The finite element analysis of the thermal stress distribution of a piston head

M.X. Calbureanu, R. Malciu, D. Tutunea, A. Ionescu and M. Lungu

***Abstract*—** Computer aided engineering (CAE) tools allow engineers to design product and to simulate these designs for residual stress, structural response, thermal effects, pre-processing and post processing fatigue on the automotive component. The main purpose of the preliminary analyses presented in the paper was to compare the behavior of the combustion engine piston made of aluminum alloys. The paper describes the mesh optimization with using finite element analysis technique to predict the higher stress and critical region on the component. As initial condition we considered a temperature on the head piston of 330°C and a total pressure of 5 MPa. There were studied two cases, a piston head and a piston, pin and connecting rod.

***Keywords*—** engine, heat transfer, pressure, temperature field of the piston.

I. INTRODUCTION

CAE analysis tools offer the tremendous advantage of enabling designers to consider virtually any molding option without incurring the expensive actual manufacturing of the machine component and machine time associated to make machine component. The ability to try new designs or concepts on the computer gives the opportunity to eliminate problems before beginning production. Additionally, designers can quickly and easily determine the sensitivity of specific molding parameters on the quality and production of the final part. The complex parts can be simulated easily by CAE tool [1,2,3,4]. Among engine components exposed to thermal effects, the piston is considered to be one of the most severely stressed, where a high amount of the heat transferred to a coolant fluid goes through it, this amount depends on the thermal conductivity of the materials employed, the average speed and the geometry of the piston and rings [5]. A piston is a component of reciprocating IC-engines. It is the moving component that is contained by a cylinder and is made gas-tight by piston rings. In an engine, its purpose is to transfer force from expanding gas in the cylinder to the crankshaft via a piston rod and/or connecting rod. As an important part in an engine, piston endures the cyclic gas pressure and the inertial forces at work, and this working condition may cause the fatigue damage of piston, such as piston side wear, piston head/crown cracks and so on. The investigations indicate that the greatest stress appears on the upper end of the piston and stress concentration is one of the mainly reason for fatigue failure. On the other hand piston overheating-seizure can only occur when something burns or scrapes away the oil film that exists between the piston and the cylinder wall [6]. In recent years, more and more efforts are made to increase horse power to weight ratio of internal combustion engines. In order to achieve the increased power to weight ratio, the necessity of design optimization of various internal combustion engine components is felt very seriously. Lighter piston reduces the dynamic balancing problem to a greater extent. So it is necessary to optimize the design of the piston to keep its weight minimum. This necessitates complete stress analysis of the piston. Analysis will help to modify the existing design for reducing the weight [7].

This paper presents the methodology for a spark ignition piston structural analysis using the finite element method. The loads from the tightening bolts process, the combustion peak pressure and thermal loading were considered as boundary conditions. It is important to determine the piston temperature distribution in order to control the thermal stresses and deformations within acceptable levels. The temperature distribution [8] enables the designer to optimize the thermal aspects of the piston design at lower cost, before the first prototype is built. Most of the internal combustion (IC) engine pistons are made of aluminium alloy which has a thermal expansion coefficient 80% higher than the cylinder made of cast iron. Therefore, the analysis of the piston thermal behaviour is extremely crucial in designing more efficient engines. The thermal analysis of such a piston is important from different point of views. First, the highest temperature of any point on piston [9] should not exceed 66% of the melting

M.X. Calbureanu, is with the University of Craiova, Faculty of Mechanics, Department of Applied Mechanic and Civil Construction, Calea Bucuresti, 107, Zipe code 200512, Craiova, Dolj (corresponding author to provide phone: 0722634340; e-mail: madalina.calbureanu@gmail.com).
R. Malciu, is with the University of Craiova, Faculty of Mechanics, Department of Applied Mechanic and Civil Construction, Calea Bucuresti, 107, Zipe code 200512, Craiova, Dolj (e-mail: ralucamalciu@yahoo.com).
D. Tutunea is with the University of Craiova, Faculty of Mechanics, Department of Vehicles, Transportation and Industrial Engineering, Calea Bucuresti, 107, Zipe code 200512, Craiova, Dolj (e-mail: dragostutunea@yahoo.com).
A. Ionescu, is with the University of Craiova, Faculty of Mechanics, Department of Applied Mechanic and Civil Construction, Calea Bucuresti, 107, Zipe code 200512, Craiova, Dolj (e-mail: adita_i@yahoo.com).
M. Lungu is with the University of Craiova, Faculty of Electrical Engineering Electric, Department of Energetic and Aerospatiale Engineering (e-mail: mlungu@elth.ucv.ro).

point temperature of the alloy. This limiting temperature for the current engine piston alloy is about 370 °C. This temperature level [9] can be increased in ceramic coating diesel engines. In our days due to technical and commercial requirements, the internal combustion engines have to operate with higher cylinder pressures and the design of the components should be optimized for the best performance. The modern calculation methods allow a precise determination of the parts stresses, as well the fatigue strength evaluation. By this way, it is possible to determine safety factors that guarantee a sufficient reliability to avoid failures and over-sizing of the components. For cylinder head stresses computation [10] it was necessary to determine the actuating loads during the critical operational cycles. As analysis load cases were considered the following: the assembly loads, combustion peak pressure and the thermal stresses [11,12,13,14].

II. FINITE ELEMENT MODEL

The entire piston was modeled in a program with finite elements (Solidworks package). We considered the piston made of AlSi and steel an alloy (Table 1). The simulation was done with the fallowing assumption:

- the effect of piston motion on the heat transfer was neglected,
- the rings and skirt were fully engulfed in oil and there were no cavitations,
- the conductive heat transfer in the oil film was neglected.

Table 1 Material properties of piston

Material	AlSi
Thermal conductivity [W/m °C]	155
Thermal expansion 10^{-6} [1/°C]	21
Density [kg/m^3]	2700
Specific heat [J/kg °C]	960
Poisson's ratio	0.3
Young's modulus [GPa]	90

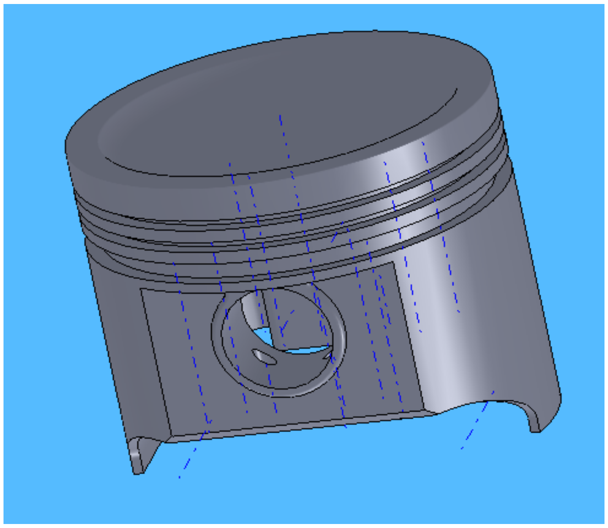

Fig. 1 Solid model for the piston in Solidworks

A. Piston Mesh and Initial Conditions

The temperature profile determination for an engine component such as a piston requires the three – dimensional heat conduction equation solving. The piston may be treated as steady and driven by an average heat flux since the penetration layers are small in the head of the piston.

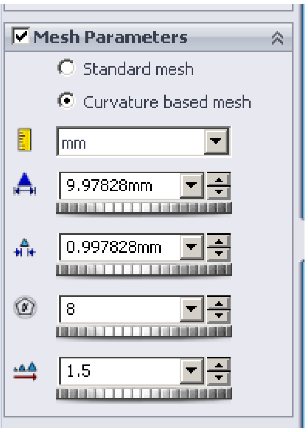

Fig.2 Mesh parameters

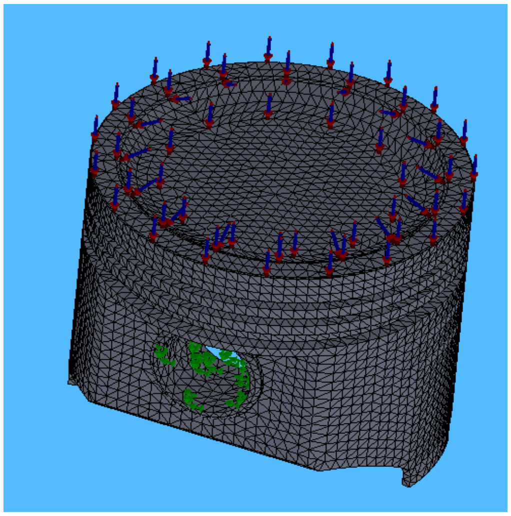

Fig. 3 Piston mesh and initial condition

Based on the research and numerous measurements on the cylinder surface found in the speciality literature, it was assumed that the inside ambient temperature could be estimated as the average of temperature values of intake, compression, combustion, expansion and exhaust temperatures during an engine cycle. As it is shown in Fig. 3 in this case we considered the piston rigid fixing in the bolt region. As initial condition, a 5MPa pressure and a 330 °C total temperature were considered on the piston head.

B. Piston, pin and conecting road condition

Automobile internal combustion engine connecting rod is a high volume production critical component. It connects reciprocating piston to rotating crankshaft, transmitting the thrust of piston to crankshaft, and is subjected to complex loading. It undergoes high cyclic loads of the order of 10^8–10^9 cycles, which range from high compressive loads because of combustion, to high tensile loads because of inertia. Therefore, durability of this component is of critical importance [15]. Connecting rods are subjected to forces generated by mass and fuel combustion. These two forces results in axial and bending stresses. Bending stresses appear due to eccentricities, crankshaft, case wall deformation, and rotational mass force. Therefore, a connecting rod must be capable of transmitting axial tension, axial compression, and bending stresses caused by the thrust and pull on the piston and by centrifugal force [16]. In the case of four stroke engines, during compression and power strokes the connecting rod is subjected to compressive loads and during the last part of the exhaust and the beginning of the suction strokes, to tensile loads. In double acting steam engines, during the forward stroke the connecting rod is subjected to compressive load and during the return stroke, to tensile load. Connecting rod materials must have good fatigue and shock resistances. Connecting rods for automotive applications are typically manufactured by forging from either wrought steel or powdered metal. They could also be cast. However, castings could have blown-holes which are detrimental from durability and fatigue points of view. The fact that forgings produce blow-hole-free and better rods gives them an advantage over cast rods. Between the forging processes, powder forged or drop forged, each process has its own pros and cons. Powder metal manufactured blanks have the advantage of being near net shape, reducing material waste. However, the cost of the blank is high due to the high material cost and sophisticated manufacturing techniques [17].

A study presented an approach to optimize shape of connecting rod subjected to a load cycle, consisting of the inertia load deducted from gas load as one extreme and peak inertia load exerted by the piston assembly mass as the other extreme, with fatigue life constraint. Fatigue life defined as the sum of the crack initiation and crack growth lives, was obtained using fracture mechanics principles. The approach used finite element routine to first calculate the displacements and stresses in the rod; these were then used in a separate routine to calculate the total life. The stresses and the life were used in an optimization routine to evaluate the objective function and constraints. The new search direction was determined using finite difference approximation with design sensitivity analysis. The author was able to reduce the weight by 28%, when compared with the original component [18]. Also was performed three dimensional finite element analysis of a high-speed diesel engine connecting rod. For this analysis they used the maximum compressive load which was measured experimentally, and the maximum tensile load which is essentially the inertia load of the piston assembly mass. The load distributions on the piston pin end and crank end were determined experimentally. They modeled the connecting rod cap separately, and also modeled the bolt pretension using beam elements and multi point constraint equations [19].

The connecting rod load calculation is determined based on the cylinder gas pressure versus crank angle curve, and the inertia forces generated due to the reciprocating masses and the engine speed of the engine. The combination of the gas force, generated due to the cylinder gas pressure, and the inertia force provide the force which is applied to the piston assembly. A FEA analysis using SolidWorks software was done in order to obtain the connecting rod strain and stress results. The below image shows the geometry of piston designed in SolidWorks.

Table 2 Geometrical dimension of the piston connecting rod assembly

Bore	mm	100
Pin length	mm	68
Pin diameter	mm	24
Connecting rod length	mm	210

Table 3 Material properties of piston, pin and connecting rod

	Material	Poissons Ratio [N/A]	Shear Modulus [N/m^2]	Density [kg/m^3]
Connecting rod	AISI 4130 Steel	0.285	8e+010	7850
Pin	AISI 4340 Steel	0.32	8e+010	7850
Piston	PA12	0.33	2.8e+01	2730

III. RESULTS AN DISCUSSION

A. Piston

A nonlinear analysis was performed on the piston model in a program using finite element method (Cosmos Flow from Solidworks) [11,12,13,14]. The thermal field, thermal stress and displacement along different directions on the head of the piston were investigated.

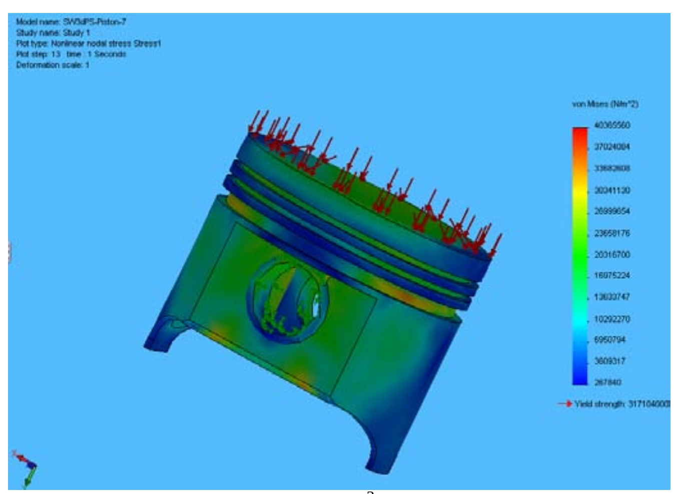

Fig. 4 Von Missses (N/m^2) stress for piston head

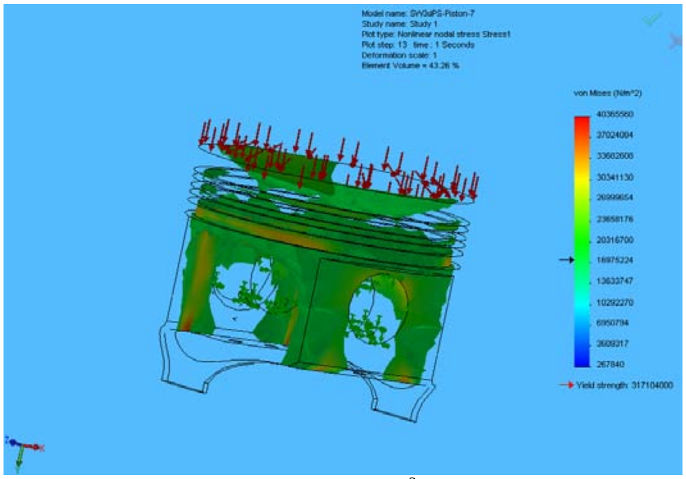

Fig. 5 Different Von Missses (N/m^2) stress for piston head

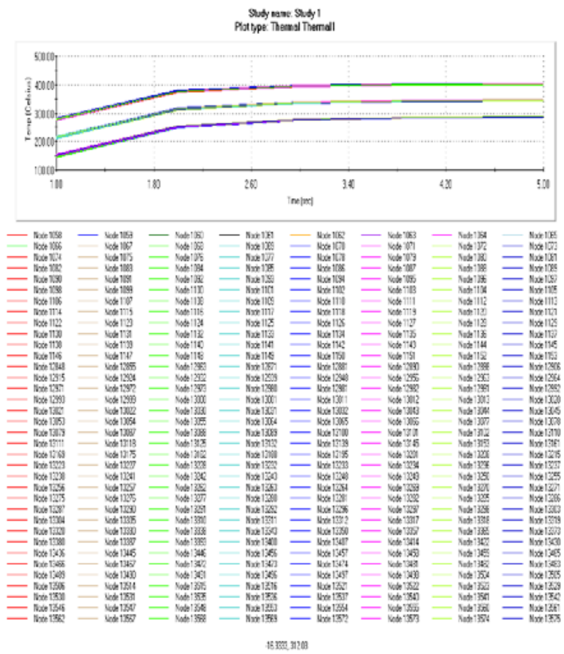

Fig. 6 The temperature scale in time [1-5] s for the first segment

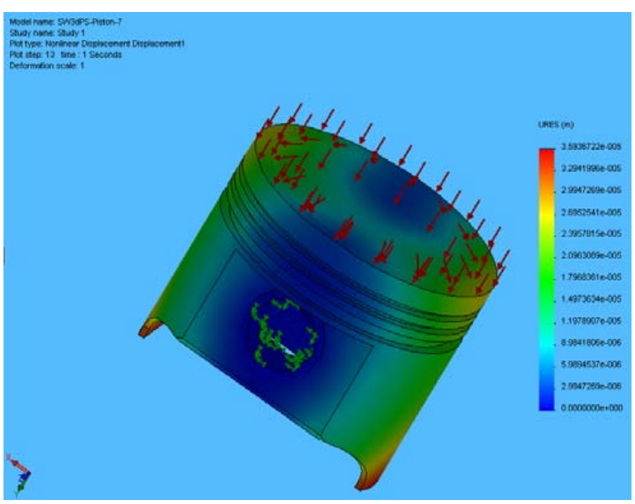

Fig. 7 Displacement URES (m) for piston

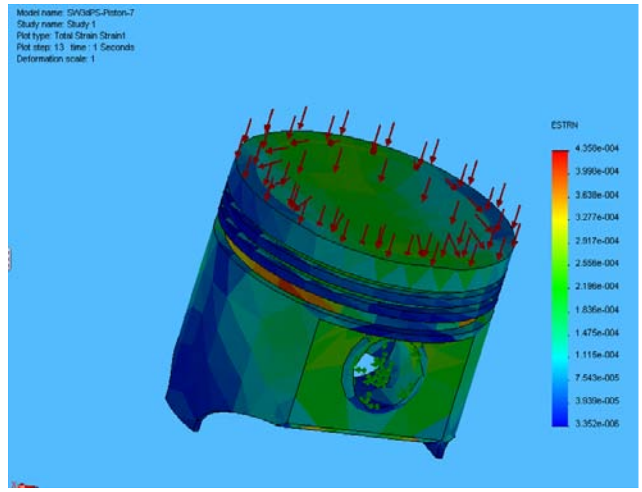

Fig. 8 Total strain ESTRN

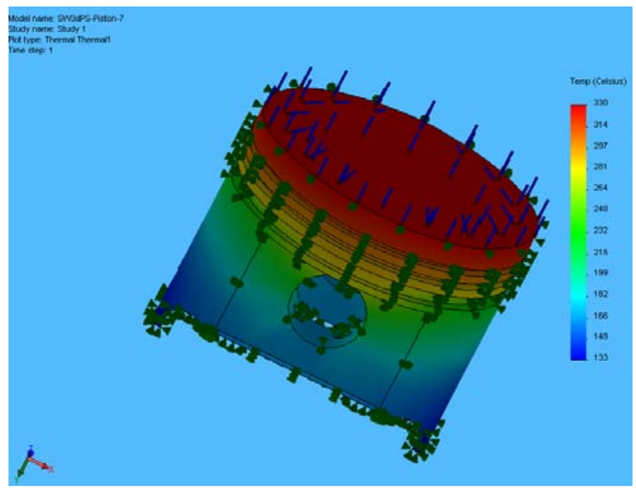

Fig. 9 Temperature field for the piston head

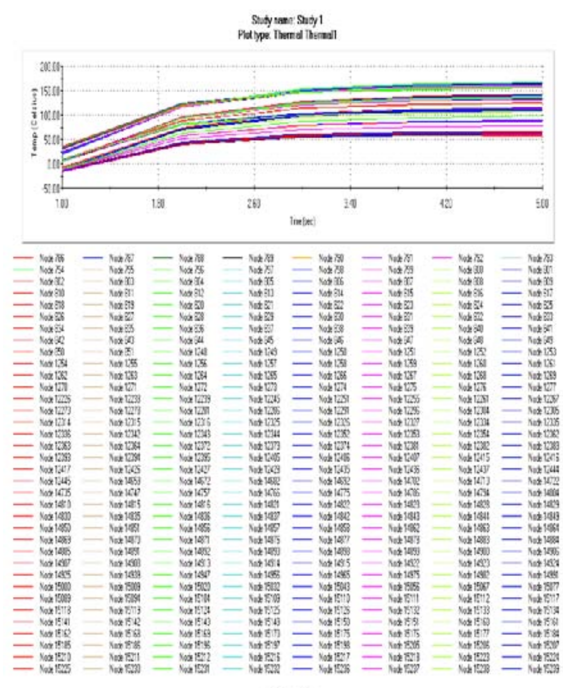

Fig. 10 The temperature scale in time [1-5]s for the third segment

The obtained data gave us the piston behaviour in different working condition during 1 second period of time. The thermal field showed a decreasing of the piston head temperature from 330°C to a low value of 133°C. For the simulation we considered only the heat transfer through convection. The piston head displacement and strain in the nonlinear simulation had low values.

B. Piston, pin and connecting rod

The connecting rod assembly is subjected to a complex state of loading. It undergoes high cyclic loads, which range from high compressive loads due to combustion, to high tensile loads due to inertia.

The structural analyses allow stresses and strains to be calculated in FEA, by using the structural model. FEA approach was adopted in structural analysis to overcome the barriers associated with the geometry and boundary conditions encountered in real experiments. For the connecting rod analyses was used a nonlinear study case from SolidWorks Nonlinear Simulation.

After performing the analysis, the results are obtained as stress fields. For the actual case, the stresses are obtained following the von Misses theory, the total stress and stress in X and Y direction. The peak stresses mostly occurred in the transition area between pin end and connecting rod. The value of stress at the middle of connecting is well below allowable limit. The maximum pressure stress was obtained between pin end and rod of connecting rod. The maximum displacement for the connecting rod has a value of 0.0187 mm.

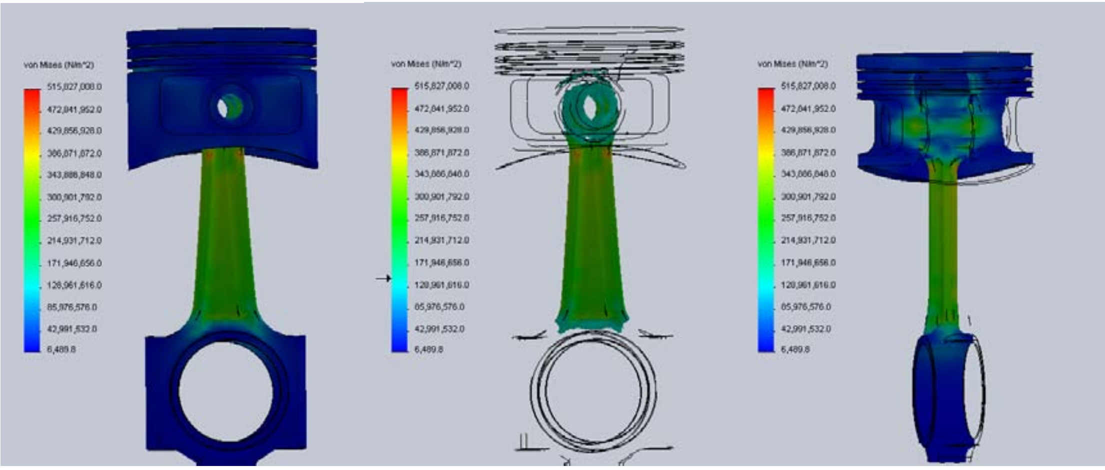

Fig. 11 Stress distribution in connecting rod, resulted from maximum pressure considering Von Misses for compressive loading

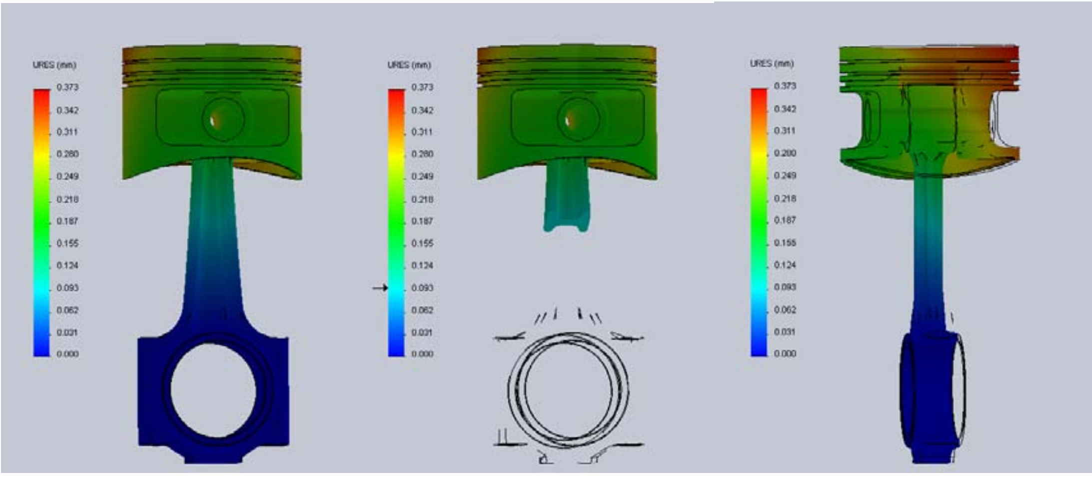

Fig. 12 Displacement (URES) in connecting rod, resulted from maximum pressure of compressive loading

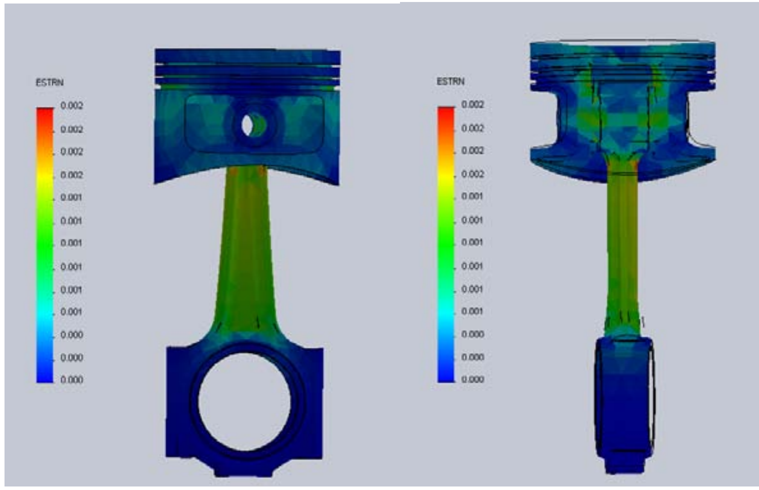

Fig. 13 Equivalent von Misses stress (ESTRN) in connecting rod, resulted from maximum pressure of compressive loading

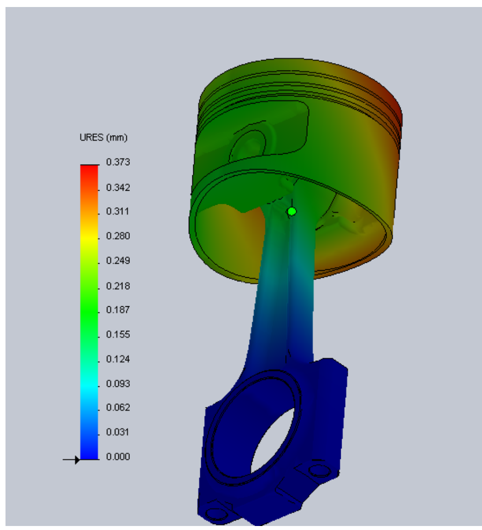

Fig. 14 Location for node 23603

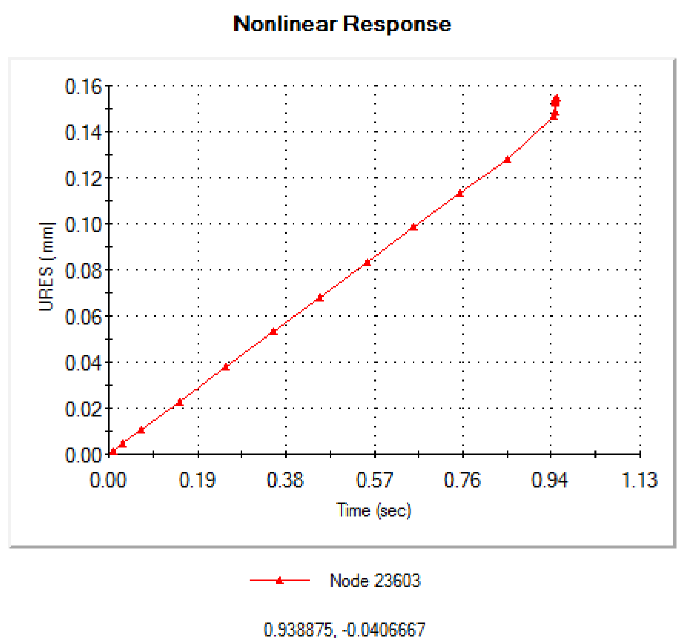

Fig. 15 Displacement function of time for node 23603

According to the Fig. 15 that shows the position of node 23603 in the piston-connecting rod assembly it is noticed the displacement of this node at different time intervals corresponding to the deformation of the connecting rod.

IV CONCLUSION

Although there are a lot of experimental studies regarding the internal combustion engines process there are a few numerical studies focused on 3-D structural and thermal analyses on a gasoline piston model. This paper presents the 3-D finite element modelling of the piston head of a conventional gasoline engine. The obtained data showed how piston worked at different moments of time, when the heat transfer through conduction was considered – fig. 9 and fig. 10. The simulation software used in this case gave valuable information regarding the material behaviour at nonlinear pressure and temperature.

A connecting rod forms a basic element of an internal combustion engine, which performs the function of converting the reciprocating motion of the piston into angular effort of the crank. In this paper, the connecting rod was designed and created in SolidWorks. The following conclusions can be drawn from this study:

- Static analysis of a connecting rod that is typically performed with nonlinear simulations provides more accurate results better suited for fatigue design and optimization analysis of this high volume production component.
- Using the overall operating load range of the connecting rod which comprises the maximum static compressive loads, can lead to an overly conservative design of the component.
- The peak stresses mostly occurred in the transition area between pin and connecting rod.
- The deformation of the connecting rod has a low value for the current load and doesn't affect the structural integrity.
- By using other materials such as micro-alloyed steels having higher yield strength and endurance limit, the connecting road can use with superior performance.

Therefore the methodology presented in this work, showed to be an important tool to be applied during the connecting rod development and design phase. Analyzing the two cases presented in the paper the best results are obtained for the second case were the piston was simulated with the pin and connecting rod. This study confirms the ability of the CFD programs to try new designs or concepts and to eliminate problems before beginning production.

REFERENCES

[1] Ghodake A. P., Patil K.N., Piston Design and Analysis by CAE Tools, IOSR Journal of Engineering (IOSRJEN) ISSN: 2250-3021 ISBN: 2878-8719 PP 33-36.

[2] Computer Aided Engineering in Compression Molding" (Antoine Rios, Bruce Davis and Paul Gramann, The Madison Group: Polymer Processing Research Corporation 505 S. Rosa Rd. Madison, WI 53719).

[3] V. Tut, A. Tulcan, C. Cosma, and I. Serban, Application of CAD/CAM/FEA, reverse engineering and rapid prototyping in manufacturing industry, International Journal of Mechanics, Issue 4, Volume 4, 2010, pp. 69-86.

[4] Robert Rajca, Lukasz Matysiak, Michal Banas, Robert Sekula, A Novel Simulation Approach for Analyzing Reactive Molding Process, INTERNATIONAL JOURNAL OF MATHEMATICS AND COMPUTERS IN SIMULATION, Issue 4, Volume 4, 2010, pp.99-106.

[5] Ahmed A. Al-Beiruti, Dr. Basim M. Al-Quraishi, Isam Ezzulddinyousif, Thermal Effects on Diesel Engine Piston and Piston Compression Rings, Eng. & Tech. Journal, Vol. 27, No.8, 2009.

[6] A. R. Bhagat1, Y. M. Jibhakate, Thermal Analysis And Optimization Of I.C. Engine Piston Using finite Element Method, International Journal of Modern Engineering Research (IJMER), Vol.2, Issue.4, July-Aug 2012 pp-2919-2921, ISSN: 2249-6645.

[7] H. V. Shete, R. A. Pasale, E. N. Eitawade, Photoelastic Stress Analysis & Finite Element Analysis of an Internal Combustion Engine Piston, International Journal of Scientific & Engineering Research Volume 3, Issue 7, July-2012 1 ISSN 2229-5518.

[8] A. S. Mendes, Structural Analysis of the Aluminum Cylinder Head for a High-Speed Diesel Engine, *SAE Paper 2007-01-2562*

[9] E. Buyukkaya, M. Cerit, Thermal analysis of a ceramic coating diesel engine piston using 3-D finite element method, *Surface & Coatings Technology 202*, 2007, pp. 398–402.

[10] K. Uzuneanu, T. Panait, M. Drăgan, Modeling the heat transfer in the piston head of a spark ignition engine supplied with ethanol - gasoline blend, *Quatrième édition du Colloque Francophone en Energie, Environnement, Economie et Thermodynamique – COFRET 08*, Nantes, Frances, 11-13 july 2008.

[11] Calbureanu M., Lungu M., Tutunea D, Malciu R.,, Dima Al. - Modeling with Finite Element the Convective Heat Transfer in Civil Building EPS Insulated Walls, 10th WSEAS International Conference on APPLIED COMPUTER SCIENCE (ACS '10), Iwate Prefectural University, Iwate, Japan, October 4~6, 2010(ISI Proceedings). ISI Web of Science, ISSN: 1792-4863, ISBN: 978-960-474-231-8, pag. 79-84. Vol. Selected Topics in Applied Computer Science.

[12] Mihai Talu , Marin Bica , Emil Albota , Stefan Talu, The dynamic visualization of the 3D thermal impression generated through the air friction with the petroleum coke plant structure, Proceedings of the 5th WSEAS/IASME international conference on Engineering education, p.349-354, July 22-24, 2008, Heraklion, Greece.

[13] Madalina Calbureanu, Mihai Talu, Carlos Manuel Travieso-González, Stefan Talu, Mihai Lungu, Raluca Malciu, The finite element analysis of water vapor diffusion in a brick with vertical holes, pag. 57-62, WSEAS International Conference on Mathematical Models for Engineering Science (MMES '10) , ISBN: 978-960-474-252-3, ISSN: 1792-6734, Puerto De La Cruz, Tenerife, November 30-December 2, 2010, Published by WSEAS Press www.wseas.org, www.wseas.us/e-library/conferences/2010/Tenerife/.../MMES-06.pdf.

[14] Madalina Xenia Calbureanu, Emil Albota, Dragos Tutunea, Sorin Dumitru, Raluca Malciu, Alexandru Dima, Contributions above the dew-point problem in civil building EPS insulated walls modeling with finite element the convective heat transfer, INTERNATIONAL JOURNAL OF MECHANICS, Issue 3, Volume 4, 2010, ISSN: 1998-4448, pag. 53-62, www.naun.org, http://www.naun.org/journals/mechanics/19-574.pdf.

[15] P. S. Shenoy, A. Fatemi, Dynamic analysis of loads and stresses in connecting rods, DOI: 10.1243/09544062JMES105.

[16] Atish Gawale, A. A. Shaikh and Vinay Patil, Nonlinear static finite element analysis and optimization of connecting rod, World Journal of Science and Technology 2012, 2(4):01-04 ISSN: 2231 – 2587.

[17] Christy V Vazhappilly, P.Sathiamurth, Stress Analysis of Connecting Rod for Weight Reduction-A Review, International Journal of Scientific and Research Publications, Volume 3, Issue 2, February 2013, ISSN 2250-3153.

[18] Pai, C. L., 1996, The shape optimization of a connecting rod with fatigue life constraint, Int. J. of Materials and Product Technology, Vol. 11, No. 5-6, pp. 357-370.

[19] Webster, W. D., Coffell R., and Alfaro D., 1983, A Three Dimensional Finite Element Analysis of a High Speed Diesel Engine Connecting Rod, SAE Technical Paper Series, Paper No. 831322.