

Design and analysis of a coffee table

Haraga Georgeta, Ghelase Daniela, Daschievici Luiza
 Engineering Graphics and Industrial Design Department
 „Politehnica” University of Bucharest
 Splaiul Independentei No.313, Street, District 6, Postal Code 060042, Bucharest
 „Dunarea de Jos” University of Galati
 Romania
 g_haraga@yahoo.com, daniela.ghelase@ugal.ro, luiza.tomulescu@ugal.ro

Abstract: This paper focuses on the design and finite element analysis (FEA) of coffee table with center hole can be used with or without umbrella. Therefore, for a better graphic performance we have used CATIA V5 that allows the 2D/3D solids modeling with geometric constraints. In CATIA, we can simulate the whole very easy thing within a virtual reality (VR) using the applications called „Part Design and Generative Structural Analysis”.

Key-Words: CATIA V5, modeling, finite element analysis, mesh, sag, coffee table.

I. INTRODUCTION

CATIA is a product of Dessault Systèmes company and which in our days, is one of the most used CAD / CAM / CAE systems, providing a great variety of integrated solutions to satisfy all aspects of design and manufacture. The innovative abilities of CATIA may allow a high level of realism for various models of aesthetic and visual capacity to experiment with the virtual design options that can help increase creativity and innovation. All these are meant to optimize the entire manufacturing process.

In this paper we will plan and execute several operations to design and finite element analysis of a coffee table.

II. MODELING PROCESS

CATIA is one of many tools used by engineers and designers and is divided in many applications such as example: the Sketcher, Part Design, Drafting and Generative Structural Analysis ones being emphasized in a special way. This software gives us a variety of new tools that allow us easy documentation of drawings at any stage of development of production. For this, at each stage it is necessary a good plan for the chosen product in the process modeling.

Modeling of a coffee table will be made in the CATIA Part Design. In this context, we will open the application Mechanical Design→Part Design. [1] At the beginning, we will sketch the first profile in the y-z plane, as shown in figure 1.

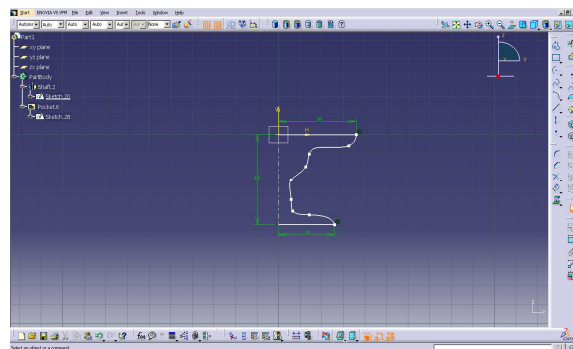


Fig.1 Creating the first feature

Moving from CATIA Sketcher application in CATIA Part Design can be done using the Exit Workbench. In the Part Design workbench, we can create a shaft which adds material. By using the Shaft command we will revolve with 360° and the first profile as shown in figure 2. [3]

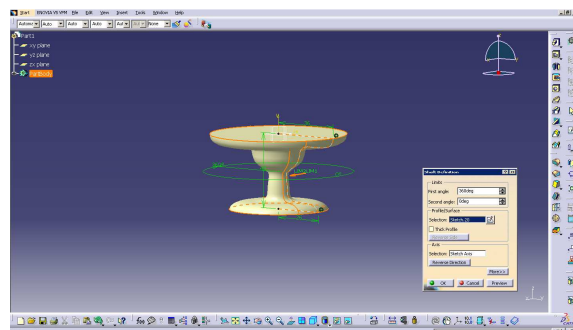


Fig.2 The Shaft command

Modeling of the coffee table is represented in figure 3 where we used several operations with Pocket, Hole and Properties commands.

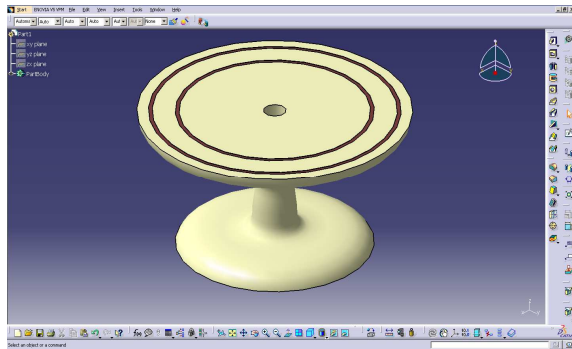


Fig.3 Modeling of the coffee table

The last step in the modeling process is the choice of material that is used. For this, we will select one of the materials that can be found in the Library dialog box and can be accessed by activating Apply Material icon. Thus, we use the Apply Material icon which opens the material database box and we choose the wood material as shown in figure 4. [3]

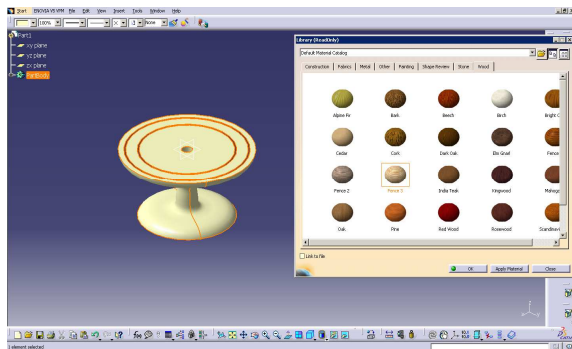


Fig. 4 Applying the wood material namely Fence 3

III. FINITE ELEMENT ANALYSIS (FEA)

In very many cases, Finite Element Analysis (FEA) is considered a powerful tool for solving specific engineering problems with complex geometry, loads, and material properties if it cannot be find analytical solutions. Use of FEA for the whole process consists of three main steps (see figure 5).

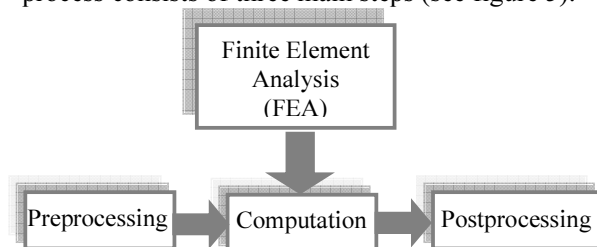


Fig. 5 Use of FEA for the whole process [2]

A. Preprocessing

In general, preprocessing is used for creating a finite element model and includes the following steps:

- ❖ Generating Mesh;
- ❖ Applying Restraints and Loads;
- ❖ Checking the Finite Element Model.

Creating a static analysis case assumes that we will analyze the static boundary conditions of the document CATAnalysis one after another. After the material choosing, we go into the menu to select Start – Analysis&Simulation - Generation Structural Analysis. Then, the "New Analysis Case" dialog box appears with Static Analysis as default option and then we click on the OK button as can be seen in the figure 6. [2], [3]

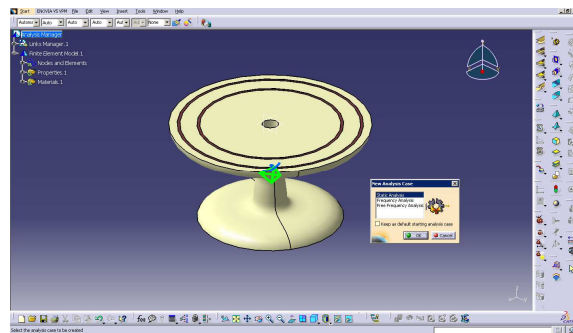


Fig.6 The New Analysis Case dialog box

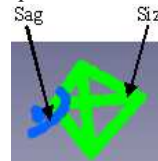
„Generative Structural analysis” is very useful to get different features of solid models in a 3D environment. Using these tools allows us to be able to analyze parts and determine their structural qualities before being made.

„Generative Structural Analysis” uses the Finite Element Method approach. Generating Mesh represents meshing the part to create Finite Element model using elements and nodes. [4], [8]

The mesh (size) and model properties (sag) are represented by two symbols in the 3D environment. The physical parameters size and sag ensure limiting the mesh size and can be found easily using the OCTREE Tetrahedron Mesh box, allowing the user to change the default values.

In FEA, restraints refer to applying displacement boundary conditions which is achieved through the Restraint toolbar. In the present problem, we can assume that the base of the longer section is clamped. The Clamp condition means that the displacements in all three directions are zero.

In FEA, loads refer to forces. The Load toolbar is used for this purpose (see in figure 7). [6]



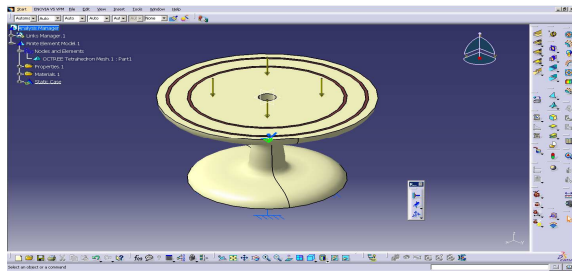


Fig.7 Applying Restraints and Loads options

B. Computation

Computation is the available solution of the finite element problem given by preprocessing to find out unknown displacement values and to run the analysis, we need to use the Compute toolbar by selecting the Compute icon, as shown in figure 8.

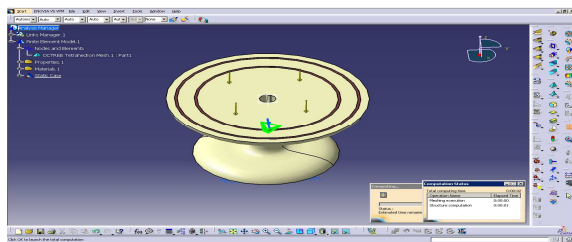


Fig.8 Starting the Computation process

Figure 9 provides information on the resources needed to complete the analysis.

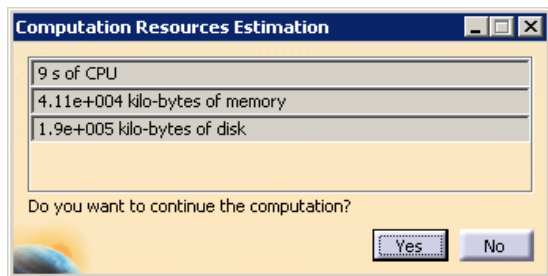


Fig. 9 Starting the Computation process and the Computation Resources Estimation box

C. Postprocessing

The last step in a finite element analysis is Postprocessing. The basic postprocessing toolbar is called Image and can be seen in figure 10.



Fig.10 The Image toolbar

The deformation image can be made using the Deformation icon as shown in figure 11.

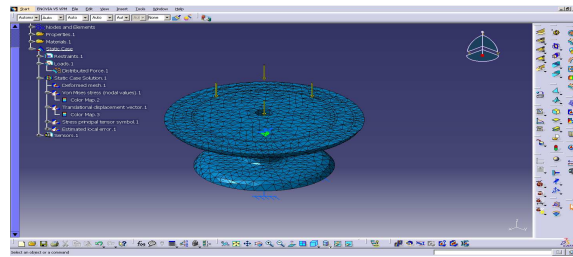


Fig. 11 The Deformation feature

Other feature is called Displacement and shows as a displacement vector field. (see figure 12)

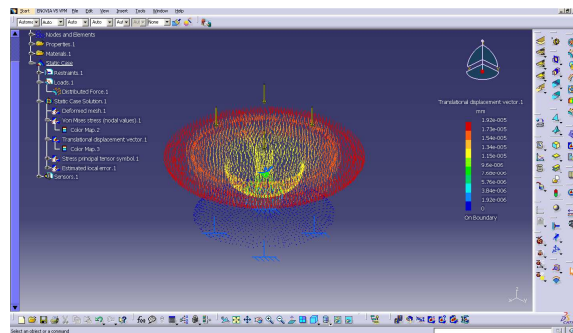


Fig. 12 The Displacement feature

The next icon from the Image toolbar is Von Mises stress. The state of stress is described by the six Cauchy stresses $\{\sigma_x, \sigma_y, \sigma_z, \tau_x, \tau_y, \tau_z\}$ which vary from one point to another as shown in figure 13.

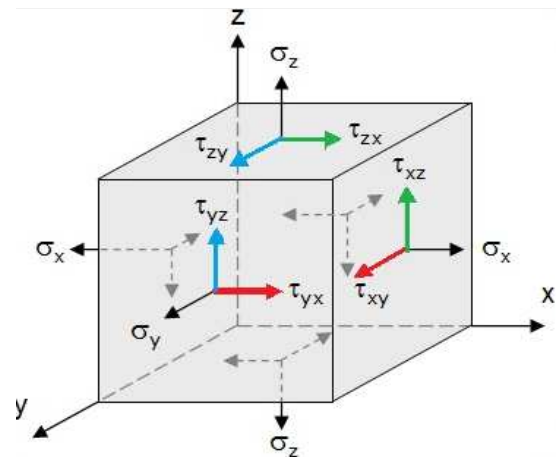


Fig. 13. Principal Stress Parameters [7]

The Von Mises stress is a combination of these and calculates values using the following formula [6]:

$$\sigma_{VM} = \sqrt{\frac{1}{2} \left[(\sigma_x - \sigma_y)^2 + (\sigma_x - \sigma_z)^2 + (\sigma_y - \sigma_z)^2 + 6(\tau_{xy} + \tau_{xz} + \tau_{yz})^2 \right]} \quad (1)$$

As a general rule, the Von Mises Stress is only a positive number and in terms of principal stresses, σ_{VM} can also be written as:

$$\sigma_{VM} = \sqrt{\frac{1}{2}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]} \quad (2)$$

where σ_1 , σ_2 , σ_3 are the principal stresses.

The following steps in the postprocessing are the Von Mises stress and Stress principal tensor features can be seen in figures 14 and 15. [6]

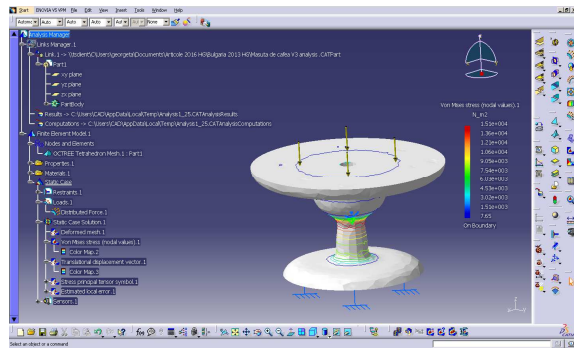


Fig.14 The Von Mises stress feature

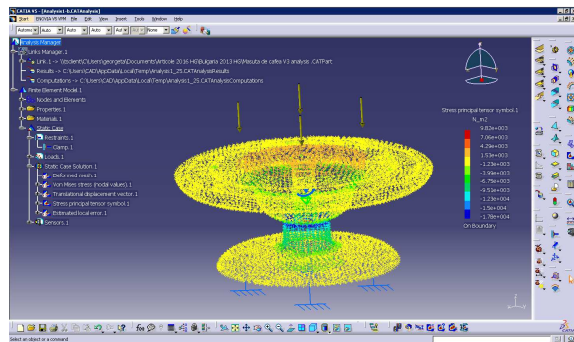


Fig.15 The Stress principal tensor feature

III. RESULTS AND DISCUSSION

In CATIA there is the sag terminology. In FEA (Finite Element Analysis), the geometry of a part is approximated with finite elements. Meshing is a main step in Finite Element Analysis. The final solution depends always on the quality of the mesh used during computation. A smaller element can lead to very precise results. However, as the mesh is used to transfer equations of stress, strain, deformation etc., a denser one means higher computing time. In conclusion, order to obtain the best results, the sag parameter value must be as small as possible. [2] There are numerous rules used to optimize the mesh for best results with an optimum computing time.

The basic one is to use a denser mesh only in areas where the results have to be accurate, where loads and restraints are applied, in thin areas or in weak points and locations of high stress gradient (like stress concentrators - bends, fillets, keyways, small holes etc.).

IV. CONCLUSION

This paper describes a variety of tools for use in the process of design and analysis of an wooden coffee table with center hole can be used with or without umbrella. The wood is considered one of the best ecological material and can be recycled in a 100%. The article outlines a structured approach to the integration of computer aided design and finite element analysis. FEA represents a method for numerical solution of field problems. One of the the most important problem of mesh generation today is access to CAD geometry in a correct and efficient manner. Starting with a solid modeling for the creation of the desired structure is used for the automatic generation of the finite element meshes. All the results of stress distribution and deformation are presented for static dynamic analysis for this chosen model. The present paper is addressed to all users interested in computer graphics.

REFERENCES

- [1] Goanță A.M., Considerations on 2D constraints for drawings made with existing design, *The Annals of "Dunarea de Jos" University of Galati*, Fascicle XIV Mechanical Engineering, Issue vol. 1, ISSN 1224-5615, 2013, pp. 9-12.
- [2] Haraga G., Ioniță E., Avramescu A-M., Analysis and modeling of Geneva mechanism, *Journal of Industrial Design and Engineering Graphics*, Vol. 10, Issue No. 2, Special Issue, fascicle 2, ISSN 1843-3766, 2015, pp. 23-26.
- [3] Haraga G., Ghelase D., Using CAD systems in ecodesign, *The Annals of "Dunarea de Jos" University of Galati*, Fascicle XIV, Mechanical Engineering, Indexed Journal BDI-CSA ISSN 1224-5615, 2012, pp.51 -54.
- [4] *CATIA V5 Fundamentals*, Version 5 Release 15, EDU-CAT-E-V5R15, May 2005.
- [5] <http://www.cadcamlab.org/courseware/PDFs/Strength%20Analysis.pdf>
- [6] <http://www.sdcpublications.com/pdfs/sample/1-58503-227-1-2.pdf>
- [7] <https://www.capinc.com/2014/02/12/frequently-asked-questions-on-von-mises-stress-explained>
- [8] <http://catia.ro/articole/mechassemb.pdf>