Multibody Dynamics: Rigid, Flexible and Everything in Between

Advances in simulation solutions for machine features accommodate more complex designs.

By Steve Pilz, Product Manager, ANSYS, Inc.

As simulation capabilities grow, an engineer's ability to simulate more complex machines increases.

Images:
motorcycle © iStockphoto.com/Paul Griffin bicycle © iStockphoto.com/Artsem Martysiok

Simpler is better — that's what we've all been told. The more complicated something is, the more ways there are for it to break. This seems logical and is something we should consider as we invent new machines. The challenge is that simple machines do simple things and often can only do one thing well. A simple bottle opener, for instance, probably isn't the best tool for anything other than opening bottles, but it does what it was designed to do. Complicated machines — both mechanical and biological — have more parts, and often can be used to do more than one thing. As an example, the adult human body typically has 206 bones and can be used for all kinds of things from opening bottles to competing in triathlons. Inventing machines that can do a variety of things requires that the machines have multiple parts that work together, preferably without failing. Simulation tools in the product portfolio from ANSYS help make designing useful machines easier and faster, as well as more fun.

Joints

When machines were simpler, there were fewer options, and multiple parts could be connected in mechanical software from ANSYS only using shared nodes, beam elements, coupling, constraint equations and node-to-node contact. These methods were adequate for many years, but eventually general surface contact was released to address the limitations. With this new functionality, parts undergoing large rotations, deformations, sticking, sliding and a host of other real-world behaviors could be modeled.

General surface contact became popular and widely used. It also became more robust and efficient with each successive ANSYS release for mechanical applications and is now considered mature, proven technology. One problem with the widespread use of general surface contact, however, is that sometimes it is more than is required. The relatively new capability to connect parts via joints has some potentially huge advantages that can be applied to many situations.

Joints were first released with the COMBIN7 element, which was used to model only pinned, or revolute, joints. At ANSYS 10.0, major advances to joint technology were made via the MPC184 element, which could be used to model multiple joint types, such as those that are translational, cylindrical, spherical, slot, universal, general or fixed. Joint elements are particularly interesting to those involved with the design of multiple-part machines because they can be used to enable large rotations and translations between parts at a very low computational expense. To illustrate the potential computational savings of using joints, a metal hinge is used as an example. (Figure 1.)

Joints: General Surface Contact vs. Revolute Joint Approach

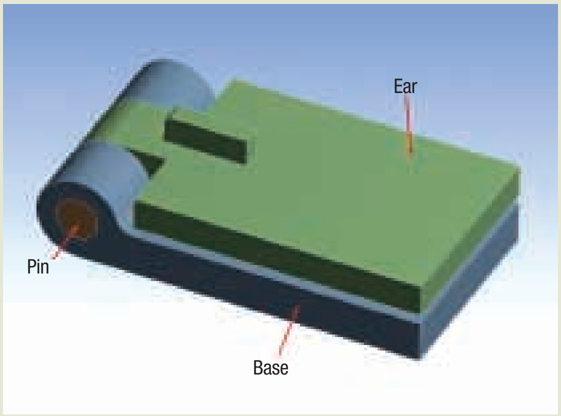

Figure 1. Hinge model

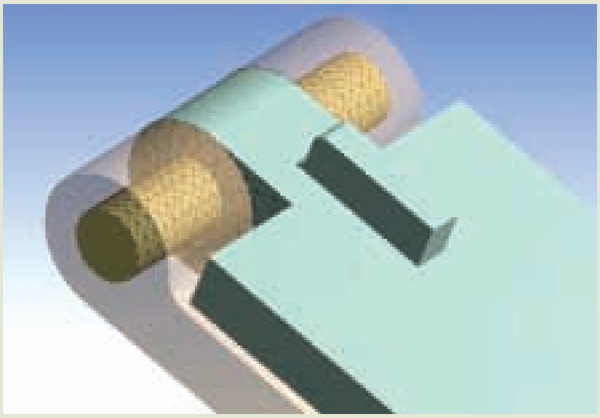

Figure 2. For this hinge model, general surface contact joints are used in three locations. First, where the ear meets the base, frictionless surfaces prevent translation along the axis of the pin and still allow rotation of the ear and base against each other at the joint. Second, bonded surfaces between the pin and the base prevent the pin from spinning or translating relative to the base. Then lastly, frictionless surfaces between the ear and the pin allow the ear to rotate freely about the pin.

There are many ways to set up a model for a metal hinge, but the two used in this investigation are a traditional general surface contact approach and a revolute joint approach (Figure 1). To simplify, the parts are set to be rigid so that problem size changes can be compared more easily. For each approach, a single CPU laptop is used to run the simulations.

In the general surface contact approach, to enable rotational freedoms but constrain all translationals except one, three contact surfaces are required (Figure 2) and one remote displacement, which rotates the hinge 90 degrees counter-clockwise. Using a few user-defined mesh specifications for surface contact size (body and edge sizing), the problem consisted of 7,188 elements (Figure 3) and took 2,249 seconds to solve.

By changing from a general surface contact approach to a revolute joint–based approach, there are three rigid parts and two joints connecting those parts to each other at the hinge: one revolute joint between the ear and the pin, and one fixed joint between the base and the pin. The pin could be suppressed since it won't perform any function once it is replaced with a revolute joint, but it is included in the model to make the run-time comparison equivalent with the general surface contact approach. The total problem size, as expected, is far smaller, uses only 14 elements (Figure 4) and requires a solution time of only 1.625 seconds.

So what have we learned? First, if detailed contact information at the hinge pin is unimportant, it is a lot more efficient to replace thousands of contact elements with a single revolute joint element. Doing that, the model can be solved in a fraction of the time it took to solve without the use of joints. Second, as can be seen from the element listing in Figure 4, even in a model in which contact surfaces are not specified, there are still contact elements — which come from use of the joint or MPC184 element — but far fewer of them.

TYPE	NUMBER	NAME
1	1	MASS21
2	1	MASS21
3	1	MASS21
4	1	CONTA176
5	1	TARGE170
6	180	CONTA174
8	1	TARGE170
9	178	CONTA174
10	180	CONTA174
11	178	TARGE170
12	576	CONTA174
13	1	CONTA176
14	1	TARGE170
15	832	CONTA174
16	1408	CONTA174
17	1408	TARGE170
18	288	CONTA174
19	832	CONTA174
20	288	CONTA174
21	832	TARGE170

Figure 3. Element description for hinge joint modeled with general surface contact

TYPE	NUMBER	NAME
1	1	MASS21
2	1	MASS21
3	1	MASS21
4	1	CONTA176
5	1	TARGE170
6	2	CONTA176
7	1	TARGE170
8	1	CONTA176
9	1	TARGE170
10	2	CONTA176
11	1	TARGE170
12	1	MPC184

Figure 4. Element description for hinge joint modeled with a revolute joint

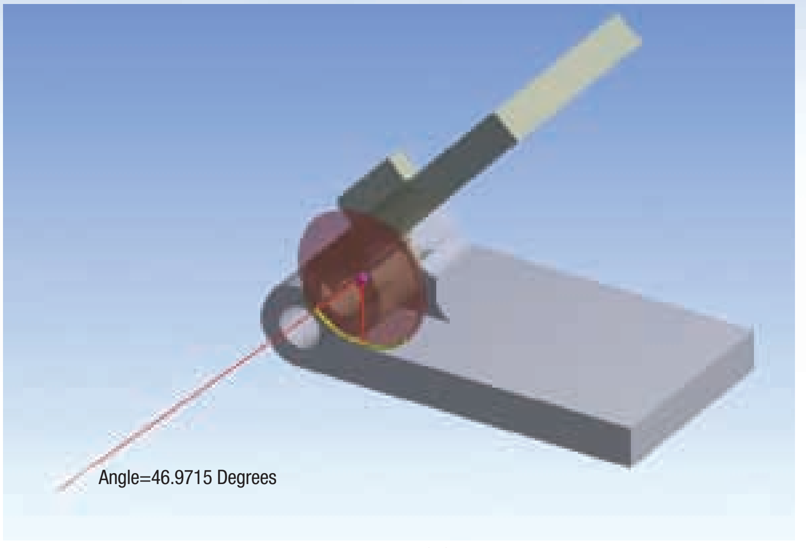

Figure 5. Interactive joint manipulation is possible within the ANSYS Rigid Dynamics module, performed on a computer screen by using the mouse to move the model.

ANSYS Rigid Dynamics

The ANSYS Rigid Dynamics module, first released at Version 11.0, makes extensive use of joints for connecting parts. This is an ANSYS Workbench add-on tool for users who have ANSYS Structural, ANSYS Mechanical or ANSYS Multiphysics licenses. The module enhances the capability of those products by adding an explicit solver that is tuned for solving purely rigid assemblies. As a result, it is significantly faster than the implicit solver for purely rigid transient dynamic simulations. The ANSYS Rigid Dynamics module also has added interactive joint manipulation and ANSYS Workbench *Simulation* interface options.

Interactive joint manipulation allows the user to solve a model essentially in real time — the explicit solver produces a kinematic solution with part positions and velocities — using the mouse to displace the parts of the model. This tool is on the menu bar in the Connections folder. New Configure, Set and Revert buttons can be used to exercise a model that is connected via joints, set a configuration to use as a starting point or revert back to the original configuration as needed. In the case shown in Figure 5, before finding a solution, the hinge has been rotated a little more than 46 degrees to verify that the joint is, in fact, behaving like a hinge.

The ANSYS Rigid Dynamics module is run using the same techniques that are used in ANSYS Workbench *Simulation* — attaching to the CAD or the ANSYS DesignModeler model, using the model tree, populating the Connections folder and inserting New Analysis, for example.

The combination of the explicit Runge–Kutta time integration scheme and a dedicated rigid body formulation creates a product that while limited to working only with completely rigid parts,

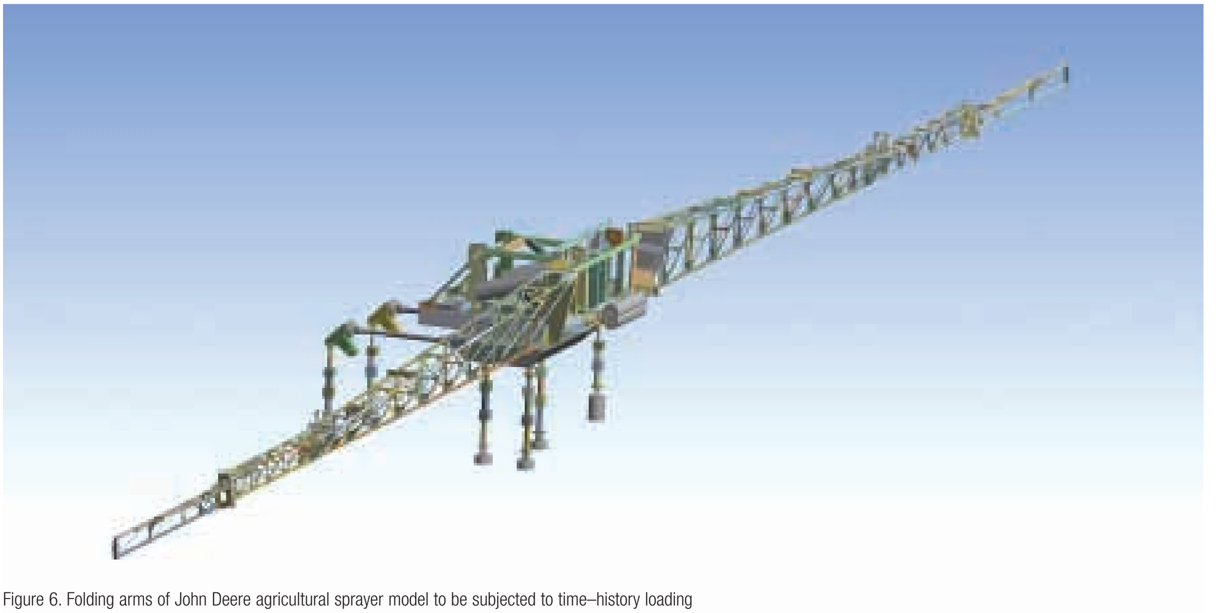

Figure 6. Folding arms of John Deere agricultural sprayer model to be subjected to time–history loading
Image courtesy Brenden L. Stephens, John Deere

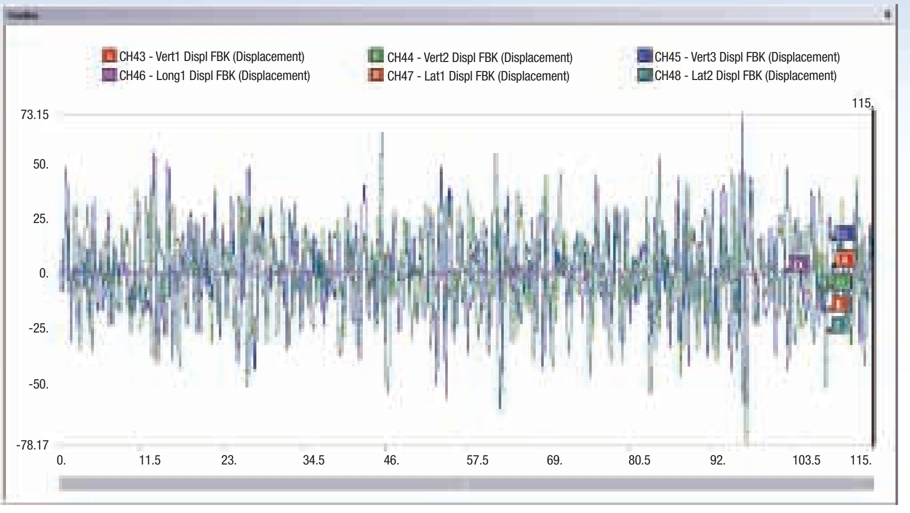

Figure 7. Time–history loading at six different geometric locations along the sprayer model in Figure 6
Image courtesy of Brenden L. Stephens, John Deere

is extremely well suited to solving multi-jointed assemblies, such as the folding arm agricultural assembly (Figure 6). This scheme is adept at handling complex time–history input (Figure 7) and is extremely fast compared to more traditional solvers. Solve time, even for complex assemblies, is typically measured in seconds and minutes rather than in hours and days. One caveat worth mentioning is that, at release 11.0, parts need to be connected with joints rather than contact when using the ANSYS Rigid Dynamics capability. If contact is required to accurately represent the part interactions, flexible dynamics simulation is required.

The ANSYS Rigid Dynamics tool should be used first on any complex, multi–part assembly with connections. Fast solution times can help users quickly find joint definition problems, inadequate boundary conditions, overconstraints and other problems. With the time saved, multiple design ideas can be analyzed in the same amount of time that it previously would have taken to simulate a single concept.

ANSYS Flexible Dynamics

Is the ANSYS Rigid Dynamics tool all that is needed to fully understand a prototype of a machine? What happens if the parts deform? Will they break? Will they fatigue and fail after a short time or only after extreme use? If parts bend, twist and flex, will the machine still perform its intended function?

The ANSYS Rigid Dynamics capability, for all its strengths, doesn't provide a complete picture of a machine's performance. In a thorough machine prototype investigation, the next step is a flexible dynamics analysis, which allows some or all of the machine's parts to behave as they would in the real world — flexing, twisting and deforming. Flexible dynamics allows users to examine parts to identify whether they are stiff and light, as they would be if made from titanium, or heavy and flexible, as they would be if made from rubber.

A more in-depth explanation of the use of ANSYS Structural, ANSYS Mechanical or ANSYS Multiphysics products running flexible nonlinear dynamics simulations is necessary to demonstrate the steps required to take an all-rigid dynamics model and turn it into a partially or completely flexible model. This translation from a rigid to a flexible model includes material assignment, meshing and solver setup. Without writing a spoiler to any future articles on this subject, this is remarkably easy to do.

The simple machines have already been invented. We don't really need a more efficient bottle opener. With the addition of more realistic and faster modeling solutions — achieved by combining the ANSYS Rigid Dynamics module and ANSYS Structural, ANSYS Mechanical or ANSYS Multiphysics software — complicated machines can be less prone to failure and produce fewer career-limiting disasters. ■