Application of Connector elements and Abaqus procedures in the Design of Driver Airbag Cover and Horn Switch Mechanism in an Automotive Passive Safety System

Sagar Venkatesh

Autoliv India Pvt. Ltd. Bangalore

Abstract:
The development of automotive passive safety system is a very complex process, as various functional requirements have to be considered. Keeping various functionalities in one module is the difficult and a challenging task for engineers. Therefore, it requires a great effort of all participating engineers to find an optimum design that satisfies all relevant disciplines. Effort from the concerned engineer should not be costly in terms of time, since the main objective of the finite element analysis is to reduce the product design life cycle and produce quick and accurate results with realistic behavior, however with some approximations.

The following salient points are discussed in detail to design a Driver Airbag Cover:-

1. *Application of connector elements*
2. *Comparing Implicit Dynamics with Static procedures for complex problems to achieve better convergence rate with more realistic behavior.*

Driver Air Bag (DAB) module consists of mainly a container, cover, horn switch, inflator, horn spring, extra travel spring and other major components.
The tear line pattern, the pre-load in the springs of the horn switch and the optimum horn effort all these functionalities to be considered and DAB module should be modeled in a very efficient manner for a faster convergence rate.
Application of connector elements in this simulation helps to reduce modeling time and realistic behavior is achieved. An approach with implicit dynamic procedure is also performed and the advantages/disadvantages are discussed briefly.

Keywords: Connectors, Driver Air Bag (DAB) module, Horn Effort, Horn Switch Mechanism, Automotive Safety, Convergence , Implicit Dynamics

1. Introduction

- Driver Air Bag (DAB) module is a Supplemental Restraint System
- DAB is a one of the Frontal Airbag Systems.
- The DAB module is an integral part of the steering wheel.

DAB module consists of mainly a container/housing, cover, horn switch unit, inflator, steering wheel armature and other major components. The DAB module resides within the steering wheel armature as shown in figure (1).

The cover should be designed for the following design criteria:

- Horn effort is optimum
- Horn switch units should take the additional load more than horn effort
- Cover should tear out in regular pre-determined pattern during deployment

The tear line pattern, the pre-load in the springs of the horn switch and the optimum horn effort all these functionalities to be considered for the design of cover in the DAB module in a very efficient manner for a faster convergence rate.

So a complete design of the DAB Cover includes the following analysis:-

- Finger Poke study – load case with ABAQUS Standard/Implicit Dynamics
- Cover flap tearing study – load case with ABAQUS/Explicit

Here we are going through the first two functionalities mentioned above, where we are using connector elements and Abaqus procedures for designing cover.
A Typical DAB module is shown in the below figure (1).

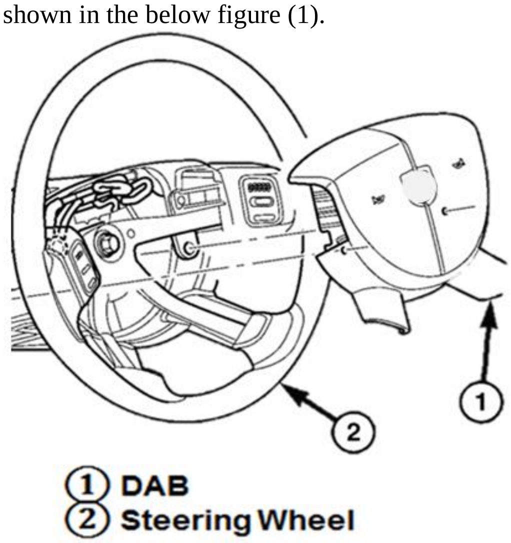

Figure 1.DAB Module

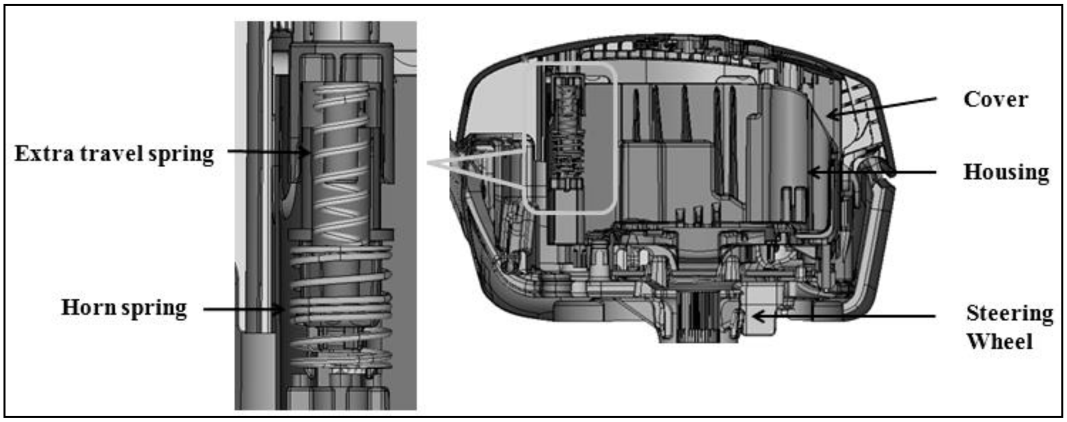

Figure 2.Detailed DAB Module and Horn Mechanism

2. Objectives of a Finger Poke Study

- To perform a finger-poke analysis on the cover assembled to the housing and taking into account the horn switches i.e. capturing pre-stress on extra travel spring and horn spring.
- To assess the relationship between pushing on the cover and the electrical contact at the horn switch (Forces and Displacements)
- To validate there is no tensile pressure stress observed on the tear line of the DAB cover due to finger-poke

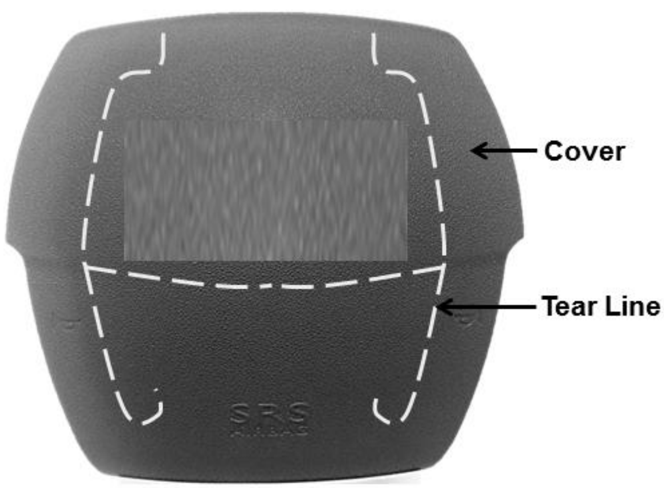

Figure 3.Cover displaying Tear line

3. Horn Mechanism

The horn switch unit mainly consists of the following components; cover, horn switch mechanism, extra travel spring, horn spring, horn track circuit. Refer to below figure (4) for simple understanding of horn mechanism.

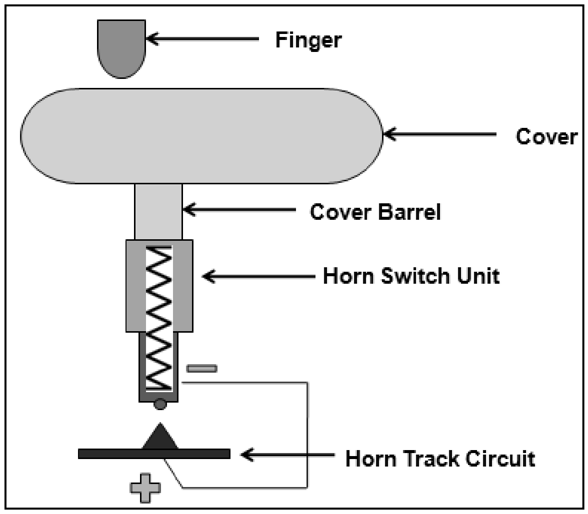

Figure 4.Simple Horn Mechanism

When the driver presses the cover unit of the steering wheel, the horn force actually pushes the cover barrel, next the horn switch unit. Once the electrical gap is closed the horn is actuated thereby causing the horn sound.

The difficult part is in simulating this mechanism in detail.

Before going through this we shall first understand horn switch unit's sub-components and its functionality, it mainly includes Horn Spring, Extra Travel Spring, Horn switch Cap and horn switch Finger. Refer figure (5).

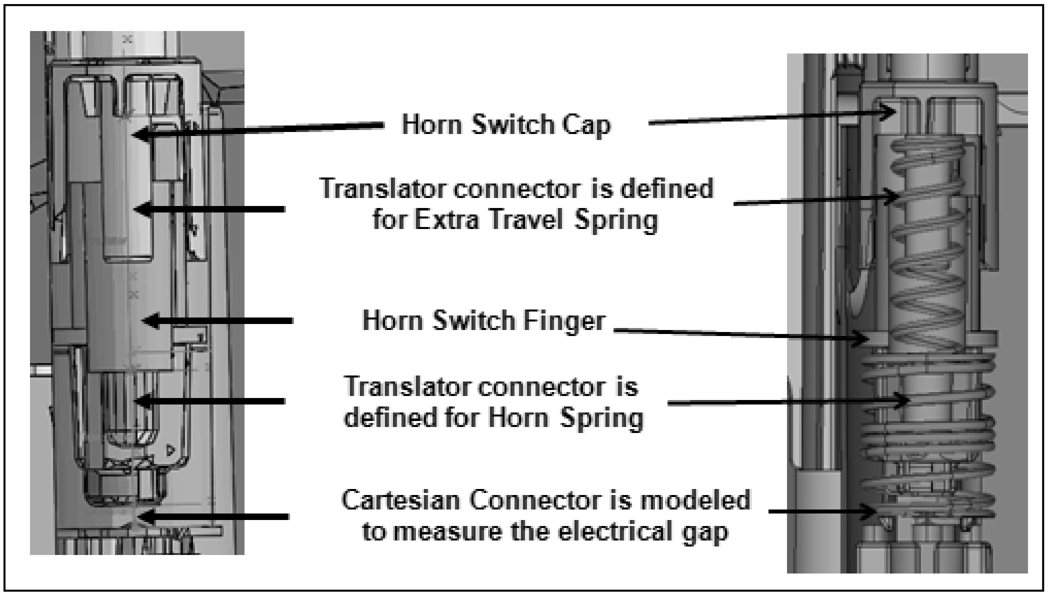

Figure 5.Horn switch unit and its sub-components.

Both the horn spring and the extra travel springs are pre-stressed and assembled in the DAB module at three different locations. The pre-stress of the springs are captured by defining “Reference length” of the connectors used. The connectors are initially modeled to the assembled length, by knowing the pre-stress load we can calculate the free length of these springs and apply “Reference length”.

The first step in the simulation is to establish the pre-stress of these two springs and establish a contact between all the components involved in the DAB module.

Once it is established the second step is to calculate the horn force required to close the electrical gap in the horn switch unit.

Firstly, when the horn force is applied by the finger, the horn spring is compressed till the electrical gap is closed, the force required for this mechanism is the actual horn force required.

Secondly, any additional force more than the required horn force exerted by the driver on the cover must be accommodated in the horn mechanism; this is done using an “Extra travel spring” with relatively high stiffness. Refer figure (5) to know how these connectors are used to replicate the functionality of the actual springs.

Once the horn spring starts travelling due to applied horn force and the electrical gap is closed, the Cartesian connector locks the horn spring not to travel further by “Connector Stop” behavior defined in Lowed bound (actual electric gap) – refer Figure (6)

Now, the additional force applied by the driver will be directed to the extra travel spring and it starts to compress gradually.

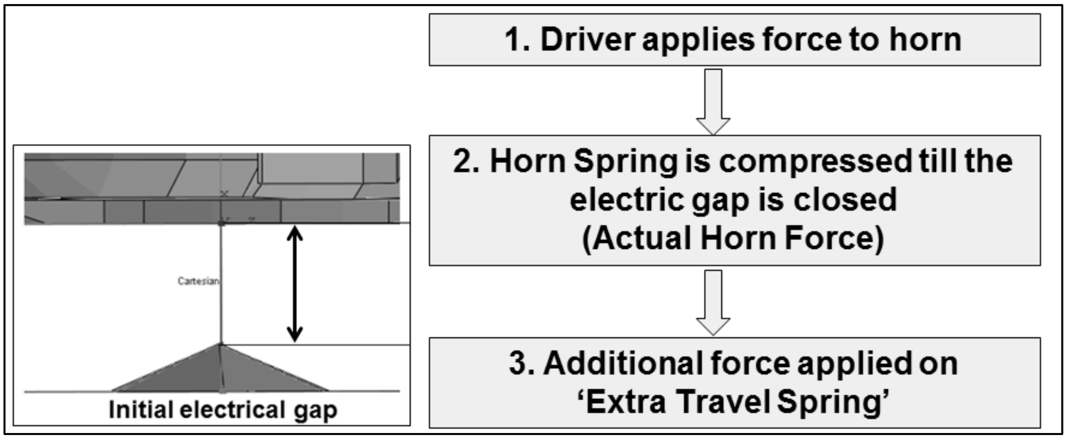

Figure 6.Initial Electric Gap and Sequence of force travel

In practical horn switch cap and the horn switch finger will not be separated because of the flange design of horn switch unit as shown in figure (7), the two components cannot move apart but can relatively move between the horn switch cap-grove.

In simulation this is captured by defining “Connector Stop” on the extra travel spring (Translator connector) by using Upper bound (actual assembled length of extra travel spring), the connector used for Extra Travel Spring will also function as the “Horn Switch Cap “ by allowing relative motion downwards but not upwards.

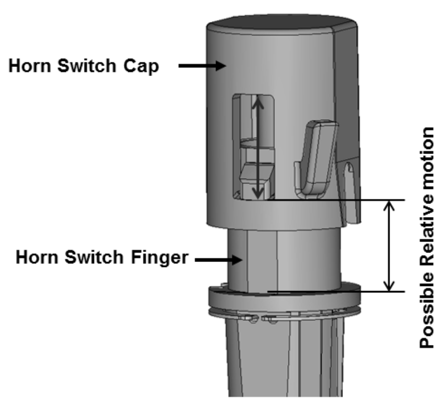

Figure 7.Possible relative motion between Horn switch Cap and Finger

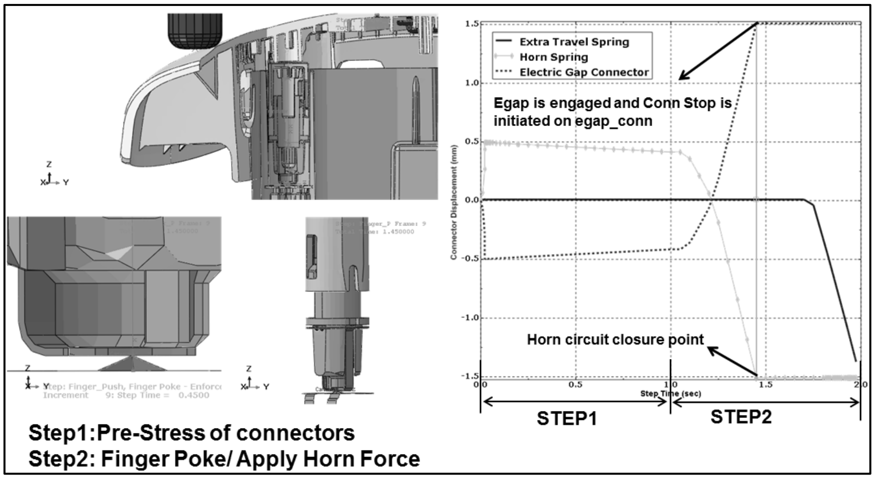

Figure 8.Outputs-Connector Displacement and Electric Gap

From figure (8), we can observe the following points:

1. The Horn Spring/Connector moves to close the electric gap thereby its movement is stopped by using connector stop
2. The electric gap Cartesian connector is used to measure the same displacement and verify that the electric gap is closed.
3. The extra travel spring is moving ahead once the horn spring is stopped i.e. additional force more than the horn force is taken by the extra travel spring (Refer figure 9)

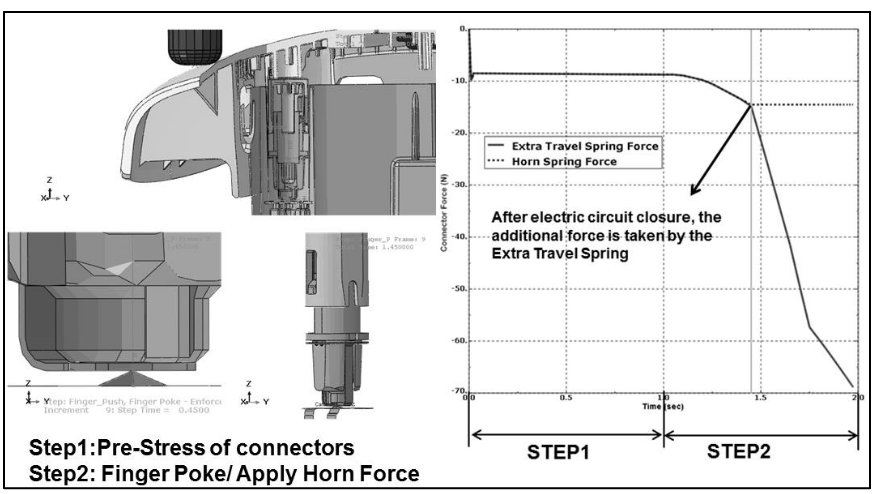

Figure 9.Connector Force and Electric Gap

The actual horn spring force can now be interpreted by the reaction force of the rigid finger used to apply enforced displacement at the instance when horn spring connector is stopped i.e. when the electric gap circuit is closed.

The Reference length, connector lock, connector displacement (CU) and connector position (CP) is extensively used in this analysis, there by capturing the actual behavior and to evaluate the horn force. Also by eliminating various contact algorithms and many simplifications.

4. Results – Finger Poke Study

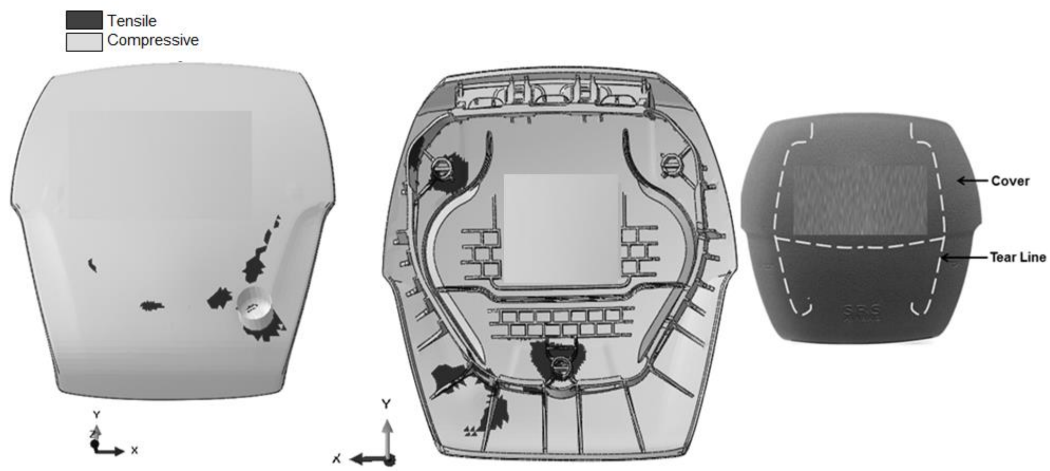

Figure 10.Finger Poke study validation

The finger poke-study is done and is cross validated for the tensile stresses near tear line region.

- No critical tensile stresses are observed near the tear line which would leads to its visibility on class-A surface
- This validates that the additional force applied is taken by the high stiffness extra travel spring.

5. Abaqus Procedures and its advantages in design of Horn Switch Mechanism

As explained earlier the first step would be to pre-stress the horn spring and extra travel spring and in turn to establish contact between the housing and cover in the DAB module.

But to establish this contact between the housing and the cover during the pre-stress is difficult since there is a small gap across all the regions of contact between these two major components.

Refer figure (11) for multiple contact regions, due to the pre-stress of the springs, the cover is lifted by the horn switch units. This leads to contact open and close at all the locations which cause convergence problems.

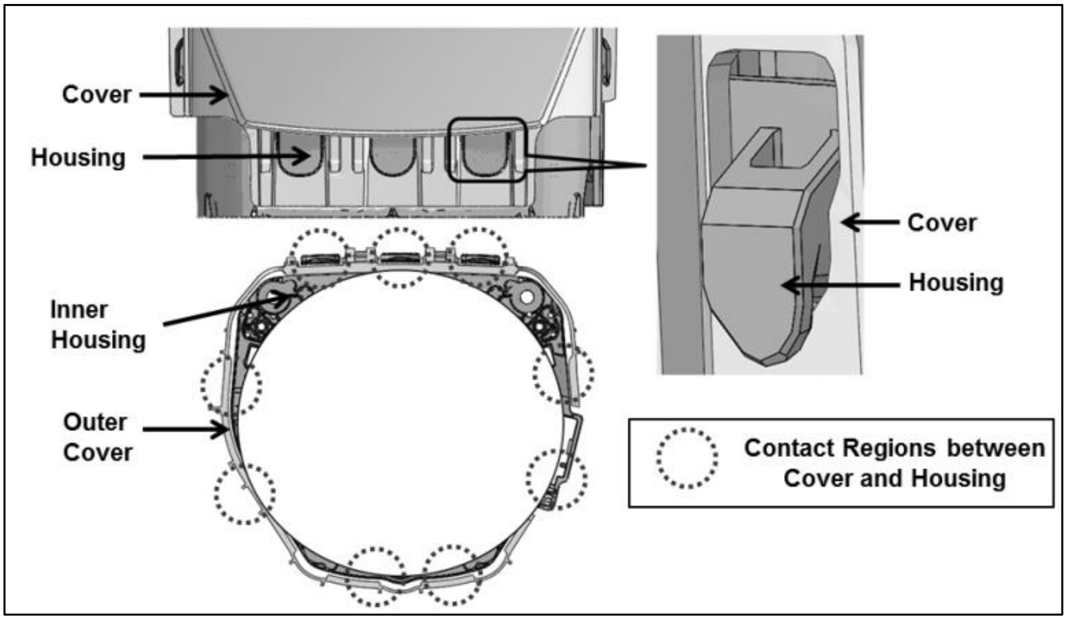

Figure 11.Contact regions between Cover and Housing

Thus using Abaqus/Standard static procedure these statically unstable contact problems will reduce the convergence rate. Hence this requires some additional convergence techniques like to introduce some damping or to use some other techniques to solve this static case. However convergence might still be a problem with fluctuating contact regions and with damping the accuracy of the obtained horn reaction forces might vary.

Thus an approach with Implicit Dynamic with Quasi Static approach used in this statically unstable problem helped the convergence rate. Inertia effects are primarily introduced to regularize this unstable behavior.

This problem was tackled both by static and implicit dynamic approaches and compared.
With Implicit Dynamic approach the solution converged at a faster rate and with less change to default settings.

Table showing the comparison between the implicit static and implicit dynamic cases taking various parameters are shown in the below table (1):

Parameters	STATIC CASE	IMPLICIT DYNAMIC
CPUS	16	16
No. of Nodes	468016	468016
Damping Used	*Static, stabilize, factor=0.0002, allsdtol=0.05	No additional damping
Run Time	7 hrs	5.5 hrs
Converged	95%	100%
Horn Force	96% Matching with Physical test	96% Matching with Physical test

Table 1.Comparison of Static and Implicit Dynamic Runs

- Large time increments are taken when possible to obtain the final solution at minimal computational cost.
- Considerable numerical dissipation may be required to obtain convergence during certain stages of the loading history in quasi static application.
- Obtained results from Quasi static approach is approximately equal to static results with 21% less CPU time.

6. Conclusion

6.1 Thus with the use of connector elements the actual behavior is captured and the horn force obtained is close to the experimental data.

Advantages:

1. Simplifies the FE model
2. Contact algorithms are reduced/removed
3. Actual stiffness and behavior of the springs could be modeled easily

6.2 Using Abaqus Implicit Dynamic Quasi Static procedure over Implicit Static approach the following advantages

1. Accurate static response could be obtained.
2. Convergence rate can be improved by introducing inertia effects
3. Statically unstable problems could be converged

7. Acknowledgements

The author would like to thank Autoliv India Design and Simulation team especially Naveen Chandra and Srivathsa Jagalur for their help and support during this activity.

8. References

1. Abaqus 6-12.1, User's Manual