CAE Simulation in Design and Development of an Automotive Sunroof System

Joe Saxon, Pan Shi
ArvinMeritor, Inc.

Abstract
The automotive industry relies heavily on CAE simulation to reduce product weight and development costs. Even “non-structural” components are engineered to withstand unusual events like collisions or unlawful forced entry. This is especially true of aperture systems like door latches and sunroofs. ArvinMeritor, a global automotive supplier, incorporates CAE in the development of its aperture products. One example is FEA verification of sunroof “push-out” strength—a sunroof’s ability to prevent occupant ejection during an accident. This paper will discuss ArvinMeritor’s use of FEA to verify push-out strength for a new generation sunroof design that replaces critical steel components with polymer. Throughout this effort, ANSYS was used to simulate the highly non-linear test-to-destruction that defines push-out strength. Discussion includes development of FEA methods that were both practical to implement and accurate when compared with tests, thus inspiring the design team’s confidence. Also discussed is the analyst’s role in not only screening, but also suggesting design changes.

Introduction
ArvinMeritor has been a leading supplier of automotive roof systems in both the North American and European markets. OEM specifications require the roof to make a significant contribution to overall vehicle stiffness, and to withstand abusive loads like the push-out test. Push-out specifications insure the ability to prevent occupant ejection during a crash, often proving the most challenging strength specification to achieve. In ArvinMeritor’s latest generation of sunroofs, critical components once made from steel have been replaced with polymers. When FEA was first introduced into the design process, its results were checked frequently against prototype tests to establish good modeling technique, correlation, and ultimately, confidence in its results. Subsequently, FEA became a prime source of design innovation and verification, reducing prototyping and the overall cost of product development.

This paper will describe FEA’s role in the design of a new generation sunroof to meet OEM push-out specs. Topics will include description of the sunroof system, modeling techniques, correlation with test results, and design evolution of the critical component.

System Description
Figure 1 shows a schematic of the load bearing components in a typical sunroof. The geometry is half-symmetric about the car’s vertical mid-plane, and the push-out load is applied to the glass panel at the mid-plane. The panel includes a steel stiffener bonded to its perimeter. Tabs on the stiffener are bolted to a slotted arm known as a colissa. The colissa is vertically and laterally supported in two places by the front and rear guides. The guides slide forward and backward in the frame to position the glass. The front guide’s position determines fore/aft location of the glass. The rear guide’s position relative to the front guide determines tilt of the glass. Thus, loads applied to the glass panel pass through the colissas, then the guides, and are finally reacted by the frame.
Element Selection

The glass panel and its steel stiffener are natural candidates for shell elements. However, the bonding material between them has a significant thickness and must be modeled with solid hex/wedge elements (ANSYS Solid95).

Historically, colissas have been flat steel strips with relatively little geometric detail. More recent polymer designs include numerous geometric features requiring detailed solid meshes to correctly capture their shape. Either tetrahedral or hex/wedge meshes are appropriate. Each has their own advantages, as will be discussed later. The same is true for front and rear guides, but emphasis is reserved for the rear guide because it carries most of the push-out load.

The sunroof frame is extruded aluminum with a cross-section not ideally suited for shells or solids, the former missing some detail and the latter requiring a huge model. Because failure of the frame is never an issue, shells have been chosen because they adequately and economically represent frame stiffness, even if they sacrifice the frame’s detailed stress distribution.

Modeling contact between the various components in the load path can be tricky. Generalized contact is one option, but it invariably leads to long solution times and convergence issues. Carefully placed gap (ANSYS Contac52) elements have proven to be an attractive alternative. Gap elements are suitable because the contact locations are precisely known and do not change significantly under load. Also, the immediate areas surrounding the gaps are not considered stress-critical. Extra pre-processing time is required to correctly place the gaps, but time saved in the solution phase more than makes up for it.
Material Properties

Material properties are critical when modeling load-to-failure. The polymer referenced here is fiber reinforced and nominally isotropic. ANSYS KINH model was adapted to stress-strain data provided by the material supplier. Likewise, correctly modeling plastic response of the steel stiffener proved to be an unexpected but essential factor in achieving good test correlation. However, this significance was not discovered until late in the analysis & correlation process. Properties assigned to the glass/stiffener adhesive were based on past experience.

Validation of FEA Results

When FEA is newly introduced in a design process, its results must be validated through comparison to experimental data. This may initially lead to refinement of the modeling techniques, but it must ultimately establish the level of confidence placed in the results. For the sunroof development described in this paper, FEA was introduced in the design process only after failure of an initial prototype. For some time thereafter, FEA and prototype testing were conducted simultaneously. Comparison of the results did lead to some modeling adjustments, and then to strong confidence in the FEA. The design process has now matured by incorporating FEA at an earlier stage, creating the opportunity to reduce prototyping and overall product development cost.

In this development program, testing was relatively simple and produced only three data points for comparison with FEA: critical location, load at failure, and deflection at the point of load application (center of the glass).

Validation of System Stiffness

Initial FEA results showed displacement at the glass center to be 37 mm, whereas a 60 mm deflection was measured in the test. Several aspects of the model were reconsidered in order to find the source of this discrepancy. The three most promising parameters were investigated thoroughly as discussed below. They were 1) Stiffness of the bonding material between the glass and its steel stiffener; 2) Stiffness of the bolted connections between the steel stiffener and the colissa; 3) Plasticity of the steel stiffener.

Stiffness of the bonding material between the glass panel and its steel stiffener is difficult to quantify. Relevant published data was not found, so the values used here are based on previous experience in validating and tuning sunroof models. However, this method has led to large variance in the estimated stiffness, so it was appropriate to investigate sensitivity to this parameter for the current case. A simple parametric approach was used, decreasing Young’s Modulus in the bond by 10%. The result was only a 1% decrease in system stiffness, indicating this parameter was not responsible for the large discrepancy observed between the FEA and test results.

Each bolted connection between the panel stiffener and colissa is modeled with rigid spiders. Initially, a completely rigid connection was enforced by using a single spider connecting both components as per Figure 2. This represented the upper bound of connection stiffness, thus creating suspicion it contributed to the overly stiff response of the model. As an alternative, the single spider was replaced with two spiders as per Figure 3, and their center nodes were coupled in translation only. This represented the lower bound on connection stiffness estimates, but its implementation decreased system stiffness by less than 2%. Clearly, system stiffness was insensitive to this parameter and it could not explain the FEA-vs.-test discrepancies.

The third parameter investigated was plasticity of the steel stiffener. Early in the analysis effort, little attention was focused on this component because the damage it incurred was not highlighted in the post-test inspection. Therefore, linear material assumptions were applied. Upon closer investigation, this material was discovered to be an extremely mild steel with Yield Strength of only 117 Mpa. Replacing the linear assumption with an appropriate elastic-plastic definition drove the model’s deflection up from 30 mm to 56 mm. This is considered good correlation with the measured value of 60 mm. Plasticity of the stiffener was clearly the factor missing from the early models that led to over-estimation of system stiffness.
Validation of Component Strength

Sunroof push-out is primarily a test of system strength rather than stiffness.

Fiber reinforced polymers are brittle at ambient temperatures, and in a typical test, the sunroof’s load capacity is sharply defined by sudden fracture of a key component. The initial prototype for this sunroof sustained only 79% of the customer’s target load capacity due to a fractured rear guide. This initial failure was the motivation for bringing FEA into the subsequent development efforts.

Reliable evaluation of component strength is much more mesh-sensitive than evaluation of system stiffness. Typical strength evaluation consists of tracking stress and/or strain results through successive design iterations, comparing them to material data, or better yet, comparing them to a baseline design already verified through test. Valid comparisons are much easier when revision-to-revision consistency is maintained in element type, size, and quality, especially in critical locations.

As previously stated, the critical component in this sunroof turned out to be the rear guide. It was initially modeled with tet10 (ANSYS Solid92) elements which gave results shown in Figure 4. Tet10 were initially chosen because they required less de-featuring of the CAD geometry, and therefore less pre-processing effort. Subsequently, the rear guide was modeled with hex20 (ANSYS Solid95) elements which gave results shown in Figure 5. Comparison of these two cases supports the following conclusions:

1. The indicated peak stress locations are similar for both cases and consistent with the fracture location observed in tests. The hex mesh location seems slightly better.

2. The peak stress values are within 4% of each other, the higher value coming from the hex mesh.

3. Stress contours on the hex mesh are much more smooth, and therefore more believable, than on the tetra mesh.

4. The tet10 mesh required less pre-processing effort while still giving acceptable results

Subsequent work was based on the hex mesh since it appears to give slightly better results.
FEA as a Design Tool

With a validated FEA approach in place, attention was turned to improving system strength to meet customer expectations. Doing so required a 21% increase in load capacity over the initial prototype. FEA plots of the critical component helped the design team visualize the deflections leading to failure. What had been imagined as a primarily vertical load was revealed to contain a significant lateral component that caused bending of the rear guide’s sidewall as seen in Figure 6. Subsequent discussion between the FE analysts and the other design team members led to modifications such as those illustrated in Figure 7. These include an increased thickness in the critical area and an added “wing” feature that limits rotation of the sidewall by establishing contact with adjacent components.

When these design modifications were evaluated, they were found to decrease peak stress by 23%. Generally, this should not be interpreted as a proportional increase in load capacity. But in this particular case, this assumption appeared valid since the polymer exhibited a nearly linear stress-strain curve. This assumption was validated in later testing when the system did show a similar load increase, its performance then being limited by fracture of a different component.

Conclusion

The contribution of FEA to development of this sunroof design has been clearly demonstrated. In terms of both failure location and load capacity, the FEA results have proven accurate. The FEA solutions were non-trivial, requiring non-linear geometric, material, and contact considerations requiring a sophisticated...
Figure 6. Deformation of original design included significant lateral bending.

Figure 7. Design Changes Illustrated on the Critical Cross-Section of the Rear Guide.
FEA solver such as ANSYS. This was not done in the absence of testing, but in conjunction with testing that provided benchmark results for correlation. Having adjusted the modeling technique to match the previously established benchmarks, solutions for untested variations were generated. Subsequent testing proved these solutions to also be accurate also. Future development of ArvinMeritor sunroof systems will incur reduced prototype costs by the confident application lessons learned from this effort. Already, a new sunroof design is underway that is fully utilizing FEA very early in the design process.