In service, an airplane is subject to many different loads, all of which contribute to the fatigue of the structure. Of these loads, wing bending cycles occur most frequently. While virtual models could be used to assess the stress corresponding to the various loads, the emergence of composite wings has introduced uncertainties both in terms of material properties and geometries. Thus, physical testing is still a key part of the design process and reliability assessment. This application brief shows how a model consisting of an assembly of rigid and deformable parts can help designers build a test rig that can reproduce the loads accurately.

**Test Rig Description**

The rig is literally a gigantic steel “cage” surrounding the airplane, with hundreds of actuators and tens of thousands of sensors. This paper focuses on the portion of the rig that bends the wings (Figure 1).

While the actual loads during flight are distributed over the wing, the load for the test rig is concentrated on a few points distributed along the lower side of a supporting structure located under the wing, in an effort to minimize the local stress hot spots that the loading will cause.

Seven hydraulic actuators bend this flexible supporting structure, which is articulated on the frame, allowing large rotations of the wing. A cable and pulley system prevents the introduction of an added mass effect from the test rig.

**Test Rig Description**

The CAD model of the test rig consists of 138 parts (Figure 2). The frame and attachment beams of the flexible supporting structure are oversized enough to be considered as rigid. The cable can be considered as inextensible, but to properly account for the effect of the balancing masses, a set of constraint equations is added to the model to keep the total length of the cable constant.
Rigid and Flexible Body Simulation of an Aircraft Wing Test Rig

The mesh of the supporting structure needs to be good enough to represent properly the stiffness, while stresses are less important at this point. Thus 20 parts corresponding to the beam plates with larger aspect ratios are declared as “flexible.”

Because the whole wing supporting structure is articulated, large displacements and rotations need to be accounted for. Nevertheless, the strain in the deformable portions remains relatively small, so reduction techniques can be applied to decrease analysis time. ANSYS Mechanical Enterprise allows deformable bodies to be included in a multibody model through Component Mode Synthesis. This method condenses the deformable substructure on a reduced basis of modal shapes and static responses, reducing a finite element model of arbitrarily large sizes to tens to hundreds of degrees of freedom. This process decreases the analysis time to only a fraction of that required for a full model analysis, while fully accounting for all effects of large rotations and displacements. While the model setup is very similar to any transient analysis, the actual solution that runs underneath consists of three steps:

1. A generation pass, based on the full finite element model of each “condensed part,” is run. This pass goes through a modal analysis, followed by a series of static analyses to generate the so-called “attachment modes.” To properly account for large rotations and displacements, extra coupling terms between small elastic deflection and global motion are computed and exported.

2. A use pass in the multibody solver is run. This process solves for all bodies motion and forces, with the inclusion of the condensed parts from the generation pass. Thanks to a consistent tangent matrix approach, the convergence of this nonlinear solution proves to be very smooth, and autotimestepping can be used.

3. Finally, stresses are recovered by reinjecting the reduced model degrees of freedom back to the full finite element model, and looping through the elements to compute strains, stresses and recovering nodal displacements.

It is worth mentioning that the generation pass is valid for multiple analyses, allowing changing of the loads while rerunning only steps 2 and 3, or even only step 2 if the stresses and strains in the deformable parts are secondary design criteria.

Design of the Control System
When applying the ultimate load on the wings, the reference load is applied over a period of 20 seconds, and the actuators must apply a force such that the displacement of the wing tip varies linearly over time. The amount of load applied in each pair of actuators is ramped, and is defined as a parameter using a “parametric” command snippet. Various design points are generated for various
distributions of loads; they are then run from ANSYS DesignXplorer's design point table. An example of final configuration is given in Figure 3. Each design point is run in about 80 seconds on a desktop computer.

While trial and error can lead to a fairly reasonable design for one tested wing, in reality a control system drives the actuators.

While during this first phase of the analysis using ANSYS Simplorer to model the control loop itself was not done, it will be included in the next step. To achieve this, some “input pins” will be defined for each of the cylinders, and “output pins” will be defined to measure the corresponding reaction forces in the actuator. Once defined in this way, the model can be directly linked to Simplorer from the schematic, where the control loop is modeled. The simulation in Simplorer will use cosimulation to couple the two sides of the system (hydraulics and control on one side, assembly of rigid and deformable parts on the other).

Summary
Quick design iterations of a complex assembly of rigid and deformable bodies can be achieved using the Component Mode Synthesis reduction of deformable bodies with ANSYS Enterprise multibody solver. This type of model can be coupled with ANSYS Simplorer to simulate the whole system.

Authors
Jean-Daniel Beley, Francis Chapuis, ANSYS France SAS
Model courtesy of DGA, French Ministry of Defense.