SIMULATION OF MECHANICAL STRESSES OF A ASSEMBLY CONNECTING ROD

Ioan GHIMBASEANU

Transylvania University of Brasov, Romania (ghimbasani@unitbv.ro)

DOI: 10.19062/1842-9238.2017.15.1.13

Abstract: *Designing a software that simulate the state of strength in the material under conditions of strength offers the possibility of creating a virtual laboratory able to make applications to different areas: medicine, constructions, aeronautics. The aim of the paper is to develop interactive software for monitoring the technological process. The paper create interactive programmers for monitoring the results obtained after the analysis with finite elements for a assembly connecting rod. The method can also be applied for monitoring of the mechanical stress nodal of aircrafts, rockets, ballistic missiles and gun barrels.*

Keywords: *strength, von misses stress, finite elements, connecting rod.*

1. INTRODUCTION

The assembly connecting rod is a component of an internal combustion engine. The assembly rod is composed of: a body, a cap and screws. The assembly straight forward movement of the piston to the crankshaft that turns into rotation.

The materials used are fatigue-resistant and high breaking strength: OLC 35,OLC 45, steel allied with: Cr, NI, Mo, OL 50B, OL 60B. [1,2,3]

The assembly connecting rod is required by compressed gas pressure and bubbles. The inertia force of a group piston requires stretching and compression. The variable size of the applied load is required as a fundamental condition: to have superior mechanical strength.

2. STATICAL ANALYSIS OF A CONNECTING ROD AT TRACTION TEST

2.1 Aim of the application

The Generative Structural analysis programming module of CATIA environment allows the simulation of the test pieces mechanical behavior. [4]

2.2 Analysis model processing

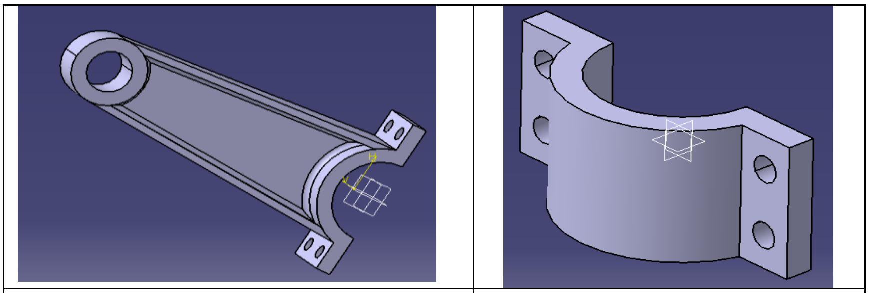
Geometrical modeling

Firstly, the solid model of the connecting rod is designed in the soft CATIA. The body(a) and the cap(b) of model is shown in fig. 1.

Modeling the characteristic o the material

The introduction of the values of the material characteristics necessary for the finite element analysis is made through using the CATIA programmer's library of materials.

The steel material is selected.



(a) (b)

FIG. 1. The model of the connecting rod: the body(a) and the cap(b)

Solving the assembly model

CATIA assembly design packed is launched for generating the assembly between the body and the cap of the connecting rod. The ensemble generate is shown in fig. 2.

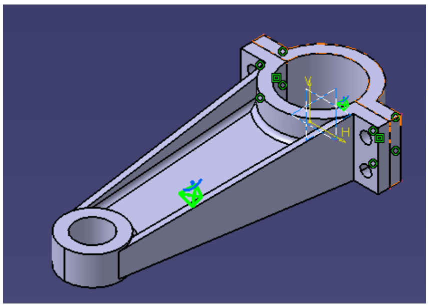


FIG.2 The ensemble between the body and the cap of the connecting rod

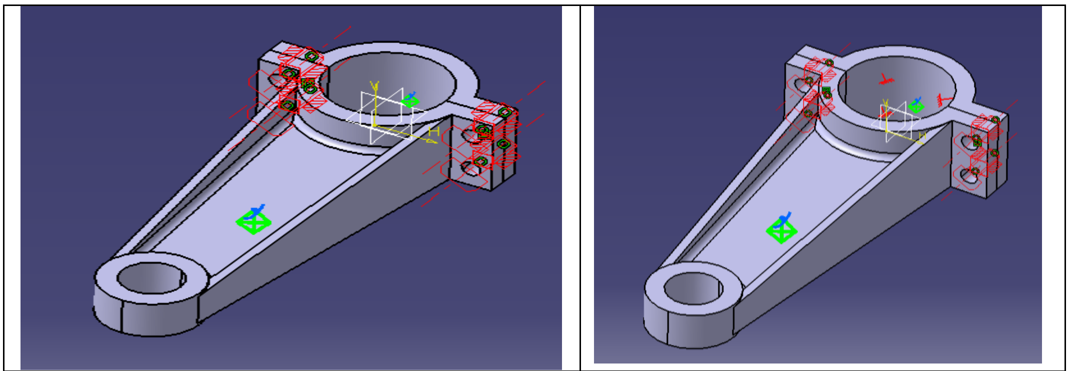
This packet makes the assembly between the body and the cap of the connecting rod when some constraints are imposed: constraints between the surfaces and constraints of co linearity.

Finite element modeling

CATIA Analysis & Simulation packed is launched for generating the finite element. This packet makes the static analysis of the structure when some constraints are imposed and when some stress is independent-time.

They're introduced constraints of bolt tightening connection for constraints of co linearity. The values for these constraints are: tightening force: -100N, orientation: opposite. These constraints are shown in fig.3.a.

They're introduced constraints for displacement: clamp. These constraints are shown in fig.3.b.



(a) (b)

FIG.3 Application of constraints: bolt tightening connection(a), clamp(b)

Load modeling

For load modeling is used: bearing load. The application of force F=3000N is shown in fig.4.

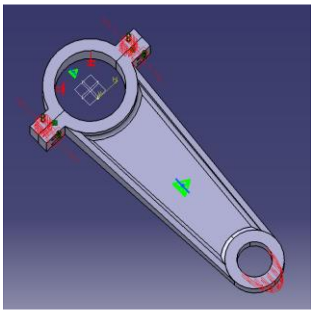


FIG. 4. Applying forces

Solving the model and post processing the result

Then the calculation model is lanched.

Figure no.5 shows the von Mises Stress.

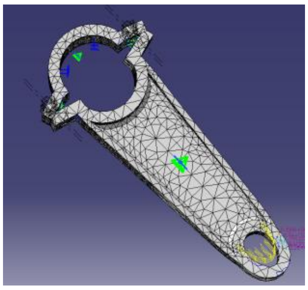


FIG.5 Von Mises Stress

Maximum value for Misses stress is 29.6 e+006 N/m2.

For simulation the traction is used a Wheatstone bridge. [5]

On the basis of the calculation of the bridge, the following equation result, as can be shown below:

$$\varepsilon^2(k^2\Delta u + k^2y^2\Delta u + 2k^2\nu\Delta u - k^2u + k^2\nu^2u) + \varepsilon(4k\Delta u - 4k\nu\Delta u - 2k\nu - 2k\nu u) + 4\Delta u = 0 \tag{1}$$

$$R = 150\Omega, \nu = 0.3, u = 10V, k = 2, \Delta u = masurabil$$

The values of results are shown in table 1.

Table 1. Data results

Δu[mv]	ε[10-5]	Δl[10-3mm]	σ[N/mm2]	F[N]
0.4	2.747	4.39	5.76	2430
0.5	4.121	6.59	8.65	3650
0.75	5.407	8.65	11.354	4791
0.8	5.494	8.79	11.537	4868
1	8.42	13.47	17.68	7461
1.2	9.616	15.38	20.19	8520
1.4	11	17.60	23.1	9748
1.8	13.73	21.96	28.83	12166

3. CONCLUSIONS

The method is a modern one and allows the simulation of the tensile test of the materials. This method permits the geometrical modeling of a test tube, the application of various materials stored in the library of the program, the modifications of these materials, making constraints and force application. The test results are graphically visualized. In this way, the user can simulate the behavior of different materials subjected to tensile test.

The simulation by using Catia software can be used to determine the tensions inside the structure of helicopters (blades), of aircraft's (fuselage, wings), and guns.

REFERENCES

[1] *Metals testing, tensil test, STAS 200-75;*

[2] *G.Gutt, Testing and characterizing metallic materials,Technical Publishing House, Bucharest, 1985;*

[3] *C.Atanasiu, Material's testing, Technical Publishing House, Bucharest, 1982;*

[4] Mogan G., Butnariu S., *Finite element analysis in engineering*, Brasov, Transylvania University of Brasov Publishing, ISBN 978-973=598-159-4, (2007);

[5] Nicolau Th., *The electronics measurements in industry*, Bucharest, Technical Publishing House, (1964) .