A Unified FE Modeling Approach to an Automobile Cockpit Module

Tae-Jung Yeo (Hyundai MOBIS) and
Jun Park (ABAQUS, Inc.)

ABSTRACT

Development process of the automobile part modules usually requires various kinds of numerical simulations to reduce the number of prototypes. In the past simulation, engineers had been forced to use several tools for different types of Finite Element (FE) based stress analysis, even though the simulation processes are well defined, since no FE based stress analysis software had not provided various capabilities required for the development process. Due to severe competition in the current market situation, automobile companies and their part suppliers should reduce the time to market and costs for the development to keep them as competitive as they can. Thus, the lesser time engineers are allowed in designs, the more need for effective tools that can handle almost all of their numerical procedures is expected.

In this paper, we are going to show how ABAQUS effectively works well for different types of simulations. The effectiveness comes mainly from using the same FE model with minor adjustments for each analysis item. Several important analyses which are mandatory in the design phase of cockpit module such as head impact analysis, natural frequency extraction, sag analysis, and creep analysis have been carried out as examples.

Introduction

A cockpit module is one of the extremely complex vehicle components. The cockpit module of automobiles usually consists of an instrument panel (IP), a steering column system, a HVAC system, a glove box, a cowl cross structure, various trays, an audio and navigation system, and decorative facials (Figure 1). Development of a cockpit module requires a lot of simulation work to verify its performance prior to prototyping. To satisfy its design goals that are so-called as quality requirements and to meet regulations such as Federal Motor Vehicle Safety Standard No. 201 (FMVSS 201), numerical analysis of passenger air bag (PAB) deployment, head impact and knee impact protection, thermal deformation, NVH, and CFD has been frequently adopted as effective means of design verification. However, the characteristics of each analysis are quite different from one to another so that even major automobile companies or related module suppliers have relied on various kinds of simulation tools even though those analysis are based on the same FE approach.(Kovacic *et al*., 2002).

In this paper, it will be shown how a unified FE modeling approach could handle head impact, natural frequency extraction, sag, and creep by a single code, ABAQUS. The feasibility of applying ABAQUS to the PAB deployment, the knee impact analysis or another kind of structural analysis that might be necessary during the development phase of cockpit can be assured with examples shown in this study unless it needs human dummy models, that is not available with the current version of ABAQUS (6.3-1).

For safety issues, the cockpit modules should be designed to fulfill performance requirements defined by safety regulations, such as FMVSS 201 in North America (NHTSA, 1995). At the initial development phase, predictions of safety-related performance of the cockpit module have a higher priority than other items and are mostly accomplished by numerical simulations before making prototypes (Gholami *et al*., 2002). Due to the newly introduced general contact capability in the version 6.3, ABAQUS can simulate the crashworthiness of structures such as the head impact protection described in FMVSS 201 as well as conventional items that have been dealt with ABAQUS for long time. A simulation of head impact protection was conducted and its results will be compared with those of other commercial codes that use explicit time integration schemes.

Natural frequency extraction analysis of structures has been widely carried out since a spectrum of the natural frequencies and its mode shapes help to understand dynamic characteristics of the modules. Natural frequencies of the cockpit module should avoid having the same natural frequencies with the other parts of automobiles, especially in lower frequency ranges. Distribution of strain energy density gives characteristics of system stiffness, which is closely related with dynamic performance. A natural frequency extraction analysis was done with only a portion of the FE model used in the head impact protection. The frequencies of important modes in the cockpit module designs will be compared with those of MSC/Nastran.

The cockpit module may sag due to the self-weight when it is in shipping to assembly lines. Excessive deflections or distortions can cause trouble in assembling the cockpit module to the other parts of automobiles. To prevent such cases, deformations of the cockpit module should be predicted in advanced. Distributions of displacements of the cockpit module will be presented with fixed boundary conditions applied on locations where jigs are set up.

A large portion of the cockpit module consists of plastic parts. It is well known that plastics are much more vulnerable to temperature variation. The temperature variations of the IP usually range from under the freezing temperature to over the glass transition temperature of the plastic material. Thus the varying temperature may bring in structural damage as well as defects in the appearances of the IP. A creep analysis that simulates a creep test in a laboratory is carried out using the same FE model already used in the head impact and sag analysis.

A Unified FE Modeling Approach

For a unified FE modeling approach, it is very important to use the same mesh in as many analyses as possible. Thus it should be considered at the initial phase of FE modeling that a representative FE model should be compatible to various structural analysis items. The unified approach that will be shown in this paper may not be effective at the very first stage of simulations. A number of iterations will be needed to decide the appropriate mesh size and so on. However, it is definitely efficient if the simulation procedures for validations of designs are well established and fall into a category of routine jobs.

Usually impact analysis needs a finer mesh than the others to catch behaviors of structures that end less than about a hundred milliseconds in the cockpit module design. Codes that use explicit time integration schemes are well known as tools to solve this type of analysis. The smaller the mesh size is, the more accurate behaviors could be obtained at the expanse of cost. Thus the structure should be meshed small enough to describe the motion of the structure during impact, but not smaller than a criteria which has been decided on the consideration of analysis cost. Contrary to impact analysis, generally speaking, not all analyses need such a fine mesh in the cockpit module design. Applying a finer mesh to other analyses will take more time to get results. However, it could be compensated by reducing time wasted in making a new mesh for different analyses. As computing power is increasing rapidly, the gains by using the same mesh will overwhelm the losses by handling bigger problems.

Regardless of the characteristics of each analysis, it is preferable to apply same unit system all over the analysis. The dimensions for length and mass are millimeter and kilogram, for all cases, respectively. However, the time scale is categorized into two cases just for the purpose of easy recognition of analysis results. An impact is an event that takes place within a very short time interval, thus a mm-Kg-msec unit system is appropriate for its analysis. For natural frequency extraction, sag, and creep analysis, a unit system of a mm-Kg-sec is applied since they occur at relatively long-term interval than impacts. Both unit systems are not self-consistent, thus great care should be taken in data conversions. However, shifting from one unit system to another is quite simple provided that two sets of material definitions based on each unit system are prepared and saved using different names. The material definition can be shifted from one unit system to another by changing a value of the parameter FILE of the keyword *INCLUDE according to the characteristics of each analysis.

A series of jobs – head impact, natural frequency extraction, sag, and creep – will be demonstrated in turn. It is obvious that element types and definition of procedure should be modified along with the analysis type in appropriate manner. Not all parts of the cockpit module are necessary for every analysis. It depends on the characteristics of each problem and experiences in simulations. For example, the steering column and wheel are excluded except for the natural frequency extraction analysis, but almost all of plastic parts are not included. Modifications of the control cards for the next analysis from the previous can be accomplished with ease. Even a basic editor available on MS-Windows or Unix supports “Find and Replace” and “Cut and Paste”.

One of the key benefits in using only one code for various analyses is that the same syntax can be applied on them. Under a single analysis code environment, it is possible within few keystrokes with an editor whatever it may be, to modify an input deck for the next analysis if each part in the cockpit module is grouped.

Head Impact Analysis (FMVSS201)

According to FMVSS 201, the standard's test procedure requires that a headform with 165mm in diameter and 6.8 Kg in mass impacts the interior at an initial velocity of 19.2 Km/h with an airbag and 24.1 Km/h without it, respectively. The standard regulates that the acceleration at the center of the headform should not exceed 80g during any 3 milliseconds interval (Yeo, 2002).

A mesh applied to the unified approach is composed according to the impact analysis standard. Figure 2 shows the FE model used in this approach. The steering column and wheel are eliminated in the assembly because they don't play a great role in the head impact analysis. The headform is considered as a rigid body.

ABAQUS/Explicit is used to simulate the behavior of the cockpit module on impact load conditions. Shell elements used in this analysis are S3RS and S4RS that are well suited to shell problems with small membrane strains and arbitrary large rotations. Many impact dynamic analyses may fall within this class, including those of shell structures undergoing large-scale buckling behavior but relatively small amounts of membrane stretching and compression (ABAQUS, Inc., 2002). The Kg-mm-msec unit system was adopted since impacts last about a half hundred milliseconds. The job ran in a single precision mode since no apparent gains in accuracy were observed when run in a double precision.

Figure 3 shows example of impact points on the surface of cockpit module. These points should be located within an impact zone defined by the regulations. Basically, any location within the impact zone can be impacted. In this study, a point near upper center part of glove box was considered. Figure 4 shows the deformation of the cockpit module just before the rigid headform bounced back.

To figure out whether a job analyzed by an explicit time integration scheme is done well or not, all energies of the structure should be monitored carefully (Figure 5), even though it is not sufficient. The ALLKE, ALLIE, ALLPD, ALLSE, ALLFD, and ETOTAL mean kinetic energy, total strain energy, energy dissipated by plastic deformations, recoverable strain energy, total energy dissipated by friction, and total energy of the system, respectively. The ALLKE decreased quadratically and bounced back in the same manner. No jump is found in any energy curves. The ETOTAL should be maintained as a constant theoretically for any event. As shown in the figure, the variation in ETOTAL is less than 0.1% during the whole impact event. It means that the results are quite reasonable.

A comparison of the magnitude of the acceleration with Pam-Crash and LS-DYNA 3D at the center of the headform is shown in Figure 6. The curve showing the ABAQUS result may look noisier than other two curves since there are more than 10 times of sampling points. Figure 6 indicates that Pam-crash reached a peak earlier than the other two. It means that the stiffness of the FE model, actually it does not make any stiffness matrix, generated by Pam-crash was softer than those of the other two. ABAQUS' result shows very close agreement with LS-DYNA 3D's result.

Natural Frequency Extraction Analysis

The FE model for this analysis is shown in Figure 7. At the initial development phase, dynamic stiffness of steering column structure that contains a steering column, a steering wheel, cowl cross members, and brackets for left/right sides and mountings should be predicted to avoid resonance with engine idling frequency. Generally, a vertical and a lateral mode of the steering column and steering wheel (Figure 8) are first or second mode of concern.

Since the FE model was constructed under the standards of impact analysis, several adjustments are necessary for this analysis. Both S3RS and S4RS elements used in the previous analysis are not suitable for natural frequency extraction analysis so that they were substituted for S3R and S4R, respectively. The unit system was changed to kg-mm-sec to get insight from the results with ease. It is also obvious to change the procedure definition and related output variables in an appropriate manner.

Table1 lists the two important natural frequencies of the simplified structure compared with those of MSC/NASTRAN. As it may be expected, no remarkable discrepancy is shown when both results are compared.

Sag Analysis

The sag due to gravity of the cockpit module was analyzed to simulate shipping, loading, and assembly conditions. Disagreement in location of even a single bolthole of the cockpit module may cause trouble in the assembling stage. Thus dimensional variations of each bolthole center should be checked with care.

The same mesh used in the head impact protection analysis was used again except the rigid headform. For a sag analysis the cockpit module was fixed at the points where jigs are set up for shipping and handling. There is no chance of plastic deformation since it only bends due to self-weight. The maximum displacement of 3.55 mm occurred at the front HVAC duct (Figure 9). The lower part of the cockpit module showed a relatively small displacement since steel made cowl cross members support the whole module.

Creep Analysis

The interior parts of the vehicle cabin are exposed to temperature variations from under freezing temperature of water in winter to over 100°C due to radiation effect of sun ray in summer. Plastic parts appear to be more thermally susceptible than steel parts in temperature variations since its thermal expansion coefficient is at least 5 times greater than steel. The incompatibilities of thermal expansion between materials cause permanent deformation of plastic parts in conjunction with constraint conditions. Therefore it is one of the key issues to minimize thermal deformation while developing the IP.

In addition to the material data for the sag analysis, viscoplastic data for each plastic part is necessary to conduct a long-term basis and temperature dependent structural problem – creep. Black dots in Figure 10 indicate the points where fully fixed mechanical boundary conditions were applied. It is assumed that the highest temperature at the surface where it was exposed on direct sun ray could reach up to 120°C (Figure 11). No creep data is necessary for the steel parts such as the cowl members since steel may creep at a much more higher temperature than the plastic materials do.

Among the creep laws that ABAQUS provides, the power-law was applied since it is easier to get material data from experiments (Yeo *et al*., 2003). In addition to the self-weight as in the sag analysis, three consecutive cycles of various temperatures with different time durations applied on the cockpit module (Figure 12).

Figure 13 shows the distribution of the residual displacement after the thermal cycle loading. The figure indicates that left and right front of the HVAC, the center facia panel, and around the cup holder deformed more than the other part since they were away from connecting locations described as fixed boundary conditions or supporting by the steel structure.

Conclusion

This paper describes a series of the essential numerical simulations in cockpit module designs with a unified FE modeling approach. The effectiveness of this approach mostly comes from recycling of the same mesh used in an analysis in other analyses with minor adjustments. Although applying a finer mesh in other analyses may take more time to get results, the gains by using the same mesh will overwhelm the losses by handling bigger problems as the computing power is increasing rapidly. As shown in examples, ABAQUS is a suitable tool available in the current market place to be able to handle almost all of structural problems in a single code environment.

References

ABAQUS, Inc., ABAQUS/Explicit Users Manual (version 6.3-1), 2002.

Gholami, T., Lescheticky, J., and Paßmann, R., Theobald, A., Jansohn, W., Niedermeyer, S., EDAG, Schrank, M., and Engelmann, B., "Finite Element Analysis of Head Impact According to Extended FMVSS 201," ABAQUS Users' Conference, Newport, Rhode Island, May 2002, pp. 1 - 11.

Kovacic, T., S. Lanka, and M. Marks, “Development of an Integrated Structural HVAC Instrument Panel Cockpit System,” SAE 2002-01-0309, SAE 2002 World Congress, Detroit, Michigan, March, 2002.

NHTSA National Highway Traffic Safety Administration, Department of Transportation, Federal Motor Vehicle Safety Standard No. 201, 1995

Yeo, T. J., “PU Head Impact Analysis ,” Internal MOBIS Report, Yongin, Korea, 2002.

Yeo, T. J., S. S. Kim, S. J. Park, C. S. Kim, Y. S. Park and S. H. Lee, “A Study on Creep Analysis of Automobile Instrument Panel,” SAE 2003-01-1173, SAE 2003 World Congress, Detroit, Michigan, March, 2003.

Tables

Table 1. Comparison of two natural frequencies

Mode (Hz)	ABAQUS/Standard	MSC/NASTRAN
Vertical mode	29.7	29.4
Lateral mode	37.9	37.0

Figures

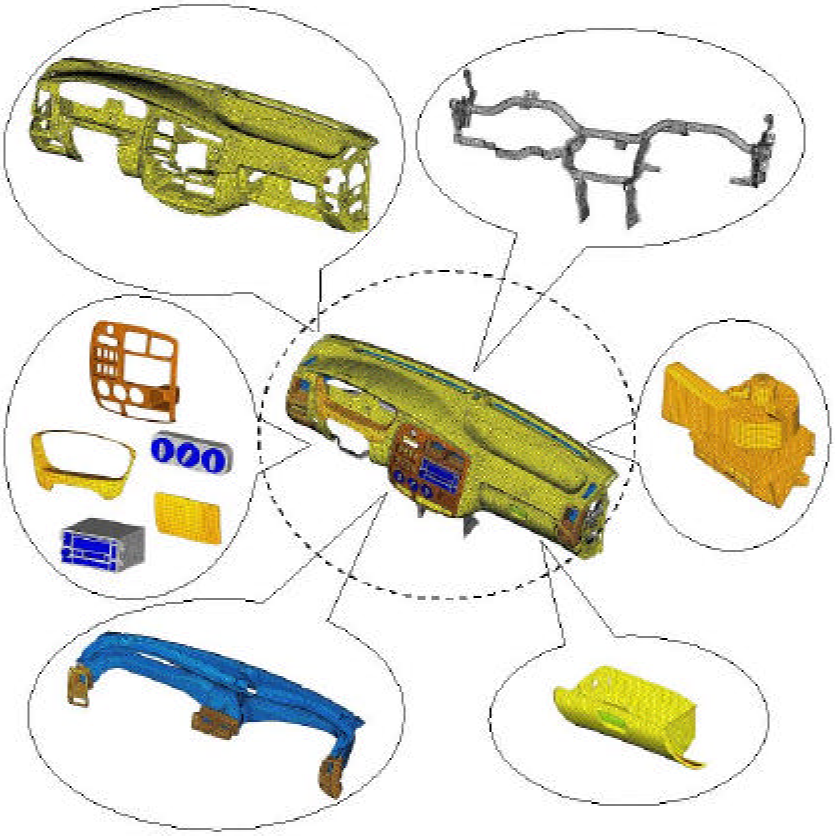


Figure 1. A typical cockpit module

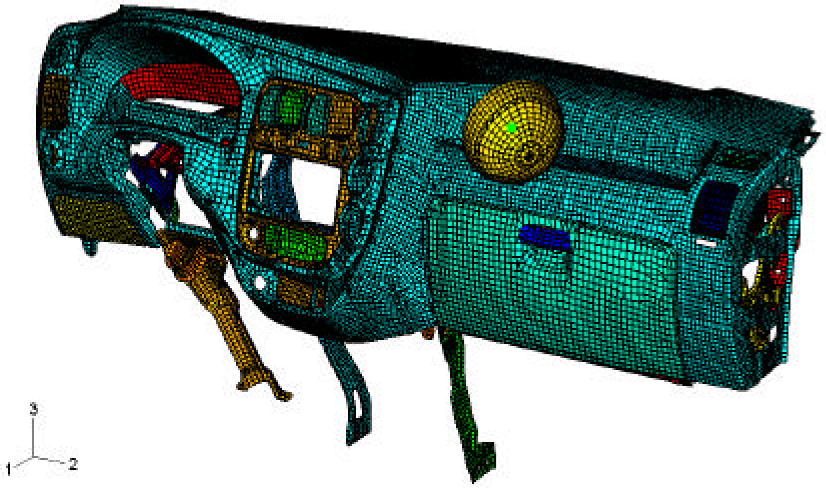


Figure 2. A mesh for the unified FE modeling approach

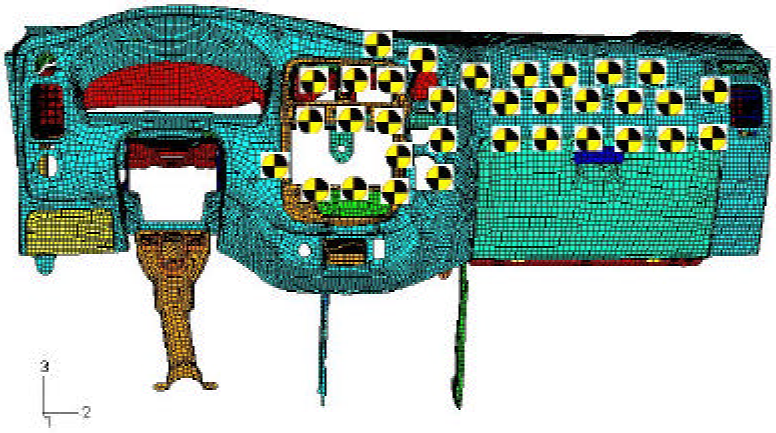


Figure 3. Locations where the headform impact should be checked

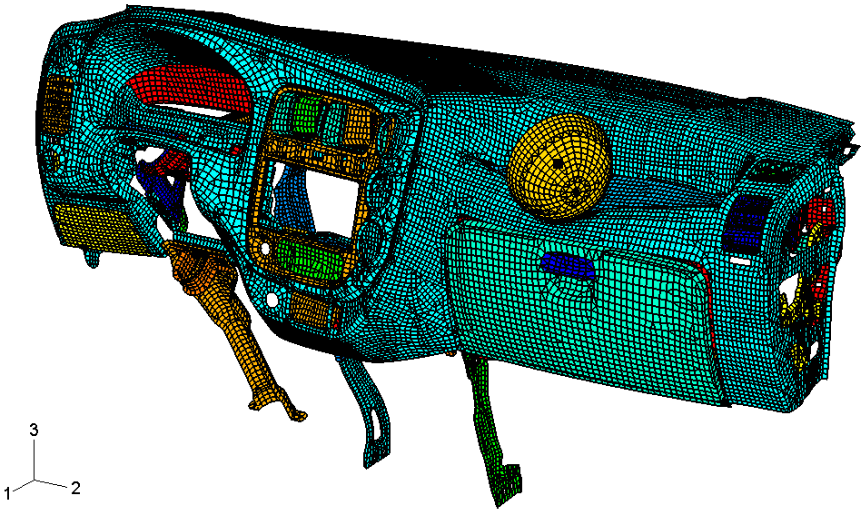


Figure 4. Deformation just before rebounding

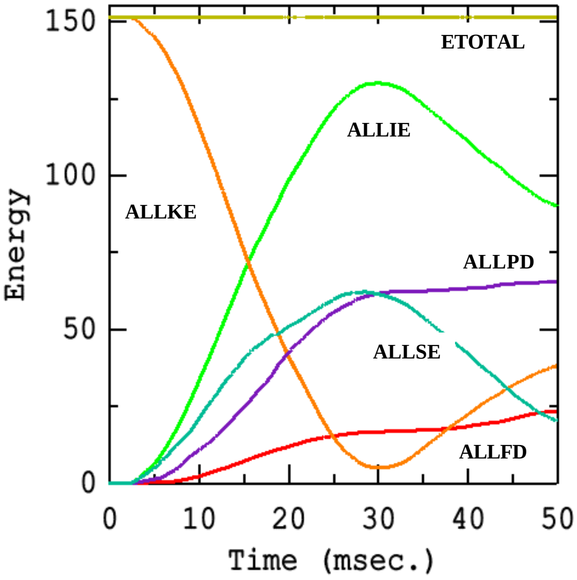


Figure 5. Variation of each energy

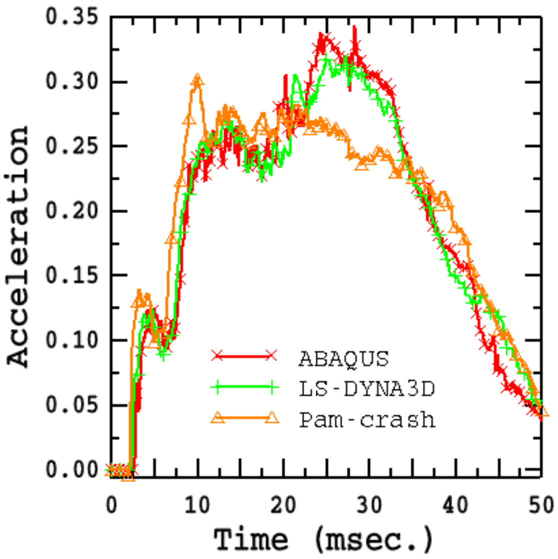


Figure 6. Acceleration curves at the center of the headform

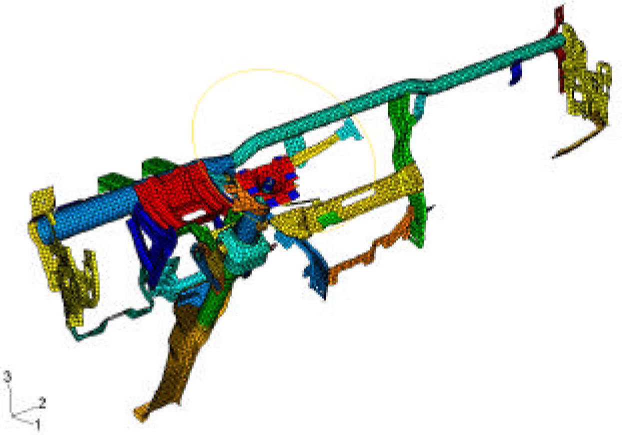
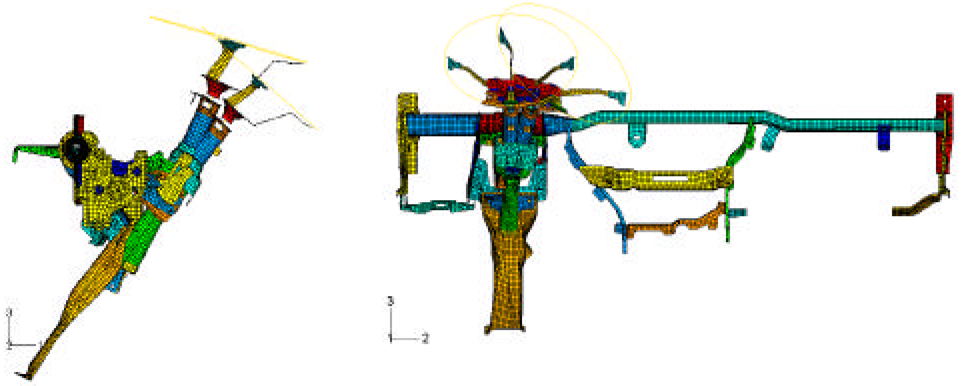


Figure 7. FE model for the natural frequency extraction analysis



(a) the vertical mode (b) the lateral mode

Figure 8. Tow important modes of the steering column system a) the vertical mode, b) the lateral mode

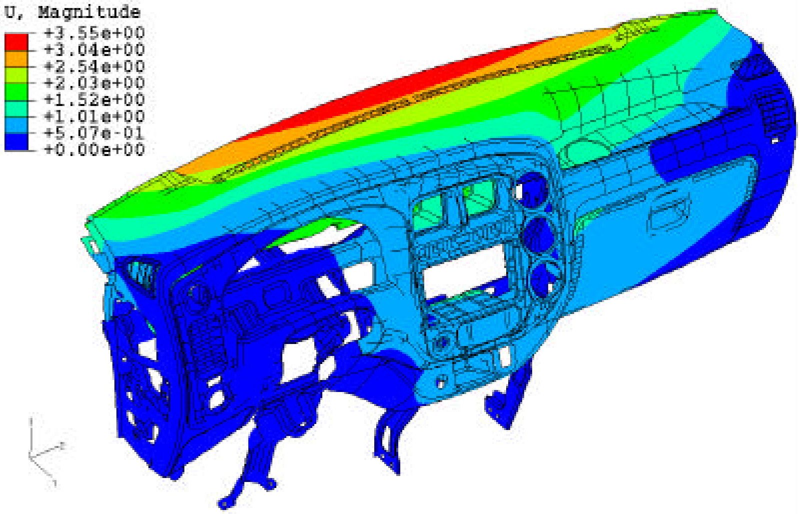


Figure 9. Contours of deformation by sag

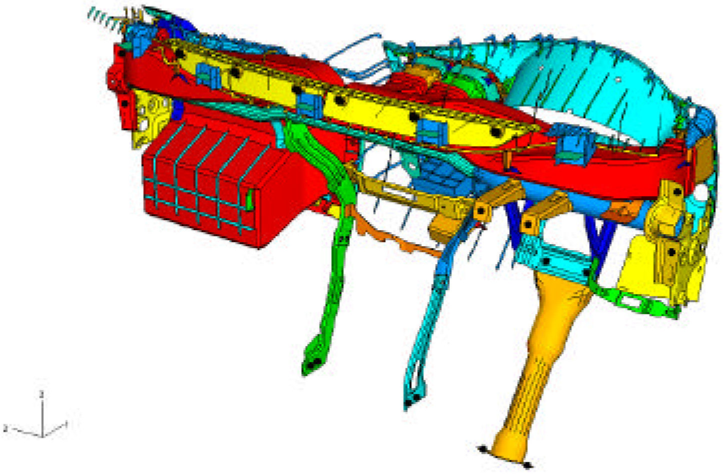


Figure 10. Fixed boundary conditions for the creep analysis (black dots)

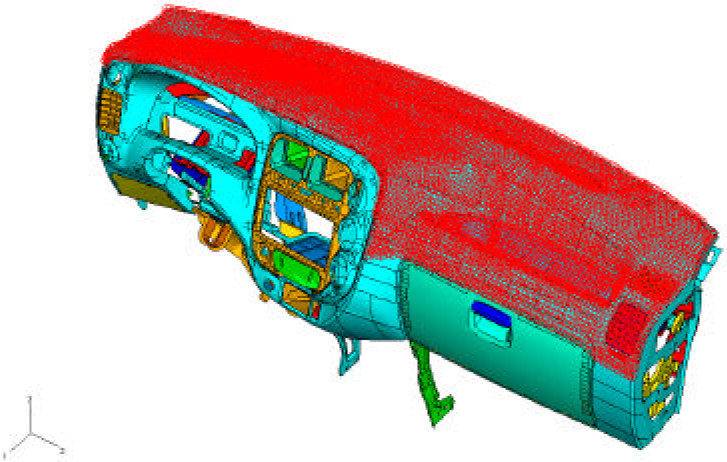


Figure 11. Surfaces on where the maximum temperature applied

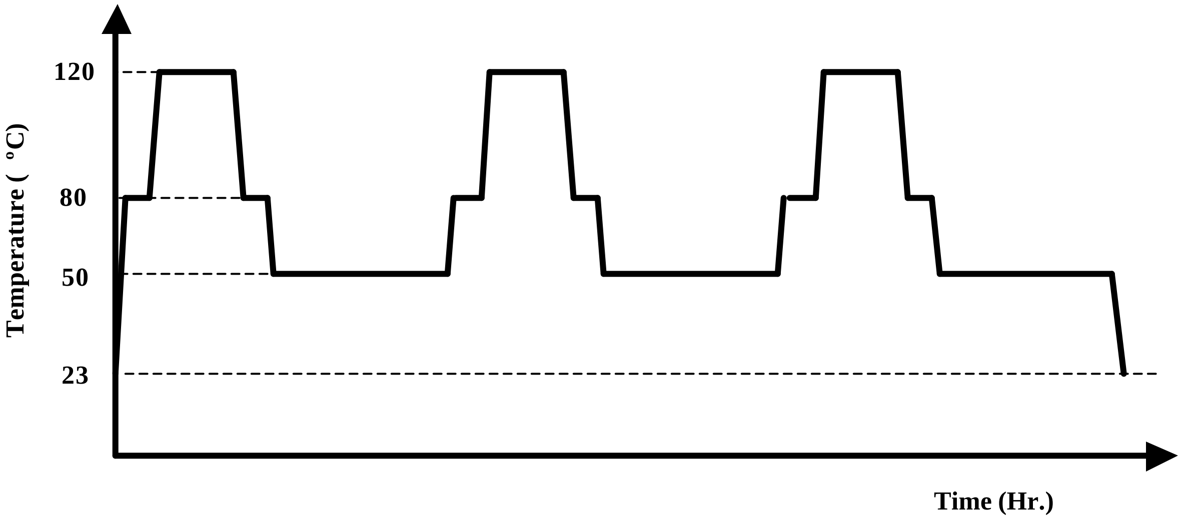


Figure 12. Thermal loading history

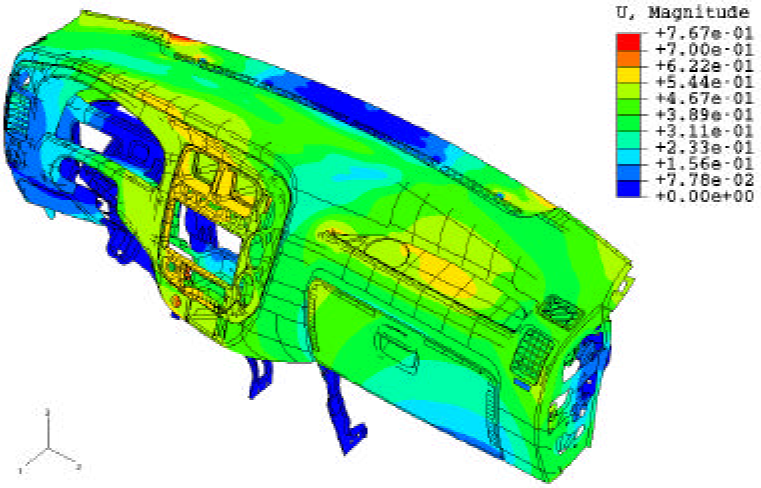


Figure 13. Residual deformation after the cyclic thermal loading