IT'S A SNAP

Audio system that uses snap-fit assembly

Valeo uses static nonlinear best practices to simulate snap-fits using ANSYS software.

By K. Vaideeswarasubramanian, Engineer, CAE; **Vinod Ryali Balaji**, Senior Engineer, CAE; and **Karthic Sethuraman**, Engineering Manager, Valeo India Private Limited, Chennai, India

Valeo produces many automotive components — such as smart antenna systems, smart keys, switches, mechanical control panels, thin film transistor (TFT) displays and electronic control unit (ECU) enclosures — that are secured and, in some cases, activated by snap-fits during the assembly process. In each case, the clipping and unclipping forces must be calculated, and risk of structural failure must be evaluated. This is achieved by performing static nonlinear simulation of snap-fits that includes multiple contacts with friction and thermoplastic materials. Valeo engineers have developed best practices for using ANSYS Mechanical in all stages of the simulation process, from geometry preparation to post-processing.

GEOMETRY PREPARATION

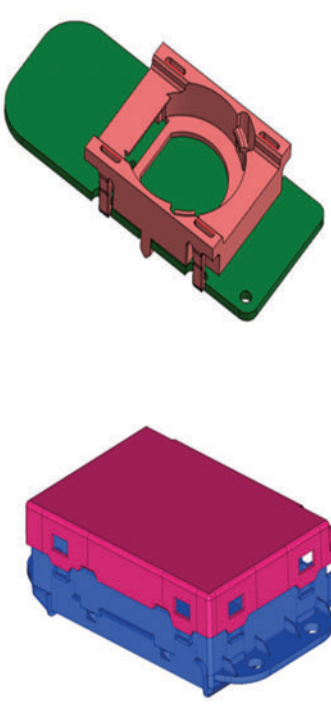
A sweepable volume has the same number of vertices per face and a smooth path from the source to the target face. One advantage of a sweepable volume is that it can be automatically meshed with hexahedron or brick elements that can fill a volume more efficiently. This leads to fewer elements and faster solution times. Another advantage is that brick meshes are more uniform, which provides greater accuracy. The option `Show - Sweepable Bodies` in ANSYS Meshing quickly identifies sweepable bodies within the assembly. Bodies that are not sweepable can be sliced into sweepable volumes and the `Form New Part` option can be used to ensure element connectivity between sliced parts.

Geometric features in the model with sharp edges close to the snapping region are common sources of nonconvergence. This problem can be addressed by adding small fillets to these specific contact regions in the simulation model.

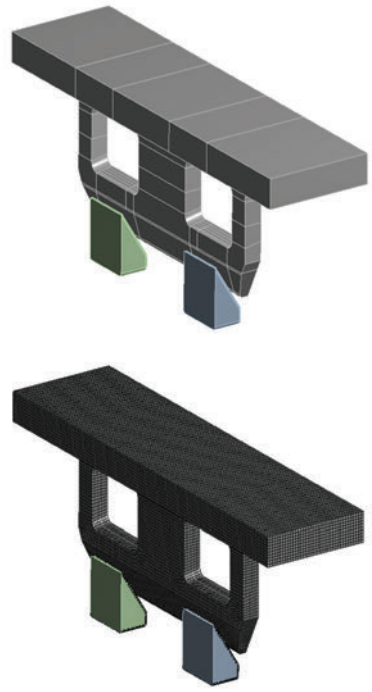
Engineers reduce computational time by defining the parts that are not of primary interest as rigid bodies, without having any significant effect on results accuracy.

MATERIAL MODELING

Thermoplastic material modeling is still much more of an art than a science, and each current method has limitations. One of the challenges is that the breaking point of many thermoplastic materials is not available in any number of commercial



▲ Typical snap-fit application



▲ Dividing geometry into sweepable volumes shown in top image. Meshed model shown below.

material databases. The absence of a breaking point can cause convergence difficulties. In some cases, Valeo engineers solved this problem by obtaining the breaking point of the material from the material supplier. However, when breaking point data is unavailable, extrapolation of the available stress–strain data is performed on a case-to-case basis to improve convergence.

A limitation of the finite element method is that when a small region of a model bears an excessive load, the elements in this region can become distorted, which has a negative impact on accuracy. The engineers avoided this problem by slicing the areas where high compressive stresses and strains occur, then assigning linear elastic properties to these slices to obtain better convergence. Generally, the results from a model with a small linear elastic region do not vary much from a nonlinear model. In addition, element distortion and resulting noise in the force displacement curve are usually eliminated.

CONTACT SETTINGS

When two or more clips are simultaneously activated in an assembly, convergence problems may occur due to contact chattering. Valeo engineers define the clips as a single contact-target pair to alleviate this problem.

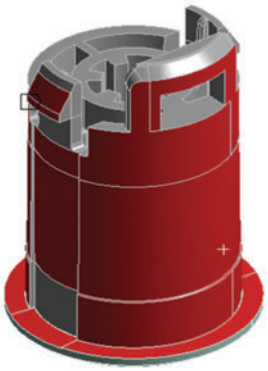
Co-efficient of friction values for interfaces in snap-fit assemblies are frequently not known. However, accurate friction values are often critical to achieve simulation results that correlate with physical tests. If the results do not correlate well with physical tests, the friction co-efficient is varied in the simulation until a good correlation is effectively achieved.

When multiple snap-fits are used in an assembly, the solution often does not converge beyond a particular point using frictional contacts. In this case, the team runs the solution until maximum force is obtained with a frictional contact. Accuracy is critical up to that point, because maximum force is often highly dependent on friction. The engineers then perform the complete simulation with a frictionless contact and use the results from frictionless contact only from that substep for which the solution with frictional contact did not converge.

MESHING

In some cases, problems such as generation of highly distorted elements may be experienced with a default surface mesh. These problems can be addressed by using the mapped face mesh option, in which the ANSYS software

Valeo engineers developed best practices for simulation of snap-fits using ANSYS Mechanical in all stages of the simulation process.



▲ Two clips defined as a single contact pair

maps a rectangular grid to a rectangular domain. The analyst can choose the number of divisions for each edge. The mapped face mesh option provides element shapes that are generally well within acceptable quality limits for the solver.

When converting geometry into sweepable volumes, it often turns out that there are some leftover areas that are not sweepable. In such cases, it is preferable to use a tetrahedral mesh. The hex-dominant mesh method should be used with great care, especially when high compressive strains on elements are expected.

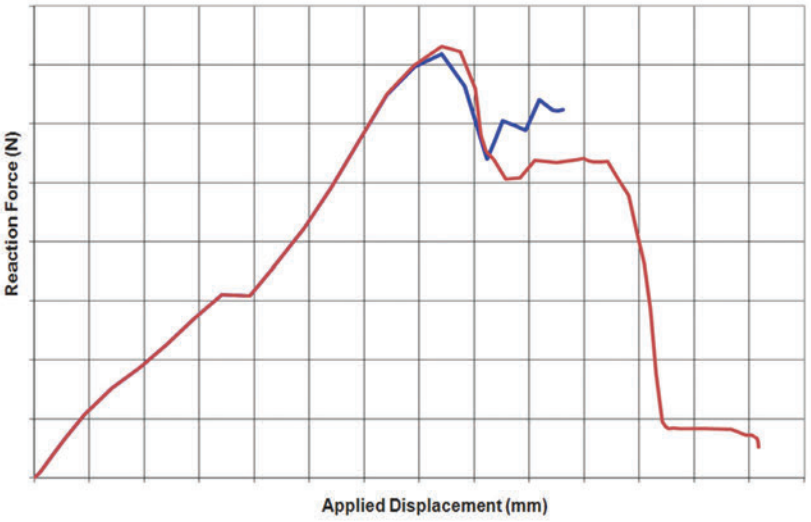
LOADING

Displacement control, rather than force control, usually provides better convergence in the snap-fit assembly.

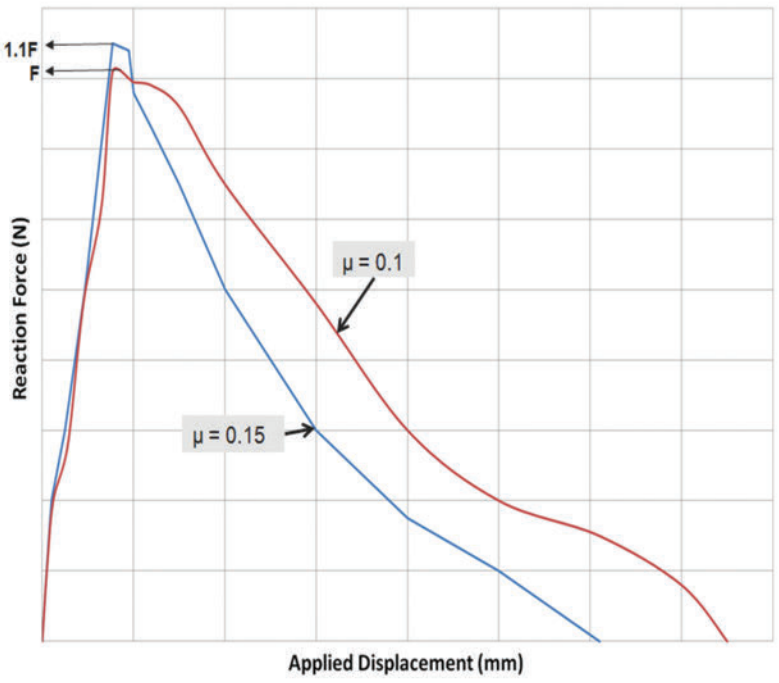
Many snap-fit assemblies experience large displacements for very small loads. Separating the loads into a number of small steps will aid in smooth convergence; it will also capture critical clipping points.

ANALYSIS SETTINGS

The distributed memory parallel solver for ANSYS Mechanical generally provides the fastest solution times. This solver decomposes the model into domains and sends each domain to a different core to be solved. A considerable amount of communications between the different cores is required. The results are automatically combined at the end of the solution. There are some cases, usually involving highly distorted elements and excessive strains, in which the distributed solver will terminate abruptly. In these cases, engineers use shared memory parallel solver.



▲ Blue line represents partially completed nonlinear solution. Red line is completed nonlinear solution with a local linear elastic region.

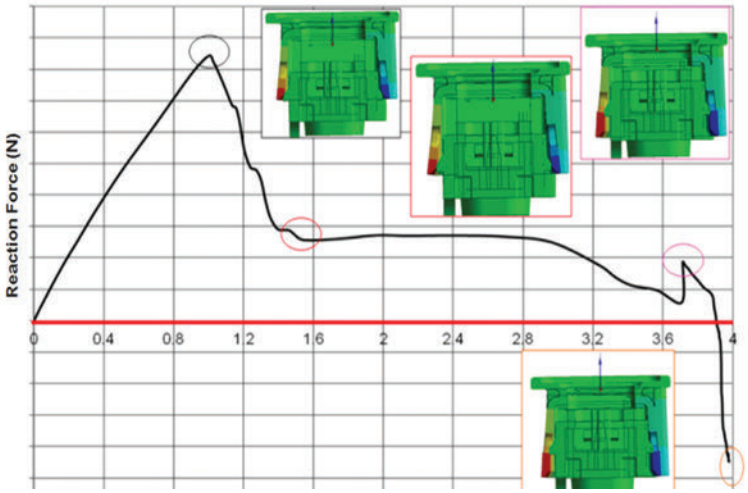


▲ Simulation results for different friction co-efficients. These results will later be compared to test results.

POST-PROCESSING

It is important to know the maximum force required for a snap-fit assembly; when multiple steps are involved, the force required in each step is also important. For better clarity, Valeo engineers overlay the corresponding deformed model alongside each peak in the reaction force curve.

Thermoplastics tend to be very strong in compression, so in most cases the results in tensile areas are most critical for the design process. However, if high stresses and strains occur in compression, there is the potential for plastic deformation to occur. In such cases, the compression results are treated on a



▲ Deformed shapes overlaid on reaction force curve showing multiple peaks

DIAGNOSING NONLINEAR STRUCTURAL SOLUTIONS IN ANSYS MECHANICAL
ansys.com/92snap

case-to-case basis depending upon the material and snap-fit design.

By using best practices and ANSYS Mechanical software for nonlinear simulation, Valeo engineers have confidence that their snap-fits will work reliably. Performing structural simulation very early in the design process helps to avoid costs associated with multiple prototypes, rework and changes to tooling. ANSYS high-performance computing has reduced simulation time by 50 percent, making it possible to complete the structural simulation for clipping and declipping processes in one week.

ANSYS HPC reduced simulation time by 50 percent, making it possible to complete structural simulation for clipping and declipping processes in one week.