17th International Conference
ENGINEERING MECHANICS 2011
Svratka, Czech Republic, 9 – 12 May 2011

FEM ANALYSES FOR DESIGN VERIFICATION OF AIRCRAFT ENGINE PARTS

M. Nesládek*, M. Španiel*

Abstract: *The presented paper documents the concept and some major results of the heat transfer and stress analyses performed on a cylinder and a cylinder head of an aircraft piston engine. Both stress and temperature fields were computed by the FEM software Abaqus/CAE. The stresses in the assembly were calculated as a consequence of major mechanical loads together with the influence of the steady state temperature field on deformation. Several modifications of geometry were considered in the sense of their influence on cooling performance and strength. It should be noted that these analyses are preliminary and were performed for the purpose of supporting the engine design optimization. In the future prospects, it is assumed that all these models will be calibrated by data measured on the engine prototype in order to get more relevant results for the planned engine series.*

Keywords: ***FEM analysis, aircraft piston engine, cylinder, cylinder head.***

1. Introduction

The analyses presented here were a part of the project based on the cooperation between ADW, s.r.o. and Josef Božek Research Centre of Engine and Automotive Engineering. The aim of this project is to design a new small four-cylinder aircraft engine. The proposed conception is diesel with supercharging and common rail fuel injection system and it is supposed to be air-cooled.

The given task was mainly to analyze stresses in the structure with geometry shown in Fig. 1, resulting from service and assembly loads. The stress analysis was also focused on the problematic contact of the cylinder liner and the inner surface of the cylinder. Problems can arise here since the temperature expansion of cast iron, which the cylinder liner is made of, is about half of the value of the aluminum alloy. Especially in places with high temperature load, this difference can cause large contact gaps having unfavourable effect on the heat transfer from the combustion chamber. However, it was not possible to design the cylinder to be made exclusively of the aluminum alloy because of its poor tribological properties.

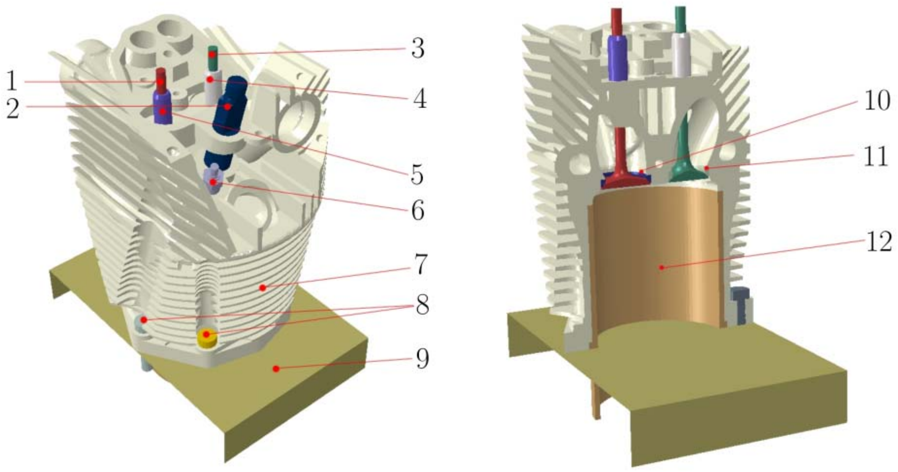

Fig. 1: CAD model of the assembly used for FEM model creation.

* Ing. Martin Nesládek and assoc. prof. Ing. Miroslav Španiel, CSc.: Faculty of Mechanical Engineering CTU Prague, Technická 4; 166 07, Prague 6; CZ, e-mails: martin.nesladek@fs.cvut.cz, miroslav.spaniel@fs.cvut.cz

Tab. 1: The list of parts included in the FEM model.

Link	*Part name*		*Link*
1	*intake valve*	*cylinder and head body*	*7*
2	*injector*	*bolts*	*8*
3	*exhaust valve*	*engine block replacement*	*9*
4	*exhaust-valve guide*	*intake seat*	*10*
5	*intake-valve guide*	*exhaust seat*	*11*
6	*heating plug*	*cylinder liner*	*12*

2. Conception of the analyses

The stress analysis of the cylinder and the cylinder head was conceived as an uncoupled problem of thermo-elasticity. This means that the first step was an evaluation of the steady-state temperature field from known boundary conditions consisting of film coefficient and temperature of gas contacting certain surface. Further, a static mechanical analysis followed where the previously calculated temperature field was applied together with mechanical loads in order to establish displacement and stress fields. The local temperature difference computed from the initial value and previously evaluated temperature field then caused additional displacements and stresses.

The above problem was solved using the Finite Element Method (FEM) implemented in the commercial code Abaqus. The preparation of numerical models was performed in the Abaqus/CAE GUI using the geometry imported from Pro/Engineer. The Python script was applied to simulate the finishing of seats and guides surfaces that are in contact with the valves. This eliminates their radial deformations after assembling them by pressing into the cylinder head.

2.1. FEM model description

Some modifications of the original geometry were performed to avoid mesh generation problems and to keep the number of nodes reasonable. This is especially related to technological radiuses and chamfers making meshing algorithm to produce either poor or high-density mesh locally.

Mostly 3D continuum elements with linear interpolation of displacements were used for the model discretization. A different mathematical formulation was chosen only in the case of the cylinder liner which was meshed by continuum-shell elements allowing again keeping the number of nodes reasonably low.

It is convenient to create an identical mesh for both stress and heat transfer analyses to avoid problems with the correct upload of temperature to the mechanical model. Nevertheless, this was not possible in this case because of the complexity of the model and the fact that it had to be rebuilt several times for different reasons. For these situations, interpolation of nodal temperatures between dissimilar meshes is needed. The algorithm is also a part of the Abaqus code and gave satisfactory results for this purpose.

The stress analysis consisted of several computation steps respecting assembling and loading sequence. In a simplistic form (skipping the initiation steps added to the model for better convergence) these steps are as follows:

1. Assembling the valve guides and seats and the cylinder liner by pressing them into the cylinder and the cylinder head.
2. Contact constraint creation among valves and the appropriate seats and guides, activation of the traction force simulating the effect of valve springs.
3. Loading by bolt pretension simulating the connection between the cylinder and the engine block (the engine block was substituted by analytically rigid surface in this model).
4. Pressure loading of the combustion chamber.
5. Temperature loading of the completely assembled and mechanically-loaded model.

Due to the nature of the structure, the plastic state can be achieved only locally, e.g. in the valve guides or seats, and is marginal problem. Thus the mechanical behaviour of the material was assumed to be linear-elastic with variation of properties with temperature. In general, this is a problematic approach since obtaining temperature-dependent material data is nowadays still a rather difficult and expensive task. That is the reason why proper data were only obtained from different sources for the aluminum alloy and cast iron.

Number of constraints was defined between contacting pairs of parts. Mostly they were based on the Abaqus contact algorithm using surface-to-surface discretization and finite-sliding tracking approach (the simpler one, small-sliding gave biased results of contact quantities in the case of temperature-loading analysis step). For the friction model, the coefficient of friction equal to 0.15 was assumed and was enforced by the penalty method.

To support convergence, kinematical constraint was applied to the valve stems that prevented rotation around the longitudinal axes. The heating plug and injector were connected to the cylinder head by the tie constraint that keeps zero gradients of the displacement at the interfaces.

The boundary conditions of the inner cylinder and cylinder-head surfaces were calculated by the GT-POWER software package. The cooling effect of airflow in the cooling ribs was supplied by the temperature of air and film coefficients. They were estimated by analytical relations based on an analogy with airflow through straight and curved channels (Doleček et al., 2010).

A more detailed description of the model conception together with boundary conditions and material data can be found in Nesládek & Španiel (2011).

2.2. Design modifications compared by the analyses

The influence of some design modifications on the strength and cooling performance of the structure was studied. These modifications are summarized in the following list:

1. Influence of the oil gallery on cooling of the cylinder bottom and the exhaust duct
2. Adjustment of the cylinder-liner wall thickness (from 2.2 to 3.4 mm).
3. Outer geometry modifications, i.e. decreased thickness and increased number of the cooling ribs in the exhaust duct area. Together with these changes, the shape of the oil gallery was modified in compliance with technological recommendations.
4. Change in the cylinder-liner pressing overlap from 0.093 to 0.073 mm per diameter.

As a result of this comparison, the optimal geometry was chosen for further design process.

3. Results

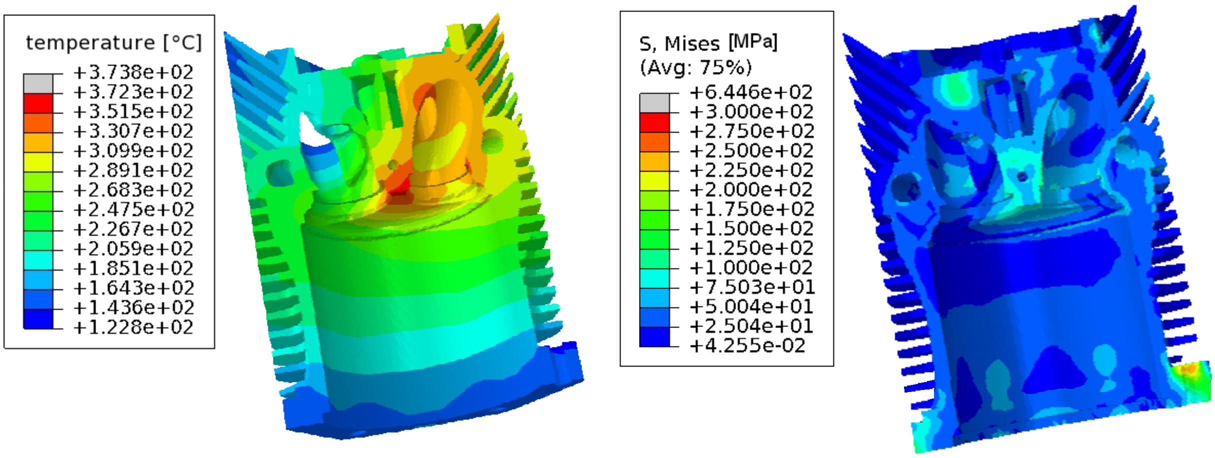

Fig. 2: An example of results – temperature field (left) and von Mises stress after all mechanical and temperature loads were applied to the structure (right).

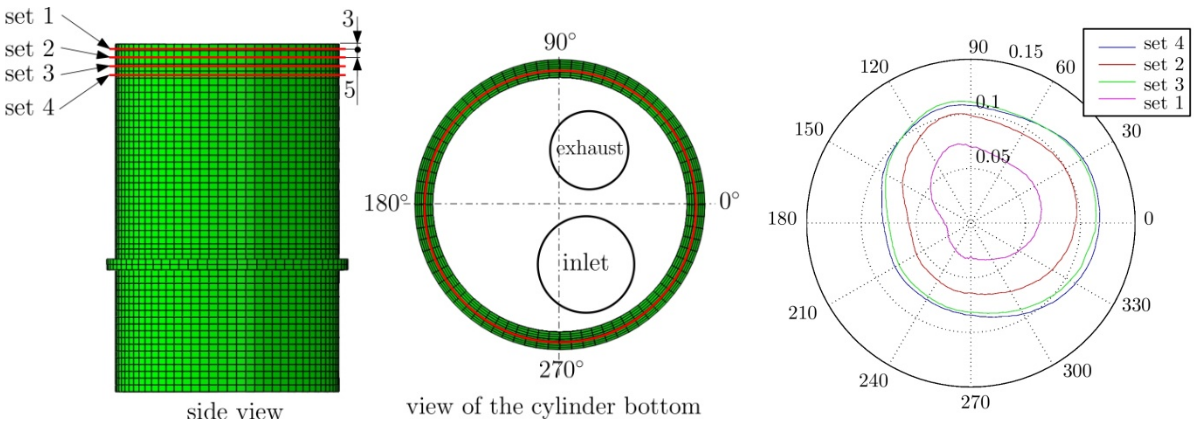

Fig. 3: A plot example of contact gaps between the cylinder and the cylinder liner (right); radial coordinate represents the contact gaps in mm, while tangential direction stands for circumferential position on the contacting surfaces; geometrical meaning is clarified by the two pictures on the left.

Sample results of the main monitored quantities (i.e. temperature, stress and contact gaps between the cylinder and the cylinder liner) are shown in Fig. 2 and 3. As can be seen from the polar plot, relatively high contact gaps are predicted by the model. They were plotted for several data sets measured near the cylinder end where the gaps are at the maximum.

The maximum stresses in the cylinder liner were close to the strength limit when all the assumed loading was applied. At this point, high tensile stress components prevailed which could be severely damaging to cast iron. Raising the wall thickness to 3.4 mm has overcome this problem but with two side effects observed. The first one results in slightly larger contact gaps and a consequence of the second one is the increased temperature of the structure which is about 5 °C higher in the monitored hot spots. This is due to significantly lower heat conductivity of the cast iron compared with the aluminum alloy.

If the oil gallery is present in the structure, the temperature of the measured hot spots is about 25 °C lower. Thus it significantly contributes to lowering the stress response.

The modification of the outer design which was focused on the heat transferring surface enlargement leads to improved cooling performance, as expected. The obtained results show that it can eliminate problems with the increased cylinder liner thickness mentioned above.

4. Conclusions

The previously described computation methodology and numerical model have the capability to predict response of the designed structure to different types of loading. Possible problems with strength and contact gaps were revealed in connection with the cylinder liner. Based on the computed values and their comparison the best-performing modifications were selected. They are mainly represented by the reinforced cylinder liner, presence of the oil gallery and enlarged outer surfaces for better cooling performance.

Acknowledgement

The authors would like to acknowledge the support of the subsidy programme Research and Development for Innovations of the Ministry of Industry and Trade of the Czech Republic.

References

Doleček, V., Tichánek, R. & Macek, J. (2010) Crankshaft modal analysis, valve-operating mechanism optimization and boundary conditions for heat transfer analysis of a small diesel aircraft engine. Technical report no. Z10-15 (in Czech), Josef Božek Research Centre of Engine and Automotive Engineering.

Nesládek, M. & Španiel, M. (2011) FEM analyses for design verification of an aircraft engine cylinder and a head of cylinder. Technical report no. 12105/10/28 (in Czech). Faculty of Mechanical Engineering CTU in Prague, Department of Mechanics, Biomechanics and Mechatronics.