ISSN 2176-5480

22nd International Congress of Mechanical Engineering (COBEM 2013)
November 3-7, 2013, Ribeirão Preto, SP, Brazil

STIFFNESS ANALYSIS OF CAPAMAN-2BIS USING FINITE ELEMENT ANALYSIS

João Marcus Teixeira Pêgo
Federal University of Uberlandia
School of Mechanical Engineering
Campus Santa Mônica
38400-902 Uberlândia /MG, Brasil
e-mail: j.marcus.gtr7@hotmail.com

Rogério Sales Gonçalves
e-mail: rsgoncalves@mecanica.ufu.br

João Carlos Mendes Carvalho
e-mail: jcmendes@mecanica.ufu.br

Giuseppe Carbone
e-mail: carbone@unicas.it
LARM, DiMSAT, University of Cassino, Cassino, Italy

Marco Ceccarelli
e-mail: ceccarelli@unicas.it

Abstract. *This paper presents the stiffness analysis of CaPaMan-2bis (Casino Parallel Manipulator version 2 bis) robotic structure using the Finite Element Analysis (FEA). First, a review of the methods for calculating the stiffness of multibody systems is presented. After is presented the kinematics model of CaPaMan-2bis. Many types of computer software can be used to model the structure and run its finite element analysis. The software chosen for this paper were the SolidWorks® for 3D modeling and ANSYS® for the finite elements modeling and analysis. Finally are presented the results of the compliant displacements of this structure.*

Keywords: *Stiffness Analysis, Compliant Displacements, FEA, CaPaMan-2bis*

1. INTRODUCTION

Multibody systems consist on a kinematic chain composed of links that can be rigid or flexible, interconnected by joints. An example of multibody system that has been widely studied in recent years is the parallel robotic structure. In spite of its several advantages, this structure presents challenges to researchers on what concerns the solution of problems such as: singularities, stiffness and accuracy (Gosselin and Angeles, 1990; Macho et al., 2008; Gonçalves and Carvalho, 2008; Gonçalves, 2009).

The influence of compliant displacements of links and joints can be obtained from the stiffness study of parallel robotic structures.

Stiffness can be defined as the capacity of a mechanical system to sustain loads without excessive changes of its geometry (Rivin, 1999). These changes on geometry, due to the applied forces, are known as deformations or compliant displacements.

Compliant displacements in a parallel robotic system produces negative effects on static and fatigue strength, wear resistance, efficiency (friction losses), accuracy, and dynamic stability (vibrations). The growing importance of high accuracy and dynamic performance for parallel robotic systems has increased the use of high strength materials and lightweight designs improving significant reduction of cross-sections and weight. Nevertheless, these solutions also increase structural deformations and may result in intense resonance and self-excited vibrations at high speed (Rivin, 1999). Therefore, the study of the stiffness becomes of primary importance to design robotic systems in order to properly choose materials, component geometry, shape and size, and interaction of each component with others.

The overall stiffness of a manipulator depends on several factors including the size and material used for links, the mechanical transmission mechanisms, actuators and the controller (Tsai, 1999). In general, to realize a high stiffness mechanism, many parts should be large and heavy. However, to achieve high speed motion, these should be small and light. Moreover, one should point out that the stiffness is greatly affected by both the position and the values of the mechanical parameters of the structure parts (Yoon et al., 2004).

There are three main methods have been used to derive the stiffness model of parallel manipulators (Gonçalves, 2009; Deblaise, 2006). These methods are based on the calculation of the Jacobian matrix (Zhang et al., 2004; Majou et al, 2004; Company et al., 2005); the Finite Element Analysis (FEA) (Bouzgarrou et al., 2005; Corradini et al., 2004) and the Matrix Structural Analysis (MSA) (Deblaise, 2006; Przemieniecki, 1985; Dong et al., 2005, Gonçalves and Carvalho, 2008).

In this paper is presented an approach to stiffness analysis of robotic structures using the FEA method.

Stiffness analysis of structures, when done through analytical methods, almost always produces solutions in the form of closed equations, which are hard to solve, and in some cases, impossible to be solved. In that case, it becomes necessary to realize a numerical approximation of calculus, so we can obtain results as close to real as possible.

The FEA method is a numerical analysis technique used to obtain approximate solutions to problems governed by differential equations. Although the method was first developed for statical analysis of structural systems, it has been used in the study of a great variety of engineering problems, in the domains of solids mechanics, fluids mechanics, heat transfer and electromagnetism (Brenner, 2008).

The FEA method can be defined as a process of discretization of the problem, which changes the infinite-dimensional condition of the problem to a finite-dimensional condition, limiting the number of unknowns. The method consists in dividing the domain in multiple linked regions, named “elements”. Each element is limited by points called “nodes”, and the group of elements with their nodes is called “mesh”. After defining elements and their respective nodes, inside each element, are admitted approximate solutions expressed by interpolation functions. Also are imposed conditions guaranteeing a continuous solution in the nodes shared by various elements. The problem’s unknowns, called “degrees of freedom” (dof), become the values of the field variables in the nodal points, and the number of these unknowns (now finite) is called number of degrees of freedom of the model. Depending on the nature of the problem, after the discretization, the governing mathematical model results of the representation of a finite number of ordinary differential equations or algebraic equations, of which numerical solving leads to the values of the nodal unknowns. Once these unknowns are determined, the values of the field variables inside the elements can be evaluated using interpolation functions (Brenner, 2008; Rade, 2001).

The main advantages of FEA are: elements of different shapes and sizes can be associated to discretize domains of complex geometry; the division of the continuous domain in regions makes the modeling of problems with non-homogeneous domains easier, where physical properties vary; and the method can completely formulated with matrices, making its computational implementation easier. The main disadvantages of the method are the uncertainties inherent of the FEA modeling resultant of simplifications of the physical model such as: not considering some physical effects like non-linearity, hysteresis, damping, etc.; discretization error; inaccurate values of physical and/or geometrical parameters (elasticity models, density, etc.); difficulty in modeling localized effects such as screwed or riveted junctions; and errors derived of the process of numerical resolutions (Brenner, 2008; Rade, 2001).

The finite element modeling is appropriate for robotic structures, since it allows us to faithfully describe the structure’s geometry. Besides, it is possible to consider the behavior of the structure’s joints and restrictions.

Many authors have been using FEA for stiffness analysis of robotic structures. Bouzgarrou et al. (2004) held a stiffness study of the parallel robot 3TR1 presented by Gogu (2002) using finite elements coupled to a CAD model. Clinton et al. in 1997 studied the stiffness of the Gough-Stewart platform. In this structure all elements are subject only to traction and compression solicitations. Each element is studied individually and then assembled in order to do the set study of the structure. In Dong et al. (2005) a study using FEA is done in order to determine the stiffness model of a parallel structure that has flexible joints.

Zhou et al. (2006) uses FEA for the modeling of the parallel manipulator 3-PRS. The moving platform is modeled by triangular plate elements and the legs by beam spatial element. The flexibility of the joints is considered by the introduction of virtual springs to the FEA model. As the joints parameters are not known the FEA numerical model is adjusted through experimental tests according to the frequency response functions.

Deblaise (2006) used FEA modeling to compare the results obtained using matrix structure analysis (MSA). This study was applied in the modeling of a Delta parallel robot, Fig. 1(a). In this paper, a first model, considering only the flexibility of the segments, Fig. 1(b), did not give compatible results with the ones from experimental tests.

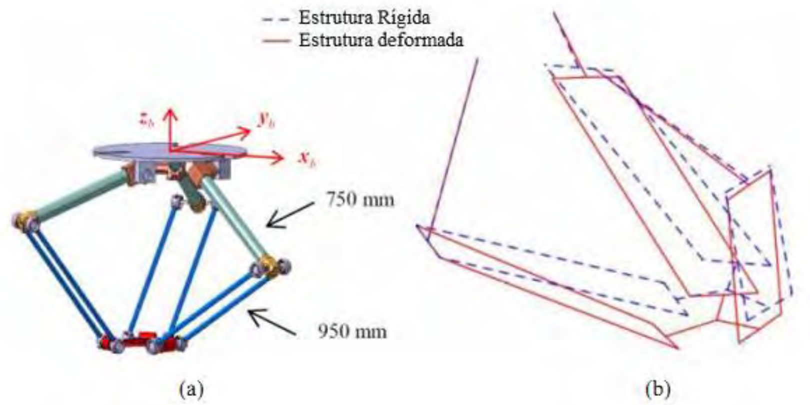


Figure 1. (a) Delta parallel robot; (b) FEA model (Deblaise, 2006).

ISSN 2176-5480

Corradine et al., (2004) used FEA to model the H4 robot, Fig. 2(a). Besides the modeling of segments, the joints in the structures, spherical and rotational, were modeled introducing “displacement relaxation” in the FEA model, which allows, for the spherical joints, that all flexible translation displacements between the segments are the same, but not the rotations. For the rotational joints all movements, excluding the rotation axis, must be the same. Figure 2(b) presents the results of the FEA modeling obtained. When comparing the results of the FEA model with those from experiments, they were close to each other.

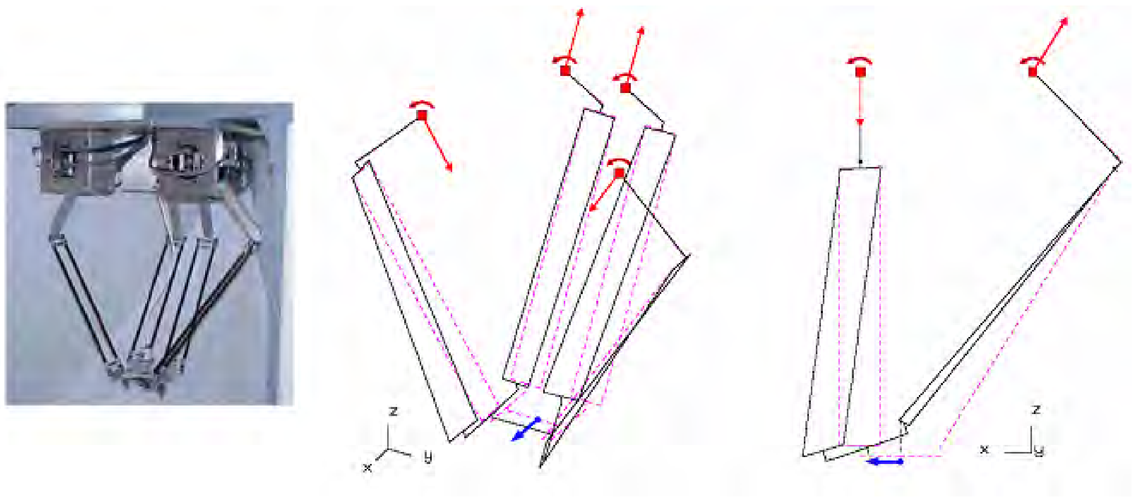


Figure 2. (a) Prototype of H4 robot; (b) FEA model of H4 robot (Corradine et al., 2004).

Aginaga et al. (2012) presented an analytical method to calculate the stiffness matrix in parallel structures, applying it to the 6-RUS structure. The finite elements method was then applied to a specific component of the structure, due to its complex geometry, in order to calculate the stiffness indexes of such component.

Hu et al. (2011) used the finite element method to evaluate the stiffness and elastic deformations of a serial-parallel manipulator 2(SP+SPR+SPU). The results for the calculus of stiffness of the finite elements simulations were found within the expected reasonable approximation, and the results for the elastic deformations simulations were practically coincident to the analytical solution.

Kobel e Clavel (2011) developed a reduced size with large workspace robot, named ΦR, for micro manipulation and assembling, using finite elements to evaluate the static behavior of the structure. The bearings in the structure were idealized using 6 springs between inner and outer rings of each revolute joint in order to account for the influence of such bearings.

Li (2009) proposed two new micro manipulator models of three degrees of freedom, using the serial parallel configuration, one of them with actuators redundancy. Finite elements in this cased were used not to evaluate the stiffness of the structures, but to find was to optimize it. Results showed that the actuator redundant structure had greater strength.

Liang et al. (2011) modeled a 6 degree-of-freedom micro manipulator based on the CPM (compliant parallel manipulator) structure, using finite elements to validate the studies done over the desired characteristics for the structure and its accuracy. Results found were satisfactory, showing that the structure can provide high accuracy and resolution movements.

Rezaei et al. (2012) held the stiffness analysis of a spatial parallel 3-PSP structure, considering the moving platform as flexible. The finite elements method was then used as comparison parameter.

Taghvaeipour et al (2010) developed an online method for calculating the stiffness matrix of structures using finite elements, and used the McGill SMG structure as a case of study.

Gonçalves and Carvalho (2008) used FEA method to validate their proposed method using Matrix Structural Analysis to find the stiffness of parallel structures applied in 6-RSS parallel manipulator.

Tian et al. (2010) presented the project and analysis of 6 degrees of freedom parallel robot, used in surgeries of artificial replacement of the cervical disc. The structural statics, singularities and workspace were evaluated using FEA.

From the papers analyzed the FEA method, generally, is used to validate analytical models (El-Khasawneh e Ferreira, 1999; Li et al., 2002; Deblaise, 2006; Liang et al., 2011; Taghvaeipour et al., 2010) and or experimental results (Corradini et al., 2004; Bouzgarrou et al., 2004; Zhou et al., 2006; Deblaise, 2006; Rezaei et al., 2012) or even for optimization of structures and parts of structures (Li, 2009; Aginaga et al., 2012).

The biggest advantage of using the FEA method is the utilization of the mechanical structure’s project with no simplification, considering its full geometry. For robotic structures with irregular geometry, like industrial serial robots

ISSN 2176-5480

and parallel robotic structures that are subject to not only axial loads, like the Gough-Stewart platform, the use of the FEA method can make the stiffness analysis for the robotic structure easier.

The disadvantage of the FEA method, when it involves commercial software for analysis, is that it requires great computational effort, because, since stiffness depends on positions, it's necessary, for each specific position, to build a finite element model (Huang et al., 2002).

In this paper the FEA method was used to evaluate the stiffness of the CaPaMan-2bis (Casino Parallel Manipulator version 2 bis) structure.

Many types of computer software can be used to model the structure and run its finite element analysis. The software chosen for this paper were the SolidWorks® for 3D modeling and ANSYS® for the finite elements modeling and analysis.

2. CAPAMAN-2BIS DESCRIPTION AND KINEMATIC MODEL

The object of study in this paper is the parallel robot CaPaMan-2bis, Fig. 3, designed and built at the University of Casino, Italy at LARM: Laboratory of Robotics and Mechatronics.

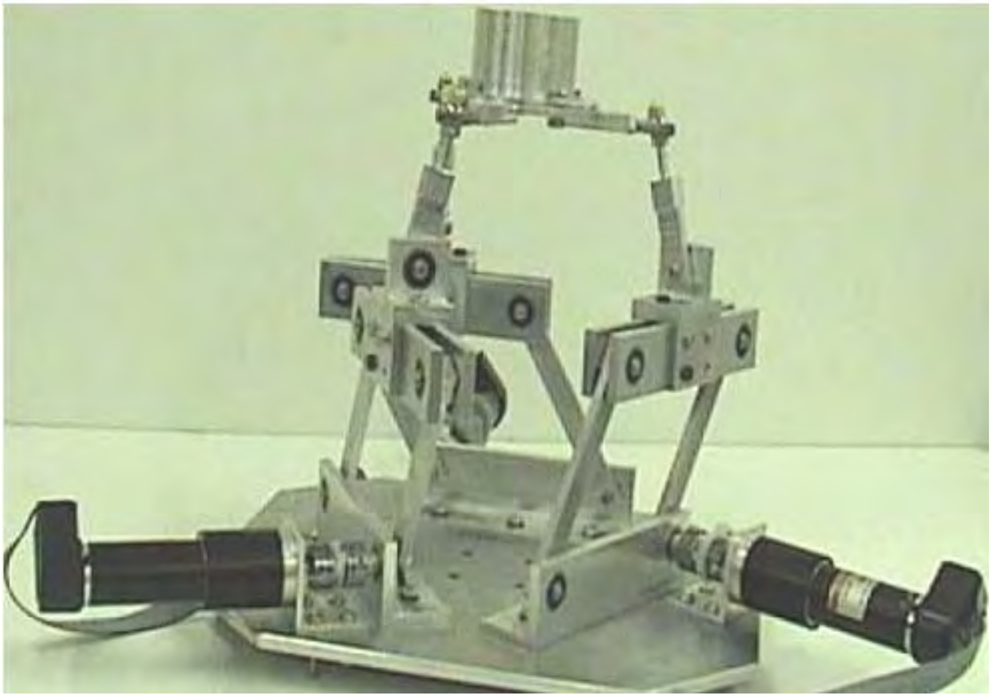


Figure 3. CaPaMan-2bis prototype at LARM in Casino (Liang et al., 2009).

Several prototypes of parallel manipulators have been built at LARM, with low-cost and easy-operation characteristics. Research activities have been carried out on both theoretical study and application aspects. In the works of Ceccarelli (1997), Ceccarelli and Figliolini (1997) and Carvalho and Ceccarelli (2001), a three D.O.F (degrees of freedom) spatial parallel manipulator named as CaPaMan was proposed with peculiar mechanism architecture. Direct and inverse kinematic algebraic equations have been formulated for it in a close-loop form. Workspace capacity and operation characteristics have been evaluated in simulation and experimental tests (Ottaviano and Ceccarelli, 2002, Ceccarelli and Carbone, 2002 and Ceccarelli et al., 2002). The platform of CaPaMan has been used as a motion generator for earthquake simulation at LARM (Ottaviano and Ceccarelli, 2006). Later, two prototypes named as CaPaMan2 and CaPaMan2-bis have been built with modified mechanism architectures. In addition, stiffness and accuracy have been improved by paying special attention on mechanical design (Liang et al., 2009).

CaPaMan-2bis has been the result of these enhancements, where the modified shape of the movable plate has been designed to properly lock a commercially available drilling tool. CaPaMan2-bis has been implemented as a part of a hybrid robotic architecture for surgical tasks (Carbone and Ceccarelli, 2005), as well as a trunk module in a humanoid robot design named as CALUMA (CAssino Low-cost hUMAnoid robot) (Nava-Rodriguez et al., 2006). Previous works were developed to analyze the CaPaMan-2bis prototype in terms of stiffness and basic kinematics (Carbone and Ceccarelli, 2006 and Aguirre et al., 2003). Since the solution of direct kinematic problem for CaPaMan-2bis can be complex, the system behavior can be conveniently simulated and experimentally measured (Hernandez-Martinez et al., 2010).

ISSN 2176-5480

Figure 4 shows a kinematic scheme of CaPaMan-2bis. It is composed of a movable platform MP, which is connected to a fixed base FP by means of three leg mechanisms. Each leg mechanism is composed of an articulated parallelogram AP whose coupler carries a revolute joint RJ, a connecting bar CB that transmits the motion from AP to MP through RJ, and a spherical joint BJ, which is installed on MP at point J. The revolute joint RJ, which is installed on the coupler of AP has the rotation axis coinciding with the parallelogram plane.

Each leg mechanism is rotated of $\pi/3$ with respect to the neighboring one so that the leg planes lie along two vertices of an equilateral triangle, giving symmetry properties to the mechanism. Design parameters of the mechanism are depicted in the Fig. 4, and they can be identified for the k-th leg mechanism (k=1, 2, 3) as: a_k is the length of the frame link; b_k is the length of input crank; c_k is the length of the coupler link; d_k is the length of follower crank and h_k is the length of connecting bar CB. The size of the movable platform MP and fixed base FP is given by the distance r_p and r_f as depicted in Fig. 4. Points H and O are the center points of MP and FP, respectively. Points O_k is the middle point of the frame link a_k, J_k is the connecting points between the k-th leg mechanism and platform MP. The design parameters of the prototype CaPaMan2 bis shown in Fig. 4 are listed in Tab. 1.

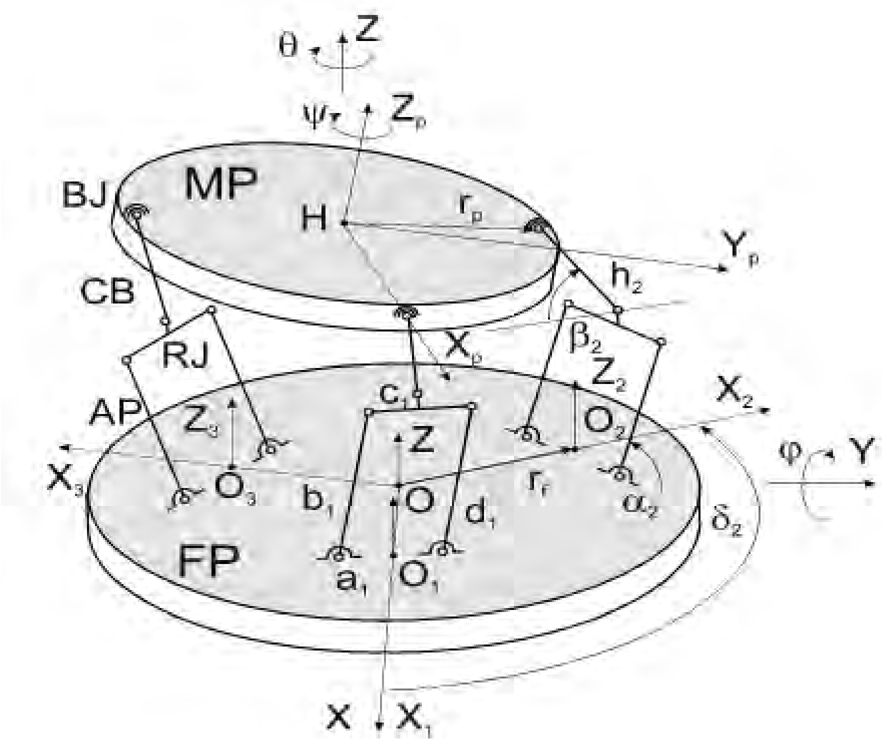


Figure 4. A scheme of mechanism for CaPaMan-2bis with design parameters (Liang et al., 2009).

Table 1 – Mechanical design parameters of CaPaMan-2bis prototype in Fig. 4 (Liang et al., 2009).

$a_k = c_k$ (mm)	$b_k = d_k$ (mm)	r_p (mm)	r_f (mm)	h_k (mm)	$I_{cxy}=I_{cxy}=I_{cyz}$ (kg.mm^2)
100	100	65	65	50	0
α_k (deg)	β_k (deg)	m (kg)	τ (N/m)	I_{czz} (kg.mm^2)	$I_{cxx}=I_{cxy}$ (kg.mm^2)
45~135	30~120	2.3	2	24600	12400

An inertial frame OXYZ has been assumed to be fixed to the base FP. A moving frame $HX_PY_PZ_P$ has been attached to the platform MP. OXYZ has been fixed with Z axis orthogonal to the FP plane and X axis as coincide with the line joining O to O_1. The moving frame $HX_PY_PZ_P$ has been fixed to the platform MP with ZP orthogonal to the MP plane and X_P axis as coincident to the line joining H to J and Y_P to give a Cartesian frame. The angle δ_k is the structure rotation angle between OX_1 and OX_k as well as between HJ_1 and HJ_k. They are equal to $\delta_1 = 0$, $\delta_2 = 2\pi/3$ and $\delta_3 = 4\pi/3$.

The position of H in the center of movable platform MP is represented by coordinates x, y and z. The orientation of MP with respect to the fixed base FP can be represented with the three Euler angles θ, φ and ψ as depicted in Fig. 4, where θ is a rotation about the Z axis; φ is a tilting rotation about the YP axis, which is the Y axis after a θ rotation; and

ISSN 2176-5480

ψ is a rotation about the ZP axis. For each leg mechanism, the kinematic variables are the input crank angles α_k and the passive revolute angle β_k. CaPaMan-2bis is a three DOF spatial mechanism. When one leg mechanism is moved, a coupled movement with orientation angles and position translation is obtained. A quasi translation movement can be prescribed in XY plane by fixing one leg mechanism and synchronizing the motions of other two leg mechanisms. The platform MP has a yaw movement that is coupled with a translation along Z axis even when the three leg mechanisms have the same input motions (Liang et al., 2009).

The shape and size of all the links has been redesigned to have higher stiffness. Ball bearings have been installed at the rotational joints in order to reduce friction (Carbone and Ceccarelli, 2005).

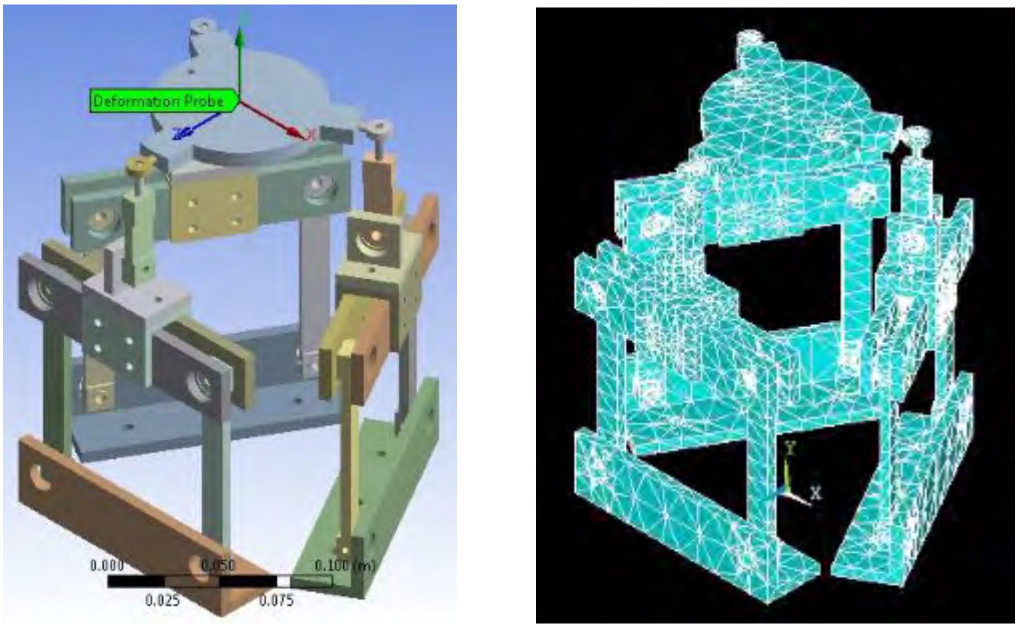
3. FINITE ELEMENT ANALYSIS OF CAPAMAN-2BIS STIFFNESS

The CaPaMan-2bis structure was built tridimensional in the SolidWorks® software and modeled in finite elements using the software ANSYS®. Both models are shown in Fig. 5.

The fixed platform was removed from the finite element analysis because it does not affect the static behavior of the structure. In the middle of the mobile plate MP were applied loads.

This model of CaPaMan-2bis structure considers the flexibility of all elements, including that of the joints. When doing the finite element modeling in the ANSYS® software the AQWA environment of the software allowed us to define all of the structure joints (rotational and spherical), as shown in Fig. 5(a), as well as the fixed constraint in the points where the rotational actuators are placed, in order to evaluate the structure statically. The FEA modeling joints is complicated by the necessity of defining the joint stiffness parameters, thus, in this paper the joints are modeled using the standard element MPC184. The properties used to segments are: aluminum EN AW-1200, E = 7e+10 N/m², G = 2.7e+10 N/m².

The structure was then meshed in low resolution, triangular element, automatically defined by the software as "medium size elements", in order to lower the processing and simulation time, given that the more complex the structure is and the higher the resolution of the mesh is, more powerful the computer needs to be and more time will be consumed. The computer used had an Intel® Core™ 2 Duo P8700 2.54 GHz processor with 4 Gb of RAM memory and an Intel® Mobile Graphics card. In such configuration, the meshing took a few seconds and the static evaluation no more than a minute, Fig. 5(b).



(a) (b)

Figure 5. (a) CAD model of CaPaMan-2bis Aqwa; (b) FEA model of CaPaMan-2bis.

The idea of this model is by the application of a force in the center of the moving plate, obtained the values of flexible linear and angular displacements for this point, and through these values of displacement, combined with the value and direction of the applied force evaluated the stiffness of the structure.

It is important to say that this analysis depends on the configuration of the structure, and in this paper, the CaPaMan-2bis was studied in the vertical position, that is, with all legs making a 90º angle with the ground, Fig. 5. Table 2 shows the results obtained for linear displacements in the central point of MP in function of applied load.

ISSN 2176-5480

Table 2. Displacement values for the center of the moving plate.

Displacement	F_x = 100N	F_y = 100N	F_z = 100N	F_x = F_y = F_z = 100N
δ_x [mm]	0.2644	0	0	0.2644
δ_y [mm]	0	0.0043	0	0.0042
δ_z [mm]	0	0	0.2644	0.2644
USUM [mm]	0.2644	0.0043	0.264	0.3739

Figure 6 shows the results obtained in simulations for the total deformation of the structure when applied the loads $F_x = F_y = F_z$ = 100N. It's notable that the structure's legs had little load, and the greatest displacements were found in the moving plate MP. Still, all displacements were very little, even in this platform. The displacement in vertical direction is minimum.

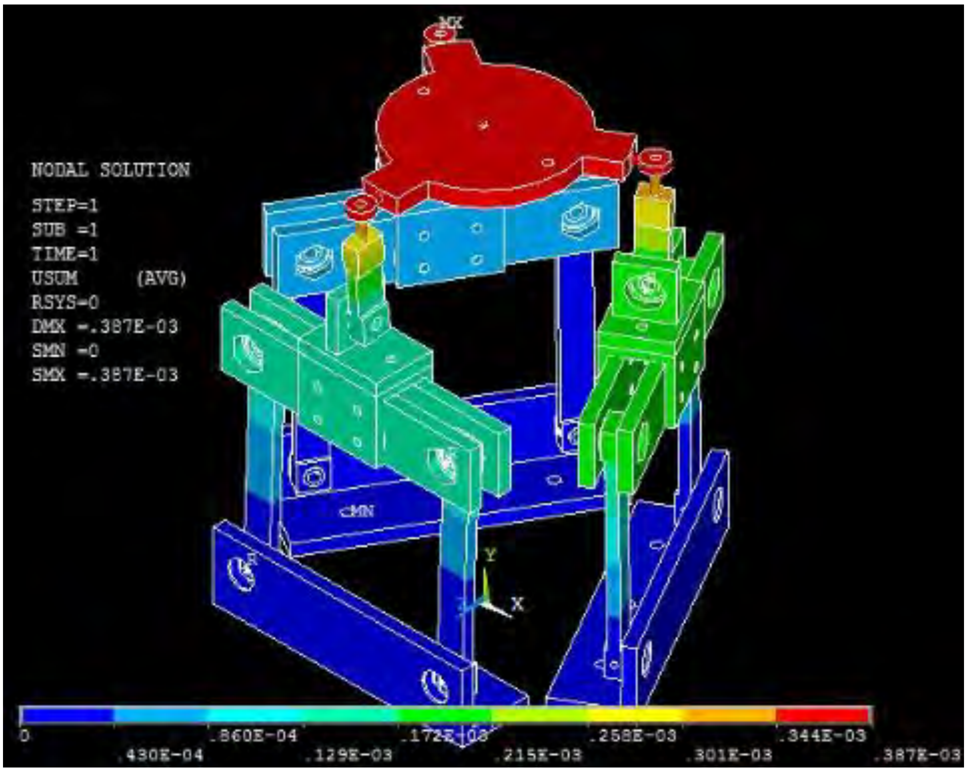


Figure 6. FEA model results of the CaPaMan-2bis when applied the loads $F_x = F_y = F_z$ = 100N.

4. CONCLUSIONS

This paper presented the stiffness analysis of CaPaMan-2bis (Casino Parallel Manipulator version 2 bis) robotic structure using the Finite Element Analysis (FEA).

In this paper was performed the review and an exposition of the advantages and disadvantages of using the FEA.

For application of the presented methodology the kinematic model of the structure and dimensions of its elements (segments and joints) must be known.

The use of *FEA* method produces satisfactory results, but its major drawback is the need to develop a model for each position to be examined and getting your mesh, and then apply a (solver) finite elements software. Thus, for example, to map the stiffness of the structure, using the *FEA* method is inconvenient, but to validate experimental results and news methods of stiffness analysis is highly recommended and used.

Future research includes the experimental tests to validate and adjust the proposed FEA model of CaPaMan-2bis considering the experimental joints parameters.

5. ACKNOWLEDGEMENTS

The authors are thankful to UFU, FEMEC, CNPq and FAPEMIG for the partial financing support of this research work.

6. REFERENCES

Aginaga, J., Zabalza, I., Altuzarra, O., Nájera, J., 2012. "Improving static stiffness of the 6-RUS parallel manipulator using inverse singularities", *Robotics and Computer-Integrated Manufacturing* 28, Spain.

Aguirre, G., Acevedo, M., Carbone, G., Ceccarelli, M., 2003. "Kinematic and Dynamic Analyses of a 3 DOF Parallel Manipulator by Symbolic Formulations", *Thematic Conference on Advances in Computational Multibody Dynamics, ECCOMAS*, Lisbon, Paper MB2003-010.

Bouzgarrou, B.C., Fauroux, J.C., Gogu, G., Heerah, Y., 2004. "Rigidity Analysis of T3R1 Parallel Robot with Uncoupled Kinematics", *Proc. Of the 35th International Symposium on Robotics*, Paris, France, March.

Brenner, S. C., Scott L. R., 2008. "The Mathematical Theory of Finite Element Methods", *Texts in applied mathematics*, Vol. 15, Springer.

Carbone, G., Ceccarelli, M., 2006. "A Procedure for Experimental Evaluation of Cartesian Stiffness Matrix of Multibody Robotic Systems", *15th CISM IFToMM Symposium on Robot Design, Dynamics and Control, Romansy*, Montreal, Paper Rom04-24.

Carbone, G., Ceccarelli, M., 2005. "A Serial Parallel Robotic Architecture for Surgical Tasks", *Robotica*, Vol. 23, pp. 345-354,.

Carvalho, J.C.M., Ceccarelli, M., 2001. "A Closed-Form Formulation for the Inverse Dynamics of a Cassino Parallel Manipulator", *Multibody System Dynamics*, No. 5, pp. 185-210.

Ceccarelli, M., 1997. "A New 3 D.O.F. Parallel Spatial Mechanism", *IFToMM Journal Mechanism and Machine Theory*, Vol. 32, No. 8, pp. 895-902.

Ceccarelli, M., Carbone, G., 2002. "A Stiffness Analysis of CaPaMan (Cassino Parallel Manipulator)", *Mechanism and Machine Theory*, Vol. 37, No. 5, pp. 427-439.

Ceccarelli, M., Figliolini, G., 1997. "Mechanical Characteristics of CaPaMan (Cassino Parallel Manipulator)", *3rd Asian Conference on Robotics and Its Application,* Tokyo, pp. 301-308.

Ceccarelli, M., Fino, P.M.D., Jimenez, J.M., 2002. "Dynamic Performance of CaPaMan by Numerical Simulations", Mechanism and Machine Theory Vol.37, pp. 241-266.

Clinton, C.M., Zhang, G., Wavering, A.J., 1997. "Stiffness Modeling of a Stewart-Platform-based milling machine", *Trans. Of the North America Manufacturing Research Institution of SME*, Vol. XXC, pp. 335-340, Lincoln, NB, US, May 20-23.

Corradini, C., Faroux, J.C., Krut, S., Company, O., 2004. "Evaluation of a 4 degree of freedom parallel manipulator stiffness", Proc. *Of the 11th World Cong. In Mechanism & Machine Science, IFTOMM'2004*, Tianjin, China, April 1-4.

Deblaise, D., 2006. "Contribution à la modélisation et à l'étalonnage élasto-géométriques des manipulateurs à structure parallèle", *Thesis (in French)*.

El-Khasawneh, B. S., Ferreira, P. M., 1999. "Computation of stiffness and stiffness bounds for parallel link manipulator", *Int. J. Machine Tools & manufacture*, vol. 39(2), pp. 321-342.

Dong, W., Du, Z., Sun, L., 2005. "Stiffness influence atlases of a novel flexure hinge-based parallel mechanism with large workspace", Proc. off IEEE ICRA: Int. Conf. on Robotics and Automation, Barcelona, Spain, April 18-25.

Gogu, G., 2002. " Structural Synthesis of Parallel Robotic Manipulators with Decoupled Motion", *Internal Report ROBEA MAX* – CNRS.

Gonçalves, R. S., 2009. "Estudo de Rigidez de Cadeias Cinemáticas Fechadas", 239f.. *Tese de Doutorado*, Universidade Federal de Uberlândia, Uberlândia.

Gonçalves, R. S., Carvalho, J. C. M, 2008. "Stiffness analysis of parallel manipulator using matrix structural analysis". EUCOMES 2008, *2-nd European Conference on Mechanism Science*, Cassino, Italy.

Hernandez-Martinez, E. E.; Conghui, L.; Carbone, G.; Ceccarelli, M.; Lopez-Cajun, C. S., 2010. "Experimental and Numerical Characterization of CaPaMan 2bis Operation", *Journal of Applied Research and Technology*, Vol. 8, Num. 1, pp. 101-119 Universidad Nacional Autónoma de México, Mexico.

Hu, B., Lu, Y., Tan, Q., Yu, J., Han, J., 2011. "Analysis of stiffness and elastic deformation of a 2(SP+SPR+SPU) serial–parallel manipulator", *Robotics and Computer-Integrated Manufacturing* 27, China.

Huang, T., Zhao, X., Whitehouse, D.J., 2002. "Stiffness estimation of a tripod-based parallel kinematic machine", *IEEE Trans. On Robotics and Automation*, Vol. 18(1).

Kobel, P., Clavel, R., 2011. "Micro Robot for Rotary Desktop Assembly Line", Switzerland.

Li, J., 2009. "Design of 3-DOF Parallel Manipulators for Micro-Motion Applications", *Master's Thesis*, University of Ontario Institute of Technology, Canada.

Li, Y.W., Wang, J.S., Wang, L.P., 2002. "Stiffness analysis of a Stewart platform-based parallel kinematic machine", *Proc. Of IEEE ICRA: Int. Conf. On Robotics and Automation*, Washington, US, May 11-15.

ISSN 2176-5480

Liang, C., Carbone, G., Ceccarelli, M., Hernandez, E., 2009. "A Comparison of Simulations and Experimental Tests on Operation Performance of CaPaMan2 bis", *Proc. Of the IEEE Int. Conference on Mechatronics and Automation*, Aug. 9-12, Changchun, China.

Liang, Q., Zhang, D., Chi, Z., Song, Q., Ge, Y., Ge, Y., 2011. "Six-DOF micro-manipulator based on compliant parallel mechanism with integrated force sensor", *Robotics and Computer-Integrated Manufacturing* 27, China.

Nava-Rodriguez, N.E., Carbone, G., Ceccarelli, M., 2006. "CaPaMan 2bis as Trunk Module in CALUMA (CAssino Low-cost hUMAnoid Robot)", *2nd IEEE International conference on Robotics, Automation and Mechatronics, RAM 06*, pp. 347-352, Bangkok.

Ottaviano, E., Ceccarelli, M., 2006. An Application of a 3-DOF Parallel Manipulator for Earthquake Simulations, *IEEE Transactions on Mechatronics*, Vol. 11, No. 2, pp.240-146.

Ottaviano, E., Ceccarelli, M., 2002. "Optimal Design of CaPaMaN (Cassino Parallel Manipulator) with Prescribed Orientation Workspace", *International Journal of Robotics*, Vol. 20, pp. 159-166.

Rade, D. A., 2001. "Método dos Elementos Finitos em Engenharia Mecânica". *Apostila*.

Rezaei, A., Akbarzadeh, A., Akbarzadeh-T., M., 2012. "An investigation on stiffness of a 3-PSP spatial parallel mechanism with flexible moving platform using invariant form", *Mechanism and Machine Theory* 51, Iran.

Taghvaeipour, A., Angeles, J., Lessard, L., 2010. "Online Computation of the Stiffness Matrix in Robotic Structures Using Finite Element Analysis" , *Department of Mechanical Engineering and Centre for Intelligent Machines*, McGill University, Canada.

Tian, H., Wu, D., Du, Z., Sun, L., 2010. "Design and Analysis of a 6-DOF Parallel Robot Used in Artificial Cervical Disc Replacement Surgery", *IEEE International Conference on Information and Automation*, June 20 - 23, Harbin, China.

Zhou, Z., XI, J., Mechefske, C .K., 2006. "Modeling of a Fully Flexible 3PRS Manipulator for Vibration Analysis", *Journal of Mechanical Design*, Vol. 128, pp 403-412.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.