

Snow Load Simulation Correlation of the Light Commercial Vehicles

Halil Bilal

Tofaş Türk Otomobil Fabrikası A.Ş.

Abstract : Today, snow load damage test is mainly used to decide on roof panel's style and its thickness. This test is much more important for a light commercial vehicle (LCV) because LCV has bigger roof area, more straight style and as a result of these two properties it has heavier roof panel than the other vehicle.

Moreover, simulation is quite difficult and its tolerance is not less than 15 % because of not only panel's post buckling behavior and material nonlinearity but also deformations on panel which highly depend on manufacturing process.

In this study, the analysis parameters have been investigated and a correlation with physical test results has been set. The main analysis parameters are as follows:

- *Solid element integration and material properties of adhesives between the crossbeam and the roof,*
- *Number of element on the thickness for the solid elements of adhesives,*
- *Solver type (Eigenvalue buckling, RIKS and Explicit),*
- *Loading velocity for the explicit dynamics solutions,*
- *Shell thickness and perturbation of the roof,*

In the end, right analysis parameters and simulation methodology are expected to be defined in order to make the simulation %5 close to physical test results.

Keywords: Snow load, buckling, riks, explicit, perturbation

1. Introduction

Snow load damage test is mainly used to decide on roof panel's style and its thickness. Generally, it is expected to avoid permanent deformation of the roof under the severe winter and snow condition (Figure 1). Therefore, snow load analysis performance is one of the most important parameter of the roof panel and the roof traverse design.



Figure 1. The cars under heavy snow conditions

In this study, first of all, the main important parameters of the snow load analysis have been decided. Then the effects of every parameter on the results have been investigated. Also the results of the virtual analyses have been compared with physical test results. At the end, the right values of every parameter for having correlated results with physical tests have been set. Investigated parameters are given below;

- Solid element integration and material properties of adhesives between the crossbeam and the roof,
- Number of element on the thickness for the solid elements of adhesives,
- Solver types (Eigenvalue buckling, RIKS and Explicit),
- Loading velocity for the explicit dynamics solutions,
- Shell thickness and perturbation of the roof,

Solution types and thickness distribution effects have been focused in this paper.

2. Finite Element Model

Figure 2 shows finite element model of the roof panel that used for the analysis. All sheet parts have been modeled with shell elements, all other adhesives have been modeled as solid elements. While spotwelds have been modeled with fastener element type.

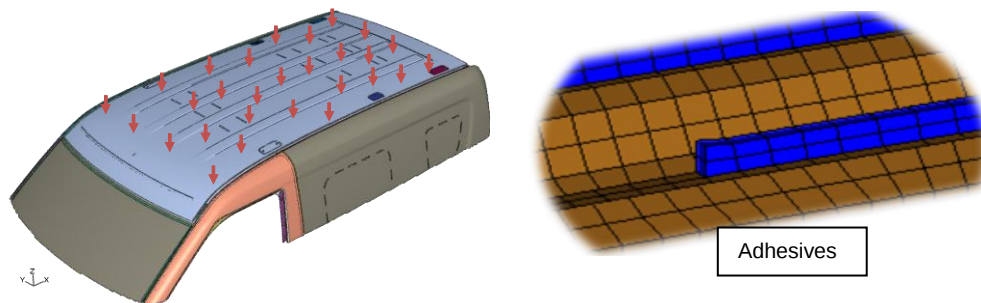


Figure 2. Finite element model

3. Solution Type Effects

Various solution types are available to simulate snow load analysis. They are;

- Eigenvalue Buckling
- RIKS
- Explicit Dynamic Analysis

3.1 Eigenvalue Buckling

Eigenvalue buckling analysis:

- is generally used to estimate the critical (bifurcation) load of “stiff” structures;
- is a linear perturbation procedure;
- can be the first step in an analysis of an unloaded structure, or it can be performed after the structure has been preloaded—if the structure has been preloaded, the buckling load from the preloaded state is calculated;
- can be used in the investigation of the imperfection sensitivity of a structure; and
- Cannot be used in a model containing substructures.

Eigenvalue buckling is generally used to estimate the critical buckling loads of stiff structures (classical eigenvalue buckling). Stiff structures carry their design loads primarily by axial or membrane action, rather than by bending action. Their response usually involves very little deformation prior to buckling. A simple example of a stiff structure is the Euler column, which responds very stiffly to a compressive axial load until a critical load is reached, when it bends suddenly and exhibits a much lower stiffness. However, even when the response of a structure is nonlinear before collapse, a general eigenvalue buckling analysis can provide useful estimates of collapse mode shapes.

Figure 3 and Figure 4 show the first 4 eigenvalue buckling loads and mode shapes of the roof panel. As seen every 2 modes are symmetrical due to the symmetrical design.

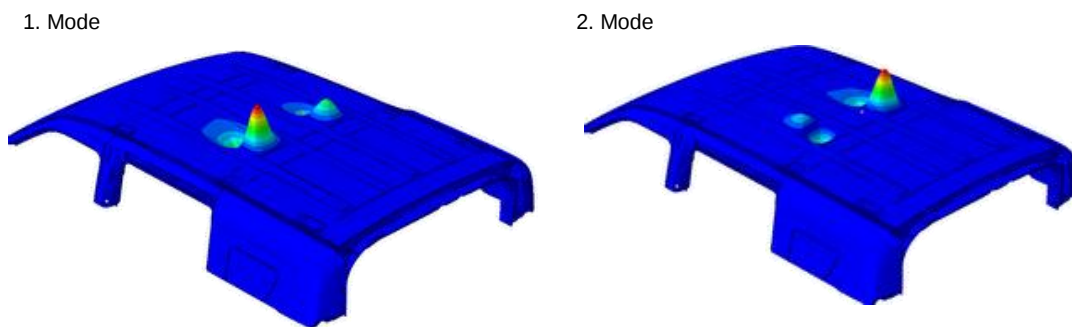
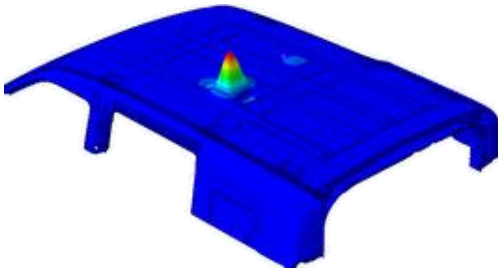


Figure 3. 1st and 2nd Eigenvalue buckling modes of the roof panel

3. Mode



4. Mode

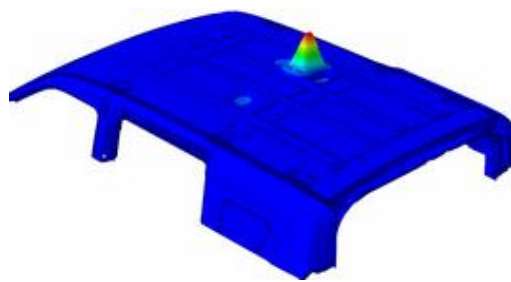


Figure 4. 3rd and 4th Eigenvalue buckling modes of the roof panel

3.2 Static Analysis with RIKS Method

Geometrically nonlinear static problems sometimes involve buckling or collapse behavior, where the load-displacement response shows a negative stiffness and the structure must release strain energy to remain in equilibrium. Several approaches are possible for modeling such behavior. One is to treat the buckling response dynamically. Alternatively, static equilibrium states during the unstable phase of the response can be found by using the “modified Riks method.” This method is used for cases where the loading is proportional; that is, where the load magnitudes are governed by a single scalar parameter. The method can provide solutions even in cases of complex, unstable response such as that shown in Figure 5.

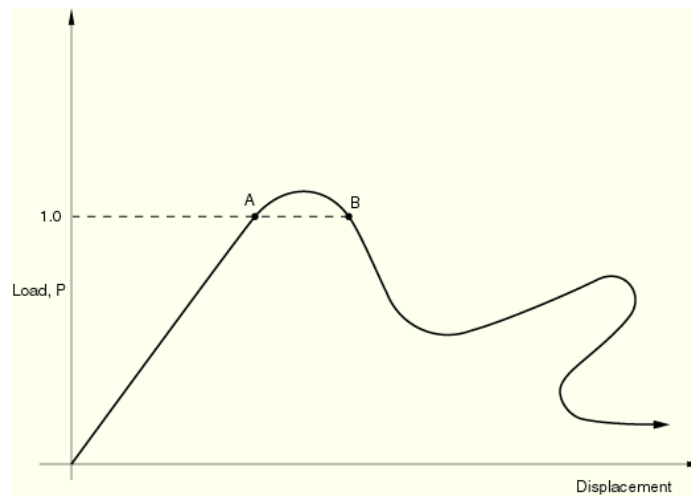


Figure 5. Proportional loading with unstable response

The Riks method uses the load magnitude as an additional unknown; it simultaneously solves loads and displacements. Therefore, another quantity must be used to measure the progress of the solution; Abaqus/Standard uses the “arc length,” l , along the static equilibrium path in load-displacement space. This approach provides solutions regardless of whether the response is stable or unstable.

Load proportionality factor can be controlled from the status file (.sta). Negative value of the load proportionality factor means, the load-displacement response shows a negative stiffness and load started to decrease. Figure 6 shows stress distribution of the roof panel. Stress concentration is not seen on the increment 55 (Figure 6a). While increment 135 (Figure 6b) shows stress concentration on the beat end. Load proportionality factor has been started to decrease at Increment 55. Load proportionality factor has been reduced until increment 135. The value of it has been reduced from 1.86 to 1.73. It means that although load scale factor is decreased, stress concentration has been occurred on the roof. Figure 7 shows plastic strain distribution on the roof at increment 135.

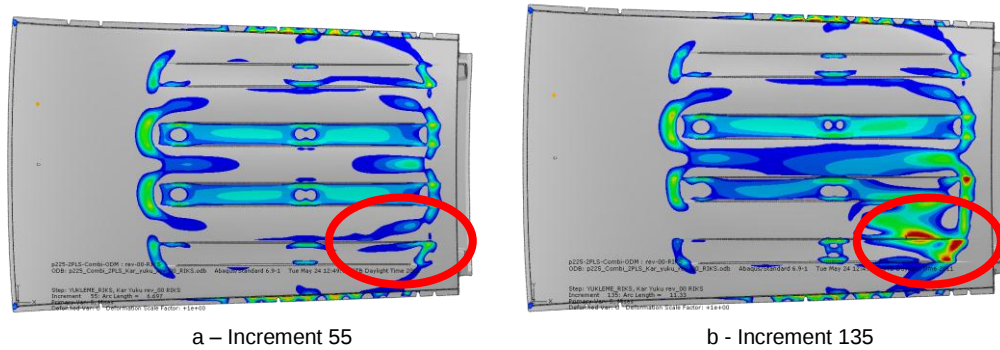


Figure 6. Stress distribution on the increments 55 (a) and 135 (b)

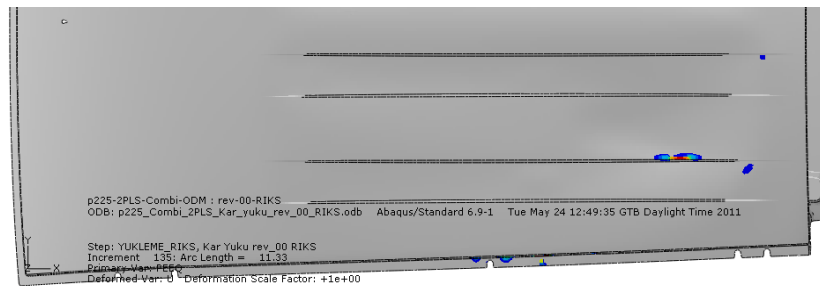


Figure 7. Plastic strain distribution at increment 135

3.3 Explicit Dynamic Analysis

The explicit dynamics procedure performs a large number of small time increments efficiently. An explicit central-difference time integration rule is used; each increment is relatively inexpensive (compared to the direct-integration dynamic analysis procedure available in Abaqus/Standard) because there is no solution for a set of simultaneous equations. The explicit central-difference operator satisfies the dynamic equilibrium equations at the beginning of the increment, t ; the accelerations calculated at time t are used to advance the velocity solution to time $t + \Delta t / 2$ and the displacement solution to time $t + \Delta t$.

Snow load has been defined incrementally as a function of the time for explicit dynamic analysis. To do that Smooth step function has been used as shown on Figure 8. Thus, the load on the roof is intended to ramp up smoothly from one amplitude value to another.

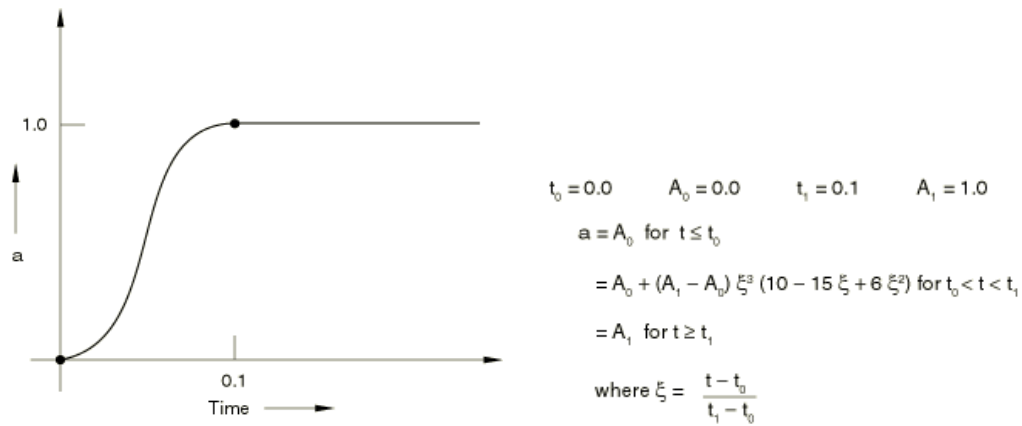


Figure 8. Smooth step function

Real physical problem is static. But is needed to simulate dynamically to solve involve buckling time. It means the load should increase by the time. Figure 9 shows two different loading rates for the same explicit dynamic analysis.

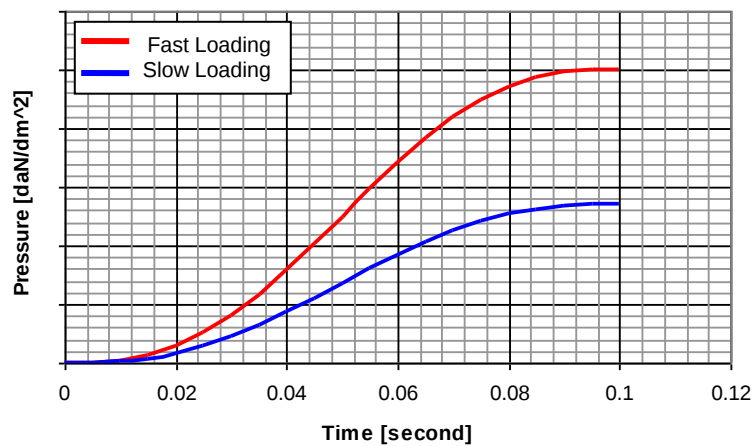
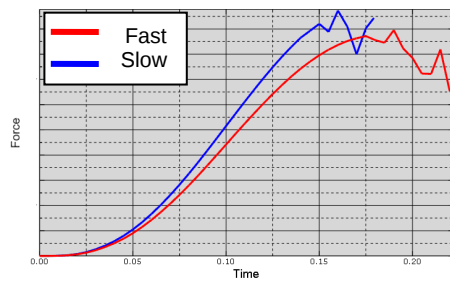
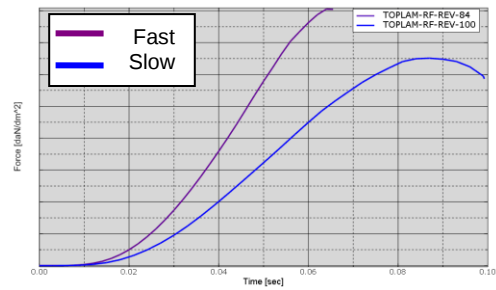


Figure 9. Different loading rate

Figure 10 shows variation of the buckling force by the loading rate for 2 different models. Accordingly, the effect of loading rate on the buckling force varies from depending on the model. Figure 11 shows also buckling modes are effected by loading rate. While buckling is starting from central beat by slow loading rate, it starts from the outer beats by fast loading rate.



a – Fiorino



b – New Doblo

Figure 10. Loading velocity effect on the buckling load

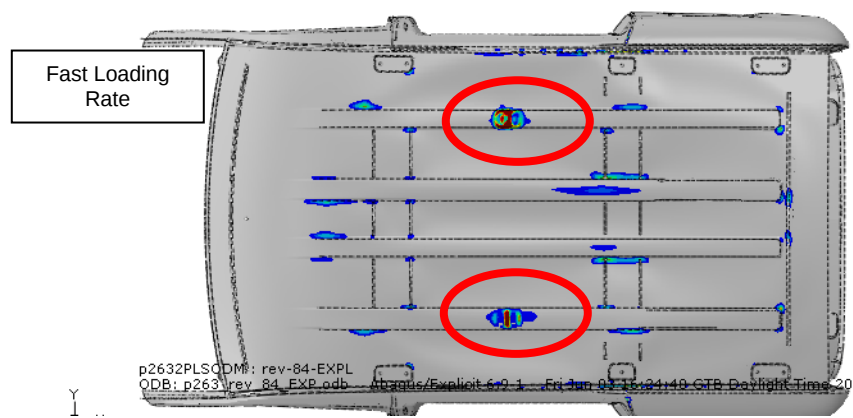
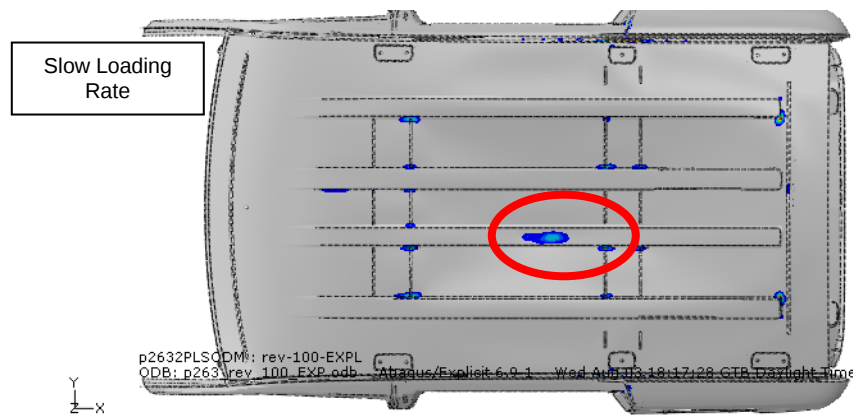


Figure 11 Plastic strains for two loading rates

Whole model kinetic energy variation also effected by loading rate. Figure 12 and Figure 13 show effect of the loading rate. Huge increase on the kinetic energy after buckling involve by the slow loading rate, that is clearly seen on Figure 12. Even then, it is not seen on Figure 13 by fast loading rate.

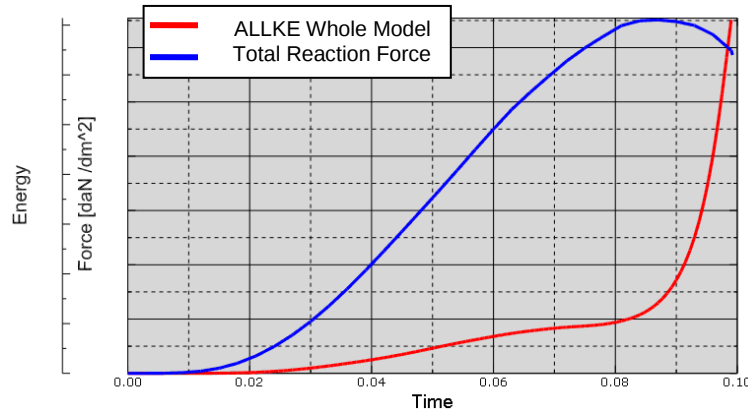


Figure 12. Kinetic Energy and Reaction Force distribution for SLOW loading rate

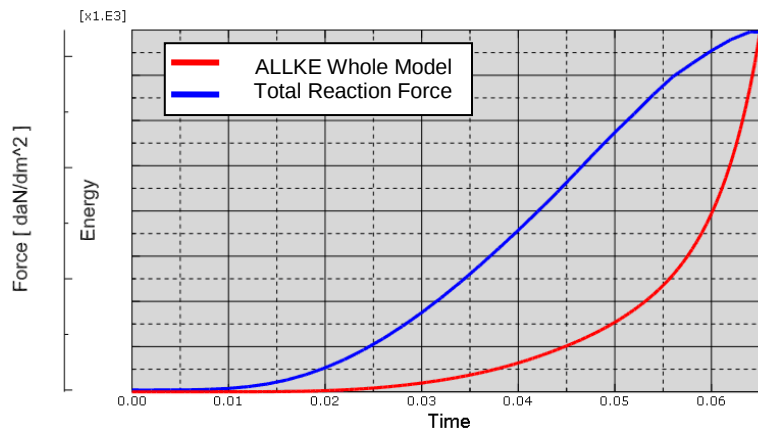


Figure 13. Kinetic Energy and Reaction Force distribution for FAST loading rate

4. Thickness Distribution Effect

Sheet parts that are used on the car haven't got constant thickness as decided during the project phase. Thicknesses change according to the forming conditions and production tolerances. Figure 14 shows thickness distribution of the roof panel after forming process. It is possible to define various thickness distribution of the shell section in Abaqus. It requires nodal thickness information of the nodes that are used in Abaqus FE model.

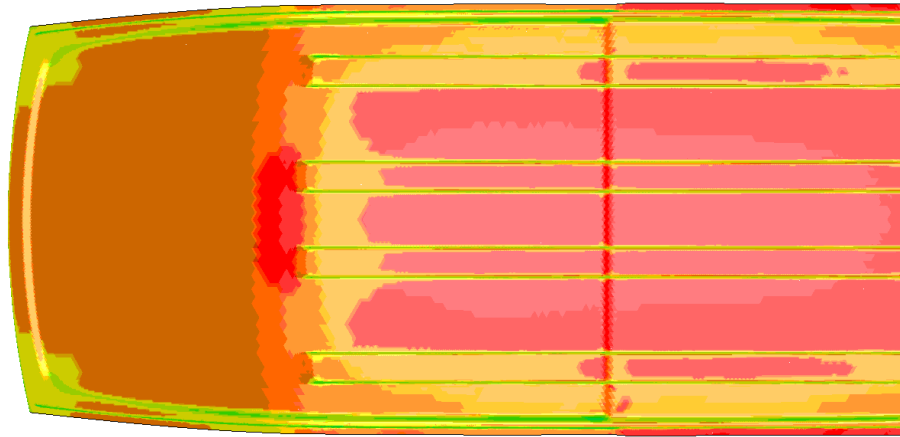


Figure 14. Thickness distribution on the roof panel

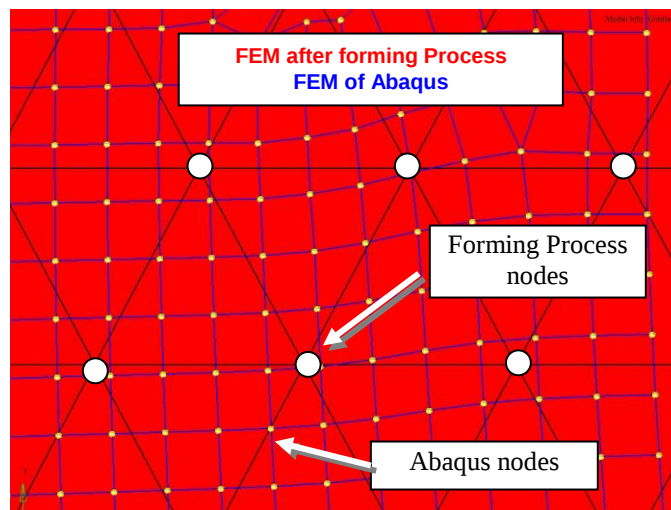


Figure 15 Comparison between forming process FEM and Abaqus FEM

But, generally FEM of the forming process and FEM of the Abaqus don't match to each other. Figure 15 shows this inconsistency between two different FE models. Thickness information on the nodes used forming process should map to the Abaqus nodes.

Due to impossibility to perform this activity with the earlier version of Abaqus/CAE, an in-house code has been written to perform this. Figure 16 shows the variation on the buckling loads by the thickness distribution. Buckling force had been decreased up to 8% by various thickness distributions. Middle column in Figure 16 shows the result of 5% decreased nominal thickness.

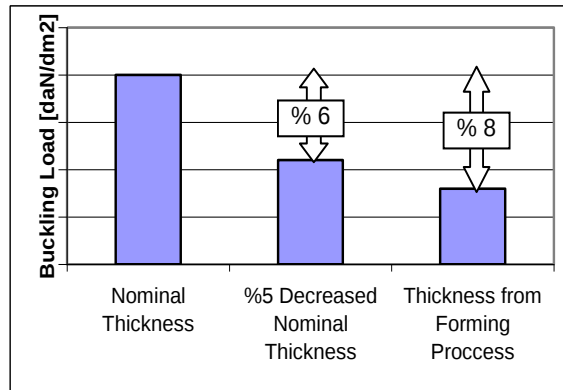


Figure 16. Thickness distribution effect on the buckling force

5. Conclusions

Every solution types have some advantages and disadvantages to each other. For example, the buckling involve can be seen obviously by explicit analysis. Explicit analysis solution time was 6 hours on 16 cpus, while the solution time of RIKS analysis is 1.5 hours on 8 cpus. It means the CPU time of explicit analysis is almost 8 times more than implicit analysis. Eigenvalue buckling analysis is quite fast, but the accuracy of the analysis is not enough. Figure 18 for comparison.

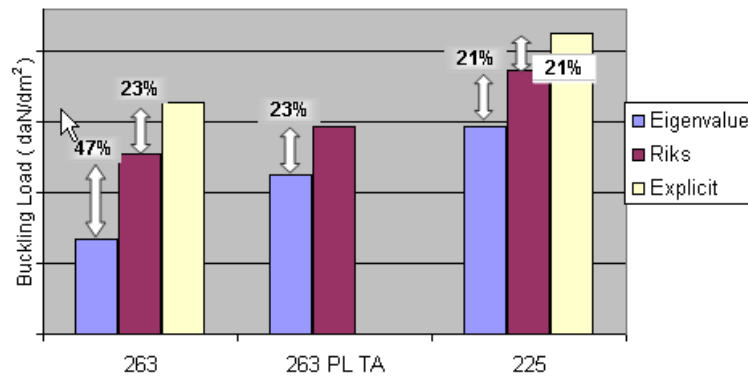


Figure 17 shows variation of the buckling forces by analysis types and FE model of different vehicles. Physical test results have been added to Figure 18 for comparison.

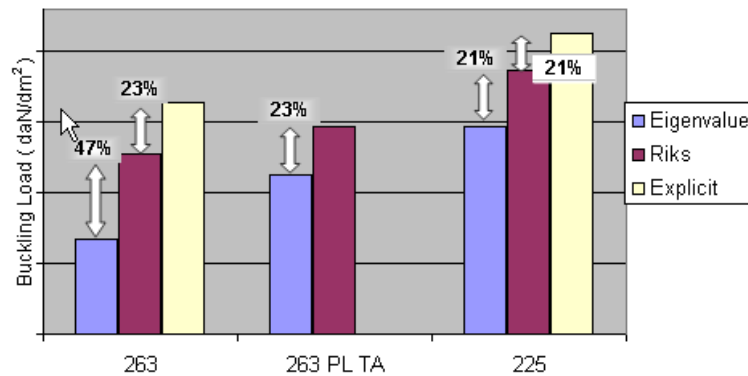


Figure 17. Different analysis types comparison for different models

Eigenvalue buckling results are quite conservative from the other results. When the RIKS and explicit solution types are compared, it is seen that the results are quite close to each other depending on the loading rate of the explicit analysis.

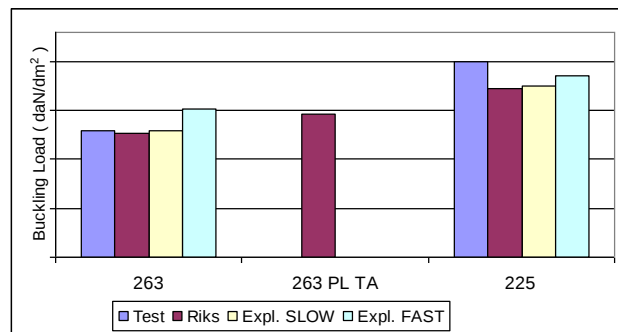


Figure 18. Comparison between physical test and FE analyses

However, it has been seen that solid element formulations and the material properties of the adhesives are very effective on the results. Especially RIKS solution method is very sensitive to the element formulation and to the number of elements on the thickness of the adhesives. Some numerical problems like hourglassing of the solid elements causes divergence of implicit solution. In the end of this study, analysis parameters have been set to get closed correlation with physical tests less than 5%.

6. References

1. Abaqus Version 6.9 Documentation
2. Related Fiat Specifications.