SOME ASPECTS OF FINITE ELEMENTS ANALYSIS METHOD APPLIED USING CAD SYSTEMS

Horia CIOBAN
Assoc. Prof., Ph.D., Eng., Faculty of Engineering, North University of Baia Mare,

***Abstract**: the paper presents some cases of study using the FEA method in the design activity; the study shows the different stages of the 3D model, the loads and constrains applied on the digital product; Stress Analysis was used to simulate the behavior of a mechanical part under structural loading conditions.*

1. Introduction

The terms of **Computer Aided Design** were used for the first time in the '60 years, but having the name of "**Computer Aided Drafting**".

In the '70, the costs were very high (around 150,000 $/workstation), but the born of PC's in 1980 reduce the costs of a CAD system to 10,000$. At the beginning, computers were used to replace the work on paper only in 2D Design. In the half of the 9'th decade of the last Century the price of PC's and software goes down and CAD became a common activity in schools and universities.

The first modules for 3D modeling on PC's were presented in 1987. The models could be obtained only by 3D faces, entering the point's coordinates in x,y,z. Two years later, Autocad Release 11 is the first version having a solid modeling module. The software running on PC's tried to obtain similar performances to *CATIA, Pro/Engineer, IDEAS* or *EUCLID* running on workstations. On the other side, many tools were developed to support designers to improve performances, like modules or applications for load and stress calculations or FEA Analysis.

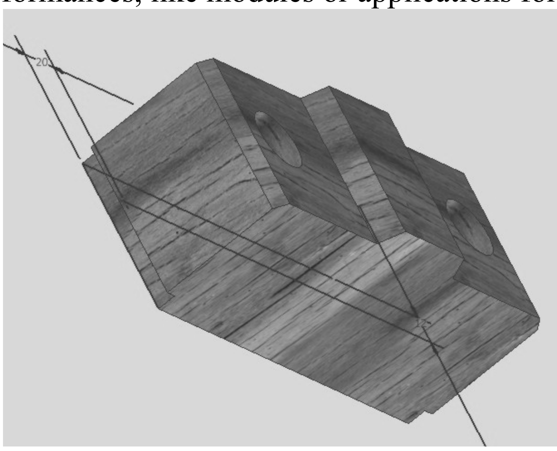

Fig.1 - 3D Model as a sum of parametric features

Instead of solids, a new type of entities is used, having similar parameters and behavior like solids, named **features.** A 3D model can be obtained by a sum of features. The most important property of the features is **parametric modeling** - figure 1.

Definition: parametric modelling uses parameters or dimensions to define a model; the parameter may be modified later, and the model will update to reflect the modification; typically, there is a relationship between parts, assemblies, and drawings; a part consists of multiple features, and an assembly consists of multiple parts. [1]

This new technology is migrating now from workstations to PC's and is used by performing software like CATIA, PRO/E and INVENTOR.

2. Actual stage of FEA

Finite Element Analysis is a computer simulation technique used in engineering analysis that uses a numerical technique called the finite element method [2]. It is one of the most advanced technologies to verify the structures, to establish the stability, durability and the behavior of a model in load and constrains conditions. A common use of FEA is for the determination of stresses and displacements in mechanical objects and systems.

First trials to solve practical problems using numerical methods were done by Hrennikoff on beams in 1941, McHenry and Courant in 1943 on 3D solid models. Clough introduced the term of "finite elements" in 1960. Using classical solutions, it takes a lot time and labor to solve the difficult equations and some software improve the performance and the accuracy of calculus.

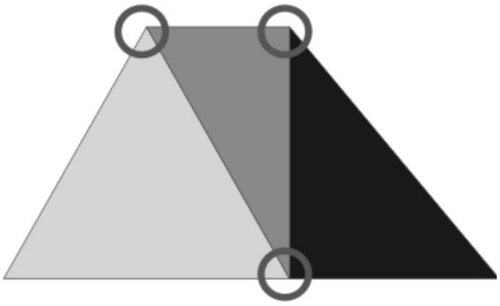

Fig.2 -Interconnections between finite elements

The principle of the method is to divide the model into a number of parts called "finite elements". Each element is connected to two or more neighbored elements by vertexes or by boundary lines or surfaces. In this way, the model is covered by a *mesh*, like in figure 2. The mesh density is not the same inside the model, but can be different depending of the geometry and material properties.

Taking into account the type of supports, the degrees of freedom of the model and the border conditions, the loads, forces and moments are also divided under physical, mechanical and mathematical laws and applied to each finite element.

At the first step, a set of equations will be obtained that solve the nodal displacements. The second set of solutions is related to efforts and loads. Depending of the value of these parameters, each finite element will take a deferent color, from blue at low values to red at high values. The entire model will be covered by a colored map, which can be easy analyzed [2].

Using the FEA method some steps are needed:

- obtaining the model using one of CAD software;
- dividing the model and choosing the type of finite element;
- choosing the displacement function;
- defining the relationships between efforts-displacements and loads-efforts;
- the matrix of equations of constrains have to be derived;
- solving the unknown degrees of freedom;
- solving the elements of efforts and loads;
- understanding the results.

COSMOS/M, NASTRAN or ANSYS are some of these applications that applies the principles of FEA but they have difficulties in obtaining 2D or 3D models. Usually, the model is designed in one of CAD applications and is imported in FEA software. On the other side, most used CAD software on PC's as CATIA, PRO/E or INVENTOR include now FEA modules.

Fig.3 - The demolition machine

3. Case study

The demolition machine for chimneys - figure 3, is a very intersting project developed by a romanian company together with German, Swedish and Holland companies.

The demolition machine is used to destroy big towers. In this moment, in the Eastern Germany many new power plants are replacing the old ones including old chimneys having up to 300 m high.

Explosions cannot demolish these because they are too much closed to very expensive installations. The equipment can be divided into two parts: a climbing stage and the demolition installation.The climbing stage - figure 4 - is a ring on two levels and was mounted at 30 m high on the chimney.

Fig.4 - The climbing stage

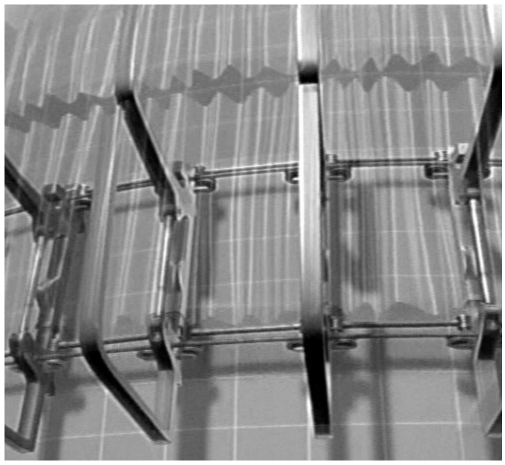

Fig.5 - The consoles and hydraulic cylinders

The mainframe of the climbing stage consists in 26 consoles on each level, is fixed to the tower by cable, and can climb up and down like a spider. At the first step, the cable sustaining the upper level is relaxed and hydraulic cylinders push up. Then the upper cable is gathering and the downer is relaxing. The cylinders pull the downer consoles and the second cable is gathering, preparing the next step, like in figure 5.

In this way, it is necessary one month for the stage to go up to the top of the tower. The climbing stage serve for personnel, materials, diesel and oil needed for the engines. An elevator solves the current access for the shifts. Three crushers mounted on three arms, having as supports two rotating platforms in star shape - figure 6, compose the second part, the demolition installation.

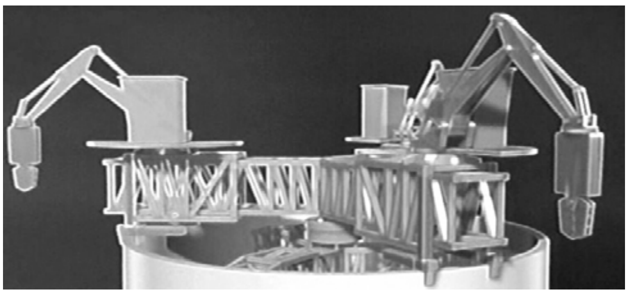

Fig.6 - The demolition equipment

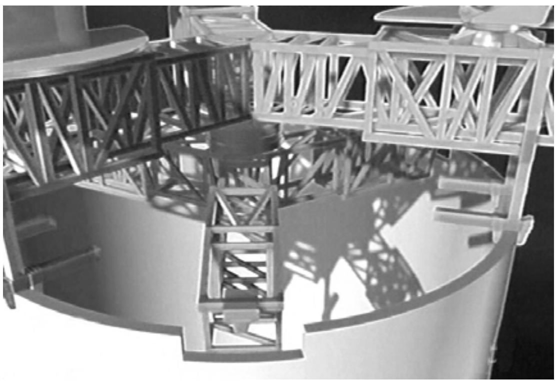

Fig.7 - The equipment is supported by the upper platform

The upper platform is relying on the top of the chimney when the crushers start the demolition, creating three holes in the wall. The crushed concrete falls down inside the tower. The supporting legs of the upper platform push up, the downer platform is rotating, and sliding until the arms of the star goes into the hole -figure 7.

Now, the equipment is sustained by the downer platform until the upper part is rotating to the next position and the demolition process will continue - figure 8. After a ring of the wall is destroyed, the climbing stage goes down one-step and starts a new cycle.

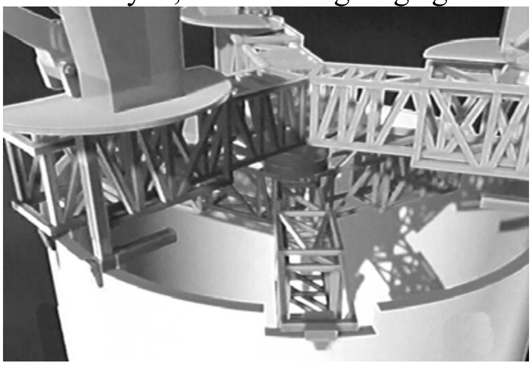

Fig.8 - The equipment is supported by the downer platform

Fig.9 - The crusher

One of the subassemblies of the demolition machine is the crusher - figure 9. Two hydraulic cylinders move the jaws acting in the pivoting point of the lever. The main effort is taking over by the torsion cassette, as in figure 9.

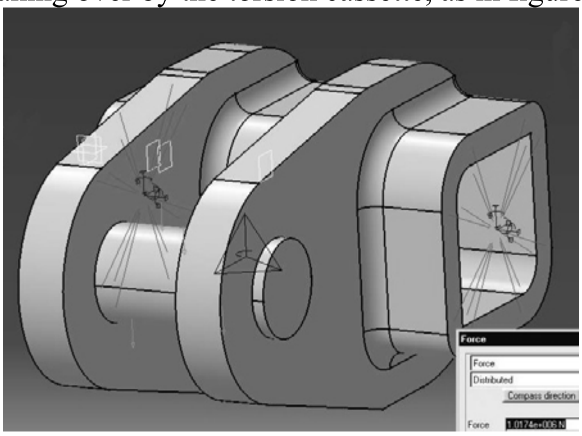

Fig.10 - The torsion cassette

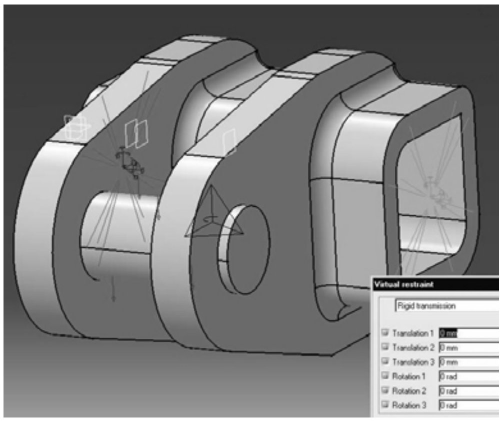

Fig.11 - Applying the constrains

The conclusions resulted after the demolition of the first tower shown a very high displacement. The study of the process started designing the 3D model. The degrees of freedom are reduced applying the constrains, like in figure 11. The model is dividing into finite elements applying the mesh procedure. The figure 12 shows the results of the analysis and using Von Misses graphical representation it can see the displacements and the distribution of the stress around the torsion cassette.

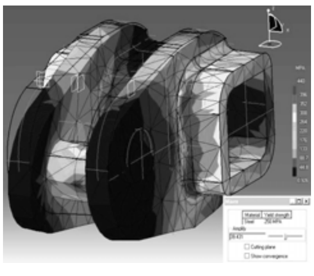

Fig.12 - The displacements and stress representation

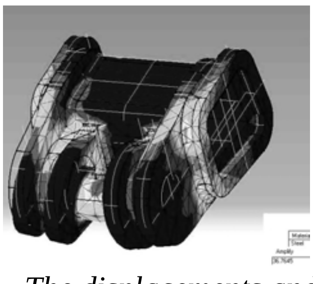

Fig.13 - The displacements and stress representation on the new model

There are two possibilities to obtain a better behavior of the part: the first one is to change the composing material or, the second to change the geometry of the model. This was the chosen solution and the square section was replaced by a trapezoidal section.

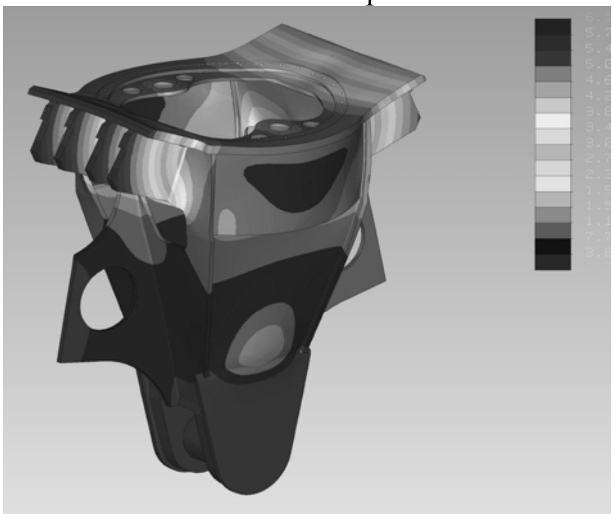

Fig.14 - The displacements in the ball-bearing area

The analysis resumed the same steps and the Von Misses graphics shown like in figure 13. In this moment, the efforts have a uniform distribution around the torsion cassette and the bending of the bolt is in a normal range.

Another example was to find a better geometry of the main body of the crusher that is linked to the excavator's arm by a ball bearing. A displacement over 0.5 mm of the upper ring is dangerous for the bearing. The weight of the entire subassembly was also limited and in this case increasing the width of the walls was not a solution. Applying the FEA method after some steps of changing the geometry of the body, the displacements were reduced to accepted values - figure 14.

From another point of view, it is interesting to compare the results obtained by different CAD systems. The welded assembly presented in figure 15 is designed in Inventor. Even this software has a FEA module, it cannot solve assembled models. The same situation can be found at PRO/Engineer. There are two possibilities: to ignore the welding and to re-design the assembly as a single part or to export the model to another software, like ANSYS or CATIA.

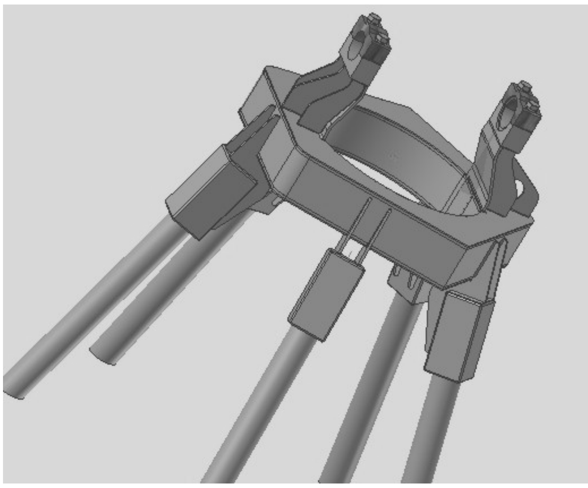

Fig.15 - The welded assembly in Inventor

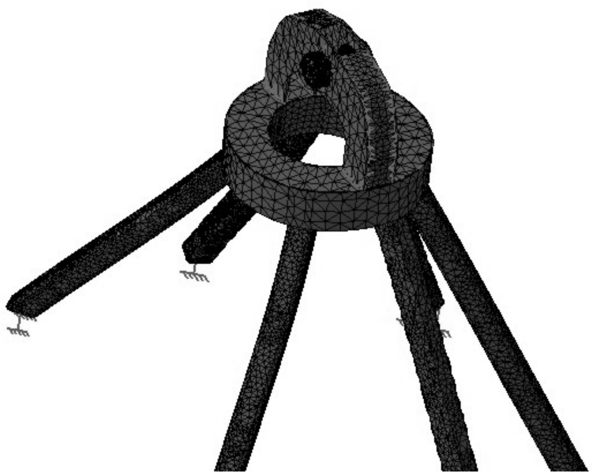

Fig.16 - Another solution in CATIA

Exporting/importing data cannot offer security and accuracy and for this reason the choused solution was to design all parts in CATIA from the beginning. This software has the capability to analyze weldments using FEA method. Figure 16 presents the applied mesh on the model.

The upper side was changed to obtain a better distribution of the loads. Von Misses graphical representation of the stress and the displacements are shown in figure 17 and 18.

Fig.17 -Von Misses Stress

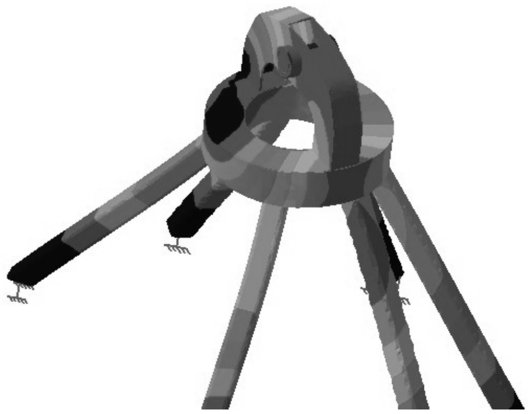

Fig.18 - Displacements

4. Conclusions

Finite Element Analysis method is a powerful tool to support the designing activity using CAD systems. A common use of FEA is for the determination of stresses and displacements in mechanical objects and systems. However, it is also routinely used in the analysis of many other types of problems, including those in heat transfer, fluid dynamics and electromagnetism.

References

Internet
1. *** Wikipedia, The Free Encyclopedia, www.en.wikipedia.org.

Books
2. Cioban, Horia, Bazele Proiectarii Asistate de Calculator, Editura Risoprint, Cluj-Napoca, 2004.
3. Cioban, Horia, Ghidul operatorului in proiectarea asistată de calculator , Editura Risoprint, Cluj-Napoca, 2006.