



ANSYS Workbench Cranks Up Simulation

Piaggio engineers an enhanced design workflow to develop a crankshaft.

By Riccardo Testi, CAE Analyst, Piaggio Group, Pontedera, Italy

A while back, the way we used simulation was different from what we do today. I can still remember looking at a 3-D CAD model of an engine block and pondering the best way to perform a structural mechanics simulation using PREP7. I was able to modify the geometry so that it could be meshed every time, but I would have to start from scratch to repeat the entire procedure with the next CAD model. In those days, beam representations in conjunction with stress intensification factors (SIFs) were the best way to model crankshafts. Other components required a similar effort using simplifications and approximations.

Simulation procedures continued to evolve, and ANSYS changed its face completely with the introduction of the ANSYS Workbench environment. At first, I thought that coping with this new interface would be a dive into a bottomless ocean, just like the first time I encountered CONTA174 with its plethora of options. But I've always been eager to try new CAE techniques, so I jumped into Workbench as quickly and as fully as I could.

My way of working changed radically — geometry import and meshing issues sharply decreased, and I could use past models as templates for new, similar analyses. This last aspect evolved dramatically with the introduction of the Workbench project schematic, in which groups of interconnected analyses form real CAE procedures that are understandable with almost no effort.

Recently, I was able to take full advantage of this new way of working when I needed to perform a crankshaft analysis. Through ANSYS Workbench, I was able to determine the load distribution at the roller bearing that connects the crankshaft and the conrod, as well as to assess the functionality of the friction coupling that transmits the motion from



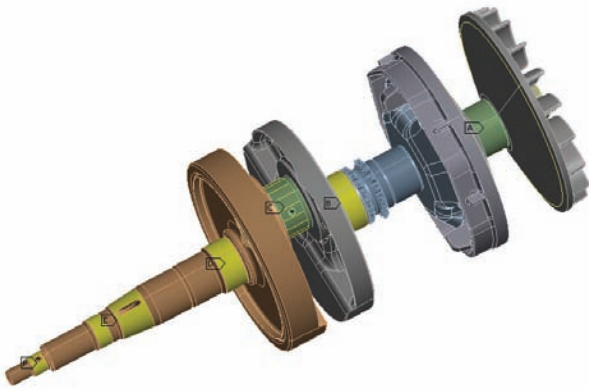
ANSYS Workbench project schematic

the crankshaft to the transmission subsystem. I also used Workbench as part of a CAE sequence that included a multi-body dynamics system and durability analysis.

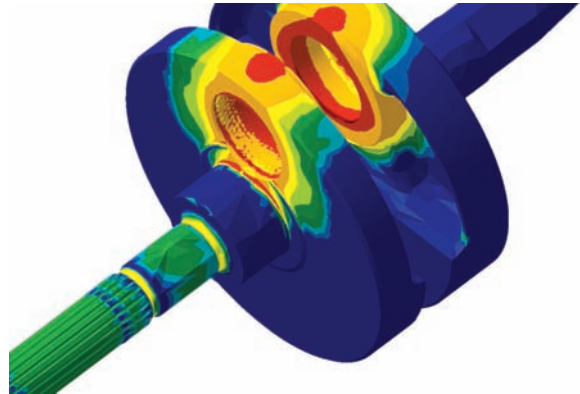
The simulations required linear and nonlinear models that involved modeling neighboring components along with the crankshaft itself. The Workbench platform allowed me to quickly set up a baseline model. The ANSYS DesignModeler tool fixed a few CAD issues, and the ANSYS Mechanical automatic contact detection feature greatly speeded the assembly setup. The CAD interface identified simplified CAD representations (subsections within the entire assembly created within the CAD tool) to allow import of only the components required to generate the structural model — a process very useful when dealing with big CAD models.

It was easy, then, at the project level to generate all the other models I needed from the baseline model. Workbench allowed duplication of analysis systems when a different topology was needed, and it provided the ability to join systems for which different load cases or analysis types were required. I could assess both the frictional load transfer

ANSYS Workbench represents a milestone in allowing simulation to be perceived as a standard and required activity in complex product development.



Model of complete crankshaft system, including remote displacement supports



Durability analysis generated from stress data computed using ANSYS Workbench

capability and the fatigue performance of the complete crankshaft assembly.

For frictional load transfer, I used nonlinear models, employing Mechanical contact features whose default settings are much more error proof than navigating among all the key options and real constants of the good old CONTA family. I could check the functionality of friction couplings with both standard and custom post-processing tools. The latter are easily definable with the aid of another feature of Mechanical, the worksheets. These provide an overview of modeling stages in a neat tabular form. I can

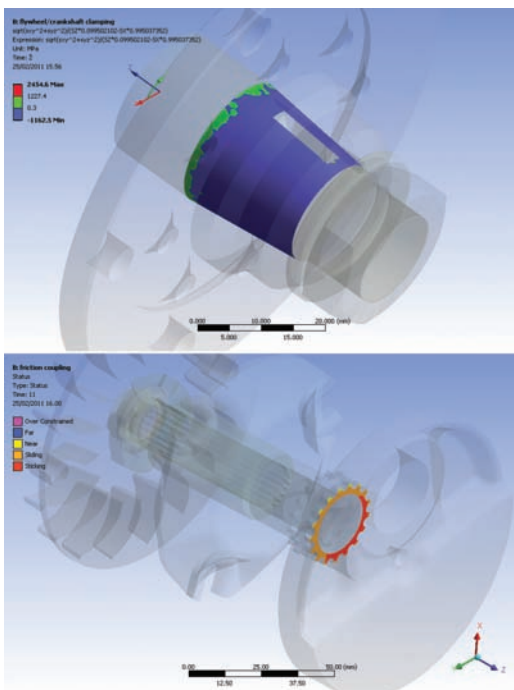
check the properties of the bodies included in an assembly, properties of the contact interfaces, or available post-processing quantities, to name a few. From a table, I can jump to the relevant analysis object with a simple click. Worksheets have proven to be a valuable tool to check what has been completed.

Fatigue analyses required computation of load/stress transfer functions and Craig–Bampton modal representation. The latter was calculated using a simple combination of rigid remote displacement features and a commands object. No more messing around with APDL commands, since Workbench did the job in a straightforward yet simple fashion.

I not only obtained fatigue safety factors and stress distributions; more important is that I had a useful outline of what I did at both project and system levels (using Workbench terminology). The next time a similar component is analyzed, it will require only an update process, rather than generating the model and simulation process from scratch. The entire procedure will be easily recognizable within the ANSYS Workbench project schematic.

In the past, CAE methods struggled to be simultaneously fast and accurate, slowing the integration of these techniques into the development process of complex systems. I think that ANSYS Workbench represents a milestone in overcoming these problems and allowing simulation to be perceived as a standard and required activity in complex product development. Piaggio implemented these new CAE techniques with the support of ANSYS channel partner Enginsoft, which has been providing Piaggio with ANSYS solutions for many years. ■

Riccardo Testi is a CAE analyst and supervisor with almost 20 years' experience in simulation for the automotive industry. He is a member of the Piaggio Group, the largest European manufacturer of two-wheeled motor vehicles and one of the world's leaders in this sector.



User-defined expressions allow users to build their own equation for post-processing (top). Contact status results help the analyst determine what regions of surfaces are in contact (bottom).