# MODELING OF A MICRO-GRIPPER COMPLIANT JOINT USING COMSOL MULTIPHYSICS SIMULATION

Mihăiță Nicolae ARDELEANU, Veronica DESPA, Ioan Alexandru IVAN
Valahia University from Targoviste

E-mail: mihai.ardeleanu@valahia.ro, dumiver@yahoo.com, ivan@valahia.ro

***Abstract.*** *COMSOL Multiphysics is a popular finite elements modeling (FEM) software tool used in by most research teams around the world. To validate its working performance, the present paper introduces a model that simulates the behavior of a micro robotics compliant arm. The device is in progress under the research project ADMAN [1]. Emphasis is laid on designing the geometrical model, on applying the mechanical loads and on observing the simulation results in terms of beam stiffness, maximum stress at compliant joint level and eigen frequencies values.*

***Keywords***: *COMSOL Multiphysics, geometrical model, simulation, compliant joint.*

## 1. INTRODUCTION. COMSOL MULTIPHYSICS MODELING.

Admittedly, mathematical modeling is an important part of the research work in the developing scientific and engineering fields [2]. The connection between idea and the concept (the prototype) is the mathematical modeling and simulation, thus enabling the rapid understanding of the quantitative and qualitative aspects of the study, from both scientific and engineering points of view.

Using languages like Java (for interfaces) and C/C$^{++}$ (for solving methods), the Comsol Multiphysics environment provides genuine performance in this sense [3]. In this case, the mathematical modeling of a compliant micro-gripper [4], [5] includes:
- The equation-based description of the studied mechanism;
- Determining the partial differential equations governing the studied model;
- Determining the mechanism’s geometry, required by the modeling and the corresponding limiting-conditions;
- Determining the analytical/numerical methods for solving the equations.

The Comsol Multiphysics modeling process uses the mathematical modeling described above and requires the following steps:
- Determining the compliant structure’s physics (Structural Mechanics Model – in this case);
- Drawing of the micro-gripper (2D/3D);
- Defining the material properties of what the solid is made of and establishing the boundary condition;
- Configuring and constructing the mesh network (free, mapped);
- Selecting and configuring the solver (static, transition, eigen frequencies);
- Post processing of the results.

## 2. MODELING STRATEGY OF A MICRO-GRIPPER COMPLIANT JOINT USING COMSOL MULTIPHYSICS SIMULATION

### 2.1. Geometric concept of the compliant joint

Figure 1 shows the compliant joint concept, whose basic idea include a rectangular free-clamped beam and a cylinder-cut compliant articulation of radius R used to render the structure flexible along the direction of interest.

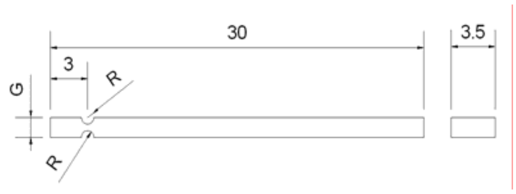


**Figure 1. The geometric concept of the compliant joint (dimensions are in mm)**

The basic parameters of the geometrical model are:
L – total length of the piece (30 mm in our case);
$L_1$ – the distance from the clamping surface to then recessed surface of the cutting cylinder (has been chosen as a 1/10 of the L beam);
H – the height of the piece;
G – the thickness of the piece;
R – the cutting cylinders radius.

### 2.2. Geometric modeling of the compliant joint

The compliant joint design performed so as to obtain an optimal geometry, requires the careful study of its functionality as part of the ensemble. From this point of view, the technical application that the compliant joint is applied in requires positioning accuracy at low operating frequency.
The manufacturing is based on a technology of high precision layer-by-layer deposition, which allows complex geometries we shell focus on the compliant joint which is a building block of a future microgripper.

In the given case, the relevant geometry is simple, based on the idea of a recessed parallelepipedic body, subjected to flexed bending produced by concentrated forces applied perpendicular to the longitudinal axis of the beam (figure 2).

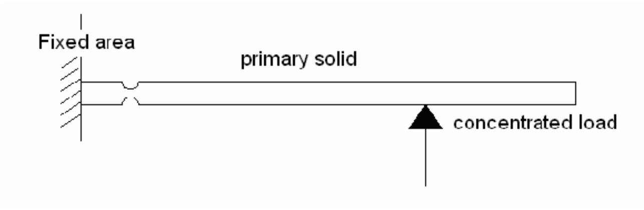


**Figure 2. The mechanical loading is applied transversally to the beam axis**

The idea of conferring elasticity in a single direction of the primary structure is by cutting-out the cylindrical areas from the initial solid piece. This reduction of the beam section is in an area close to the clamping zone. Figure 3 presents the geometric method of cutting to obtain the required primary structure elasticity.

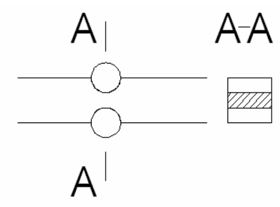


**Figure 3. Elasticity cutting of the primary structure**

The working parameters, in order to get closer to the optimal iterative configuration of the compliant joint, are: radius R and thickness G.
Following the variation of the two parameters, there will result a plan of simulated values of the elastic constant parameter, calculated based on the expression:

$$K = \frac{F}{\delta_{\max}} \left[\frac{N}{m}\right]$$

The force F will be imposed to a constant value throughout all the simulations (10mN), the maximum displacement provide the beam stiffness according to the simple formula above.

The value of parameters R and were set within the following intervals:

$G \in [1,3; 1,5]$ (mm)
$R \in [0,16; 0,5]$ (mm)

The intervals will be covered in three points for each parameter.

The results obtained will be shown in a three-dimensional chart as shown in figure 4:

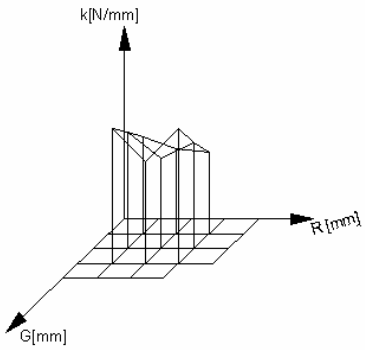


**Figure 4. Compliant structure elasticity FEM results**

### 2.3. The modeling-simulation procedure in COMSOL Multiphysics for the compliant arm

The piece geometry is imposed drawing functions under COMSOL or importing from other CAD environments. After launching the application, a physical model or a predefined model may be selected. In this case we used the "Box" and "Cylinder" functions to define the boundary geometries of the composite structure. From the "Box" element there were "subtracted" the two straight cylinders placed on the sides of the central axis, at a fixed distance at 1/10 from the steady end. The function used to "subtract" the 3D structures is very intuitive, thus obtaining modeling complex structures.
Figure 5 presents the compliant joint which makes the object of modeling in this paper.

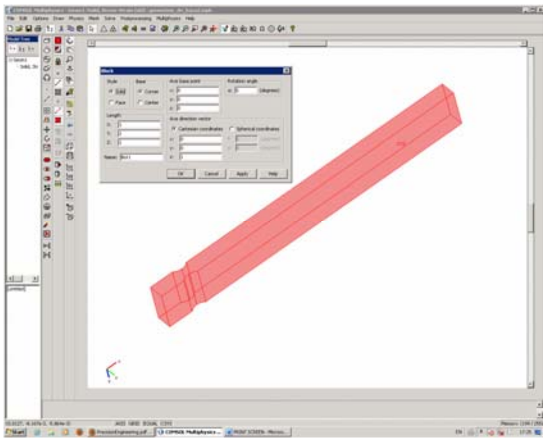

**Figure 5. COMSOL Multiphysics – Drawing Geometry**

Figure 6 shows a screenshot of the operation used to actually select/set the material of which the piece is made of. COMSOL Multiphysics keeps on extensive database listing the properties of the materials used to make the parts.
There is also the possibility of extending the standard database, by entering new material tensors in the working format for which the physical-chemical-mechanical properties are particular. In this case we used a titanium alloy, existing in the database.

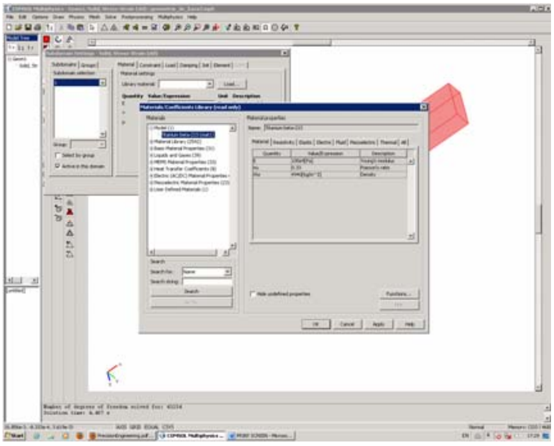

**Figure 6. COMSOL Multiphysics – Stating the physical material properties**

In order to evaluate the geometry, a concentrated external force and a target parameter to be checked through the variation of geometry definition are required.

The parameter watched is the elastic constant obtained for each geometrical configuration modeled. This constant is the force-displacement ratio (the quantitative cause-effect ratio).

The force is a mechanical cause that produces the corresponding maximum tip displacement in the piece of a specific geometry.

In this case, Figure 7 shows the numerical setting of this force at the value of 10mN.

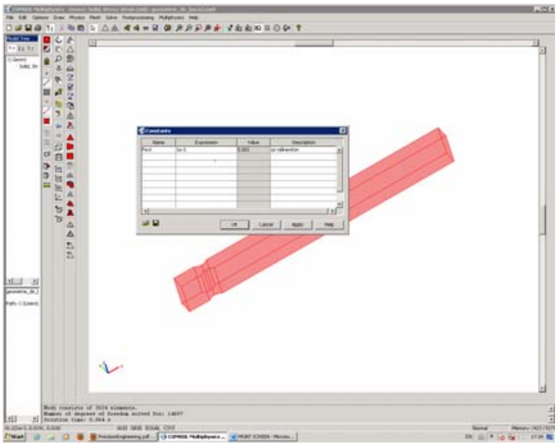

**Figure 7. COMSOL Multiphysics – Stating the constants (end tip applied force)**

The developed mechanical load requires a fixed base for the structure. We will consider the piece as being embedded, which means that the displacement is null on the three coordinates (x, y, z) for the left area in Figure 8.

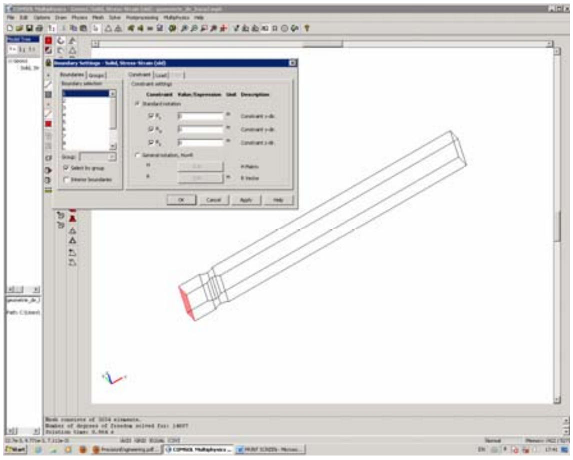

**Figure 8. COMSOL Multiphysics – Setting the boundary and point conditions**

Figure 9 shows the application of the 1 mN force, spread evenly across the two points that border the edge considered. Effectively, in points no. 17 and 18, the total force is split into two equal parts, of 5mN parallel and of same direction.

For simulation purposes, the solid structure should be digitized into a dense mesh. The program has set several basic element geometries triangular, rectangular, which generate the mesh network. In this case, the element used is of triangular type. The mesh digitization may be refined by decreasing the basic elements use and multiplying them by default so as to cover the full volume of the part. In this case, the accuracy in rendering the loads/deformations is higher, but increases accordingly the computing time of the simulated model, as the number of mathematical equations (degrees of freedom) is expanding the number of calculations.

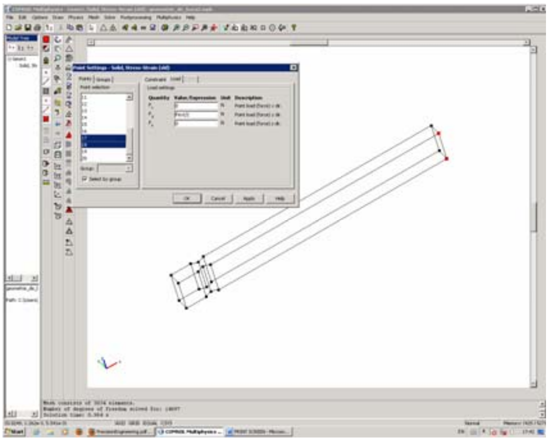

**Figure 9. COMSOL Multiphysics – Setting the transverse force $F_{ext}$ into two points equally distributed**

Figure 10 shows the compliant joint mesh digitization, a mesh size of about 10.000 is sufficient for the given model.

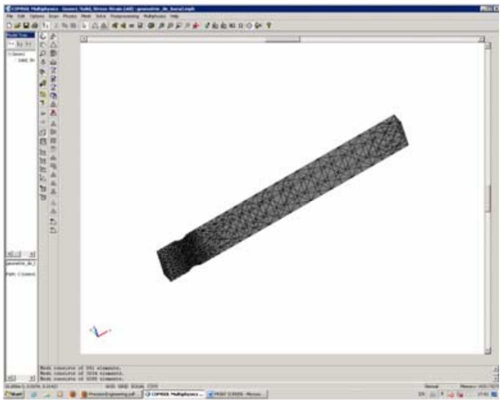

**Figure 10. COMSOL Multiphysics – Meshing the object (free mesh, trapezoidal with imposed maximal size and “growing” ratio)**

Figure 11 shows a snapshot of the “math” behind the model.
The simulation results (post processing) are presented numerical and in rainbow color mode. The highest numerical values tend to Red, while the minimum values tend to Blue.

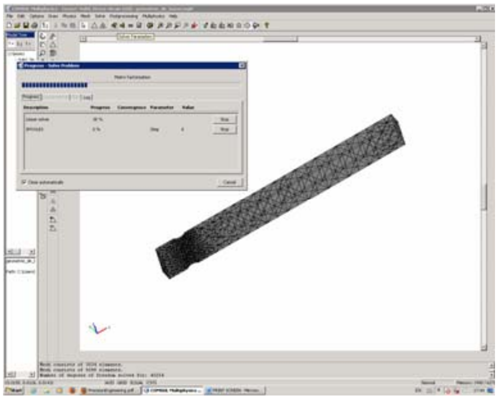

**Figure 11. COMSOL Multiphysics – Solving the model**

Figure 12 shows a screen capture of some results obtained for the compliant joint single geometry. From the right scale there may be calculated the elastic constant from the maximum displacements value of the upon applying the 10 mN of external force.

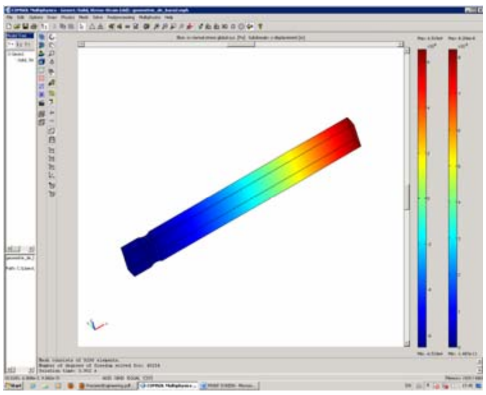

**Figure 12. COMSOL Multiphysics – Post processing of the transverse displacement**

Another important mechanical parameter to optimize the dynamics [6] of the final geometry of the compliant joint is the evolution of Eigen_frequency. Figure 13 shows the area from the menu where the mathematical functions are set to perform the Eigen_frequency calculations.

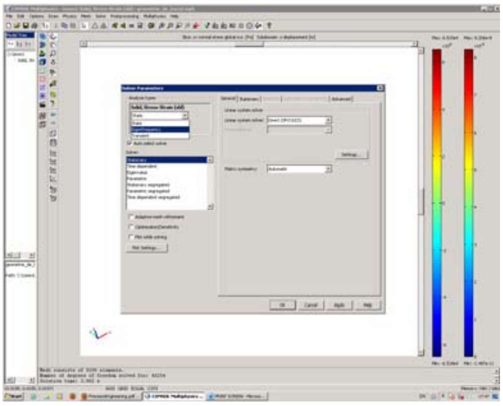

**Figure 13. COMSOL Multiphysics – Setting the solver from “static” to “Eigen_frequency” calculation**

The window in Figure 14 shows the values of the first six Eigen_frequency values of the modeled structure.

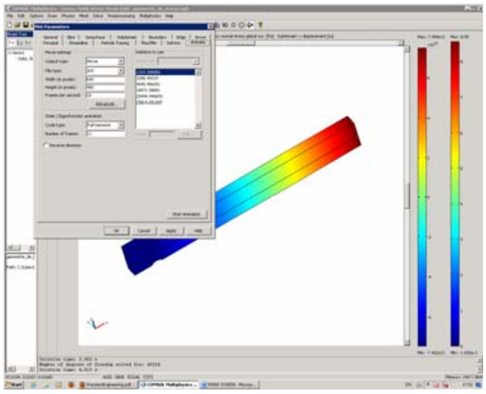

**Figure 14. COMSOL Multiphysics – Eigen_frequency list evaluation**

The simulation process ends with an animation showing how the compliant joint solid vibrates, in the vibration mode 1. Figure 15 shows a screenshot of the relevant menu windows.

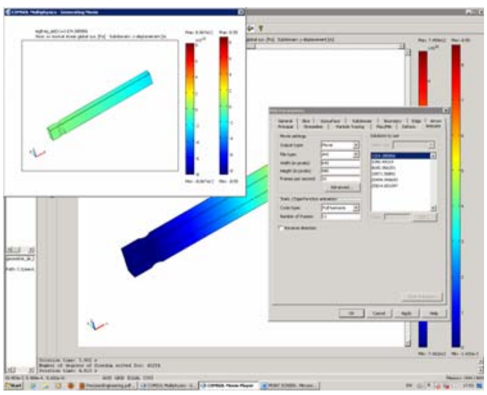

**Figure 15. COMSOL Multiphysics – Post processing interface – animation video rendering**

## 3. SIMULATION RESULTS AND THEIR INTERPRETATION

The Eigen frequency of the system is a primary indicator of its dynamics. The higher the Eigen frequency, the more improved the system's dynamics, and, possibly, the more insulated from external noise and vibration.

In this case, the system's vibration is not a rigorous value, as the system's geometry was not modeled entirely, but it provides us primary, valuable information of the system's future dynamics.

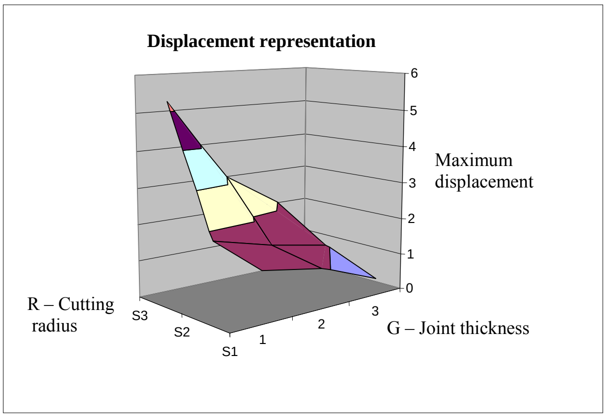


**Figure 16. 3D plot of the maximum displacement the cutting cylinder radius and beam thickness**

## 4. CONCLUSIONS

We extracted from simulation sessions, 9 numerical results in the same experimental force conditions and different geometries for element section. Our final goal was to obtain an optimal compliance of this element, following an imposed 1000 Hz limit value for natural frequency.
In figure 16, are shown the maximal deformation related to geometrical parameters G and R corresponding to an applied force of 10 mN.

We observed that the deformation did not exceed the 1 micron value for the section corresponding to G = 1,5 mm and R = 0,16 mm. Natural frequency for this case is around of 2000 Hz value.

The maximal deformation was obtained for G = 1,3 mm and R = 0,5 mm, and the corresponding displacement value was 5,3 μm. The natural frequency corresponding for this case is 1326 Hz.

In conclusion, the established compliant joint parameter values for the first experimental prototype are the pair (G = 1,3 mm and R = 0,5 mm).

## ACKNOWLEDGEMENT

National Project, Young Research Teams, PN-II-RU-TE-2011-3-0299, no. 85/05.10.2011, *"Advanced Devices for Micro and Nanoscale Manipulation and Characterization (ADMAN)"*.

## REFERENCES

[1] Romanian Project, Young Research Teams, PN-II-RU-TE-2011-3-0299: *"Advanced Devices for Micro and Nanoscale Manipulation and Characterization (ADMAN)"*.

[2] Neil A. Gershenfeld, *"The Nature of Mathematical Modeling"*, Cambridge University Press, 1999.

[3] William B. J. Zimmerman, *"Multiphysics Modeling With Finite Element Methods"*, Series on Stability, Vibration and Control of Systems, Series A: Volume 18, World Scientific, 2006.

[4] Simona NOVEANU, Dan MANDRU, Ioan Alexandru IVAN and Vencel CSIBI, *"Design and Modelling a Mini-System with Piezoelectric Actuation"*, 3rd European Conference on Mechanism Science (EUCOMES), Cluj-Napoca, Romania, 2010. Published into Mechanisms and Machine Science Vol. 5 (New trends in Mechanism Science) Springer Ed., pp. 125-133, ISBN 978-90-481-9688-3 , DOI 10.1007/978-90-481-9689-0_15, 2010.

[5] Simona NOVEANU, Dan MANDRU, Ioan Alexandru IVAN and Vencel CSIBI, *"Research Concerning the Ramp and Sinusoidal Command Signals of the Piezoelectric Miniactuators"*, Solid State Phenomena, Vols.166-167, pp. 321-326, 2010.

[6] N. Lobontiu, *"Dynamics of Microelectromechanical systems"*, Springer, 1st Edition, 2007.