ANNALS OF THE FACULTY OF ENGINEERING HUNEDOARA
2004, Tome II, Fascicole 3

USE OF THE FINITE ELEMENTS METHOD IN ORDER TO DETERMINE THE STRESS FIELDS FOR AN INTERNAL COMBUSTION ENGINE

Claudiu OBOGEANU, Ilie DUMITRU

UNIVERSITY OF CRAIOVA

ABSTRACT

A numerical analysis with finite elements method has been performed in order to obtain the stress fields for a four-stroke engine assembly. We have evolved and analysed the components response of the engine mechanism. The steady-state analysis has been developed for two running rotation conditions. The numerical analysis has been evolved for maximum acceleration condition. The research results will be marked out in a proper graphical type of Ansys software.

KEYWORDS:
engine assembly, stress field

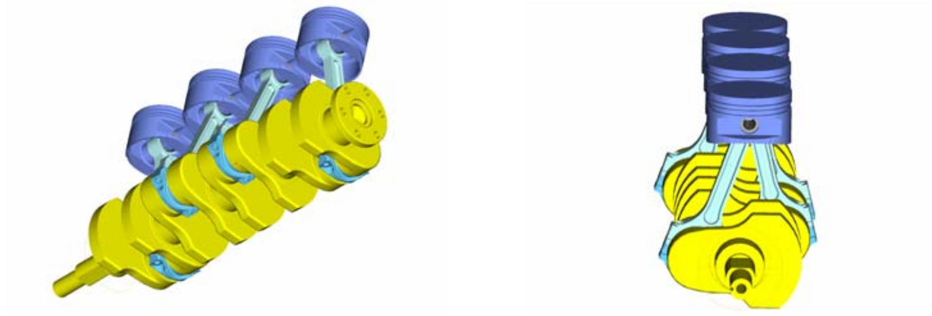
1. ENGINE MECHANISM MODELING

We have researched the mechanical behaviour of a internal combustion engine mechanism for two analysis conditions, a medium rotation state of the engine crankshaft, $n_{med} = 0,5 \cdot n_n = 2600$ rot/min, and a maximum rotation state of the engine crankshaft, $n_{max} = 1,2 \cdot n_n = 6240$ rot/min. It was considered the nominal rotative speed as $n_n = 5200$ rot/min.

For these two rotation states, it is performed a numerical steady-state analysis for maximum acceleration condition.

We have built the three-dimensional solid model of the engine mechanism, with the ProEngineer software, figure 1. The model of the engine mechanism which is inserted in Ansys, is presented in figure 2. In the figure 3, it is presented the discretization model with finite elements of the engine assembly.

The discretized model includes 47710 finite elements, considering that the model indicates just a fourth from the entire assembly. The loadings and the boundary conditions for the mechanical analysis are presented in figure 4.



(a) (b)

Figure 1: Engine mechanism - 3D views, ProEngineer

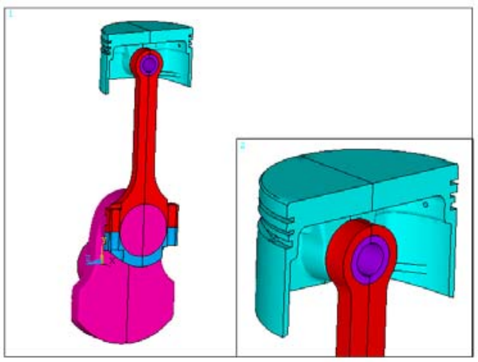


Figure 2: 3D model inserted in Ansys

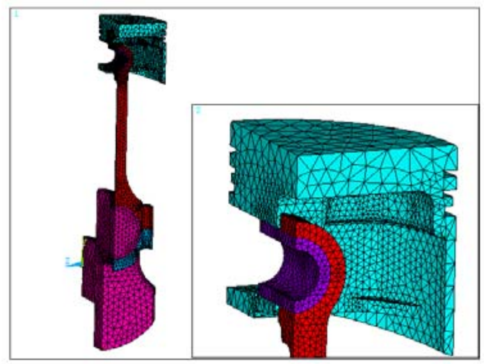
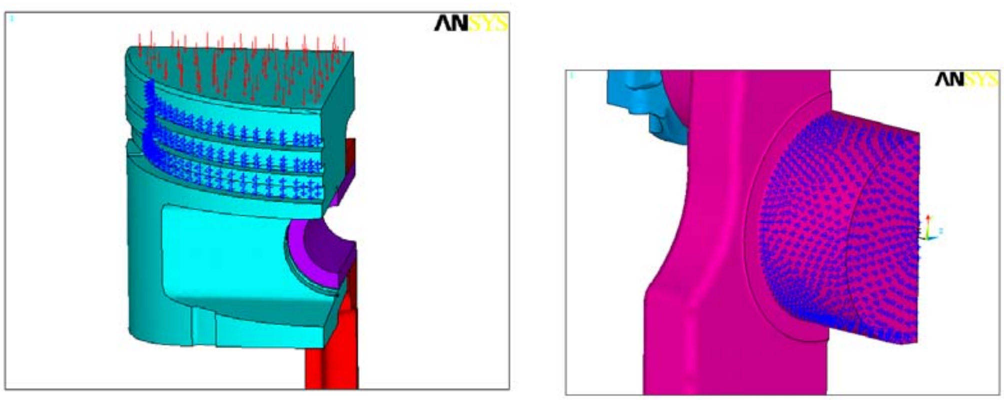


Figure 3: Discretization with SOLID 92 type finite elements



(a) (b)

Figure 4: Loadings and boundary conditions for mechanical field

3. Mathematical model with finite elements method

The proper functional for a finite element method solution, is Π_p, the expression of the potential energy. We estimate Π_p related to nodal degrees of freedom. Using the principle of the stationary potential energy, we can write $d\Pi_p = 0$, from which it is obtained the system of algebraic equations.

The expression of the potential energy is,

$$\Pi_p = \int_V \left(\frac{1}{2} \{\varepsilon\}^T [E] \{\varepsilon\} - \{\varepsilon\}^T [E] \{\varepsilon_0\} + \{\varepsilon\}^T \{\sigma_0\} \right) dV - \int_V \{u\}^T \{F\} dV - \int_S \{u\}^T \{\Phi\} dS - \{D\}^T \{P\} \tag{1}$$

in which, inside the round bracket from the expression of the first integral, the deformation energy on unit volume, W_0 ,

$\{u\} = [u \quad v \quad w]^T$ represents the displacement field,

$\{\varepsilon\} = [\varepsilon_x \quad \varepsilon_y \quad \varepsilon_z \quad \gamma_{xy} \quad \gamma_{yz} \quad \gamma_{zx}]^T$ - strains field,

$[E]$ - material properties matrix,

$\{\varepsilon_0\}, \{\sigma_0\}$ - initial strains and stresses,

$\{F\} = [F_x \quad F_y \quad F_z]^T$ - massic forces,

$\{\Phi\} = [\Phi_x \quad \Phi_y \quad \Phi_z]^T$ - surface forces,

$\{D\}$ - structural nodal degrees of freedom,

$\{P\}$ - external applied loadings.

The elemental displacements are interpolated from nodal degrees of freedom $\{u^e\}$,

$$\{u\} = [N]\{u^e\} \tag{2}$$

and the strains can be written as

$$\{\varepsilon\} = [\partial]\{u\} = [\partial][N]\{u^e\} = [B]\{u^e\} \tag{3}$$

Finally it is obtained,

$$\Pi_p = \frac{1}{2} \sum_n \{u^e\}_n^T [k]_n \{u^e\}_n - \sum_n \{u^e\}_n^T \{r_e\}_n - \{D\}^T \{P\} \tag{4}$$

where the sum symbol indicates the fact that we have taken in consideration the entire contributions of the whole structure elements, or,

$$\Pi_p = \frac{1}{2} \{D\}^T [K][D] - \{D\}^T \{R\} \tag{5}$$

where,

$$[K] = \sum_n [k]_n \quad \text{and} \quad \{R\} = \{P\} + \sum_n \{r_e\}_n \; . \tag{6}$$

Doing Π_p stationary, we obtain, $\left\{ \frac{\partial \Pi_p}{\partial D} \right\} = \{0\}$, which becomes,

$$[K]\{D\} = \{R\} \tag{7}$$

The matrix equation represents a group of algebraic equations which are resolving according to the degrees of freedom $\{D\}$, then can be determined the stress field.

4. Numerical steady-state analysis

The numerical modeling and simulating of the stresses fields are performed by obtaining of the equivalent stress field σ_{ech}. The equivalent stress σ_{ech} is determined by the expression of the fifth modified theory of Huber-Hencky-von Mises.

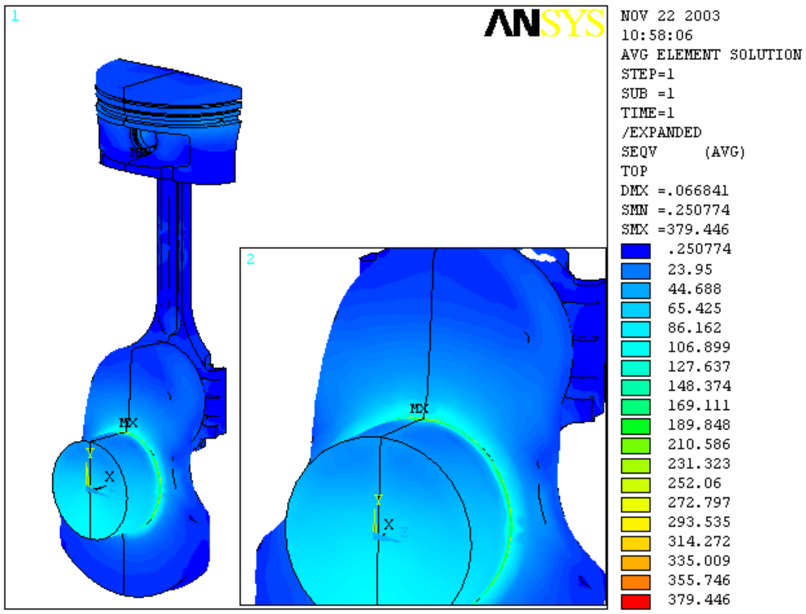


Figure 5: The equivalent stress field for the medium rotative speed condition [MPa]

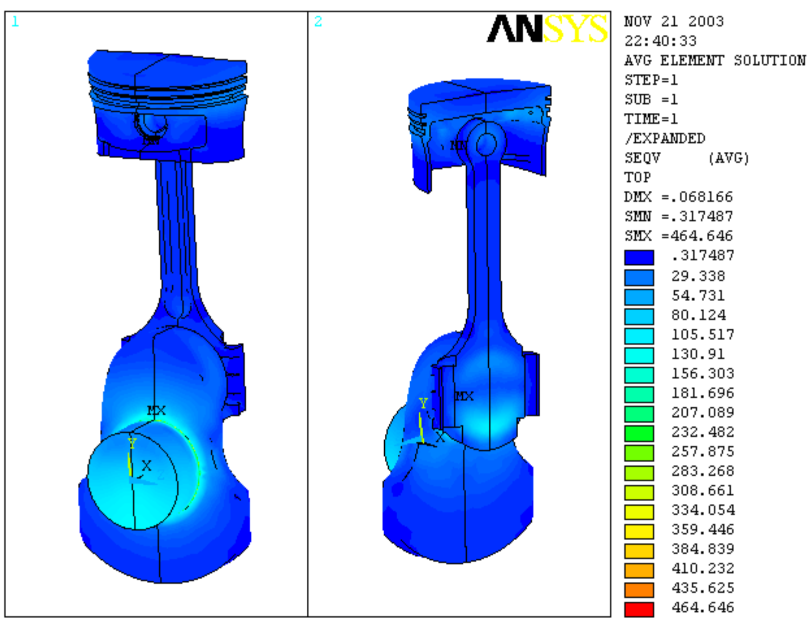


Figure 6: The equivalent stress field for the maximum rotative speed condition [MPa]

The figures 5 and 6 describe the internal loading mode of the engine assembly, for the two rotative speed conditions researched, relating the equivalent stress field distribution with the maximum values, for the medium rotative speed, $\sigma_{ech,max}$ = 379.446 [MPa] and for the maximum rotative speed, $\sigma_{ech,max}$ = 464.646 [MPa].

We have also determined the stress field dispersion on the Oy axle, which represents the cylinder axle as presented in figure 7.

The numerical analysis performed for determining of the internal stresses field of the engine mechanism, indicates the researched dynamic components behaviour, according to external loadings which operate on these components.

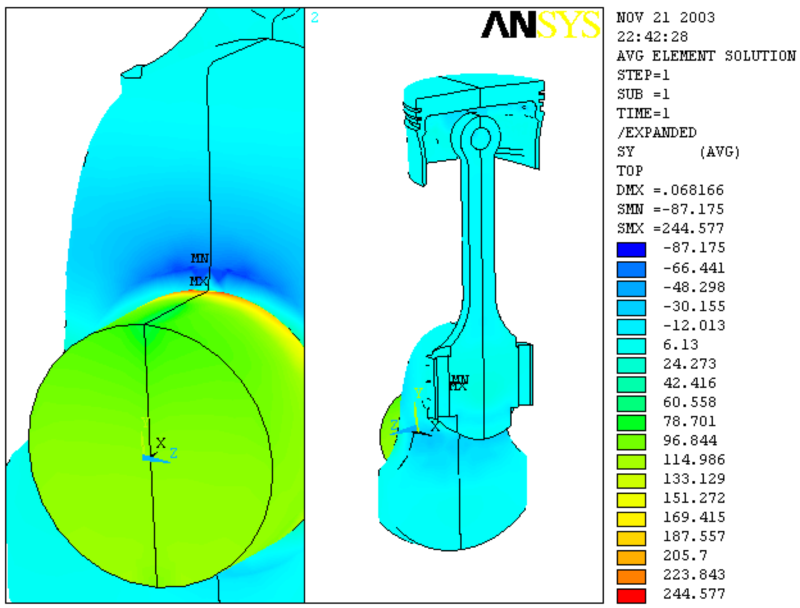


Figure 7: The stress field for the maximum rotative speed condition on the Oy axle [MPa]

5. Conclusions

The finite elements method is firmly established as a powerful analysis tool, which is applied to many different problems of continua, but is most widely used for structural mechanics.

In this paper, it was performed a finite elements procedure which produces many simultaneous algebraic equations, which are generated and solved on a digital computer using the Ansys computer program, mentioning that the engine virtual solid model has been developed by using the ProEngineer software.

The finite elements method originated as a method of stress analysis, aspect used also in this research for an internal combustion engine mechanism.

6. References

[1] Cook, R.D., Concepts and applications of finite element analysis, John Wiley & Sons Press, New York, 1989.
[2] Obogeanu, C., Modeling and simulating with the finite elements method of the thermomechanical loadings in the internal combustion engine components, Ph.D. Thesis, Polytechnic University of Bucharest, 2004 (in Romanian).
[3] Olariu, V., Bratianu, C., Finite element numerical modeling, Editura Tehnica, Bucharest, 1986 (in Romanian).
[4] Washizu, K., Variational methods in elasticity and plasticity, Pergamon Press, Oxford, 1975.
[5] *** Theory Reference, Users Manual & Seminars©, v.5.0-7.1 ANSYS®.