

# ABAQUS USED IN SOLVING ELASTICITY PROBLEMS OF LENS MOUNTS

Horea HORA<sup>1</sup>, Cristina HORA<sup>2</sup>

<sup>1</sup> Faculty of Management and Technological Engineering, University of Oradea, e-mail:horeahora@yahoo.com

<sup>2</sup> Faculty of Energy Engineering and Industrial Management, University of Oradea, e-mail:chora@uoradea.ro

**Abstract** — Complex state of stresses and strains lens requires resorting to numerical analysis methods accessible through dedicated programs. Most suitable are those that practice the finite element method for solving problems of elasticity. The paper's goal is to solved elasticity problems of lens mounts using Abaqus software. The first part presented the model design, followed by post-processing of results.

**Keywords** — Optical glass, lens mounts, stress and strain, Abaqus software

## I. INTRODUCTION

THE Abaqus / CAE program, created by Hibbitt, Karlsson & Sorensen Inc., Version 6.3 is the finite element method [1] for solving various problems, covering a wide range of applications designed for elastic mechanical structures, elastic-plastic, hyperelastic - linear or nonlinear - in static or dynamic states, with the possibility of extending the study on thermal effects. The program also offers solutions to problems of heat transfer, fluid mechanics, or of electrical domain [2], [3].

Further, the examples will refer to solving a mechanical application, regarding the stress state of a set consisting of two parts, which's materials behave elastically.

## II. MODEL DESIGN

Model design, solving the system of equations, and post processing the results requires several of sequentially modules.

"Part", the first module, allows sketching a piece or importing its drawing from a graphics program such as AutoCAD, in specific formats (\*. dxf for plane sketches of some sections, or \*. sat for tridimensional bodies) or from other finite element programs. Abaqus software is compatible with models generated in versions of MSC Nastran, Cosmos, Catia software

etc., which can convert your own models to folders accessible through Abaqus.

All the created models were based on \*. sat files made in AutoCAD and imported into Abaqus.

Fig. 1 shows a printscreen of the "Part" module after importing 3D drawing of a converging lens. All the parts to form an analyzed aggregate are imported or drawn in this module.

In "Property" module, each piece is assigned a material and a section characteristic.

Fig. 2 shows the window through which a model for the convergent lens was chosen and the specific material characteristics for glass sort BK7, were introduced [4], [5].

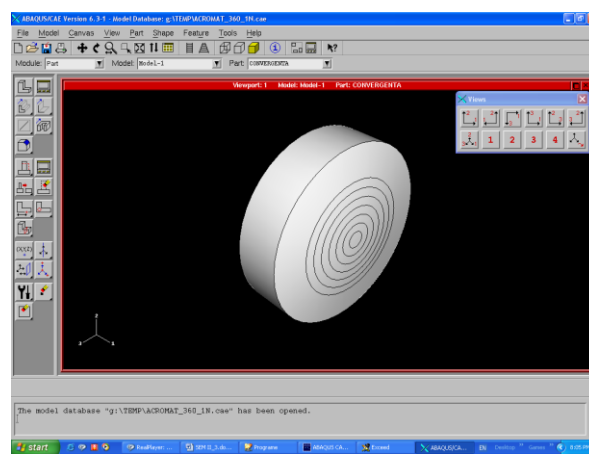


Fig. 1. Generating the lens with "Part" module

In the "Section Manager" window, in Fig. 3, it is seen that the two lenses have been assigned solid and homogeneous sections.

In the "Assembly" module instances of the components defined in module "Part" are created, resulting geometrically, the assembly to be analyzed. Fig. 4 presents a print screen where, for the converging and diverging lenses forming a doublet, instances were created and the assembly was formed.

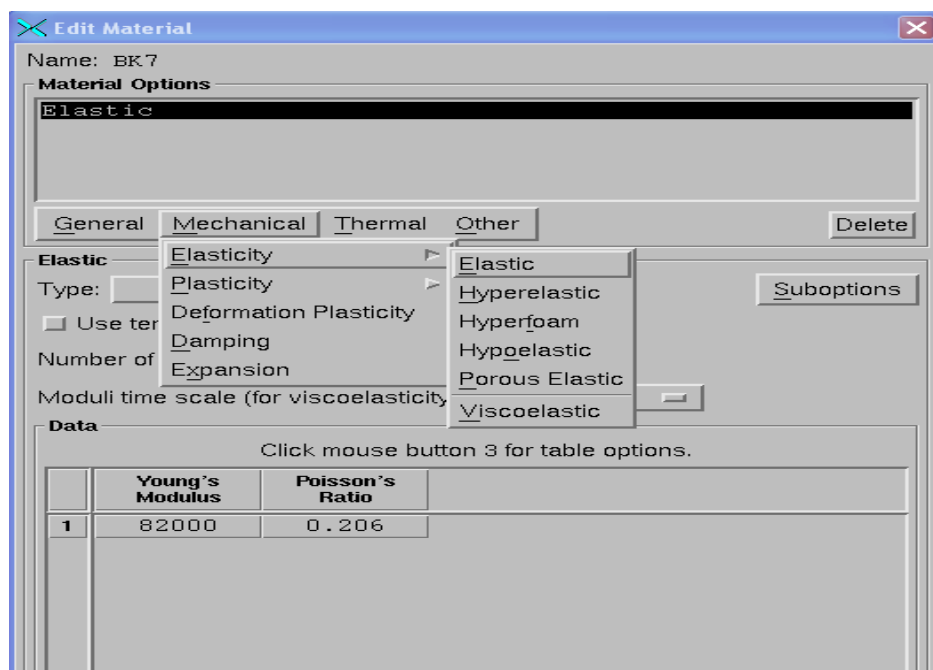


Fig. 2. Assigning the material constants using the "Property" module

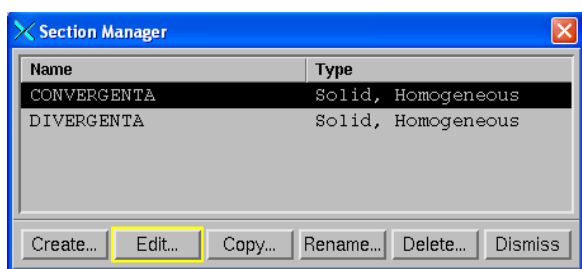


Fig. 3. Assigning the section with the "Property" module

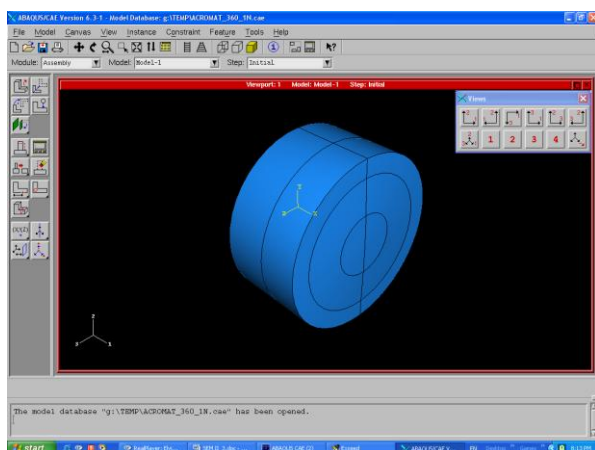


Fig. 4. Generating structure by creating courts in the "Assembly" module and partitioning to highlight the generatrix in a meridian plane

In addition, we performed a partitioning of both lenses, in a meridian plane (the track of the plan is visible on the spherical and lateral surfaces, cylindrical). Partitioning is necessary to bind the program that generates the sequence of tracks so as to divide the lenses so that on the meridian plane lines (which are geometrically part of the surface of the

dioptr curve generator) to be mesh nodes. The analysis actually aims the dioptr's deformation, respectively the nodes shifting on its surface. By partitioning, the assembly was virtually separated into two halves, which are separately discredited, but that provide that the adjacent elements have common sides and nodes [6].

The "Step" module requires baseline of applying external forces, the boundary conditions. Applying forces takes the form of a ramp signal, discredited in value increments imposed by the user.

In Fig. 5 the increasing increment of forces has the value 0.1 out of the maximum value (1). Since the entire calculation takes place iteratively, it is possible to decrease the increment, its minimum value can be indicated by the user (0.01 in Fig. 5), for if there are discontinuities and incompatibilities that cannot be exceeded mathematically.

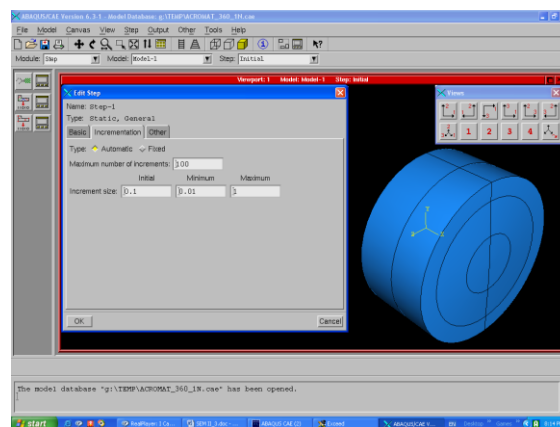


Fig. 5. Setting number and extreme values admitted to operate in the "Step" module

It is also noted that the maximum number of iterations may be setting (100 in Fig. 5) where the solution turns from start as being divergent and the procedure of calculation is started, but incompatibility issues might appear on the way. In this case, the number of iterations per step is increasing (limit 10) all smaller increments (up to 0.00001), but the solution is not converged and the time and memory consumption becomes useless.

A field of output variables and a "history" of them are created on increments, still in the "Step" module. In Fig. 6 is shown a window where the user has opted for determining stress, specific strains, displacements, reaction forces, contact pressure and volume variation, thickness and coordinates for the entire model.

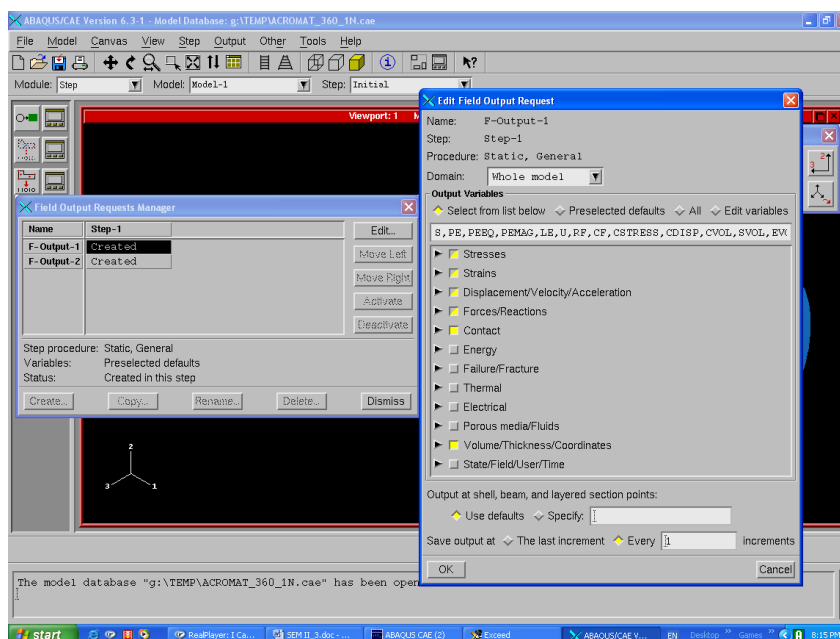


Fig. 6 Creating database files results according to user options in "Step" module

In "Interaction" module a condition about lens bonding was setting, by choosing a "tie" type constraint. In Fig. 7 the common surface of the bonded lenses is highlighted as well as the window in which the operator setting certain conditions on how the connection between the surfaces is shaped.

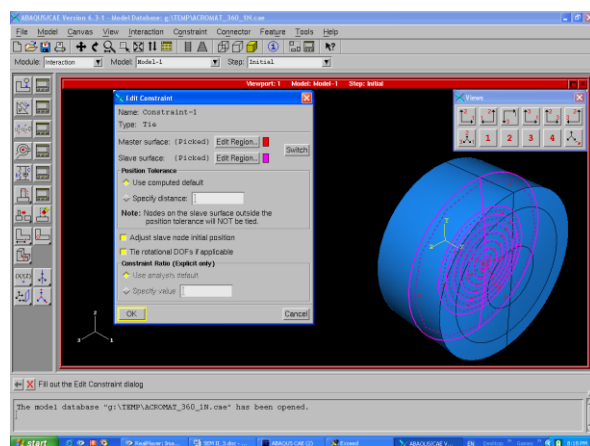


Fig. 7. Lens bonding simulation in "Interaction" module

The selection of the option "Adjust slave node initial position" is observed, which requires the design module of mesh to generate elements whose nodes should be common on the two surfaces in the initial

stage, and after the requests' application these links must be kept in such a way that no gaps between lenses appear.

In "Load" module the external forces and the boundary conditions are given. Fig. 8 shows the printscreen when is creating the fixed element of the structure.

This was associated with axial surface of the converging lens on the metal mount (the area is marked with red). The fixation corresponds to the "Encastre" option which setting null values for linear displacements and rotations at nodes under the area indicated.

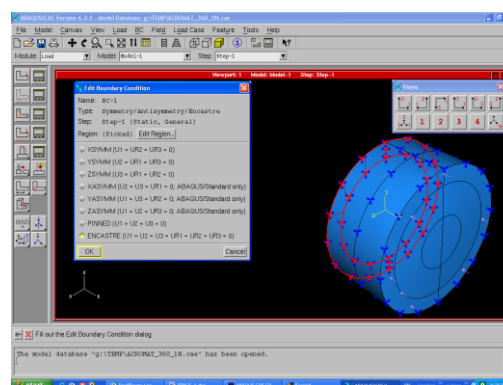


Fig. 8. Setting the boundary conditions in "Load" module

A second set of boundary conditions was created which bans cylindrical outer surface rotations (only linear displacements along the three axes are allowed) [7].

The value of these linear displacements hasn't been required, since the extension is not hindered by anything along the optical axis, and after y and z axis, the predicted values of the displacement are below the fitting motion.

On the contact surface between diverging lens ring and screw tightening its effect was modeled by applying a contact pressure on the related surface, marked in red in Fig. 9.

The dialog box that opens allows you to enter a numeric value for pressure in MPa.

In "Mesh" module, the first operation consists of choosing the type of finite element, which the operator proposes to discretize each created parts instance.

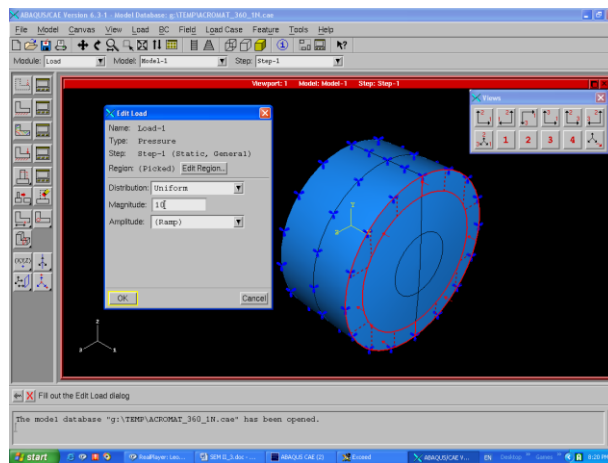


Fig. 9. Setting distributed load in "Load" module

In Fig. 10 the option for a standard, linear tetraedric element type ("tet"), meant for 3D analysis, is observed.

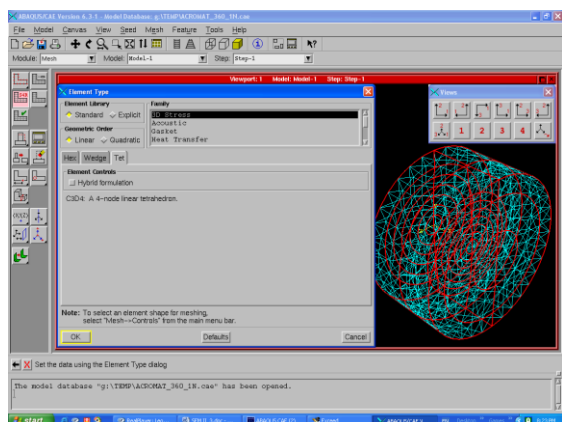


Fig. 10. Setting finite element type to be used for meshing in "Mesh" module

The mesh network is generated automatically. The operator can check its appearance through direct

visualization or mathematical tests of "Verify mesh", which indicate the presence of mathematically inappropriate elements (concave tetrahedra with dihedral angles greater than 120° or acute dihedral angles less than 30°).

Through "Query" menu and direct selection with the mouse you can view, in the "Text" section at the bottom of the window, the item type, the automatically assigned serial number and the nodes' numbers in the mesh (Fig. 11). Optionally, you can display all the elements' and nodes' names.

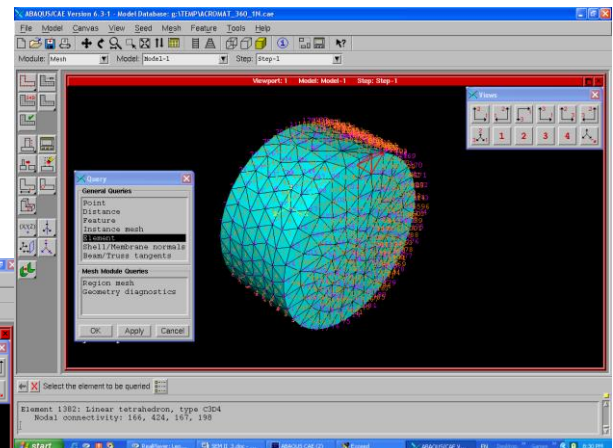


Fig. 11. Optional operations on individual elements' view, elements' and nodes' serial number display in the "Mesh" module

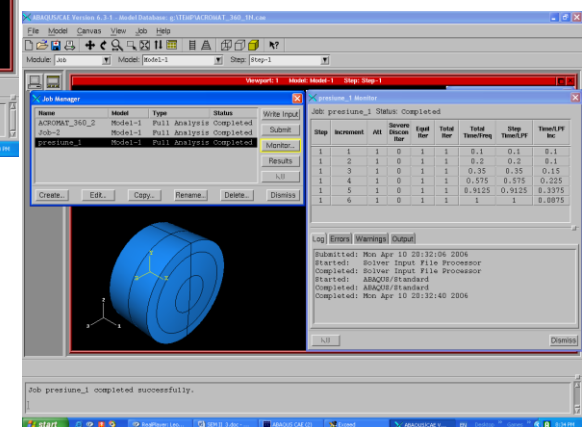


Fig. 12. Initialization and monitoring the calculation evolution in "Job" module

In "Job" module the finite system's analysis is initiated. Monitoring the evolution of solving the equations system can be traced in a special window stating real time increment, the number of iterations / increment and the equivalent "time" of achieving step (Fig. 12).

At the bottom of the monitor-window operator shall notify data on development and problem solving of the system by dialing messages appearing on "Tab" s "Log", "Errors", "Warnings" and "Output".

The "Results" button in "Job Manager" window allows the transition to post-processing results after a successful solving the system of equations.

Besides viewing through graphical interfaces, the Abaqus program allows the access to a computer file that contains a complete description of all operations performed in the program's modules.

### III. POST PROCESSING RESULTS

Solving the linear system of equations can be done. The post processing of the results involves determining specific parameters required by the operator. The deformation energy of the elements and von Mises stress are being automatically calculated. (Fig. 13).

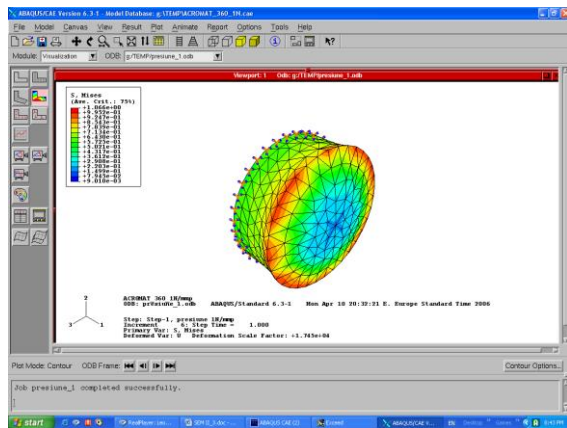


Fig. 13. Von Mises stress in "Visualization" module

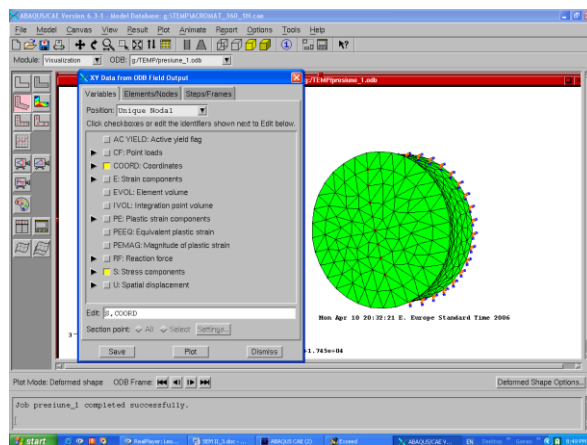


Fig. 14. Post processing example by setting global nodal coordinates' and stress components' calculus in the "Visualization" module

For further information check dedicated subroutines which demand elements' or nodes' of interest specification and sizes to be determined. In Fig. 14 for example, the calculation of voltage components and of the coordinates in deformed state is required.

In Fig. 15 the requirement is detailed through indicating von Mises stress.

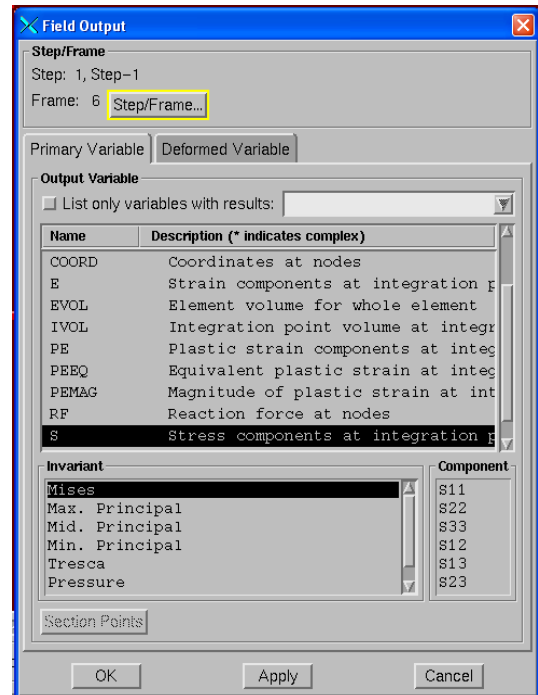


Fig. 15. Window detail of the requirement of stress components

Fig. 16 and 17 show figures related to the windows which require calculating the x coordinate corresponding to the optical axis and the nodal displacements in the same direction.

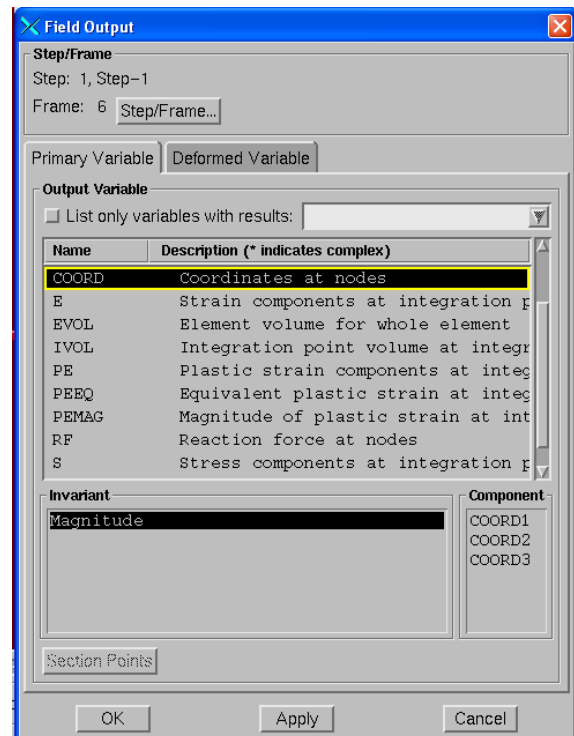


Fig. 16. Detailing the requirement of nodal coordinates display



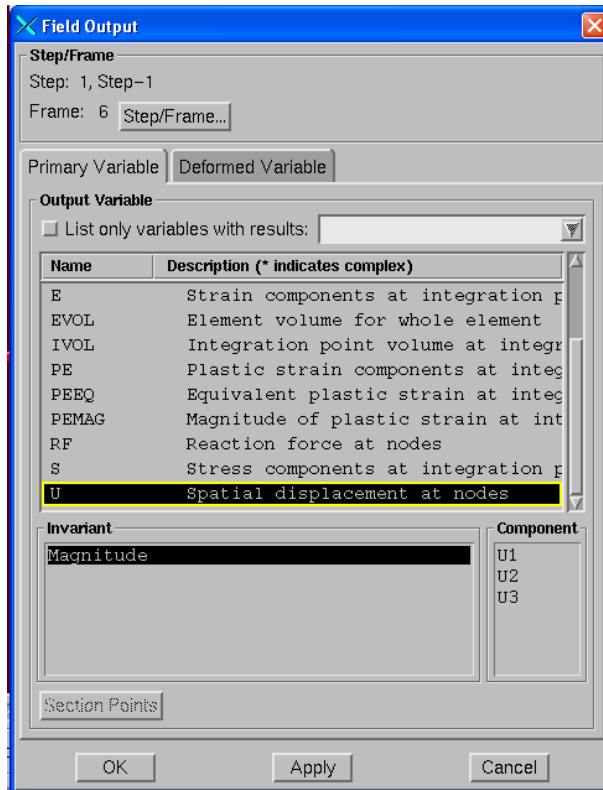


Fig. 17. Detailing the requirements for the nodal displacements display

As an immediate result of these requirements graphical results are obtained, consisting of representation of the deformed system (Fig. 18), having attached a chromatic scale or numerical values on predefined sets of nodes or nodes, in the "Text" at the bottom of the window.

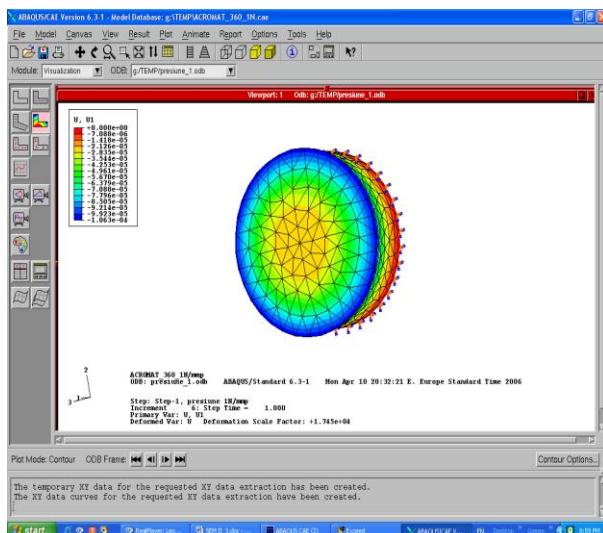


Fig. 18. Graphic results obtained from post-processing resulted from the requirements of the operator

#### IV. CONCLUSION

The theoretical foundations of the finite element method and characteristic of the method applied to elasticity problems specific to mechanical structures are summarized.

The fundamental part which is going to be used in the analysis of the optomechanic subassemblies is developed. It refers to the analysis of the triaxial state of stress and strain, and, given the particular forms of the lens type components it leads the descriptive approach to solving a problem of elasticity using linear tetrahedral finite elements.

In order to analyze the problem of lenses fixed in mounts, the specialized program Abaqus is proposed for being used, whose description is made on stages, as a concrete, definite example.

Running the Abaqus program illustrate the working mode and facilities on structural modeling and results post-processing.

#### REFERENCES

- [1] Zienkiewicz, O.C., *The Finite Element Method*, McGraw-Hill Co., 1977
- [2] Okruhlik, M., *Finite Element Homework and Programming Applications*, IFTOM, p. 3161-3165, Politehnica di Milano, 1995
- [3] Sinclair, D.C., *Optical Design Software*, in Handbook of Optics, vol.I, ch. 34, McGraw Hill Inc., NY,1995
- [4] Kingslake, R., *"Lens design fundamentals"*, Academic Press, N.Y.,1978
- [5] Shannon, R.R., *Optical Specification*, in Handbook of Optics, vol.I, ch.34, McGraw Hill Inc., NY,1995
- [6] Hecht, E., *Optics 3<sup>rd</sup> ed.*, Eddison Wesley Longman, Inc., NY, 1998
- [7] Gruescu C., Balaban, G., *"Analyse of transverse spherical aberration of a tilted lens"*, The 8.th Symposium on Mechanisms and Mechanical Transmissions with international participation, Timișoara, 2000, p.143-148