by combining simulation by means of solid objects (Multi Body Systems – MBS) with the finite element method was clearly ascertained [5].

The procedure is that the initial flexible sub-models of the robot have to be imported in the FEM program, which will be used later for the entire construction of model in the MBS solver for the kinematic and dynamic analysis. It is also possible to perform a static analysis directly into the MBS, as well as to transfer the data obtained from the simulation of the MBS in the FEM module for a specific analysis.

A combined modeling and simulation procedure using FEM and MBS modules may have some restrictions that come in particular from the fact that the deformations of the structural elements are relatively small in respect to the movements of the bodies. In addition, the structure is characterized by a number of prime modes. On the other hand, the model of the FEM structures cannot be changed during the simulation. However, this integrated approach of the two types of models remains the best solution in terms of the accuracy and the effectiveness of the simulation results, with all the disadvantages mentioned above.

Mixed methods of modeling and simulation currently used for industrial robots are presented in Table 1.

2. GEOMETRY AND 3D MODEL

The design concept of a robot has always to be optimized by means of CAD and FEM with regard to cost-effective lightweight construction and high torsional and flexural rigidity to assure a good dynamic performance with high resistance to vibration [6].

A draft model of a six-axis industrial robot was developed using the general configuration and the exterior design of an industrial KUKA robot. The general areas of application are: handling, assembling, application of adhesives, machining etc. [6].

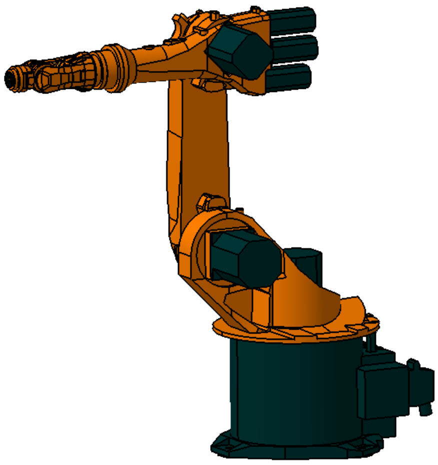


Fig. 1. 3D model of the robot.

The KUKA type 3D model was created in CATIA V5, with jointed-arm kinematics for all point-to-point and continuous-path tasks (Fig. 1). The assigned material of all the main bodies of the principal moving assemblies are of cast light alloy. The 3D model of the robot is shown in Fig. 1.

3. INITIAL STATIC ANALYSIS

In order to take into account the static structural behavior of the robot on the modal characteristics an initial static analysis was done. A maximum payload of 40 kg placed on the mounting flange of the end-wrist was considered.

3.1. Model preparation

The geometry of the robot was imported in the solver using a neutral file. Defeaturing, as well as model simplification and checks were performed in order to obtain a clean topology. The model was meshed using dominant hexahedral elements with 8 nodes. An element size of 10 mm was used (Fig. 2). Details regarding the mesh are plot in Fig. 3. Figure 4a shows the loads considered in the initial static analysis. The total calculated displacements were below 0.001 mm and the maximum equivalent stress was 5.73 MPa in this initial static case (Fig. 4,b).

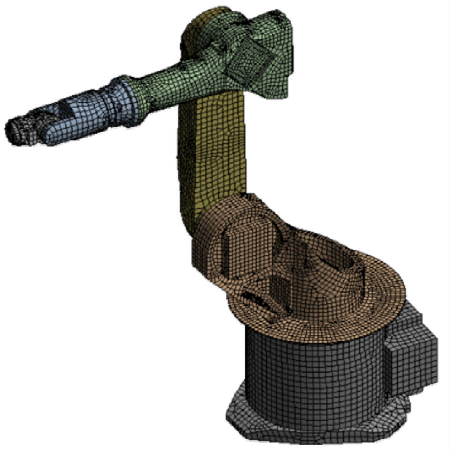


Fig. 2. FEM model of the robot.

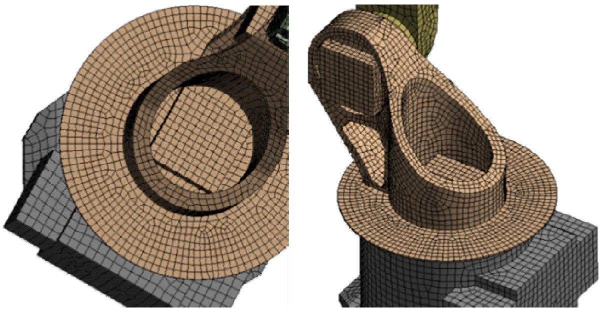


Fig. 3. Mesh. Details.

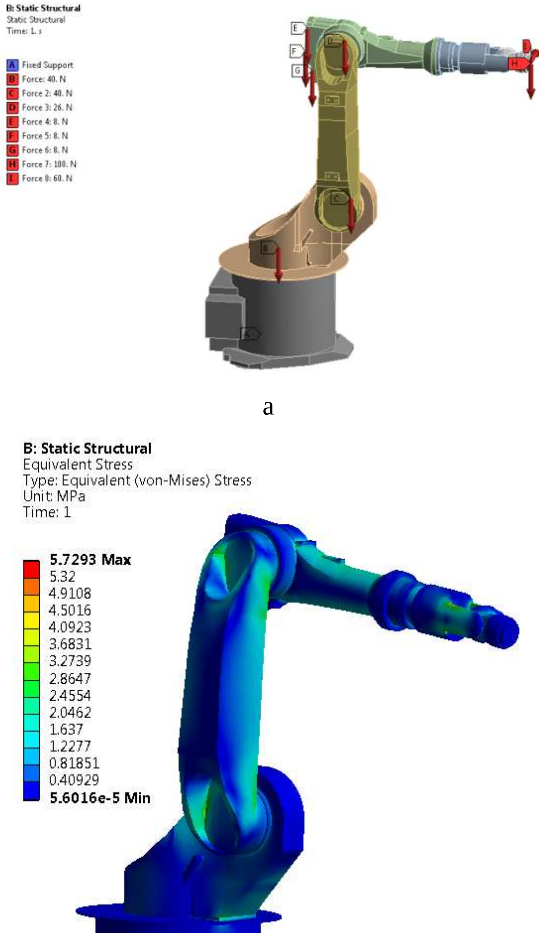


Fig. 4. Initial static analysis: *a* - Loading; *b* - Maximum Von Misses stress.

Neither the displacements, nor the maximum stress have high values that could affect the robot accuracy for this preliminary calculation.

4. MODAL ANALYSIS

The modal analysis determines the vibration characteristics: natural frequencies, mode shapes of the structure and participation factors. It is also the starting point for any another, more detailed, dynamic analysis. The natural frequencies and the mode shapes are important parameters in the design of a robot regarding the dynamic loading conditions. When the modal analysis uses the results of a static analysis as inputs, the modal analysis is named modal-prestressed. This is the approach used in the present study, where the first six natural frequencies where computed (Fig. 5).

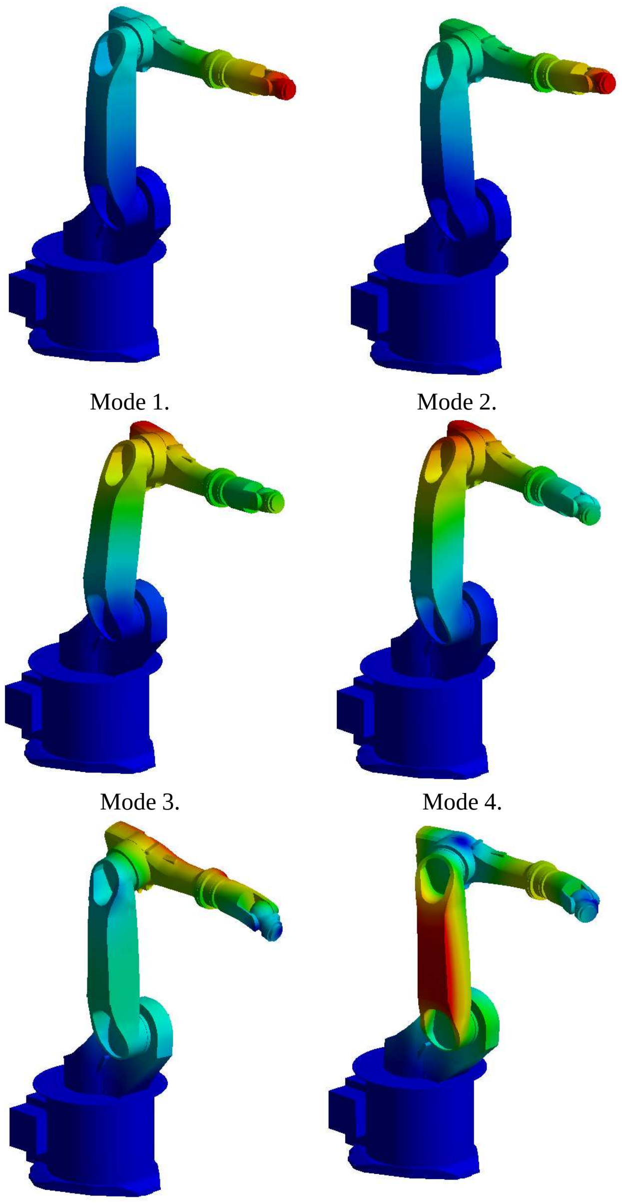


Fig. 5. Mode shapes of the robot.

The first two modes ($f_1 = 38$ Hz and $f_2 = 50.76$ Hz) are bending modes of the wrist-arm-link arm assembly in the horizontal plane, and in the vertical plane respectively, caused by the wrist. The next two modes are torsional vibrational modes of the same assembly caused by the arm ($f_3 = 114$ Hz and $f_4 = 128.9$ Hz), while the fifth and the sixth modes ($f_5 = 436.89$ Hz and $f_6 = 498.20$ Hz), are complex bending modes involving also the rotating column, caused by the vibration of the arm and the link arm, respectively. These two last modes have high natural frequencies. Therefore their influence on the robot positioning accuracy is low.

5. KINEMATIC STUDY

The kinematic analysis is a fundamental study on the behavior of mechanical systems. To determine the effector position of a robot, the analysis of intermediate joint positions has to be performed. The kinematic analysis also determines the joint loading and checks the robot displacements, velocities and accelerations in the work-

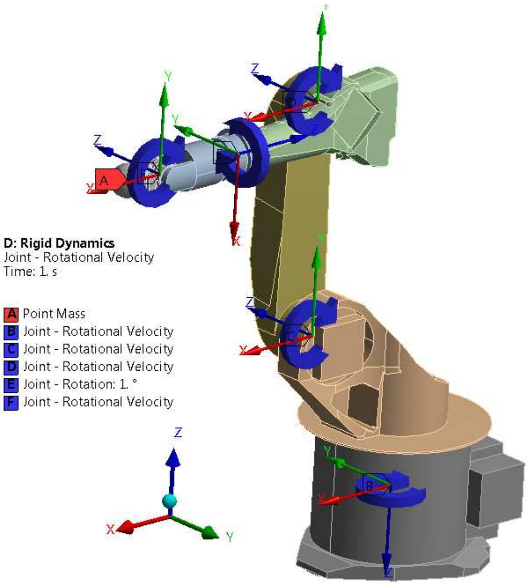


Fig. 6. Joint definition.

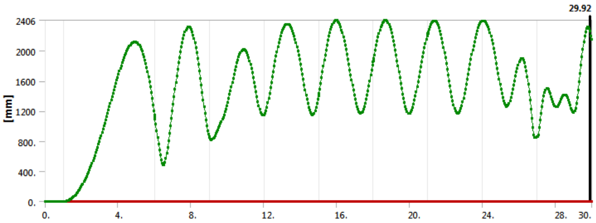


Fig. 7. Total displacements during the RBD analysis.

space. This type of analysis precedes any static or dynamic calculation and has a dedicated solver, which is the Rigid Dynamics module. The degrees of freedom are the displacements produced by the joint movement.

When reading the geometry, the solver automatically creates local reference systems in the center of gravity for each part. Each joint is also associated with a proper reference system, placed in the geometric center of the joint (Fig. 6).

The kinematic analysis was performed for hard operating conditions. The scenario supposed that the robot moves in the whole workspace following the total displacement described in Fig. 7 during a 30 s loading cycle. The maximum payload was 40 kg. The equations were solved using Runge-Kuta 4 Integration method.

For this case two critical positions were identified, with the maximum acceleration peaks at 26.2 s and 29.47 s, respectively, as shown in Fig. 8. Fig. 9 shows the force reaction over time in the joint reference system for the Joint 2.

The energy probe (Fig. 10) proved that the kinetic, potential and external energy at all the integration points were balanced.

The forces at the two pick load moments were transferred and a static analysis was performed for each part. The maximum equivalent stress after 30 seconds was plot in Fig. 11 for the robot arm. This new static analysis proved that the dynamic effect of the loading brings an increase of the static stress values more than 5 times. Although it is still not a fully dynamic simulation it gives a more realistic plot of the stress and strain peak levels during the operation of the robot.

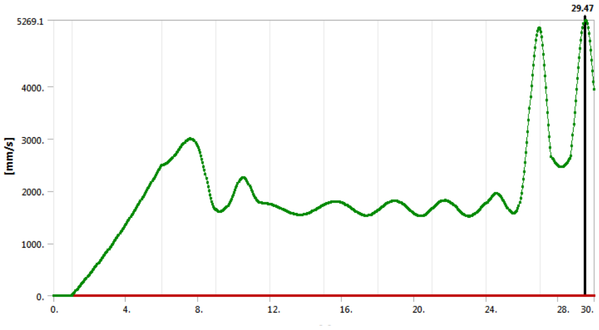


Fig. 8. Total acceleration plot.

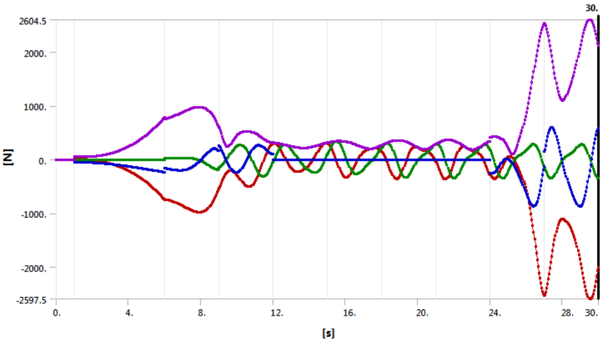


Fig. 9. Force reactions of Joint 2over time.

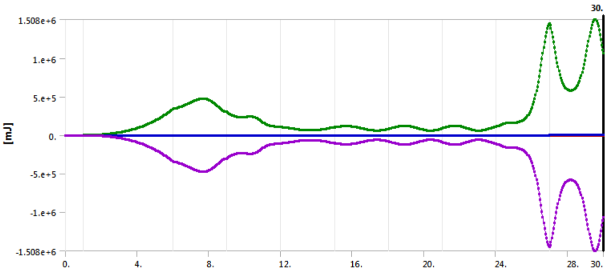


Fig. 10. Energy probe.

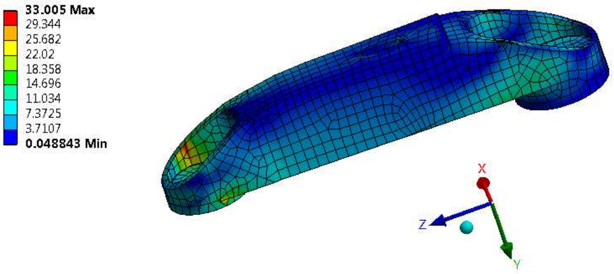


Fig. 11. The maximum stress of the arm after 30 s.

6. OPTIMIZATION STUDY

Optimization algorithms can effectively automate the iterative and time-consuming process of the design, finding suitable shapes of the parts, or the appropriate ratio between geometrical parameters. Traditional methods, such as topology optimization and advanced multi-criteria procedures using genetic algorithms are more and more accessed during the design chain.

Although topology optimization is a non-parametric procedure and efficient in most initial design stages, the geometry recovery process, the redefinition of the manufacturable form of the components and all the consecu-

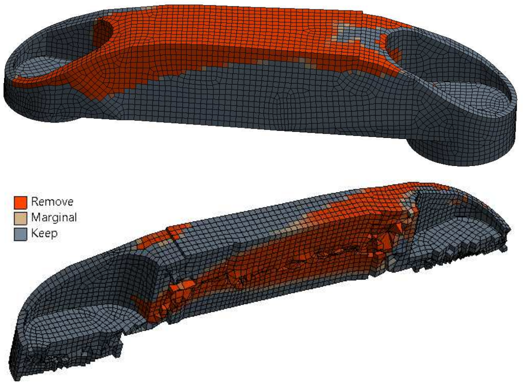
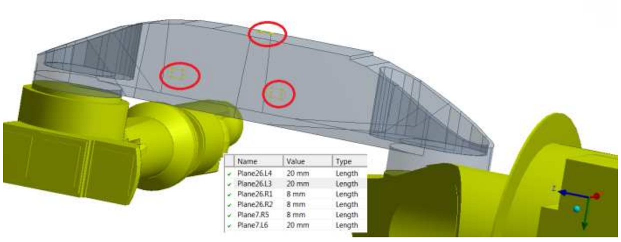


Fig. 12. Topology optimization variants without manufacturable conditions.



Name	Value	Type
✓ Plane26.L4	20 mm	Length
✓ Plane26.L3	20 mm	Length
✓ Plane26.R1	8 mm	Length
✓ Plane26.R2	8 mm	Length
✓ Plane7.R5	8 mm	Length
✓ Plane7.L6	20 mm	Length

Fig. 13. Parameter definition and initial values.

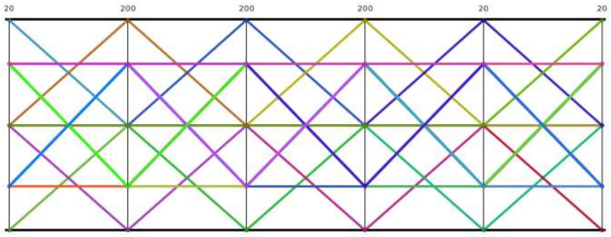


Fig. 14. Sampling points in the design space.

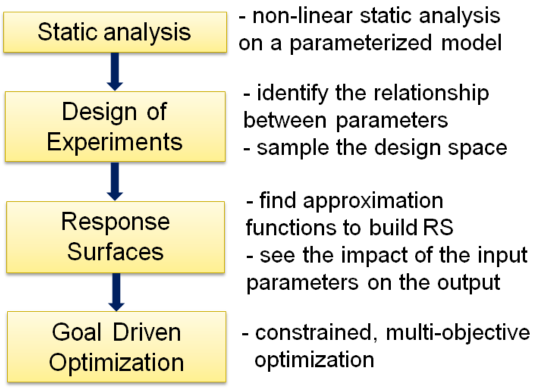


Fig. 15. Optimization stages and strategy.

tive verifications of the results may lead to inefficient computing times. Figure 12 shows two topology attempts without frozen domains and no manufacturable conditions.

To avoid the reverse engineering stages after topology optimization the robot arm shape was optimized using a multi-criteria optimization procedure. The study was focused on the weight reduction of the robot arm, while increasing the stiffness of the component. The loads applied were the output of the kinematic study at the extreme position of the robot after 30 s.

The design input parameters for the optimization study were the radius and the length of three initial oval holes, introduced in the model in order to initiate the automatic material removal and the material redistribution (Fig. 13). The optimization objectives for the robot arm were the mass and the equivalent stress minimization, when the maximum allowable displacement and the maximum upper bound of the equivalent stress were set to 0.004 mm, and 52 MPa, respectively.

The procedure combined manufacturable constraints with a genetic algorithm performed on a Kriging Response Surface. A central composite design was followed by an optimal space filling method, to sample the design space (Fig. 14). The number of the initial samples was 100 and the maximum allowable Pareto percentage was set to 70. In order to accurately detect the best design set several refinements and multiple searches in the design space were done.

The relationship between the design variables and the performance of the component can be identified using Experiments combined with Response Surfaces techniques [7]. These tools provide all information needed in the multi-criteria optimization using FEM. The Response Surfaces show interactively the performance variations due to the design variable changes and therefore it is easier to understand and to identify the improvements needed for a product to meet the requirements.

The identification of the best design set was carried on, reducing the number of the Pareto fronts. The block diagram of the optimization strategy is represented in Fig. 15. One of the analyzed Response Surface is plot in Fig. 16.

The sensitivity analysis (Fig. 17) confirmed the importance of the parameter $L6$, the length of the oval hole placed on the upper face of the arm, on the output parameters: the total deformation, the maximum equivalent stress and the geometry mass of the robot arm.

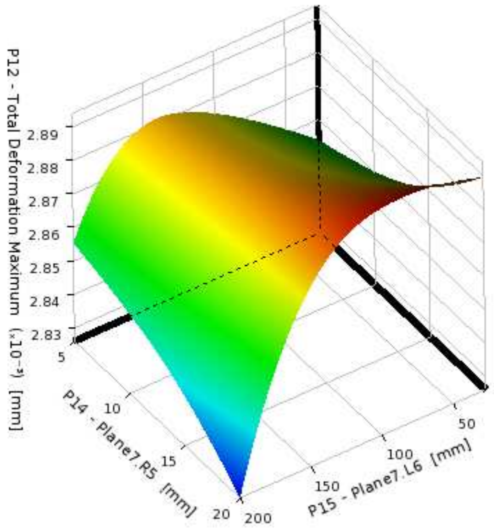


Fig. 16. Response surface of the total deformation.

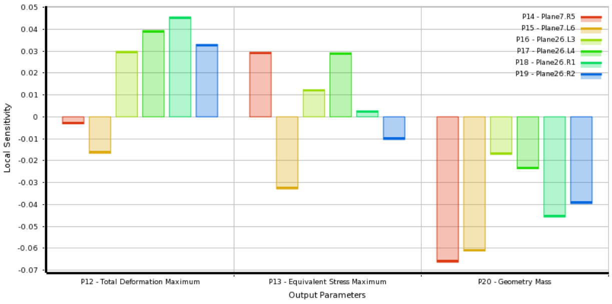


Fig. 17. Sensitivity analysis results.

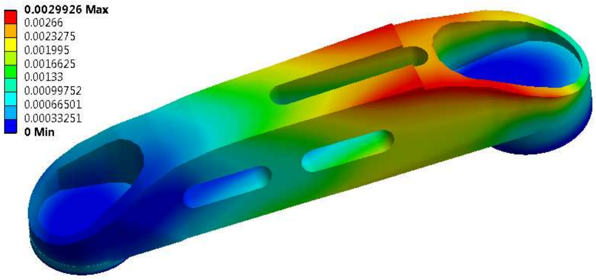


Fig. 18. Optimized shape of the root arm and minimum displacements.

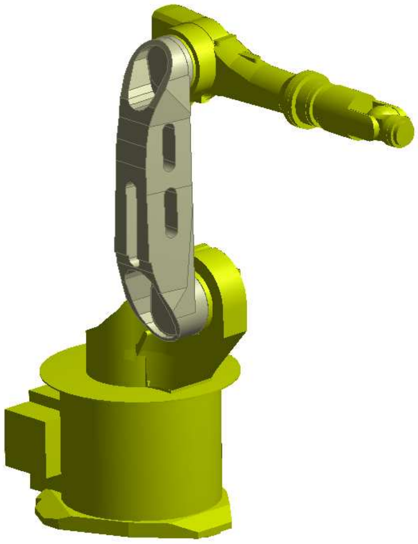


Fig. 19. Integration of an optimized component in the initial assembly.

The optimized model of the robot arm has a 20% reduced weight. The value of the maximum equivalent stress was decreased from an initial value of 33 MPa (see Fig. 11) to 18 MPa, less than 55% of the initial value. The solution converged after 1 172 evaluations. Because the optimization objectives were minimum mass and minimum stress level, the procedure finds a compromise or a trade-off between the targets and constraints, leading to a "best" set of parameters that satisfy the design requirements.

7. CONCLUSIONS

This research is a design by analysis study for an industrial robot. It doesn't prove that this is the way that the structural elements of an industrial robot have to be designed, but it shows that actually there are efficient design tools which help the designer all over the design stage. These procedures, like kinematic simulation, design by analysis and FEM optimization, drive the designer in an integrated CAD-CAE environment. They don't replace, but enforce the traditional design or the design-by-rules procedures, and allow knowledge-based solution to be taken into account.

The used CAE procedures were coupled, such as modal-prestressed, kinematic simulation and FEM optimization and multiple loading effects were taken into account. Using a limited number of simulations, the CAE optimization allowed the exploration of an extended design space, taking into account a large number of variants and identifying the best design through-out a ranking and sorting scheme.

Parameterization in the CAE environment was considered a good choice, avoiding the loose of information between different systems and giving full access to parametrical optimization procedures.

Because the simulations and the optimization procedures suppose a reduced, but still important number of FEM solutions, a special attention has been paid to the model preparation procedure. As such, Boolean operations have been avoided and the mesh generation was fast, of good quality and easily to be restored whenever the design parameters changed.

The CAE study was performed on a simplified model of an industrial robot. Further research is in progress on a more detailed model and information regarding the stiffness of the joints will be considered. A transient analysis and a response spectrum simulation will also be included in the integrated research environment using ANSYS Workbench.

Business today is racing to improve product quality, innovate and minimize time and costs. CAE procedures are certainly a solution to address these challenges.

REFERENCES

[1] M. Akdağ, *Design and analysis of robot manipulators by integrated CAE procedures*, DEÜ Fen Bilimleri Enstitüsü, Doctoral Thesis, 2008.

[2] A. Kazi, G. Merk, M. Otter and H. Fan, *Design optimization of industrial robots using the Modelica multi-physics modeling language*, Proceedings of the 33rd ISR (International Symposium on Robotics), October 7-11, 2002, Vol. 7, p. 11.

[3] S. Clark, and Y.J. Lin, *CAD tools integration for robot kinematics design assurance with case studies on PUMA robots*, Industrial Robot, 34 (3), 2007, pp. 240-248.

[4] B. Reddy, R.G. Brioso, *Automated and Generic Finite Element Analysis for Industrial Robot Design*, Thesis, Linköping University, 2011.

[5] M. Zglinska, G. Orzechowski and J. Fraczek, *Dynamical analysis of the deformable power transmission mechanism using mixed FEM and MBS method*, Journal of KONES Powertrain and Transport, Vol. 15, No. 1 2008, pp. 235–245.

[6] *** KUKA Roboter, *Spezifikation* (KUKA Robots, Specifications), KR6-2, KUKA Roboter GmbH, Deutchland, Spetember 2009.

[7] *** ANSYS Inc., *Design Exploration User's Guide*, Release 15.0, ANSYS Inc., Southpointe, 275 Technology Drive, Canonsburg, USA, February 2014.