Impact Experimental Analysis and Computer Simulation  
Yucheng Liu  
Department of Mechanical Engineering, University of Louisville

Abstract
In this paper, an automotive bumper system (a bumper connected to a frame through joints/connections) is set up for small impact tests (nondestructive tests). During the tests, a steel bar is raised to certain highs then released from there and hit onto the bumper system, the bumper system is fixed to the ground through fully constraining the end of two shocks, the reaction forces at the end of shocks during the whole impact process are measured and investigated in order to determine how they are transferred to the joints then to the rest part of the frame. Also, other important experimental data are collected and studied to reflect the actual impact process. A finite element model is created for the automotive bumper system and the impact test is also simulated on the computer using ANSYS. The results from the experimental tests are compared and correlated to the finite element simulation. From the comparison, it is found that the experiment results and FEA results matches very well and the validity of the computer model is then verified. Additionally, this paper also includes a hand calculation of the impact problem, where the bumper system is modeled as a simple spring-damping system and solved using classical dynamic theory; the hand solutions are also compared to the experimental results and the FEA results to verify the correctness and reliability of the impact tests and computer simulation.

Introduction
This project is to compare and correlate the experimental results, the computer FEA results and the theoretical results for a small impact case, then to show that for any small impact case a reasonable computer model can be found to simulate it. In the project an automotive bumper system is used to perform the impact tests, some important experimental data such as reaction forces and displacements at certain locations are recorded, and a computer FEA model is created for the bumper system. A transient dynamic analysis is performed to simulate the impact tests, and the corresponding computer results are obtained. Finally, the bumper system is modeled as a simple spring-damping system and a hand calculation is done to get the theoretical results, all the three results are compared and correlated, and the strengths and the weaknesses of the project are discussed.

The objectives of the project are to verify the efficiency of the computer model and the computer simulation, and to prove that there always existed a computer model that can be used to simulate any small impact cases. At the same time, the behavior of the bumper system during the impact process is studied and how the impact forces are transmitted to the joints and how they are transferred from the bumper to the frame are explained through the project.

Literature Review
Before starting this project, some of published literatures and previous researches have been reviewed build up a solid background in the area of experimental analysis and finite element analysis.
Toshiyuki Sawa, Yoshihito Suzuki, and Shoichi Kido used finite element method to analyze the stress wave propagations in adhesive joints of similar hollow cylinders under static and impact tensile loadings in elastic deformation range. They used DYNA3D to start the analysis and applied the impact loading to the joint by dropping a weight. The effects of the Young’s modulus of the adhesive on the stress wave propagation at the interfaces were examined and finally they found that the characteristics of the joints subjected to impact loadings were opposite to those subjected to static loadings. [9]

Thomas J. Trella, Randa Radwan, Samaha (1995) described the development and validation of a computer based model of the moving deformable barrier developed for side impact safety performance simulations using LS-DYNA3D. They investigated the effects of important factors central to FEA modeling such as
material node merging, mesh density, and element type and then found that the material damping coefficient and compacted Young’s modulus both had a strong influence on the simulated impact responses.[7]

David H. Johnson, Richard B. Englund, Brian C. McAnlis, Kevin C. Sari, and David Colombet presented a modeling technique used to create a “mostly-brick” meshed 3D model of a nut and bolt joint using ANSYS and the created 3D modeling can simulate the conditions of joint tightening and sliding along the helical thread flanks when the nut is turned.[4]

Ford engineers (2001) developed a target-vehicle model used for computer simulation of vehicle crash compatibility. For the target-vehicle model they chose five frontal impact modes to test it, which included full frontal impact and corner frontal impact. After running the analysis the model would provide the vehicle responses and component characteristics such as compression, tension, bending stiffness and rate effects which were used to compare with the results of vehicle-to-vehicle test. The target-vehicle model was then be calibrated and optimized based on the results of comparison until an ideal target-vehicle was reached in the end. The methods and ideas utilized in the modeling process were kind of enlightening.[5]

S. W. Kirkpatrick, J. W. Simons, and T. H. Antoun (2000) developed and validated a high fidelity finite element model of a full size car for crashworthiness analysis, which was part of an overall program to develop a set of detailed finite element models for various vehicles. In the program, they selected the Ford Crown Victoria as the representative full size car and briefly described the modeling procedure including the vehicle teardown and digitization and model generation. The techniques used in vehicle digitization; the mesh and the element type used in the FE model were introduced, and the developed FE model was presented too. The authors performed the component crash tests and vehicle crash tests separately and obtained a set of data from the tests for validating the crash model. In their paper, they introduced the test conditions and analyzed and compared the test results thereby concluded the overall collision response of the vehicle and verified the validity of the developed model.[5]

From above demonstration, tremendous advancements have been made on the computer simulation of impact analysis and the FEA methods and the CAE tools had been intensively applied for solving such problems. And in this project, the path put forward in those previous literatures is followed to create and validate a computer model and it is proved that the methodology used in the project could be spread into other impact problems.

**Methods and Materials**

**Experimental Analysis**

**Experiment setup**

In the experimental part a series of small impact tests have been performed on the bumper system. During the tests, the reaction forces at the end of two shocks are measured, and the displacements at the connections between the shocks and the bumper are recorded either. The whole experiment system is set up like this: a bumper system is obtained and held by a bracket, a long steel pipe is held vertically to serve as a guide through which the hitting object can vertically drop onto the bumper. A 10.5”, 8.4 lb iron bar with a long nylon string tied on its end is used as the hitting object, the nylon string is marked at different locations such as 3”, 4”, etc. in order to control the iron bar’s initial position. Thus, with the initial position of the iron bar, h, the bar’s initial velocity when it hit on the bumper can be easily determined using the formula $v^2 = 2gh$. The bumper system used in the tests is detached from a 1980 Volkswagen. Since the original shocks are almost failed so they have been taken away and two solid steel cylinders are used to replace the shocks that connect the bumper and the frame together. Two dynamic load transducers, whose type is 208 A03 and limit is 800lb, are used to measure the reaction forces; the load transducers are installed at the end of the shocks and are fixed to the bottom plate of the bracket using bolts and nuts. A sensitive displacement sensor (DVRT) is applied for capturing the displacements at the connections between the bumper and the shocks, the tip of the sensor just contacts the bumper’s surface where the displacement would be measured, and then is glued to the surface using epoxy to avoid it separating from the surface during the impact process. All the transducers go to an amplifier, and the output signals come
out as voltage. A data acquisition system is used to collect all the experimental data, and a special labview program (Bumper.vi) is created to display these signals on a computer and store them on the disk. The last experimental data displayed on the reports should have been transferred from the volts to corresponding pounds (for reaction forces) or inches (for displacements) by a certain scale. Figures 1 to 3 show the bumper system and the experimental setup.

Figure 1. Bumper system installed on the bracket

Figure 2. Experimental set up for impact test
**Impact tests**

First of all, the iron bar is dropped onto the center of the bumper surface from three heights 3”, 4”, and 6” to measure the reaction forces. Different initial heights are chosen here so that the effects of initial velocities on the impact test can be studied while it has to be promised that the maximum impact force would not exceed the transducer’s limit. At each height, the iron bar is dropped three times to remove the unexpected external interferences. Special Labview program is running during the tests and for each dropping the reaction forces curve is displayed on the computer and saved to the disk. Thus, overall 9 plots are obtained from the tests, and these plots will be compared and used later. After measuring the reaction force, the force transducers are removed and the DVRT is installed. The locations where the displacement is measured are marked and the DVRT is glued to the point to make the sensor keep touching with the measured locations during the whole test. This time the iron bar is only dropped from 10” high and similarly, it is dropped three times at both positions. After all dropping tests finished, the experimental data and related curves are saved and documented for further analysis.

**Experimental results and analysis**

Because the bumper system is a symmetric structure, so the reaction forces should be the same for either end of shocks, therefore, the reaction forces at either side can be taken for analysis. Here the left hand side reaction forces are taken as samples. Figure 4 shows one of the reaction force plots when the bar is dropped from 6 inches high. As compared to other plots, it can be found that for different initial positions, both of the shape of the force curves and the times at which the peak reaction force appeared are similar, and only the peak force values changed. This means that in the low velocity impact cases, the velocity doesn’t change the bumper’s behavior but changes the impact force. (The comparisons are shown in table 1). Also, as compared the reaction force plots that dropped from the same initial position, it is observed that the peak force appears at the same time, and the peak force values are closed to each other (varied within ± 20 lbs range because of the uncontrollable factors during the impact).
After measuring the reaction forces, the load transducers are removed and the displacement transducer DVRT is mounted to measure the displacements. In order to have an obvious displacement, the impact bar is raised to 10” high and dropped from there. The displacements at both connections between the bumper and the shocks are measured and similarly, the object is dropped three times at each position to verify the reproducibility of the impact. From the resulting plots, it can be seen that the displacements at either connections are almost the same because the bumper system is perfectly symmetric about its center axis. Thus, the displacement measured at left connections is selected for further study, which is shown in figure 5.
Figure 5 displays the displacement variation at the left connection during the impact process. The first peak displacement is 0.0178”, which happened at 0.0046 second, and the second peak displacement is 0.0026” that happened at 0.0466 second. The big decrease between the first two peak values shows that the bumper system had a large damping ratio value (the damping value will be calculated later), and the increase in the fourth peak value is because that the bar rebounded after it impacted onto the bumper and hit the bumper again. Thus, from the figure 4 and figure 5, the bumper’s behavior during the impact test can be roughly outlined. During the impact tests, as the dropping bar impacted to the center of the bumper, the bumper is compressed by the impact force during the first 0.006-0.007 seconds and moved down, then returned back through the static position by a much smaller amplitude in another shorter time. After that the bumper experienced several small oscillations until it reached its static position. However, the DVRT measured the maximum displacement at 0.0178”, which might be doubted and will be explained later.

Computer Simulation

**CAD Model**

In computer simulation, a CAD bumper model is first created using IDEAS. All the geometric dimensions of the bumper system including the two shocks are measured and special modeling techniques such as reflecting, lofting, sweeping, and shell are applied to create the structure with a long curve surface. Figure 6 shows the bumper’s CAD model.

![Figure 6. Bumper's CAD model](image)

**Finite Element Model (FE Model)**

After finishing the CAD model, the bumper model’s database is then transported to an IGS file and imported into ANSYS environment to provide the original geometric information for the new finite element model. In creating the finite element model, the 2D shell element, shell 181, is used to mesh the bumper surface, and the 3D elastic beam element, beam 188, is used to mesh the two shocks. The impact bar can be meshed with 3D solid elements, since the impact bar’s deformations or stress distributions are not concentrated in this problem, so the element shell 181 can be still used to mesh the iron bar. The material of the bumper surface is aluminum, and the materials of the impact bar and shocks are steel. The real constants correspond to each element is determined from the measured dimensions. Figure 7 shows the created FE model, which included 230 elements and 357 nodes.
Computer Analysis

The whole impact process will be simulated on a computer by running a transient analysis. In the transient analysis, the impact test that the bar was raised to 6” high then dropped onto the bumper system is simulated to get the reaction force. As set up in the experiment, both ends of shocks are fully constrained; the initial velocity of the impact bar is assigned as 68.16 in/sec, and acceleration 386 in/sec² is applied on the entire environment to simulate the gravity. As a contact problem, the front surface of the bumper is defined as the target surface, and the surface of the impact bar is defined as the contact surface. The solution time is set to 0.04 seconds according to the sampling time in the experiments so that both results can be easily compared. At last the stepped load is used and the number of the substeps is set to 80. To find the displacements, the same settings are still used except that the initial velocity is reset to 87.86 in/sec to simulate the effect of a 10” high dropping. Accordingly the solution time is set to 0.05 seconds and the number of the substeps is set to 100. In order to truly reflect the actual experiments, during the computer analysis, the large deformation option (nlgeom) is turned on, a numerical damping ratio is added, and the elasto-plastic model is used in defining the real constants. After all the analyses finished, the dynamic reaction force at the end of the left shock and the dynamic displacements at the connection between the left shock and the bumper are plotted out, and listed in the time-history postprocessor. Figures 8 and 9 display the reaction force curves and the displacement curves respectively.
Comparisons and Analysis

As compare the computer results to the experimental results, it can be seen that for the reaction forces of 6” dropping, the computer analysis yields the maximum force 468 lb that appeared at 0.002 sec, which matches to the experimental results very well. The comparison verifies that the computer model can predict the peak force value and its appearing time accurately, which is very meaningful in studying how the impact force is transferred from the bumper to the rest part of a vehicle during crashes. While the weakness of the computer simulation is that it did not truly reflect the entire impact process, instead of many small oscillation curves shown in the experimental results, the computer model only yields a smooth curve and omits all the small oscillations and waving.

However, the figure 9 gives out the maximum displacement as 1.33E-4 inches at 0.0035 second, which significantly differs from what has been obtained in the experiment (0.0178”). To evaluate the actual displacement, two extreme conditions are considered. First of all, the shock is looked as a spring whose stiffness is EA/L. If the shock is applied by a static force that equals to the maximum reaction force during the 10” dropping which is 608.94lb (obtained from the computer analysis), the maximum displacement will be $d = F / (EA/L) = 4.59E-5$ inches. Also, if the shock drops from 10” high to the ground, the maximum deformation can be calculated using $d = \frac{v^2}{a}$. Where the $v$ is the impact velocity which can be calculated as 87.86 in/sec, and $a$ is defined by the shock’s young’s modulus and density as $a = \sqrt{E/\rho}$. Then the $d$ can be calculated as 3.56E-3 inches. Based on the evaluations, the actual displacement should lay within 4.59E-5 and 3.56E-3 inches and it can be found that the computer result is in this range while the experimental result seems much higher. The possible reason maybe due to the shaking of the bumper system during the impact process, since the displacements measured in the tests are very small, so any tiny shakings of the bumper system can cause big differences. To overcome this shortcoming, an obvious displacement such as the deflection at the center of the bumper needs to be measured.

Theoretical Solution

The hand calculation of the impact problem is accomplished by applying classical dynamics theories. At first, the bumper system is modeled as a simplified support beam structure with both ends fully constrained. From the mechanics of materials, the stiffness of the beam is $48 \frac{EI}{L^3}$ when the external loads applied on its center. Where the young’s modulus, $E$ of the aluminum bumper is 10E6 psi, the total length $L$ is measured as 55 in, and the moment of inertia $I$ is read from the ANSYS output file as 1.457 in⁴. So the stiffness $K$ of the bumper system can be evaluated as 4203.52 lb/in.
Thus, the whole bumper system can be considered as a spring damping system whose stiffness is 4203.52 lb/in, and the impact problem can be modeled as figure 11.

The governing equation for figure 11 is

$$m \ddot{x} + c \dot{x} + kx = W$$ (1)

Where the W is the weight of the impact bar and the bumper itself, which is 24.4 lb in this project.

Now this problem is ready to be solved to find the maximum reaction force, the maximum displacement and the corresponding time. Actually such kind of problem has been mostly solved by V. I. Babitsky [17], from his theories the maximum displacement is solved as

$$X(t_1) = \frac{v}{\omega_n \sqrt{1 - \xi^2}} \exp[-\frac{\xi}{\sqrt{1 - \xi^2}} \tan^{-1} \frac{\sqrt{1 - \xi^2}}{\xi}] \sin \left[ \tan^{-1} \frac{\sqrt{1 - \xi^2}}{\xi} \right]$$ (2)

And the corresponding time $t_1$ is

$$t_1 = \frac{1}{\omega_d} \tan^{-1} \frac{\omega_d}{\xi \omega_n}$$ (3)

Starting from (2) and (3), the reaction force then can be derived. From equation (1), the reaction force formula can be written as

$$F = c \dot{x} + kx$$ (4)

And from the force balance theory, the function of displacement x can be expressed by the equation (2) as $X(t)$ except to use general time t instead of the special time $t_1$. In order to find the maximum reaction force F, it is only need to solve the differential equation $dF/dt = 0$ to find the corresponding time $t_2$, then substitute the $t_2$ back into the equation (4) to find the maximum reaction force. Following these steps the time $t_2$ is solved as
\[ t_2 = \frac{1}{\omega_d} \tan^{-1} \left( \frac{-C \zeta \omega_n + k}{C \omega_d + \frac{\xi \omega_n k}{\omega_d}} \right) \quad (5) \]

And the maximum reaction force \( F \) is

\[ F = \frac{Cv}{\omega_d} \exp\left(\frac{-\xi \omega_n t_2}{\omega_d} \right) (\omega_d \cos \omega_d t_2 - \xi \omega_n \sin \omega_d t_2) + \frac{kv}{\omega_d} \exp\left(\frac{-\xi \omega_n t_2}{\omega_d} \right) \sin \omega_d t_2 + \frac{W}{k} \quad (6) \]

Thus, the equations (2), (3), (5), (6) will be the governing equations in this hand calculation, where

\[ \omega_n = \sqrt{\frac{k}{m}}, \quad C = 2 \xi \omega_n, \quad \text{and} \quad \omega_d = \omega_n \sqrt{1 - \xi^2}. \]

The parameters that will be used in the equations are:

\[ K = 4203.52 \text{ lb/in}; \quad W = 24.4 \text{ lb}; \quad g = 386 \text{ in/sec}^2; \quad V \] is the initial velocity that listed in table 1; and for the damping ratio, \( \zeta \) it can be calculated from the displacement plots (see figure 5) by using the logarithmic decrement method, which is

\[ \delta = \ln \frac{y_1}{y_2} = \frac{2\pi \xi}{\sqrt{1 - \xi^2}}, \]

where the \( y_1 \) and \( y_2 \) are the two successive peak amplitudes which are measured from the figure 5. Then the damping ratio value is determined as 0.28 (as mentioned before, because the experiment may not give the correct displacement result, the damping ratio value used here is still need to be verified). Substitute all the parameters into the equations and all the results can be calculated using Microsoft Excel. Table 2 lists the hand calculation results and the corresponding experimental results for reaction forces. From the table it can be seen that the reaction forces obtained from the impact tests and calculated in this section are correlated very well (with maximum error only about 5.5%), while the appearing times show some differences, which means that the simplified numerical model can represent the actual impact system in a certain level considering all the simplifications and assumptions taken in the modeling process.

### Table 2: The theoretical results and the experimental results for reaction force

<table>
<thead>
<tr>
<th>Initial position (in)</th>
<th>Peak force (lb)</th>
<th>Time (sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Experimental</td>
<td>Theoretical</td>
</tr>
<tr>
<td>3</td>
<td>311.69</td>
<td>317.01</td>
</tr>
<tr>
<td>4</td>
<td>370.72</td>
<td>373.96</td>
</tr>
<tr>
<td>6</td>
<td>483.15</td>
<td>456.46</td>
</tr>
</tbody>
</table>

For those impact cases that have not been done in the experiment, the maximum reaction force value can also be evaluated by using the hand calculation as well as computer simulation. For example, even the reaction force of dropping the bar from 10 inches high has not been measured because of the limitation of the load transducers, it still can be evaluated on the computer and be calculated using above equations as well. The computer model gives the maximum force was 608.94 lb at 0.002 second while the theoretical results is 587.26 lb at 0.003 second, the error of the maximum force value is only about 3.4%.

### Parametric Study

As demonstrated in theoretical solution, it can be seen that the system stiffness \( k \) and the system’s damping ratio \( \zeta \) are the two most important parameters, which determine the maximum reaction force and its appearing time. To find how they affect the results separately, different \( k \) and \( \zeta \) values are substituted into above equations, and related maximum reaction forces and times then can be calculated using Excel. For an
example, in the 6” dropping test, the stiffness is firstly kept unchanged, and the $\xi$ value is set to change from 0.05 to 0.5 with equal increment of 0.05 then calculate the maximum reaction forces and the appearing times. Figures 12 and 13 show the relationships of $\xi$ - time and $\xi$ - reaction force.

![Graph 1](image1.png)

**Figure 12. Relationship between damping ratio and peak value time**

![Graph 2](image2.png)

**Figure 13. Relationship between damping ratio and maximum reaction force**

From the figure 12 and 13, it can be seen that the increase in the $\xi$ will decrease the time, which means that the larger is the $\xi$, the sooner the maximum force appears. Nevertheless, the $\xi$ value does not obviously affect the maximum force. From figure 13, it can be found that the $\xi$-force plot is a smooth curve that varies gently. The peak force value goes down with the increasing $\xi$ till it reaches the lowest value at about 457 lb when the $\xi$ equals to 0.3 and then it rises up with the increasing $\xi$.

After that, the $\xi$ value is fixed at 0.28, as the value used in the theoretical calculation, and the stiffness value is changed from 1000 lb/in to 10000 lb/in with the same increment 1000 lb/in. Figures 14 and 15 show the relationship of k- time and k- reaction force.
From figure 14, it can be seen that similar to the damping ratio, the increase in the k will decrease the time too. However, unlike the damping ratio, when the stiffness increases from 1000 lb/in to 10000 lb/in, the maximum reaction force value also increases from about 200 lb to above 700 lb, which increases by 3.5 times.

Conclusions and Evaluations

Based on above demonstrations, it is concluded that the computer model can correctly predict the reaction force in nondestructive impact test. The hypothesis of the project is then testified through comparing the experimental results and computer results, and also is confirmed by theoretical evaluation. Tables 3, 4, 5 display the results of comparisons among the experimental results, the computer results and the theoretical results of the reaction forces of 3”, 4”, and 6” dropping tests. From these tables it can be seen that the experiment and computer model yield very close results including the force and the time, and the reaction force can also be verified through the theoretical solution.

<table>
<thead>
<tr>
<th>Table3. Reaction force of 3”dropping test</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental results</td>
</tr>
<tr>
<td>Computer results</td>
</tr>
<tr>
<td>Theoretical results</td>
</tr>
</tbody>
</table>
Table 4. Reaction force of 4” dropping test

<table>
<thead>
<tr>
<th></th>
<th>Maximum force (lb)</th>
<th>Error (%)</th>
<th>Time (sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental results</td>
<td>370.72</td>
<td>0.002</td>
<td>0.002</td>
</tr>
<tr>
<td>Computer results</td>
<td>367.87</td>
<td>-0.8</td>
<td>0.0024</td>
</tr>
<tr>
<td>Theoretical results</td>
<td>373.96</td>
<td>0.9</td>
<td>0.0030</td>
</tr>
</tbody>
</table>

Table 5. Reaction force of 6” dropping test

<table>
<thead>
<tr>
<th></th>
<th>Maximum force (lb)</th>
<th>Error (%)</th>
<th>Time (sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental results</td>
<td>483.15</td>
<td>0.002</td>
<td></td>
</tr>
<tr>
<td>Computer results</td>
<td>468</td>
<td>-3.1</td>
<td>0.002</td>
</tr>
<tr>
<td>Theoretical results</td>
<td>456.46</td>
<td>-5.5</td>
<td>0.0030</td>
</tr>
</tbody>
</table>

From above results, it can be concluded that the reaction forces obtained from the computer model, the experiment, and the theoretical solution correlate very well. As for the peak value time, the computer model and the experimental give very closed values and the differences caused by the theoretical solution may due to the simplifications and assumptions that have been taken during the calculations.

Through the experiments and the computer analysis, the behavior of the bumper under a low velocity impact can be determined. During the impact, the transferred force reaches its peak value shortly after the impact then reduces down and undergoes a series of smaller fluctuations until achieves to its static value.

Though the effectiveness of the computer model has been verified, there still exist some disadvantages in the project except the weakness of the experimental displacement plot. That is, as mentioned before, the computer model can expect the maximum reaction force well but it is not good in simulating the entire impact process. Therefore, the created computer model is still need to be modified and refined, and some advanced modeling techniques are required to improve the quality of the current computer model. On the other side, as presented in the paper, the force that transferred from the bumper to the frame under low velocity impact is studied, but how it is transferred into the entire vehicle? What its response might be under a destructive impact such as buckling? And is it possible for us to develop a generous modeling methodology that can be used for developing the computer model for any impact cases instead of only for vehicle crashworthiness? All these questions need to be answered in future works.

**References**

1. Abdullatif K. Zaouk, Nabih E. Bedewi, Cing-Dao Kan, Dhafer Marzougui, Development and evaluation of a C-1500 pick-up truck model for roadside hardware impact simulation.


14. Mario Paz, Structural dynamics (theory and computation).


17. V. I. Babitsky, Theory of vibro-impact systems and applications.

