Development of simplified safety assessment procedure for paratransit buses

Lesław Kwaśniewski¹, Jerry W. Wekezer², Bronisław Gepner² and Jeff Siervogel*¹

¹Faculty of Civil Engineering, Warsaw University of Technology, Al. Armii Ludowej 16, 00-637 Warsaw, Poland
e-mail: l.kwasniewski@it.pw.edu.pl
²Department of Civil and Environmental Engineering, FAMU-FSU College of Engineering, 2525 Pottsdamer Street, Tallahassee, FL 32310-6046, USA
e-mail: wekezer@eng.fsu.edu (J.W. Wekezer)

Abstract

This paper describes development of a simplified procedure for the safety assessment of paratransit buses. The principal concept of the bus crashworthiness evaluation is adopted from the United Nations (UN), and ECE Regulation 66 [13]. The main objective of this study is to replace expensive and impractical full scale tests with a series of simple and well defined experiments on structural components. The research methodology utilized for development of a new procedure is based on two tracks of parallel research activities. The first research track consists of experimental tests conducted on three conceptual levels: material characterization, component testing, and full scale tests. The second track contains virtual testing representing the experiments. The virtual tests are performed using the nonlinear explicit dynamic code LS-DYNA® and well developed, verified and validated Finite Element models. The numerical results are used for extensive parametric studies and “what if” investigations. The FE models of the entire bus and its tested components are validated through comparison with the experimental data derived from several laboratory tests of the bus connections.

Keywords: crashworthiness, impact, industrial problems, numerical analysis, structural mechanics

1. Introduction

Paratransit buses form a unique segment of vehicles in the USA. They are commercial vehicles designed to transport up to 22 passengers short distances, complementing regularly scheduled bus routes. The majority have a Gross Vehicle Weight Rating (GVWR) between 6 to 9 tons, which places Paratransit buses between the classes of lighter passenger vehicles and heavier coaches. GVWR is recognized as the main criteria for application of numerous federal standards which specify requirements and approval procedures. Unfortunately none of the regulations specifically addresses paratransit buses and especially their roof crush resistance. The lack of applicable crashworthiness standards is compounded by way the paratransit buses are constructed. The production of paratransit buses is carried out in two different stages. First, a major, reputable auto manufacturer constructs a vehicle with a driver cab and chassis under strict quality guidelines. Then a smaller company, a “body builder” attaches passenger compartments with all additional equipment, as required by a customer. The manufacturing process raises many technical problems, especially regarding the bus connections, while a large number of manufacturers results in variety of technical solutions.

The problems described above have been addressed in a series of research projects conducted since 1999 at the Crashworthiness and Impact Analysis Laboratory (CIAL) and resulted in the first version of the FDOT Crash and Safety Standard included in the Florida Vehicle Procurement Program [14]. The principal concept of the bus crashworthiness evaluation was adopted from the United Nations ECE Regulation 66 which refers to the integrity of the roof and is based on a full scale rollover test [13]. The purpose of this procedure is to ensure a superstructure of sufficient strength so that the precisely defined residual space stays intact (is not intruded upon) during and after the rollover test. The current version of Florida Standard [14] provides two assessment methods by either experimental full-scale crash tests, or by the computational analysis using a validated Finite Element (FE) method. This safety assessment program is described in detail in recent publication [9]. Although the numerical approval procedure can be recognized as an easier, more thorough and conclusive, both methods have been found to be expensive due to the need to either destroy a new bus or the time/money required to develop a validated FE model.

To overcome this inefficiency a new simplified procedure is currently under development. The main objective of this investigation is to replace the impractically expensive full scale rollover with a series of simple and well defined experiments on structural components. The new procedure is intended to estimate with a high level of confidence whether the bus is likely to pass the full scale rollover test.

The research methodology applied for the development of the new procedure is based on two parallel sets of activities. The first research track consists of experimental tests conducted at three conceptual levels: material characterization, component testing, and full scale tests. The second track contains virtual testing performed using LS-DYNA®, the nonlinear explicit dynamic code and verified and validated FE models. The numerical results are used for extensive parametric studies. Their goal is to develop detailed test procedures, expected limiting values, and approval criteria. This paper describes the computational part of the research with FE model development, and its verification and validation.

2. Full scale rollover test

A safety assessment procedure can be understood as a tool used to differentiate unsafe buses from the rest of population. Nobody can guarantee that every passenger in any case will be saved but a safe vehicle can reduce the consequences of an...
accident. Such procedure should be relatively simple, repeatable and should represent a good balance between economical and safety requirements. The main objective of a safety assessment is to determine when a structure is considered as safe by providing testing procedure and limit values.

As it is impossible to consider all different accident scenarios, the safety assessment is usually based on simplified assumptions derived from the statistic analysis. Detailed accident records regarding the performance of paratransit buses is scarce due to their common inclusion within a more general bus category in overall crash statistics. However, European bus rollover accident statistics show that while they do not occur very often, 42% of bus accidents with at least one severe injury or passenger fatality result from rollovers or tipovers [1]. Similar tendency is reported in the USA for motorcoaches where fatalities as an aftermath of rare rollover accidents comprise 34% [11].

The ECE Regulation 66, adopted in this research, refers to the integrity of the roof and is based on a rollover test [13, 14]. The test setup from R66 is shown in Figure 1. The purpose of the ECE R.66 is to ensure a superstructure of sufficient strength so that the residual space, defined in Figure 2, stays intact both during and after the rollover test. It stipulates that vehicle resting on a tilting table is first quasi-statically rotate onto one of its side. When the center of gravity reaches a critical point, gravitation causes a free falling off the bus onto the ditch. Concrete flooring of the ditch is placed 800 mm beneath the tilting table horizontal position. The bus is considered to pass the test when a defined survival (residual) space remains intact in the deformation process (see Figure 2 for residual space definition [13]). Figure 3 shows the test on an actual paratransit bus conducted by CIAL at the FDOT Springhill Facility [6].

![Figure 1: Roll-over test setup according to ECE R.66 [13]](image1)

![Figure 2: Definition of the residual space according to ECE R.66 [13]](image2)

![Figure 3: Rollover test conducted at the FDOT Springhill Facility [6]](image3)

### 3. FE model development

Development process of a finite element model of a bus closely replicates the actual bus manufacturing process. In the first stage, the FE model of the cutaway chassis was virtually extracted from a public domain FE model of the Ford Econoline Van (see Figure 4), which was developed earlier by the National Crash Analysis Center (NCAC) at George Washington University [5].

![Figure 4: Modified FE model of chassis cut of the FE model of Ford Econoline Van, developed by NCAC. [5]](image4)

In the second stage, a 3D FE model of the passenger compartment (bus body) was built based on the centerline dimensions of the profiles, provided by a manufacturer. All structural and some non-structural components of the interior including passenger weights were included in the model to fully replicate the mass distribution and inertia properties of the bus.

The model of the bus body was developed from AutoCAD drawings provided by the manufacturer, and later combined with the chassis FE model. The 3D geometry and FE mesh of the bus body was primarily developed using the Altair Hyperworks graphical preprocessor, though some later modifications were made using LS-PrePost. The process of building the FE model was based on the “short step – check” method, which divides the model development into numerous steps, each followed by a verification test. During this process many questions arose, primarily concerning connections between the structural elements and their representation in the FE model. In addition, the critical connections of the bus structure components, such as the wall-to-floor and wall-to-roof
connections were thoroughly investigated using empirical testing.

Once finished, the FE model of the body was imported and connected to the model of the cutaway chassis using LS-PrePost. Contact between all parts was defined by their inclusion in an automatic single surface contact [7]. Welded connections were modeled using rigid (spotweld) links between adjacent nodes or merged nodes. The total mass and the CG position of the complete paratransit model were determined and compared with the data included in the technical specifications. At this stage of the FE model development, the mass of the model is underestimated because of unaccounted non-structural components such as wiring, insulation, plastic covers, and many others. It is corrected by changing the mass density of some selected parts until the model is in close agreement with the specifications. Numerical simulations (primarily drop tests) were performed using LS-DYNA in order to verify structural integrity of the entire FE model.

![Figure 5: An isometric view of a complete FE bus model](image)

Several different structural materials were used to construct the actual bus. Most important was the mild steel used for the passenger cage, frame rails, suspension components, doors, and for the most of the front cab. Constitutive models and material characteristics were established based on laboratory tests [8], and where these were not available - MatWeb [10] data was used. For the majority of the bus materials, the LS-DYNA material model MAT_PIECEWISE_LINEAR_PLASTICITY was adopted [7]. Materials with failure and element erosion are often used to model local damage and material separation in crash simulations. Ultimate plastic strain was used as the failure criterion for all non-steel materials during this study. For the steel used in the bus body a failure strain of 0.2 was investigated and implemented [4]. No failure is defined in the NCAC parts of the chassis model. The strain rate dependence of steel was incorporated into the FE bus model using the experimental Cowper and Symonds model [12] and the original values of the required coefficients, established by NCAC for Ford Econoline. An example of the bus FE model, consisting of almost a million of finite elements, is shown in Figure 5. Statistical information of the FE model of the paratransit bus developed is provided in Table 1.

<table>
<thead>
<tr>
<th>Number of:</th>
<th>Chassis model</th>
<th>Bus body</th>
<th>Whole model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elements</td>
<td>189,079</td>
<td>735,407</td>
<td>924,486</td>
</tr>
<tr>
<td>Nodes</td>
<td>204,998</td>
<td>658,028</td>
<td>773,026</td>
</tr>
<tr>
<td>Parts</td>
<td>295</td>
<td>64</td>
<td>359</td>
</tr>
<tr>
<td>1D elements</td>
<td>2</td>
<td>0</td>
<td>2</td>
</tr>
<tr>
<td>2D elements</td>
<td>173,401</td>
<td>735,407</td>
<td>908,808</td>
</tr>
<tr>
<td>3D elements</td>
<td>15,676</td>
<td>0</td>
<td>15,676</td>
</tr>
</tbody>
</table>

4. Energy balance

A consideration of transition of mechanical energy during a rollover test serves two objectives. The first goal is a verification of the FE model through check of the energy balance for every time instance. The energy balance is checked and compared with simple hand calculations as one of the verification procedures. All non-physical forms of energy, i.e. hourglass and contact energy, should be kept at minimum. The second objective is determination of energy absorption among different structural parts.

Numerical solution errors in FE simulations are mainly attributable to the discretization approximation. However, there are other multiple factors influencing correctness and stability of the solution. These quantities can be checked based on the energy balance during the FE model development process (see Figure 6). All components defining the total energy should satisfy the principle of energy conservation. Energy should also be verified against hand calculations as a first check of the simulation.

![Figure 6: Energy balance in the rollover simulation](image)

An identification process of key structural components responsible for bus structure behavior during the rollover test is based on the energy approach. As estimated earlier in [9], [3], the energy delivered to the bus structure during the rollover experiment is approximately equal to:

\[ E_r = 0.75 M g \Delta h \]  \( \cdots \) \( \text{(1)} \)

Where:

- \( M \) = is the total mass of the bus,
- \( g \) = is the acceleration due to gravity, and
- \( \Delta h \) = is the vertical distance from the highest, unstable position of the bus CG to its final location.
The remaining part of potential energy (25%) is mostly transformed to the ground and dissipated by the friction and damping effects. Figure 7 shows the energy balance obtained from the LS-DYNA FE rollover simulation of the calibrated FE bus model. Energy balance at any time \( t > 0 \) is given by Equation 2:

\[
E_{\text{total}} = E_{\text{int}} + E_{\text{kin}} + E_{\text{rw}} + E_{\text{damp}} + E_{\text{wall}} + W_{\text{ext}} \quad (2)
\]

Where:

- \( E_{\text{total}} \): Total Energy defined by LS-DYNA as consisting of all physical energy components associated with a model.
- \( E_{\text{int}} \): internal energy (energy absorbed by the structure)
- \( E_{\text{kin}} \): kinetic energy
- \( E_{\text{rw}} \): stone-wall energy (energy dissipated by the rigid wall)
- \( E_{\text{damp}} \): system damping energy (energy dissipated by damping)
- \( E_{\text{wall}} \): hourglass energy (non-physical form of energy resulting from damping hourglass deformation modes)
- \( W_{\text{ext}} \): external work from applied loads.

In preparation for the development of the new simplified procedure an internal energy assessment was performed for the selected bus. Global internal energy in the simulation (energy absorbed by the bus during the impact) is distributed to the separate part groups as shown in Figure 7 and Table 2. It has been identified that the major components responsible for bus structure behavior during the rollover are: cage, skin, front fiberglass cap, and bus chassis cab. These parts were chosen for further investigation to establish energy absorption requirements for simplified procedure.

Table 2: Distribution of internal energy among major components of the bus structure

<table>
<thead>
<tr>
<th>Component</th>
<th>% of total</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 Bus steel cage</td>
<td>57.22</td>
</tr>
<tr>
<td>2 Bus skin</td>
<td>19.66</td>
</tr>
<tr>
<td>3 Front fiberglass cap</td>
<td>6.97</td>
</tr>
<tr>
<td>4 Front chassis cab</td>
<td>6.03</td>
</tr>
<tr>
<td>5 Bus doors and windows</td>
<td>4.26</td>
</tr>
<tr>
<td>6 Other components of the bus</td>
<td>5.86</td>
</tr>
<tr>
<td><strong>TOTAL:</strong></td>
<td><strong>100.00</strong></td>
</tr>
</tbody>
</table>

5. **Experimental tests on structural components**

A laboratory testing program was developed and implemented in support of the verification and validation of the FE model [9]. Material characterization tests are carried out at the lowest level of the validation hierarchy [8]. Bending of steel tubes and skin composite samples can be categorized as testing at the component level [9]. The moment-rotation connections tests and the impact test on the side wall panels are the higher level tests on the subsystems of the structure [9]. A center of gravity (CG) check is performed at the complete system level, [9]. The proposed tests comprise only the required minimum that provide information about the behavior of the main structural components. Depending on time and budget constraints, additional tests shall be conducted for improved results and increased model reliability. The most complete models should be used for connection tests with actual connectors including adhesive, welds and bolts, as appropriate.

The component tests, identified as the second level of the multi-scale testing program, are the most important for the development of the simplified safety assessment procedure and are presented below in more detail.

5.1. **Bending tests of structural tubes**

The main structural elements in Paratransit buses are usually build from rectangular tubes. A four point bending test was selected as the direct measure of the strength of the tubes and the validation of the model. The testing apparatus for the four point bending is shown in Figure 8. The distance between the external (moveable) supports is equal to 900 mm and 300 mm between internal supports. The internal supports were connected to the grip through the hinge. The diameter of supports was equal to 30 mm. The INSTRON 8802 testing machine with FastTrack software was used for the tests. The displacement was applied with the rate of 20 mm/min. The displacement of the bottom (moveable) traverse is denoted as \( d_{0c} \). Additionally, deflection of the beam in points \( d_1 \) and \( d_2 \) (under the internal supports) and \( d_3 \) (middle of the beam) were...
recorded. The quantitative results of the tests for profiles coming from three different buses are shown in Figure 9. Figure 10 presents averaged exerted load plotted against the displacement of the point for three types of tested tubes. The same test for the bus-1 was also simulated using the LS-DYNA software. The load–displacement curve from the FE analysis is also shown in Figure 10.

Figure 8: Testing apparatus for four point bending test

Figure 9: Deformed tubes as a result of the four point bending tests

Figure 10: Load-displacement characteristics for tubes tested in bending

5.2. Impact tests on tubes

If the steel frame is properly welded and firmly connected to the chassis, most of the energy is dissipated during a roll-over test in plastic hinges generated in the steel tubes. A linear impact tower was built to assess energy dissipation capability of tubes used in the bus structure (Figure 11). A rigid frame was built out of 4"x4"x1/4" HSS steel components to create a stiff testing platform. Two linear bearing shafts were fixed to the frame at the base, using a steel bearing plate, with adjustable fixity at the top, to allow unconstrained vertical movement of the impact hammer system. The hammer itself was designed with an aluminum body and semi-circular aluminum head fixed together and sliding on ceramic bearings capable of delivering a wide range of energy onto the tested sample. Two accelerometers are installed onto the impact hammer to allow the computation of velocity and displacement using accelerations recorded during the impact.

Figure 11: Impact apparatus (left), Impact hammer (right)

5.3. Bending of connections

The bus body is constructed by first assembling the major components (floor, sidewalls, backwall, roof) individually and then welding and/or bolting them together. This process is used for major connections between the subsections. Dynamic performance of these connections does not only depend on the material properties but even more significantly on the selected connection design which is affected by the bus assembly process.

Figure 12 shows location of the wall-to-floor WF (1) and roof-to-wall RW (2) connections selected for connection testing.

Figure 12: Location of components for connection testing in the bus structure

Representative samples of the connections were obtained from the manufacturer for the study of the WF and RW connections. Connections are tested in bending where one half
is clamped and the other is pulled quasi-statically to decrease the angle between both sides. A testing apparatus shown in Figure 13 was designed to measure the resistance response of the connections. It allowed for data acquisition of a rotation angle of the connection as a function of the force (or: equivalent moment) applied.

Figure 13: WF connection without skin fixed for bending testing

5.4. Side wall impact test

The fourth group of the tests comprises of side wall panels, with and without skin, subjected to impact loading using a larger impact hammer. The tests are used to validate welded connections between vertical and horizontal tubes, and to determine the amount of energy absorbed by skin. A dynamic impact test on the side wall panel was developed for additional model validation [9]. Location of the side wall panel used for the testing in the bus structure is shown as an item (3) in Figure 12. The panel is cut off from the wall and extends from the cantrail to the level of the floor. Its width spans two major vertical beams (which are included in the panel) thus both dimensions (height and width) differ for every bus model. Initial conditions for the test are shown in Figure 14.

Figure 14: Impact hammer designed for side wall impact test

The panel is resting horizontally on raised tubular supports with 150 mm in diameter. The two supports are at adjustable distance, which was set at 1600 mm. The impacting device is comprised of impacting square tube, perpendicular rectangular arms and crossing rectangular beams. The location of the impact zone is selected to be below the waistrail level, which is close to the middle of the panel. Due to the short duration of the event only the final results of the experiment are captured. The deformation pattern and maximum deflection are recorded and then used for comparison with numerical results.

6. Program for parametric study using virtual testing

Figure 15 depicts the main idea for ongoing parametric study. The whole investigation is based on the virtual testing using verified and validated FE models of the entire bus structure and its selected components. The goal is to find relationships between performance of selected components and the behavior of the entire bus during a roll-over test. Four groups of component tests have been designed thus far, as described in the previous section.

A numerical parametric study is devised to deliver data which will be used to improve test procedures and establish limit values with relation to the basic bus structure characteristics. A strong correlation between the performance of the bus connections and the overall crashworthiness of the bus should allow for derivation of simple analytical formulae for quick evaluation of component tests. It will be therefore possible to determine if the bus is likely to pass the rollover test based on the performance of its connections. Consequently, it will also be possible to determine the minimum and target energy levels of the connection to meet the desirable bus performance.

Figure 15: Concept for parametric study using virtual testing

6.1. Virtual bending of structural tubes

Figure 16 shows initial conditions for the simulation of the four point bending test. The tube rests on two rigid cylinders. LS-DYNA option RGDWAL_GEOMETRIC_CYLINDER_MOTION is used for that purpose which allows for defining cylindrical shapes with prescribed motion. Vertical displacement is assigned to the rigid cylinders in order to simulate the actual loading. Since the test is quasi static, and a kinetic energy is relatively low in comparison with internal energy, the density of the materials was artificially increased. Such approach is common and it allows for increasing the time step for the explicit solver in LS-DYNA, and for reducing computational cost of the simulation. A fully integrated element formulation 16 was used for all shell elements.
The top support was fixed in the space and only rotation in the plane of the beam was allowed during the test. A comparison of experimental and computational results is shown in Figure 17.

6.2. Virtual impact tests on tubes

A computational model of the linear impact tower shown in Figure 11 was developed. Contact option CONTACT_AUTOMATIC_SURFACE_TO_SURFACE was defined between the impact head and the beam, and between the beam and cylindrical supports. This allowed for tracing of all separate contact forces between the individual parts. The same contact option was selected for the beam itself to detect any self contacting surfaces. Such behavior can be noticed when a local plastic hinge and local buckling develop with large deformations (Figure 18 right).

6.3. Virtual bending of connections

Figure 19 shows deformations in the connections tested in the experiments and corresponding deformations in the FE simulations. All figures reveal poor design of the original connections. Major deformations occurred not in the structural beams but in the transition; members like C-channel in the WF connection and L-shape in the RW connection. In order to increase the connection strength, the elements should have additional welds and/or bolts preventing unnecessary and excessive deformation.

Figure 20 and Figure 21 present characteristic curves obtained numerically for two connections – WF and RW respectively. Curves from corresponding LS-Dyna simulations are shown along with the experimental results. The FE simulations were carried out for two cases of different tubes thickness – 100% of nominal and 93% of nominal thickness, which was equal to the measured thickness of the walls.
6.4. Virtual impact test on side wall panels

In the design of the side wall used for the test, the waistrail beam was continuous throughout the length of the bus and the vertical beams were welded to it at the top and the bottom. Discontinuity of the vertical pillars resulted in the excessive local deformations in the waistrail beam as shown in Figure 22b. Figure 22a shows corresponding deformations in the FE simulation for comparison.

![Figure 22a: Local deformations in the tested panels.](image)

The basic model with two finite elements across the beam width was not capable of capturing such severe deformation. The mesh density had to be increased to fully reflect real deformation pattern. With the increased mesh density, obtained deflection in the FE simulation was equal to 298.8 mm. The deflection was measured as 312.0 mm during the experiment, which gave the relative error for the FE simulation of 4.2 %. Such design should be avoided in the bus structure since the capacity of it depends only on the strength of the single thin wall of the waistrail beam.

The number of elements along the edge of the main tube in the entire bus model was increased to 4 after the side wall panel tests. Still it is lower than 8 elements used in the FE simulation of the side wall impact test.

Other factors were also found as significantly contributing to the overall response of the bus in the rollover test. The bonding strength of the adhesive used between the skin and the frame is one of them. Yet, the most crucial, steel cage can be assumed to be fully validated.

7. Summary

The paper presents a numerical development of a simplified method to assess crashworthiness and safety of paratransit buses. The method is based on multi-scale experimental tests. Each test is designed to determine actual amount of dissipated internal energy and compare it with minimum and target energy. Buses with connections meeting the minimum energy standards will be judged as likely of meeting the crashworthiness standard [13].

References


