Numerical simulations for the structural performance assessments: a connecting rod's case study

The Piaggio Group is the largest European manufacturer of two-wheel motor vehicles and one of the world leaders in its sector. The Group is also a major international player in the three-and four-wheel light transport sector.

Project Objectives

The connecting rod is one of the most important components in the powertrain systems, very careful structural analysis are required; its failure implies serious damage to the entire engine. Piaggio is developing a new twin engine for motorbike applications and consequently a particular attention has been paid to the connecting rod development, as shown in Figure 1.

Figure 1 - 3D geometry of the connecting rod

In this project, the design soundness of this component has been investigated through the employment of CAE simulations. Firstly, a multibody model has been implemented to evaluate the dynamic loads over the time and, afterwards, it has been possible to verify the safe design of the part by the followings structural performance assessments:

1. FE analyses to:
   a. investigate buckling issues;
   b. compute the natural mode shapes and frequencies of the system (modal analysis);
   c. perform and analyze the connecting rod's eyes deformation;
2. Durability analysis.

Modelling and simulations

Multibody model

The multibody model has been implemented to simulate the bench test conditions of the powertrain system according to Piaggio's standards. The dynamical behaviour of the system has been analyzed under the effect of the engine's combustion pressure (at maximum torque, maximum power and maximum speed), evaluated using

Figure 2 - MBS model of the system

CFD simulations, and the test rig reaction. For each corresponding speed, in stationary conditions, has been computed the load time history acting on the connecting rod (and on the other components). It has also been assessed the maximum tensile and compressive conditions of the connecting rod, used for the following analysis.

FE analyses

Buckling analysis

The buckling analysis has been performed using a simplified model, with only the connecting rod assembly, without piston, piston pin and crankshaft. The assembly has been constrained with a spherical joint on the big eye, allowing the rotation of the internal surfaces and, on the small eye, locking only the displacements along the transversal directions of the connection rod. An explorative compressive (bearing) load has been applied to the small eye; as output, it has been obtained a load multiplier and consequently the buckling critical conditions. The first buckling critical condition has been compared with the maximum compressive load during the operate. The geometry and boundary conditions are shown in Figure 3.

Figure 3 - Boundary conditions for the buckling analysis

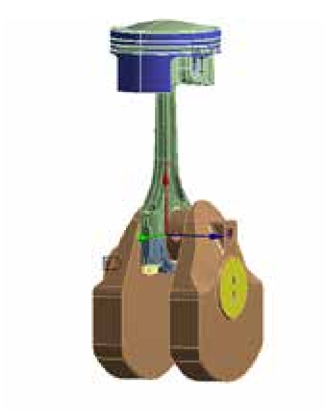

Figure 4 - Constrains for the modal analysis

Modal analysis

The modal analysis has been performed to estimate the mode shapes and natural frequencies of the part, considering all the components of the constrained system. Therefore, the crankshaft, sliced at the main bearing, has been considered fixed on these cutting surfaces, whereas, the piston has been considered free to move along the axial and tangential directions. The first natural frequencies of the system (associated to the twisting, bending, and axial modes) have been assessed.

Figure 5 - Path where deformations have been measured

Eyes deformation analysis

In this analysis, it has been used the same geometry and constrains of modal analysis (Figure 4). Each component's material has been assumed to have linear elastic behaviour, except for the connecting rod (cap and rod) and screws, for which an elastoplastic material with bilinear approximation has been considered. In the interface zones between the components, it has been assumed frictional contacts, considering interferences and clearances. Both static and dynamic loads act on the system. The static load concerning the bolt pretension and the interference on the (frictional) contacts. Regarding the dynamic loads, the maximum tensile and compressive forces have been considered and applied on the piston's upper surface, and the relative acceleration and angular velocity field too. The computing has been performed with a multistep analysis, related to static load condition, maximum tensile condition and maximum compressive condition. In order to ensure the "closure" of the contacts and achieve the convergence, the analysis has required some intermediate steps. The eyes' deformations have been computed on the middle plane of the connecting rod (Figure 5), in the maximum tensile and compressive conditions. The results have been compared with the maximum allowable value according to Piaggio standards.

Durability analyses

The durability analysis has been performed, for each test condition, with two different methods:

- Quasi-static superposition analysis;
- Transient (multistep) analysis.

Quasi-static superposition analysis

This analysis has been performed considering the effective stress tensors time histories during the engine cycle, obtained combining the results of a multibody simulation and another FEM analysis. In particular, It has been implemented a simple model, with the same geometry of the buckling analysis (without piston pin, piston and crankshaft), used in order to compute unit load/stress transfer functions.

The structural continuity has been assumed between the components due to the linearity request of the model. Each transfer function has been computed applying a unit (bearing) load at con-rod interfaces without constraining the system (it has been exploited the Ansys/Workbench's Inertia Relief feature); it has been considered also the stress field due to angular velocity field. As output, it has been obtained the i-th stress field σ_i (x,y,z) associated to each load channel. The same model, only the variation to consider "frictional" the contacts, it has been used to compute the static stress field $\sigma_s(x,y,z)$ due to interferences and bolts pretension. Therefore, the stress tensors time histories $\sigma(x,y,z,t)$ have been obtained with a linear combination of the transfer functions with the associated load time histories $F_i(t)$ (computing and extracting by multibody simulations), added to the (constant) static stress field: $\sigma(x,y,z,t) = \sigma_s(x,y,z) + \Sigma_i(F_i(t) \cdot \sigma_i(x,y,z))$

Transient analysis

The stress tensors time histories have been obtained considering only the maximum tensile and compressive conditions in the transient durability analysis. Therefore, the same FE model of deformation analysis, with the only variation of considering linear elastic materials' behaviour, has been used to estimate the stress field required (possible local plasticity has been taken into account with Neuber's rule, setted in the durability solver parameters). This analysis allows to consider the contacts' nonlinearity and consequently it enables to estimate the performance near connecting rod's eyes area in most severe conditions.
Both the analysis have been performed using the strain life approach and the same solver parameters. The material properties have been defined using the internal Piaggio's procedures. The overall fatigue safety factor has been computed and compared with the minimum allowable value according to Piaggio standards.

Results

For confidentiality reasons, the numerical values of the followings results cannot be provided.

Buckling analysis

By the assessment of the critical buckling conditions P_c and by the evaluation of the maximum compressive load P_{max} acting on the connecting rod, It has been possible to compute the buckling safety factor:

$$SF_{bukcling} = \frac{P_c}{P_{max}(max\ torque)}$$

The minimum of buckling safety factor has shown inline values compared with a similar connecting rod currently operating without criticalities.

Modal analysis

If the frequency of excitation nears with any of the natural frequencies, resonance could occur and it that may result in the mechanical failure of the system. Therefore, once the first frequencies of the first modes has been computed, it has been verified that there is no chance of resonance while operating.

Eyes deformation

Connecting rod's eyes deformation has been computed in the middle plane of the eyes, to the maximum tensile and compressive conditions. On the big eye, it has been also computed the deformation due to static load (interference between the half bearings and the connecting rod) to make a proper appraisal during operation. The results are shown in Figure 6 and Figure 7. These deformations are contained in the Piaggio's standard limits.

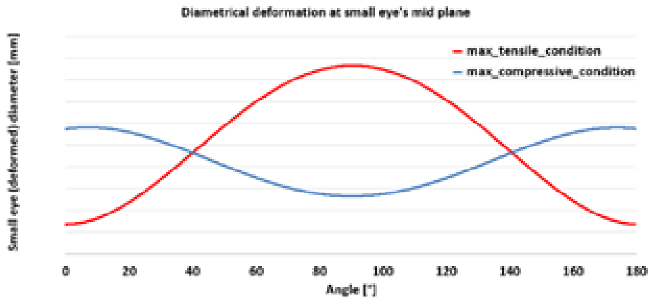

Figure 6.- Diametrical deformation of the small eye

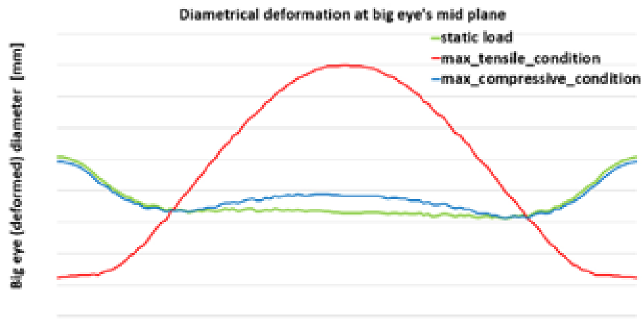

Figure 7 -Diametrical deformation of the big eye

Durability Analyses

The following figures show the results, expressed in terms of safety factor distribution in the most severe condition (maximum speed). The minimum value of safety factor has been turned out above the minimum Piaggio allowables into both analysis.

Conclusions

In the early stages of development of a new idea, the simulation is the only way to correct the design mistakes. In this project, CAE tools have been exploited to predict the mechanical performance of a connecting rod to be used in a new twin engine for motorcycle applications. The implementation of an accurate numerical models, an efficient integration between CAE tools, improve the products competitiveness, speed up the process, reducing, for example, the cycles of physical experimentation saving considerable time and therefore money.

Pasquale Viola
2-Wheeler Product Development - Piaggio Group

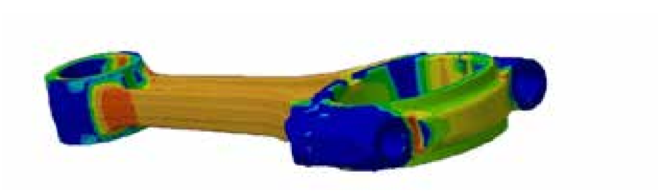

Figure 8 - Safety factor distribution with transient analysis

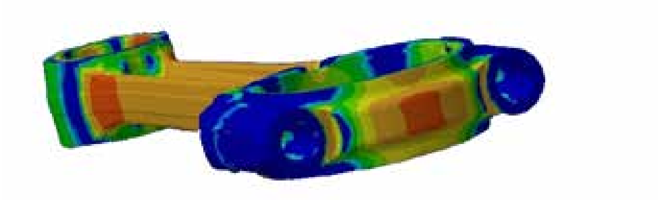

Figure 9 - Safety factor distribution with quasi-static superposition analysis