

CAE Applications and Techniques used in Calculating the Snaps Insertions and Retentions Efforts in Automotive Trims

Mohammed K Billal and B V Moorthy
Chrysler India Automotive Pvt, Ltd.

Dan Aquilina and Steven Schenten
Chrysler LLC

ABSTRACT

A snap-fit is a form-fitting joint, which is used to assemble plastic parts together. Snap-fits are available in different forms like a projecting clip, thicker section or legs in one part, and it is assembled to another part through holes, undercuts or recesses. The main function of the snap-fit is to hold the mating components, and it should withstand the vibration and durability loads. Snap-fits are easy to assemble, and should not fail during the assembling process. Based on the design, these joints may be separable or non-separable. The non-separable joints will withstand the loads till failure, while separable joints will withstand only for the design load. The insertion and the retention force calculation for the snaps are very essential for snap-fit design. The finite element analysis plays a very important role in finding the insertion and the retention force values, and also to predict the failure of the snaps and the mating components during this process. The snap insertion and retention simulation is highly non-linear, due to the non-linear material behavior and contact between the mating components. This paper explains the various techniques and methodology used to predict the snaps insertion, and the retention force for different types of snap-fits.

CITATION: K Billal, M., Moorthy, B., Aquilina, D., and Schenten, S., "CAE Applications and Techniques used in Calculating the Snaps Insertions and Retentions Efforts in Automotive Trims," *SAE Int. J. Passeng. Cars - Mech. Syst.* 7(2):2014, doi:10.4271/2014-01-1032.

INTRODUCTION

The plastic trims are assembled with their neighbor components through snap joint features or using special fasteners like clips, pins, and rivets. An integral attachment snap-fit feature is commonly used in the plastic parts and act as an interface between the assembly parts to enhance the attachments. Integral attachment features can provide benefits to the part design, by removing tool requirements and reducing the part count [1]. In automotive industries, most of the interior and exterior trims have snap-fit features, which are used to connect the neighbor components. The clip and pin is an individual part, which is made of either plastic or steel, and it works on the principle of the snap-fit.

The snap-fit joints are simple in design, economical, and a rapid way to join two different parts. The common principle of the snap-fit joint is the protruding feature available in one of the parts like a hook, bead or stud, which is deflected during the insertion process, and catches the undercut or depression in the mating part [2]. It requires some effort to insert the snap features in to the mating part and after the insertion, the snap feature returns to stress-free condition. These snap-fits may be separable or non-separable joints based on the undercut and

the snap design. The separable joint will take some retention load, and then separate from the mating component, without any damage. The non-separable joint will take the retention load, and prevent the parts from separating apart till a failure occurs [3].

In this paper, the overview of CAE techniques used, like solver selection and contact formulation is discussed initially. The snap-fit is classified under two categories; integral snap-fit joints and the panel fasteners/clip. In snap-fit simulations, the overview about the snap-fit modeling, CAE setup, the deformation behavior of each snap-fits and comparison between the CAE results and the test results are discussed.

CAE APPROACHES

The snap-fit simulation is highly non-linear, and involves the geometric, material and boundary/contact non-linearity. The selection of the appropriate CAE methods and contact formulations will help to reduce the time and cost, and improve the accuracy. For the steel and plastic components, the elastic-plastic material model is used in the snap-fit simulations.

Solver Selection

Nonlinear simulations can be solved by using implicit or explicit methods based on the complexities and nature of the problems. The abaqus solver is used for these simulations, and it has the capabilities of handling both implicit and explicit methods, with different material models and contact formulations. The implicit static method is more efficient for solving the smooth nonlinear problems, and for the snap-fit simulations is usually solved by this method. But this method may have difficulty sometimes, due to local instabilities that occurred in the models during the analysis. The origin of the local instabilities is from the geometry, such as the local buckling; contact, such as the sudden separation of bodies; and friction, such as stick-slip behavior. These local instability problems can be solved, by introducing damping in the implicit static analysis. Otherwise, it can be solved dynamically, by using the implicit dynamic solver or the explicit solver.

Contacts Formulations

In snap-fit simulations, contact is involved between two or more components. During the contact, a normal force will act on the contact surfaces between the two bodies, when they touch each other. If there is friction between the surfaces, shear forces will be created, that force will resist the tangential motion (sliding) of the bodies. In finite element analysis, the contact conditions are a special class of discontinuous constraint, which will allow the forces to be transmitted from one part of the model to another. The constraint is discontinuous, because it is applied only when the two surfaces are in contact. When the two surfaces separate, no constraint is applied. The friction is defined between the contact pairs, which will resist the relative sliding between the surfaces. The coulomb friction is a common friction model, used to describe the interaction of the contacting surfaces. The model characterizes the frictional behavior between the surfaces, by using a coefficient of friction. For the snap-fit simulation, an appropriate coefficient of friction value is defined based on the materials.

The implicit solver provides several contact formulations and each formulation is based on the choice of a contact discretization, a tracking approach, and assignment of "master" and "slave" roles to the contact surfaces. Once the contact is detected, the solver will calculate the relative sliding between the two surfaces, which is a very complex calculation. Based on the relative sliding, two tracking approaches are available; one is a small sliding formulation, which is less expensive computationally, and the other is a finite sliding formulation, which is more expensive. In snap-fit simulations, a finite sliding motion will always occur between the snaps and the mating part. So the finite sliding contact formulation is used for the snap-fit contacts. Based on the contact pair surface discretization, two options are available; the node-to-surface and the surface-to-surface discretization. In the node-to-

surface, the contact conditions are established such that each "slave" node on one side of a contact interface, effectively interacts with a point of projection on the "master" surface, on the opposite side of the contact interface. Thus, each contact condition involves a single slave node and a group of nearby master nodes, from which the values are interpolated to the projection point. In surface-to-surface discretization, the shape of both the slave and master surfaces will be considered in the region of contact constraints. The surface-to-surface discretization provides more accurate stress and pressure results, than the node-to-surface discretization. For the snap-fit simulation, mostly the surface-to-surface discretization is used between the contact surfaces. But, if there is an edge contact, then the node-to-surface is used, because the node-based surfaces will be used to represent the edges, and it is not applicable to the surface-to-surface discretization.

The explicit solver provides the general ("automatic") contact algorithm, which is very simple to define the contact between many or all regions of a model with a single interaction. In general contact, the interactions are typically defined by specifying self-contact for a default; the element-based surface defined automatically, includes all bodies in the model. To define the contact domain, the specific surfaces pairs can be included or excluded. The edge-to-edge contact is available in the general contact, which will consider the perimeter edges and geometric edges.

INTEGRAL SNAP-FIT JOINTS

The snap-fit joint is an economical and quick method to join the plastic trim parts. A snap-fit is so designed, that it is easy to assemble and disassemble without breaking the components. The strength of the snap-fit depends on the material used, its geometry, and the load acting on it [4]. The integral plastic snaps are used to hold the trim components along each other, and also it will be easy to remove for services and maintenance purposes. The load withstanding capacity of the integral snap is less compared to the metal clips.

Sun Visor Snaps

A sun visor is an automotive part, used to shade the eyes of the drivers and passengers, to avoid the glaring of sunlight. It is located on the interior of the vehicle, just above the windshield, and is designed with a hinged flap which is adjustable. The sun visor assembly consists of a bent metal rod which is covered with a plastic mold, and this plastic mold is snapped to a mounting bracket, which is mounted on the sheet metal above the headliner. The sun visor flap is covered with foam and fabric materials, and inserted in to the center bracket, which is attached to the headliner and sheet metal part. [Figure 1](#) shows the sun visor assembly.

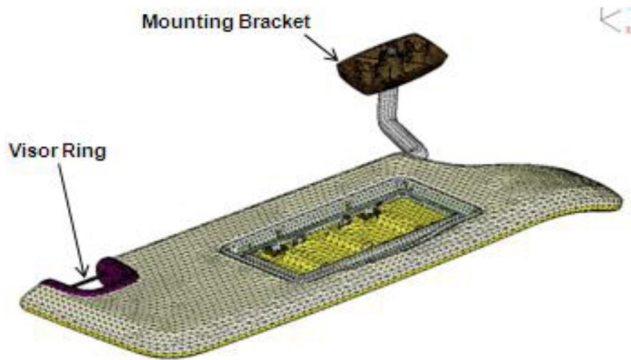


Figure 1. Sun Visor Assembly

Snap-Fit Simulation at Mounting Bracket

The steel rod with the plastic mold in the sun visor is assembled with the plastic mounting bracket separately, and then the mounting bracket with the sun visor assembly is attached to the sheet metal in the vehicle. During this assembling process, the mounting bracket is hot, so that it is easy to insert the sun visor rod. The plastic mold and the mounting bracket should not fail during this assembling (insertion) process.

In the CAE, the mounting bracket and the portion of the steel rod with the plastic mold are considered for the insertion analysis, as shown in [figure 2](#). Tie contact is defined between the steel rod and the plastic mold. The sliding contact is defined between the mounting bracket and the plastic mold. The appropriate thermal effect is considered for the simulation, and an equivalent thermal material property is used. The load is applied through MPC, which is connected to the steel rod and plastic mold assembly, as shown in [figure 3](#). The implicit solver is used to solve this insertion simulation. [Figure 4](#) shows the snap deformation steps during the insertion analysis. [Figure 5](#) shows the comparison of the CAE and test results.

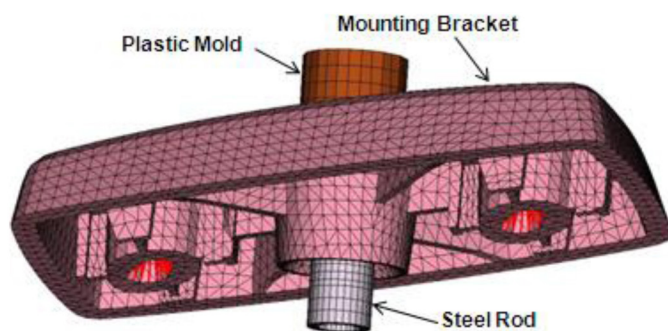


Figure 2. Sun Visor - Mounting Bracket: CAE Model

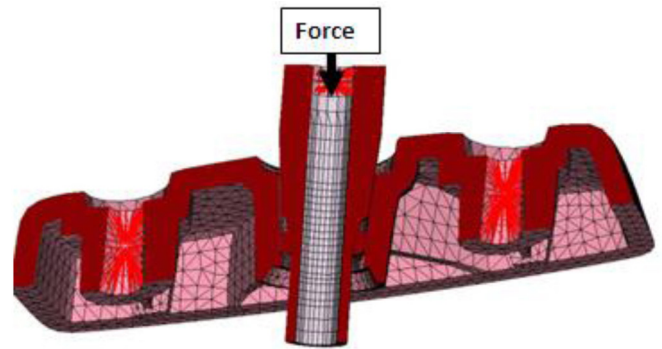


Figure 3. Sectional View: Loading Details

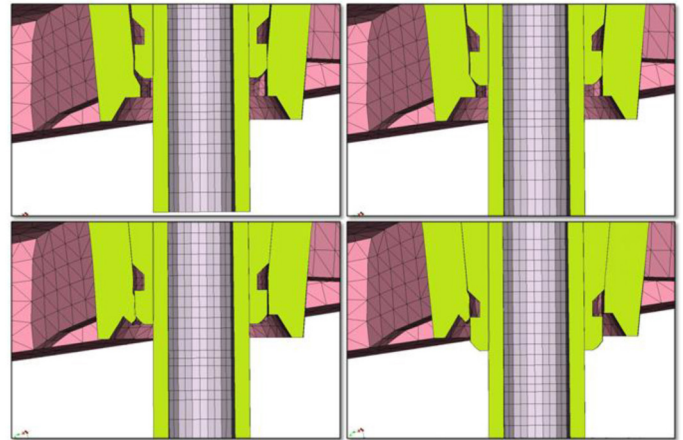


Figure 4. CAE Results: Deformation Steps at Insertion

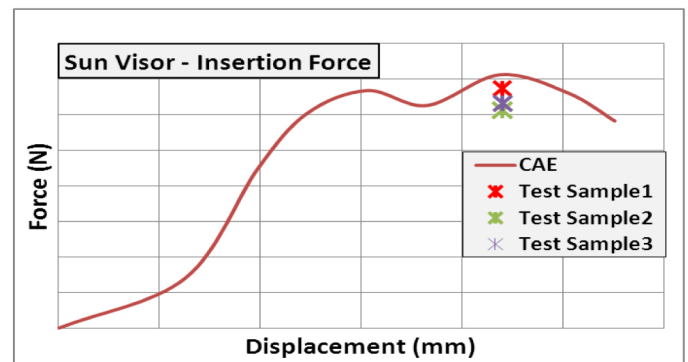


Figure 5. FD Curve: Test Vs CAE Results

Snap-Fit Simulation at Center Support

In the sun visor, one end is attached to the mounting bracket which is a non-separable snap fit, and the other end is assembled with the center support bracket, which is a separable snap-fit. The sun visor is inserted and extracted from the center support bracket several times by the passenger or driver. It is very critical that the snap-fit is easy to insert and extract from the center support bracket; also, it should hold the sun visor during normal driving conditions.