Finite Element Model of the Bus

Florida Agricultural and Mechanical University-Florida State University (FAMU-FSU) is performing numerical simulations of paratransit bus rollover accident on TRACC's cluster computer in conjunction with experimental rollover tests at FAMU/FSU. FAMU/FSU is developing finite element (FE) models for paratransit buses. The original FE model [1] of the bus was developed in the two separate stages First, a model of a cutaway chassis was extracted from a public domain Ford Econoline Van FE model, developed by the National Crash Analysis Center (NCAC) at George Washington University [2] The FE model was modified to match specifications of the chassis used for the given paratransit bus – from the van E-150 to the heavy duty E-450. Major changes have been made to the main chassis beams and suspension elements. During the second stage of FE model development, the 3D geometry and FE models of separate bus body walls (side walls, back wall, roof and floor) were created from CAD drawings supplied by the bus manufacturer. Finally the bus body cage was assembled with the chassis using LS-PrePost [3].

Florida Department of Transportation is acquiring decommissioned paratransit buses for experimental testing and FE models validation. The recently acquired buses were similar to the FE model described in the paper. The paratransit buses are custom made and for each case the structure may be slightly different although the buses have the same make and model. Modifications had to be done to the FE model in order to be able to compare experimental and computational results. Also the mesh density was nearly doubled in the new model. The original bus model, containing ~538,000 finite elements, has been expanded to almost 925,000 finite elements. The major structural components had minimum four shell elements across their width. Fully integrated shells (ELFOR 16) were used in the whole model. The statistics of the final FE model are shown in Table 6.1. The full scale paratransit bus and its model are shown in Figure 6.2.

Table 6.1: Statistics of developed paratransit bus FE model

	Chassis model	Bus body	Whole model
# of elements	189,079	735,407	924,486
# of nodes	204,998	658,028	773,026
# of parts	295	64	359
# of 1-D elements	2	0	2
# of 2-D elements	173,401	735,407	908,808
# of 3-D elements	15,676	0	15,676


Figure 1.2: The bus selected for a rollover test (left) and its FE model (right)

A full rollover test was performed at Florida Department of Transportation testing facility in 2010. The results were used to validate the FE model of the bus and were partially described in the previous reports. Only the results of the simulations with the validated model are presented here.

Rollover Test Simulation According to ECE-R66 and FDOT Standard

In the rollover test procedure, a vehicle resting on a tilting platform (as shown in Figure 6.2) is quasi-statically rotated onto its weaker side. Depending on the attachments of the staircase and the door frame to the bus frame, it is usually the road side of the bus. When the center of gravity reaches the highest (critical) point, the rotation of the table is stopped. Further the gravity causes the bus to free-fall into a concrete ditch. The flooring in the ditch is located 800 mm beneath the tilt table horizontal position. LS-DYNA [3] simulations involved simplified case where the bus was positioned in the configuration just before the impact with the ground. Initial velocities were applied to the structure of the bus to simulate appropriate conditions of the real test. Such approach saved computational time needed for each run.

The bus passes the rollover test if the residual space is not compromised during the tests (4), (5). The shape of the residual space is defined in Figure 6.3. The FDOT standard is based on the ECE-R66 but it contains several extensions. An additional quantity called Deformation Index (DI) was proposed in the FDOT standard for quantitative comparison of the results (6). Consistently with the concept of the residual space - the DI is only providing information about the passenger compartment and not the driver's cabin. The DI is based on the assumption that during the rollover-induced impact, the angular deformations develop only in hypothetical plastic hinges located at vulnerable connections in the bus cross section. The rotations in these hinges are marked on the bus cross section as through in Figure 6.3a. The elastic deformations of the walls are neglected in this definition. Based on the geometry of the failure mode (Figure 6.3b), DI is defined as:

$$DI = \frac{l}{d} \cdot \tan(\Delta\alpha_1) + \frac{(h-l)}{d} \cdot \tan(\Delta\alpha_2) \qquad 6.1$$

For acceptable designs, *DI* is in the range 0 < *DI* <1. Once the deforming walls start to touch the residual space - the *DI* is equal to 1. When *DI* > 1, the structure of the wall intrudes into the residual space and the bus fails the test.

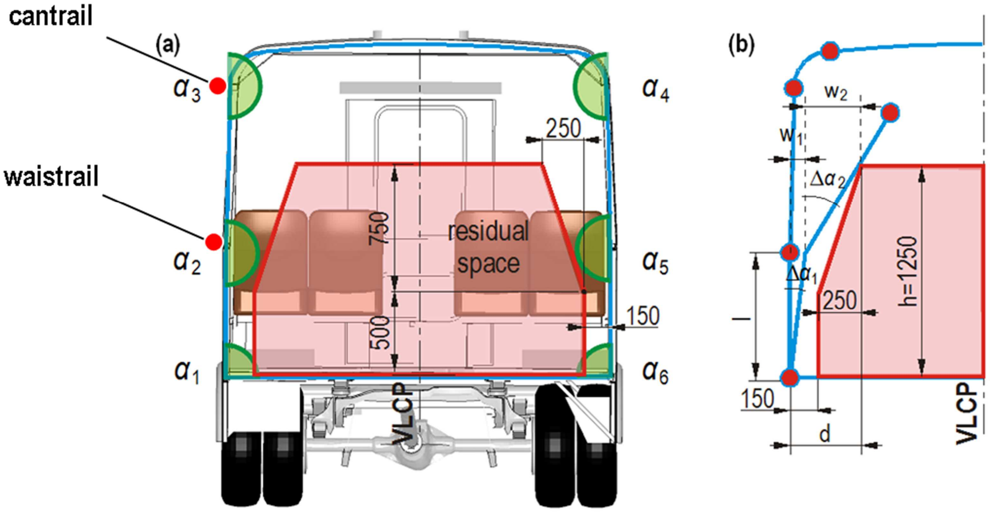

Figure 6.3: (a) Definition of the residual space (b) geometry of the failure mode (6)

The verified and validated FE model was used to simulate rollover test according to the ECE-R66. The deformations of the bus due to the impact are presented in Figure 6.4. The residual space is visibly penetrated by the wall pillars. Figure 6.5 shows history of the DI calculated using Equation 6.1. The bus significantly fails the test with DI reaching value of 2.1 at about 0.4 sec of simulation.

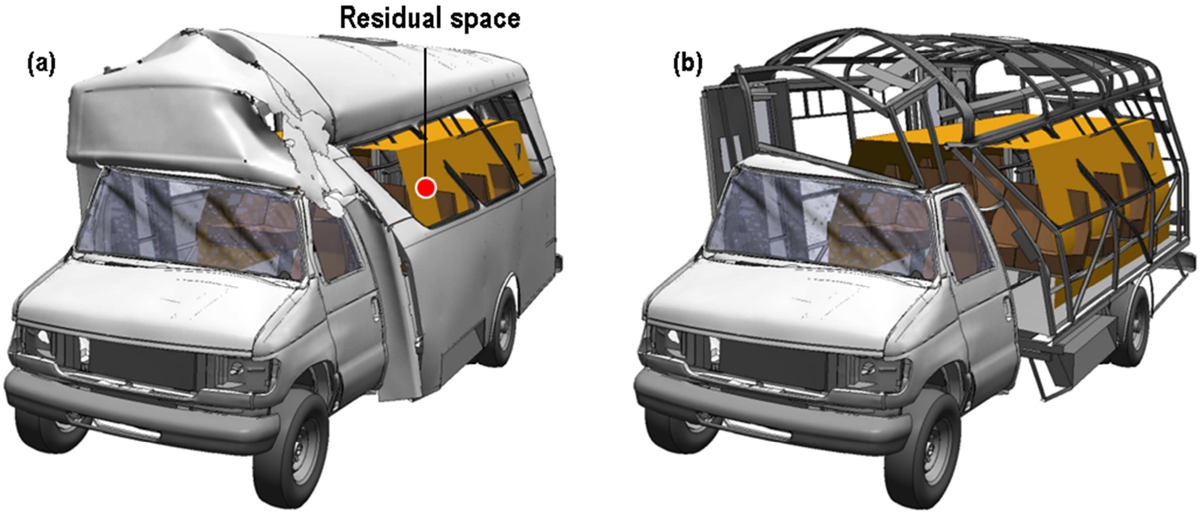

Figure 6.4: Residual space compromised by the bus structure. View of complete bus (a), view without skin (b)

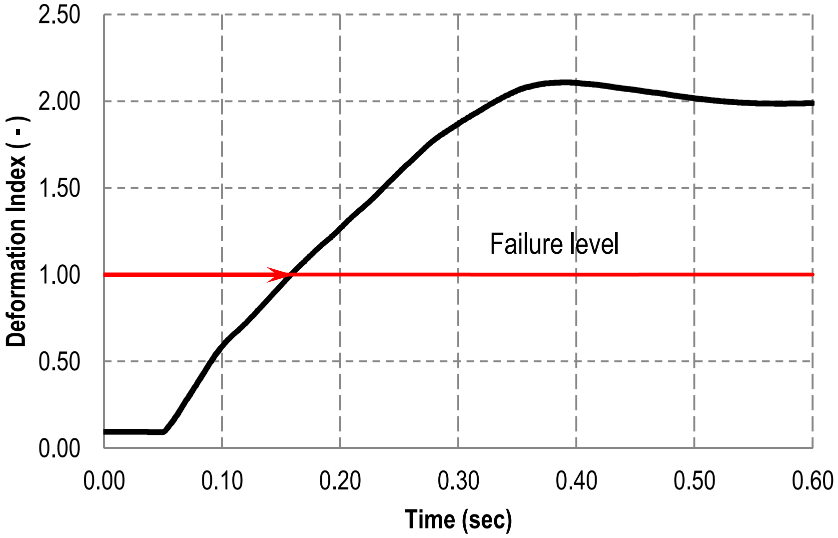

Figure 6.5: History of the Deformation Index measured in the rollover test simulation per FDOT standard

The bus is deformed in the torsional mode with rear part being considerably less deformed. As an outcome of the impact, the plastic deformations were developed at the front cap structure and at the waistrail beam. The cantrail beam was also deformed locally at the connections of the roof bows to the walls. Taking a closer look at the design of the front cap structure, one can find some obvious reasons of its weakness. The actual connections between the body and the driver's cabin are shown in Figure 6.6 and Figure 6.7. The bus body is only connected by two flat pieces of steel on the road side of the bus. On the curb side, the driver's cabin is welded to the staircase in two spots (see Figure 6.7). The cap rests on the remainder of cabin roof and is connected with only a few additional welds.

Figure 6.6: Connection of bus body to driver's cabin road side

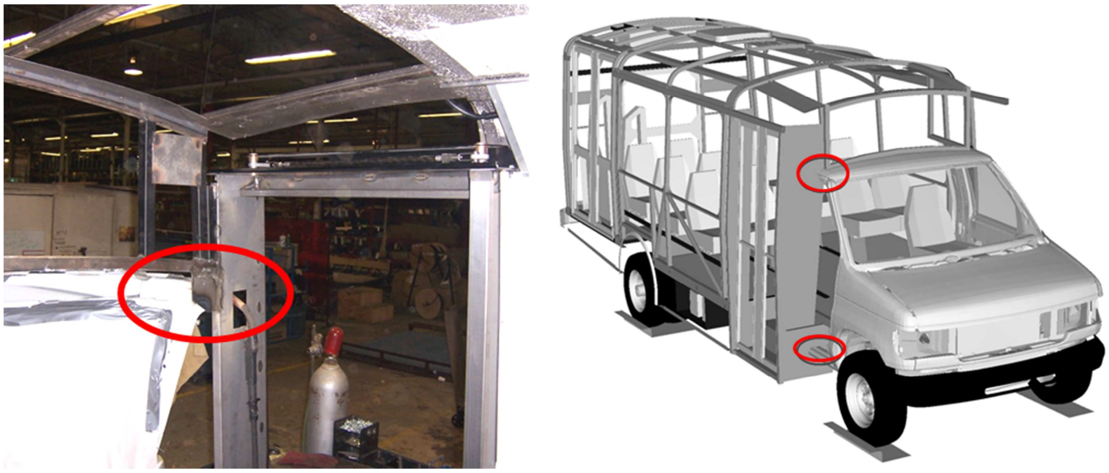

Figure 6.7: Connection of bus body to driver's cabin curb side

Roof Crush Resistance Test Simulation According to FMVSS 220 Standard

The same FE model, as used before for the ECE-R66 rollover simulation, was utilized to simulate the testing procedure of FMVSS 220 for roof crush resistance. An equivalent of 1.5 of Unloaded Vehicle Weight (UVW) is applied quasi-statically in this test procedure to the roof structure of the bus through a rigid plate. During the test, the resistance force and the displacement of the plate are recorded. This force should cause a roof deformation smaller than 130.2 mm (5.125 in) in order to pass the testing procedure. The bus chassis beams are directly supported so the deflection of the suspension is not taken into account in the test. Mass of the tested bus was equal to 4,636 kg (10,221 lb). Thus, the 1.5 of UVW was equivalent to the force of 68,219 N. The plate dimensions differ depending on the vehicle weight and in the FE model they followed the directions for the vehicle with GVWR of more than 4540 kg (10,000 lb). The load was applied in two phases as specified in the FMVSS 220 standard. First, the pre-loading of 2,227 N (500 lbf) was applied to reduce slack in the system. In the computer simulation the loading was generated through the prescribed vertical displacement applied to the center of the plate. The plate was free to rotate about this point. The coefficient of friction for contact between the plate and roof structure was set as 0.15 (steel to steel) in AUTOMATIC_SURFACE_TO_SURFACE type of contact. In the LS-DYNA simulation the loading phase was shortened to 1 sec, and the mass of the application was reduced in order to eliminate inertial effects. Additional simulations were performed with lower loading rate. It turned out that the results were similar and the lower loading rate is not needed for further simulations.

Figure 6.8: Residual space compromised by the bus structure. View of complete bus (a), view without skin (b)

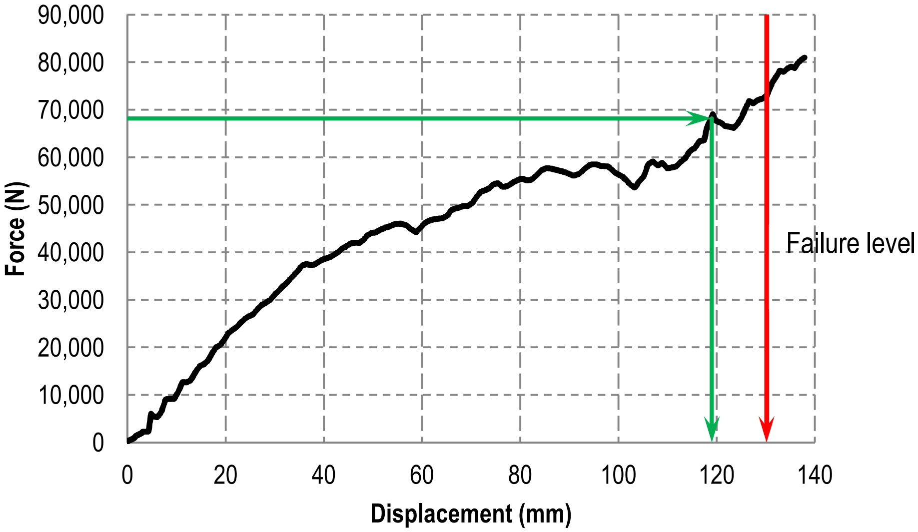

Figure 6.9: Time histories of displacements of the loading plate with zero displacement corresponding to the 2227 N (500 lbf) load

Figure 6.8 shows a view on the deformed structure of the bus. Figure 6.9 shows history plot of a force vs. loading plate displacement. The 1.5 UVW limit (equal to 68,219 N) was reached at 119 mm of penetration. Thus, unlike in the case of the ECE-R66, the bus considered passed the FMVSS 220 testing procedure.

References

1. Bojanowski C. Verification, *Validation and Optimization of Finite Element Model of Bus Structure for Rollover Test, Ph.D. dissertation*, Florida State University, May 2009
2. NCAC, Finite Element Model Archive. 2008, http://www.ncac.gwu.edu/vml/models.html.
3. Link to TRACC's ls-dyna web page
4. Ref (1) in quarterly
5. Ref (5) in quarterly
6. Ref (6) in quarterly