IOP Conf. Series: Materials Science and Engineering **163** (2017) 012019 doi:10.1088/1757-899X/163/1/012019

Dynamic simulation of road vehicle door window regulator mechanism of cross arm type

I Zs Miklos, C Miklos and C Alic

Politehnica University of Timisoara, Department of Engineering and Management, 5 Revolution Street, Hunedoara, 331128, Romania

E-mail: imre.miklos@fih.upt.ro

Abstract. The paper presents issues related to the dynamic simulation of a motor-drive operating mechanism of cross arm type, for the manipulation of road vehicle door windows, using Autodesk Inventor Professional software. The dynamic simulation of the mechanism involves a 3D modelling, kinematic coupling, drive motion parameters and external loads, as well as the graphically view of the kinematic and kinetostatic results for the various elements and kinematic couplings of the mechanism, under real operating conditions. Also, based on the results, the analysis of the mechanism components has been carried out using the finite element method.

1. Introduction

With the development of the car making industry, in addition to an increase in car technical performance, as well as the development of engines & transmissions and improvement of car bodywork aerodynamics, electronic systems (on-board computer) or passive safety features, the automobile manufacturers insist increasingly on the driver and passenger comfort, cabin ergonomics, and on the idea that, in particular the driver but also the passengers, must manipulate the various parts of vehicle systems with minimal effort. This can be achieved through the "mechanisation" of these systems, as well as by placing the actuation commands within reach of the driver. One of these systems is the window drive mechanism [1].

A study method, described below, is the dynamic simulation, which is a component of the computer-aided design, applied to assemblies for showing the movement of the moving parts, taking into account their mass. In this respect, the movable elements of the assembly that need to be defined are the kinematic couplings of various classes and the internal and external loads. By these settings, the assembly becomes a mechanism that can be studied using the graphical or analytical methods provided by the theory of mechanisms and machines. [2]

Given the widespread deployment of computer in design, the analytical analysis methods based on mathematical algorithms are mainly used, because they ensure maximum precision of the results. The analysis can be performed either using specialised software implemented on the platform of a programming language, or high-performance software (MultiBody Systems - MBS), based on the theory of multi-body mechanical systems.

In the case of MBS applications, the equations of motion are self-formulated by the software, based on the geometric model for elasticity of the mechanism, taking into account the restrictions on element movements [3]. Also, true virtual prototypes can be created using the computer in view of obtaining products that meet the high functional requirements imposed by the market.

This means that we can obtain the faithful modelling of the system components, as well as the operating conditions thereof, which enables the rapid testing of numerous geometric-constructive variants for optimising the mechanism. This removes much of the experimental testing stage, which is an expensive and time consuming process [4].

2. Dynamic simulation of the car window regulator mechanism

2.1. Considerations on Autodesk Inventor application (MBS)

The dynamic simulation of the road vehicle door window regulator mechanism was carried out, based on the principle of MultiBody Systems, using the Autodesk Inventor Professional application.

The Autodesk Inventor products provide professional grade engineering solutions for 3D mechanical design, simulation, tooling creation, and design communication which helps create advantage of a Digital Prototyping workflow to make great products, cost-effectively, in less time. Autodesk Inventor software is the foundation of the Autodesk solution for Digital Prototyping. The Inventor model is an accurate 3D digital prototype that enables you to validate the form, fit, and function of a design as you work, minimizing the need to test the design with physical prototypes. By enabling to design, visualise, and simulate your products digitally, Inventor software helps you connect more effectively, reduce errors, and deliver great product designs faster. [5]

Inventor software provides powerful parametric modelling, robust direct editing tools and advanced surfacing modelling capabilities with T-splines. You can automate the advanced geometry creation of intelligent components, such as plastic parts, steel frames, rotating machinery, tube and pipe runs, and electrical cable and wire harnesses. Inventor software also helps reduce the geometry burden so it can rapidly build and refine digital prototypes that validate design functions and help minimize manufacturing costs. Traditionally, validating the operating characteristics of a design before it was built meant hiring expensive specialists. Inventor products include easy-to-use and tightly integrated part and assembly-level motion simulation and stress analysis functionality. By simulating stress, deflection, and motion, you can optimize and validate the design under real-world conditions, before the product or part is ever built. [6]

In this regard, the analysis and simulation of the studied mechanism by using the multibody system method by means of Autodesk Inventor, involves the following stages:

- Modelling of the mechanism as a multibody system (modelling the components and defining the assembly constraints);
- Defining the kinematic couplings of the mechanism, either through self-formulating or by individual definition;
- Defining the mass properties of the kinematic elements, i.e. the force or momentum-type loads;
- Carrying out the actual simulation, graphical display of results and their interpretation;
- Analysis of mechanism elements using the finite element method;
- Graphical display of results and their interpretation after the analysis by finite element method.

2.2. 3D modelling of the mechanism used to manipulate the car window

The 3D modelling of road vehicle door window regulator mechanism was carried out by individual modelling of its kinematic elements, followed by defining the assembly constraints. The model of the mechanism can be seen in Figure 1 [7].

It is very important that the assembly constraints, as geometrical relationship among the kinematic elements, to be properly defined, because when determining the necessary parameters for dynamic simulation, they can be auto-converted into kinematic couplings of various classes. In the Figures 2.a, 2.b, 2.c, 2.d, we can see details of the application of the constraints imposed to the various elements of the mechanism, as follows:

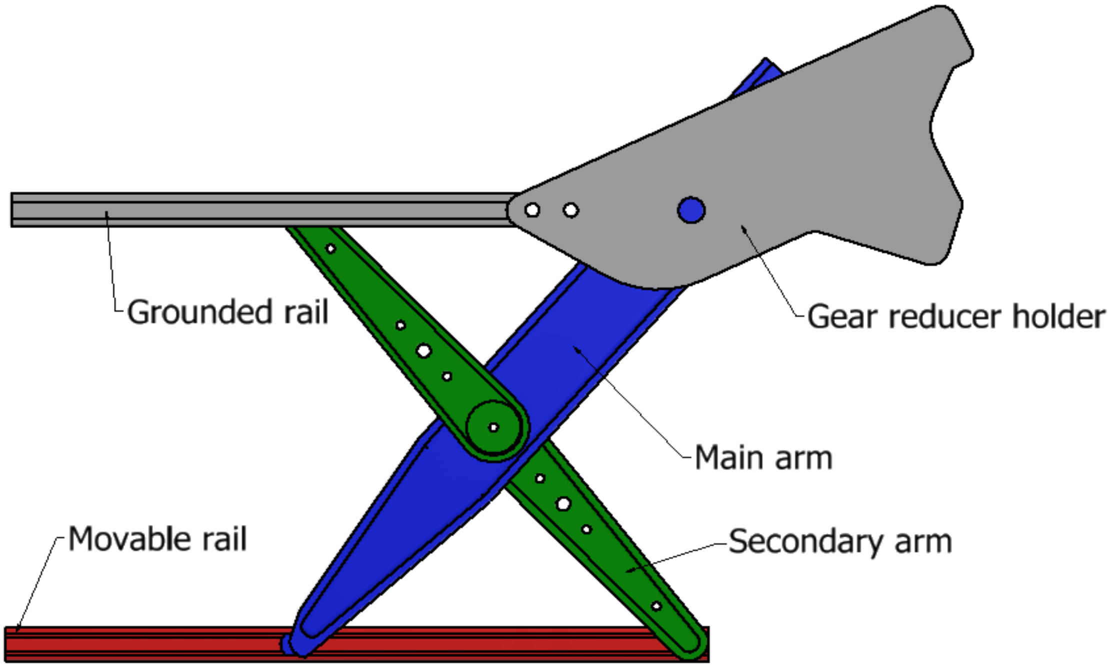
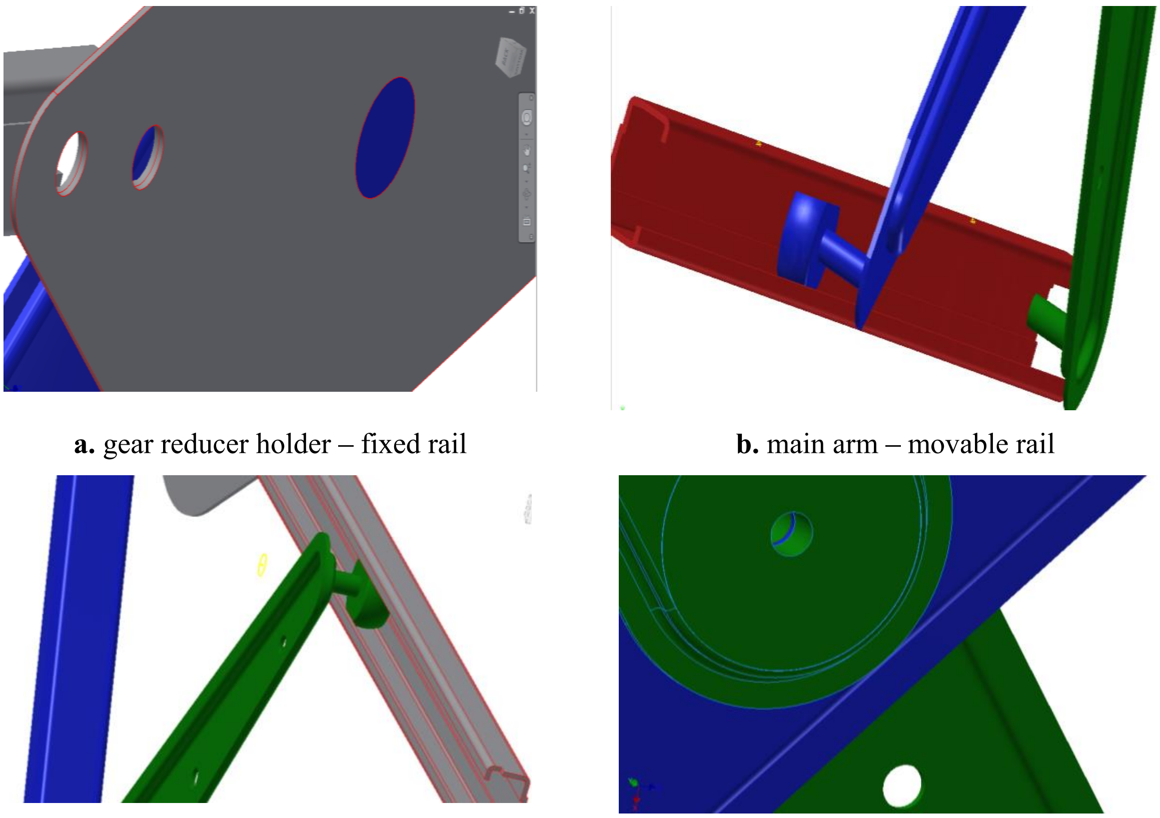


Figure 1. 3D model of the mechanism



a. gear reducer holder – fixed rail

b. main arm – movable rail

c. secondary arm – fixed rail

d. main arm – secondary arm

Figure 2. Application of assembly constraints

2.3. Analysis and dynamic simulation of the window regulator mechanism

The simulation of road vehicle door window regulator mechanism was carried out on the basis of the following parameters:

- Power transmitted: P = 0.3 kW
- Rotational speed of the driving element: n = 1.6 rot/min
- Up/down stroke – 510 mm
- Stroke length for lifting / lowering the window – t = 10s
- Payload (weight of the window) – F = 80 N

As stated in the preceding paragraph, the assembly constraints among the elements of the system can be auto-converted into kinematic couplings (Figure 3). In the case of the interface shown in Figure 3, we chose to carry out the analysis through the method of finite elements, using the AIP Stress Analysis module.

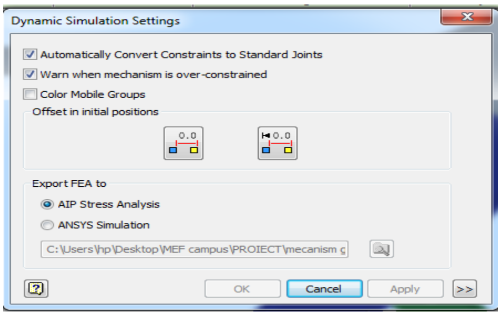


Figure 3. Self-conversion of constraints

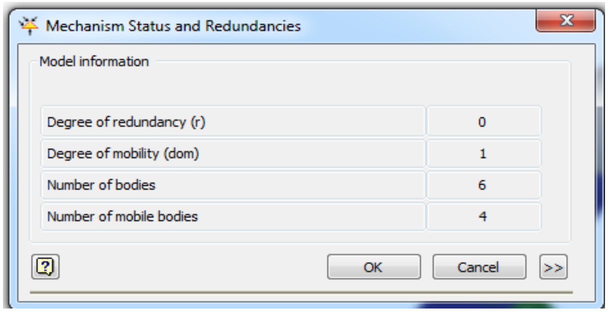
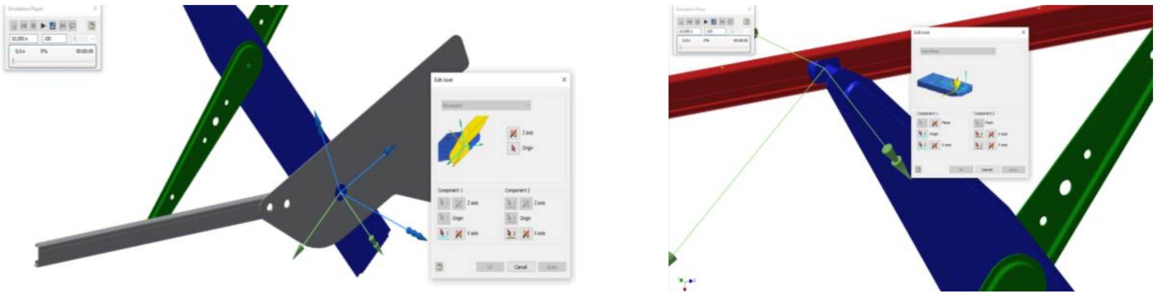


Figure 4. Structure of the mechanism

The articulated, cross arm type mechanism taken into account (Figure 1), parametrically modelled using Autodesk Inventor, by properly applying the assembly constraints, includes a kinematic chain which transmits the motion of the main arm to the movable rail, the geometric constraints being realised by rotational and translational couplings.

In these circumstances, the check of topology model leads obviously to a kinematic system without redundant joints (super type constraints), with r = 0 and the number of degrees of mobility dom = 1 (Figure 4). The indicated number of kinematic elements is 6, but it should be noted that the fixed element consists of two elements: the gear reducer holder and the fixed rail, stiffened one to another, and the secondary arm consists also of two components stiffened one to another, by properly applying the assembly constraints. Thus, the number of movable kinematic elements is equal to 3.

To ensure that the assembly constraints among the elements of the mechanism have been correctly self-converted into kinematic couplings, there is the possibility of viewing and redefining them. Thus, Figure 5.a shows the rotational coupling between the fixed element and the main arm, and Figure 5.b shows the translational coupling between the main arm and the movable rail.



a. Fixed element – Main arm **b.** Main arm – Movable rail

Figure 5. Redefining of the kinematic couplings

The driving motion has been defined in the kinematic coupling, based on the initial data, by introducing the value of angular velocity (9.7 degrees/sec). In the same kinematic coupling, we defined the torque value, known in the theory of mechanisms as *momentum balancing*.

For the mechanism to be regarded as a multibody system, as has been stated in the first paragraph, besides defining the kinematic couplings of the components, it is necessary to define the external loads, i.e. we must take into account the mass of the elements.

The external loads are given, on the one hand, by the weight of the manipulated window (they have been defined by distribution, as two concentrated forces acting on the movable rail, which actually serves as window holder - Figure 6) and, on the other hand, by the tare weight of the kinematic elements of the mechanism. The masses of the elements are determined automatically, by means of the volume or density of the materials selected accordingly. The tare weight results from defining the value and direction of the gravitational acceleration, based on the orientation of the reference system associated to the 3D model of the mechanism. The load pattern can be visualized in Figure 6.

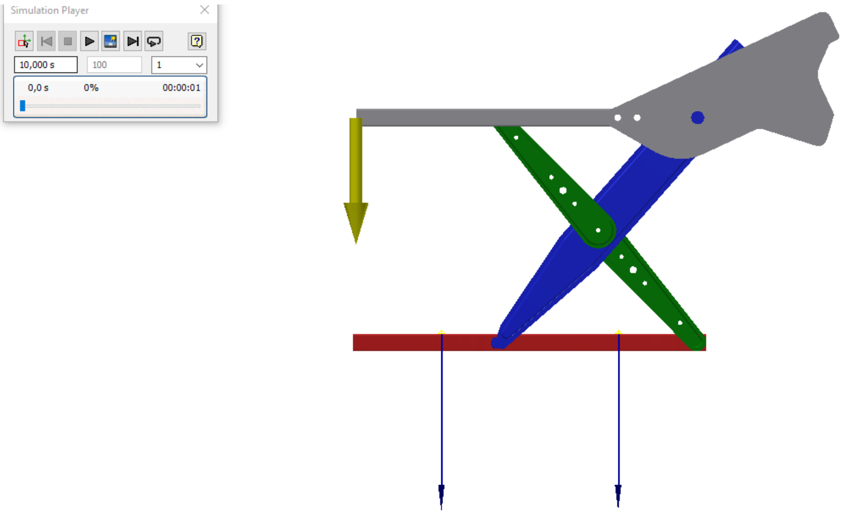


Figure 6. Load pattern of the mechanism

After determining the loading scheme, i.e. defining the motion parameters, the actual simulation of the mechanism has been carried out. As simulation parameter, we settled 100 time steps. Therefore, the results can be seen every 0.1 seconds throughout the kinematic cycle. As results, we can obtain numerical values or graphical variations of the various kinematic and kinetostatic parameters afferent to the couplings and kinematic elements.

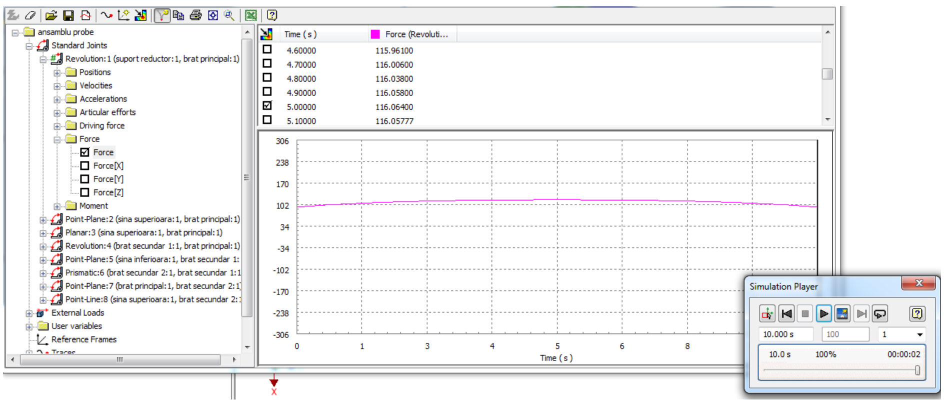


Figure 7. The reaction force in the kinematic driving coupling

Figure 7 shows the variation of reaction force in the kinematic driving coupling, between the fixed element (gear reducer holder) and the main arm of the mechanism, throughout the kinematic cycle. After viewing the result, it was found that the maximum reaction force, 116.06 N, was placed in the mid of kinematic cycle, i.e. at the time step t = 5 sec. It is important to specify this identified time step because it will be taken into account along with the external loads defined and the internal loads resulted from the finite element analysis of the mechanism elements.

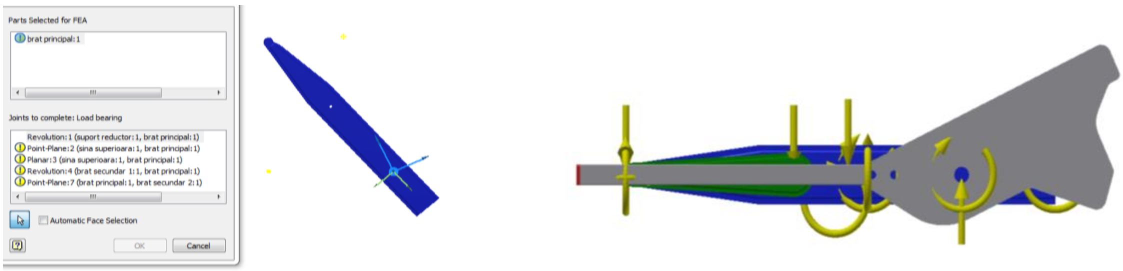
3. Analysis of the main arm using the finite element method

The finite element analysis is a method to study the state of stress and deformation in the cases of bearing structures with very complex geometry, for which the calculation is made easier if the whole is divided into simpler areas.

The finite element method is a mathematical method of numerical integration of partial differential equations in variational form. We mean that all the calculation problems of the elastic structures will be reduced to a system of partial differential equations, which usually cannot be solved analytically. This inconvenient has been eliminated by using digital computers which, using dedicated software applications, provide rapid means of making a large volume of calculations involved in the finite element analysis, actually making this method to be easily applicable. [8]

For the road vehicle door window regulator mechanism, i.e. for its components, an analysis based on the finite element method is required to determine the state of stresses and deformations in real operating conditions, in order to assist the sizing process.

Basically, the finite element analysis assumes the completion of specific stages, such as: building the geometric model, discretisation, selection of materials, settling boundary conditions and external loads. But, since the finite element analysis of the studied mechanism was performed based on the results obtained from the dynamic simulation, i.e. maximum reaction force in the kinematic coupling between the fixed element and the main arm, at the time step t = 5 sec, it is sufficient that the kinematic element examined (i.e. the main arm) to be isolated from the rest of the mechanism by specifying its movable couplings with the adjacent elements (Figure 8a). Following this operation, all the external loads defined and the internal loads resulted, as well as the boundary conditions resulted from defining the kinematic couplings, will be taken over automatically, the loading diagram of the mechanism being shown in Figure 8b.



a. Isolation of the main arm – Boundary conditions

b. External and internal loads

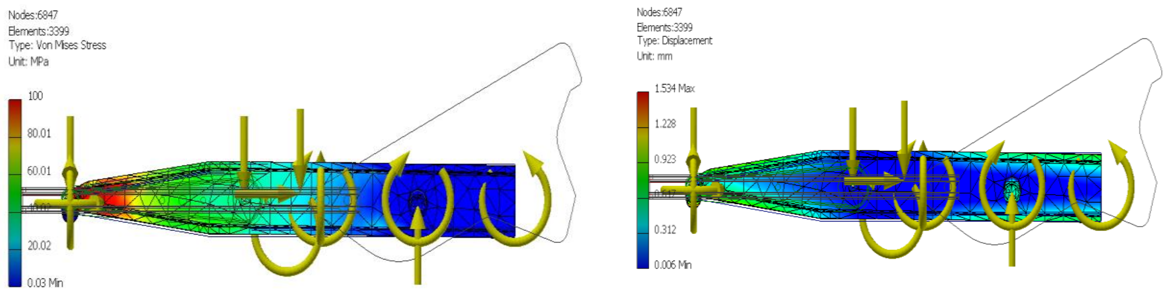
Figure 8. Establishing of the loading diagram

As material for the studied element, but also for the other components, we opted for a general purpose steel grade, whose specific properties are shown in Table 1.

Table 1. Properties of the chosen material

Mass Density (g/ cm^3)	Yield Strength (MPa)	Ultimate Tensile Strength (MPa)	Young's Modulus (GPa)	Poisson's Ratio	Shear Modulus (GPa)
7.85	207	345	210	0.3	80.7692

Following the actual analysis, the obtained results are visualised and interpreted, i.e. the Von Mises stress state (Figure 9.a) and the deformation state (Figure 9.b) in the main arm.



a. Von Mises stress state **b.** Deformation state

Figure 9. The results of the finite element analysis

It appears that the maximum stress value is 100 MPa, found in the area of movable connection between the main arm with the movable rail, and the maximum deformation is 1.53 mm, both results being able to be considered satisfactory in terms of resistance calculation.

Conclusion

The dynamic simulation is a component of the computer-aided design and a method of analysing a product under real operating conditions. It can be likened to a virtual prototyping based on a 3D model, but without the expenses required to build a real prototype.

Following the dynamic simulation of the road vehicle door window regulator mechanism, it was found that the maximum reaction force in the kinematic coupling is between the fixed element and the main arm, in the mid of the kinematic cycle, and the maximum stress in the main arm is in the area of connection with the movable rail.

References

[1] Miklos I Zs, Miklos C and Alic C 2014 Drive mechanism analysis of door windows to road vehicles, *Acta Technica Corviniensis – Bulletin of Engineering* **4** 61-64

[2] Stăncescu C, Manolache D, Pârvu C, Ghionea I and Tarbă C 2012 *Proiectare asistată cu Autodesk Inventor*, Fast, Bucureşti, Romania

[3] Rusu-Anghel S, Lihaciu I C and Rusu-Anghel N 2015 State feedback control of a robotic arm, *Annals of Faculty Engineering Hunedoara – International Journal of Engineering* **13**(1) 75-82

[4] Alexandru C 2006 Aspecte privind analiza mecanismelor ştergătoarelor de parbriz considerate sisteme multicorp, *Annals of the University of Oradea, Fascicle of Management and Technological Engineering* **V**(XV) 592-593

[5] * * * www.manandmachine.ro

[6] Dolga L, Revencu M, Maci C A and Giuchici M 2004 *Parametric and feature based modelling with applications in Catia and Inventor*, Politehnica, Timişoara, Romania

[7] Miklos I Zs, Miklos C and Alic C 2015 Articulated bar mechanism design for manipulating the doors windows of road vehicles, *Annals of Faculty Engineering Hunedoara - International Journal of Engineering* **13**(3) 159-163

[8] Spyrakos C Ch 1994 *Finite element modeling in engineering practice*, West Virginia University Press, Morgantown, USA