Figure 1. Flexible Dynamic analysis setup with all nine original assembly parts as rigid components
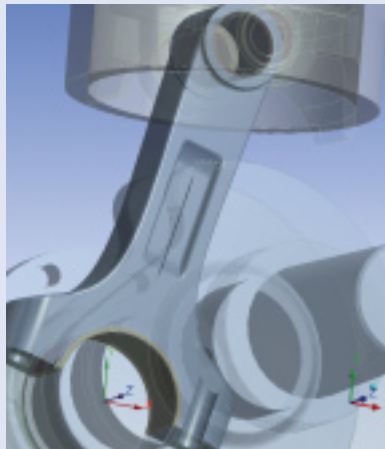

Figure 2. System-level analysis with joints defined
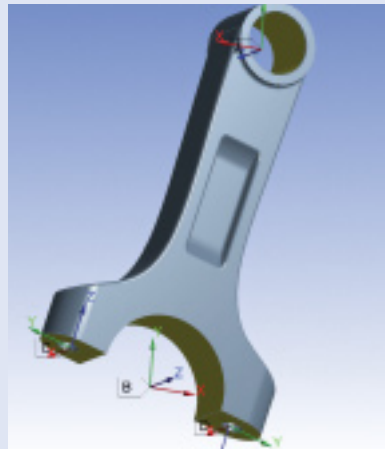

Figure 3. Reference Coordinate System location and orientation

Component Mode Synthesis in **ANSYS Workbench Simulation**

CMS superelements provide flexibility of simulation models while reducing the number of degrees of freedom for highly efficient solutions.

By Sheldon Imaoka, ANSYS, Inc.

At ANSYS Workbench 11.0, the ANSYS Rigid Dynamics add-on module enables users to define joint connections in complex kinematic assemblies.

This ability to model rigid and flexible parts in ANSYS Workbench Simulation via joints is beneficial since the rigid parts are modeled with mass and rigid links (that is, rigid contact elements), thus providing load transfer capabilities and allowing users to evaluate system-level performance with less computational cost. However, the solution time is dictated by the size of the flexible mesh, and in some cases the rigid assumption may not be accurate enough for specific applications.

A better approach in these cases is the use of Component Mode Synthesis (CMS) superelements, in which the flexibility of the model is retained yet the number of degrees of freedom (DOF) is reduced, thereby providing efficient solutions.

A superelement has a reduced number of DOF compared with the "full" model, yet it still can accurately model the flexibility of structures. Superelements can be created by regular substructuring or by component mode synthesis, in which DOF at interface nodes are retained while all other DOF are eliminated by condensation of the matrices. CMS appends a regular substructure with generalized coordinates, thereby improving the accuracy of the superelement response in dynamic applications.

Since CMS superelements can be used in large-deflection nonlinear analyses, they are especially attractive for nonlinear transient problems. This is because of the low number of DOF (that is, interface nodes and generalized coordinates) and the accurate dynamic representation. CMS superelements, however, also can be used in static, modal, harmonic and response spectrum analyses.

If contact is used with superelements (regular substructure or CMS), the number of interface nodes increases dramatically, depending on the size of the contacting region. Hence, CMS superelements should be used in Flexible Dynamic analyses in ANSYS Workbench Simulation with joints. Joints are defined at a single node, so the number of interface nodes used in CMS is reduced to a bare minimum.

Example Case

A sample Autodesk® Inventor® assembly was used for this example. A Flexible Dynamic analysis was set up with all nine parts as rigid, as shown in Figure 1. Interaction between the parts was defined by joints using rigid behavior for the associated surfaces, and two Joint Conditions were used for loading of the sample engine.

In a separate model, the connecting rod was set up in order to create a CMS superelement. Figure 2 shows the original system-level analysis with the joints. The individual model needs to reproduce the Joint Reference
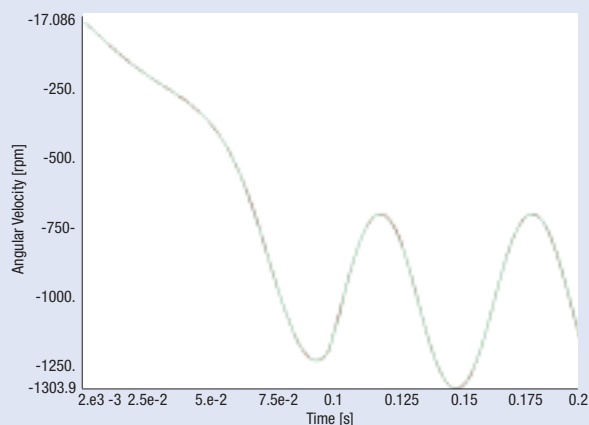
Figure 4. Comparison of simulation results for the rigid and CMS cases match for low loading. The green line indicates CMS results while the red line represents the rigid case results.
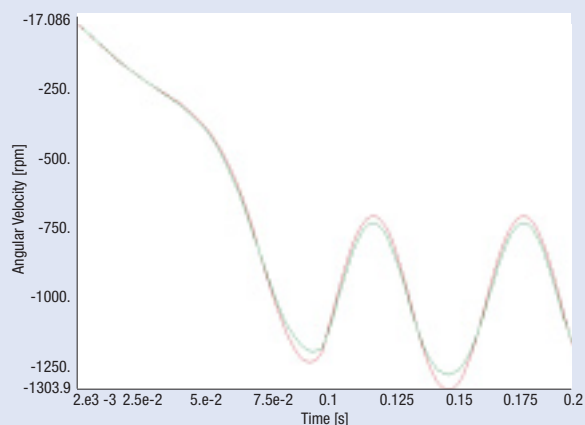

Figure 5. For higher loading, the rigid case and the CMS one differ due to the flexibility of CMS superelement components. The green line indicates the CMS results while the red line represents the rigid case results.

Coordinate System location and orientation with a Remote Displacement support, as shown in Figure 3. With that completed, the model can be meshed, and a superelement can be created.

To accomplish the generation of the superelement, a Commands object is inserted that selects constrained nodes (from the Remote Displacement support), deletes the constraints and designates those nodes as master DOF. A CMS generation pass is performed with a user-specified number of requested modes, and the resulting file.sub is the superelement file needed for the system-level analysis.

This procedure is repeated for each rigid part that will be converted to a superelement. Note that since multiple superelements may exist, each .sub superelement file should be renamed to have a unique name. Lastly, the user should ensure that the same unit system is selected as is used in the system-level assembly.

Once the superelements are generated, they can be incorporated into the system-level model. The author prefers doing this by adding two Commands objects, one under the part to be replaced and another under the Flexible Dynamic branch. The first has a single line, PART1_ = MATID, allowing the user to reference the element type ID a priori. The second Commands object would delete all rigid links of the rigid part except one, add the superelement and then couple coincident nodes together.

The rigid body is connected to joints via rigid links (that is, rigid contact elements). The reason the author prefers to delete all but one rigid link is so post-processing can be conducted as usual. In ANSYS Workbench Simulation, the original rigid mass is used to track the position of the part, so if only one of the rigid links is left, the mass element moves with the connected joint, and the user still can visualize the response of the system.

Figures 4 and 5 show relative angular velocity at the crankshaft–connecting rod joint for both the rigid assembly and the assembly that used CMS superelements to represent the piston, connecting rod and crankshaft, respectively. For relatively low loading, as illustrated in Figure 4, both results match well, as expected. For higher loading in which there is some relative deformation, as shown in Figure 5, the rigid-only and CMS cases start to show differences due to the flexibility of the parts.

It is important to note that the solution times for both system-level runs were relatively quick. The rigid-only case ran for 303 iterations, which required 19 seconds on a 3.2 GHz PC. The assembly with three CMS components ran for 1,340 iterations that required a total of 191 seconds. The CMS assembly case required more iterations to account for the flexibility behavior that was included in the analysis with the additions of the three CMS superelement components.

Once overall results (deformation, joint results, spring results) are reviewed, the deformation, stresses and strains for each part can be reviewed by expanding the superelement solution, then using "Tools menu > Read ANSYS Result Files." The user can expand either results at particular points in time or all of the results, then post-process as usual using the ANSYS Workbench platform, as shown in Figure 6. ■

Contact the author at sheldon.imaoka@ansys.com for the full article from which this was excerpted as well as demonstration input files.

Figure 6. Results, with CMS parts, are post-processed as they normally would be in ANSYS Workbench Simulation.