Finite Element Simulation Study on Circular Concrete Bridge Piers Subjected to Vehicle Collision

by

Ee Cheng Ziong
15116

Dissertation submitted in partial fulfilment of
the requirements for the
Bachelor of Engineering (Hons)
(Civil)

MAY 2014

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzua

CERTIFICATION OF APPROVAL

Finite Element Simulation Study on Circular Concrete Bridge Piers Subjected to Vehicle Collision

by

Ee Cheng Ziong
15116

A project dissertation submitted to the
Civil Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(CIVIL)

Approved by,

____
(Dr. Teo Wee)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

May 2014

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

____

(EE CHENG ZIONG)

Abstract

Bridge piers collapse has been one of the potential hazards to human lives as well as economic loses. Numerous accident records show that bridge piers are vulnerable and produce a great threat due to accidental collision with heavy truck vehicles. Further study has suggested that the current code of assessment for bridge pier design has not taking into consideration of the implication of dynamic impact. The suitability of the code to reflect the exact collision condition of the pier is being reviewed. Thus, this research has been conducted to further investigate the effect of impact collision from the heavy truck vehicle to a bridge pier and subsequent consequences it may cause. Research is being carry out using finite element simulation, LS-DYNA and LS-PrePost, in order to conduct and analyze complex impact scenarios between the pier and the vehicle. The scope of this research is however, being narrowed down to only applying circular bridge pier designed as the object of impact and a verified model of Ford F800 Series 8-tons heavy truck as well as a HGV 16-ton Truck are used in the scenario. Different parameters will be further included to analyze the impact collision in depth details.

Acknowledgement

I would like to express my sincere appreciation to each and everyone who has guided me throughout the whole Final Year Project period in Universiti Teknologi PETRONAS (UTP).

Being accepted and recognized by Dr. Teo Wee, Senior Lecturer of the Civil Engineering Department in UTP, I am entrusted with the project title Finite Element Simulation Analyses on Circular Bridge Pier Subjected to Heavy Vehicle Collision. Together with my colleague, Kilian Lau, we are given many information and guidance form Dr. Teo himself. Before the project, Dr. Teo has provided us with the LS-DYNA and LS-PrePost software training sessions in order to ease our simulation study of the project. The host company of the LS-DYNA software is kind enough to assist us with the training as well as guiding us with any difficulties arose. Dr. Teo has also been trying his best to guide us with the project's objectives of generating the actual crash simulation, putting effort in table discussion with many other parties associated.

Apart from that, I would like to thank my senior, Yin Hor, a Civil Engineering Master student for assisting us during the project initialization. Yin Hor has helped a lot in terms of providing relevant information regarding the usage of LS-DYNA software.

Last but not least, I would like to show my token of appreciation to UTP for providing a platform for me to excel in my research and to gain my experience as well as knowledge of understanding in the field of study. UTP has offered and bought the license required for the verification of LS-DYNA software solely for our project progression. Thus, we are very grateful for being the first batch to come up with the simulation using this software provided.

Table of Contents

List of Figures

List of Tables

CHAPTER 1

INTRODUCTION

Studies have shown many catastrophic accidents occurred in the collision between heavy vehicles and bridge piers which caused many serious implications both in terms of loss of human lives and damage to the transportation system as well as economy. Those piers were initially designed according to the design and assessment codes which however only taking into account the static load and not concentrating on the accidental dynamic impacts. Thus, this research is initiated to investigate the relationship between the reinforced concrete bridge piers when subjected to heavy vehicles collision. The research will focus on the simulation using nonlinear finite element to record and analyze thoroughly every details during the whole impact scenario.

1. BACKGROUND

Many disastrous accidents in the past between heavy vehicles and bridge piers have cause the loss of human lives. One of the notable tragedies took place in the year of 2003, where a semitrailer crashed head on into the median support of a bridge crossing I-80 near Big Springs, Nebraska (El-Tawil et al, 2004). The bridge collapsed after the impact and killed a person while traffic was severely disrupted on May 23rd that day.

Another tragic accident took place on September 9th of 2002. According to Dallas-News, a tractor trailer hit a concrete support column at the highway and caused the collapse of the bridge. One person was killed during the accident and also affected the economy of the country since the bridge was built with a large amount of money.

From the information retrieved regarding the design of the piers, it was found that these piers were actually constructed using the same design codes which have not been amended for years. The reliability of the codes has seen to be uncertain since the design is only taking a few parts of design consideration and ignored most of the vital parts especially dynamic impact or shear failure of the pier.

2. PROBLEM STATEMENT

Most of the bridges in Malaysia are constructed in reinforced concrete and are almost similar to the ones in the accidents mentioned above. Thus, almost all of the bridges in Malaysia are actually facing a great threat towards dynamic impact since the construction follows the same code of practice.

In this research, the problems covered are as follow:

- Review of the current design and assessment codes on the design of bridge piers.
- Factors other than the equivalent static load to the dynamic impact in the code causing piers failure.
- Proper realistic simulation of the actual crash scenario using LS-DYNA for testing and analysis.

3. OBJECTIVE AND SCOPE OF STUDY

This research is initiated with intention to verify and analyze the consistency of the current code of practice for the design of piers and bridges in Malaysia. The constants defined in the code including the speed, mass and type of vehicles, load and reinforcement of the structures will be reviewed for the suit of current technology and traffic behavior in Malaysia. In addition, factors regarding the failure of the pier will be further investigated and discussed during the impact simulation.

Simulation of the exact crash scenario will be conducted using LS-DYNA software. This software analyzes the in-depth details of the crashing scenario thoroughly with the advantage of nonlinear finite element analysis. The crashing will involve an 8-tons heavy truck crashing into a circular fixed-end pier and the information regarding the impact will be computed and evaluated for further investigation.

CHAPTER 2

LITERATURE REVIEW

There are many different types of impact which are very complicated by means of various parameters such as the effect of higher modes of vibration, changes in the failure mode due to propagating stress waves, and localized damage as well as its effect on overall stability and strength. Throughout the years, not much concentration has been focused on the effect of heavy vehicle impact on structural members like bridge piers. Thus, it represents the motivation of this research being initiated. Impact simulation will be conducted instead of actual experiment work. Simulation will be executed using LS-DYNA, a finite element simulating software.

1. LS-DYNA AND LS-PREPOST SOFTWARE

i. LS-DYNA

LS-DYNA is a simulating software developed by the Livermore Software Technology Corporation (LSTC). The advanced general-purpose multiphysics simulation software contains more and more possibilities for the calculation of many complex, real world problems. In addition, its origins and core-competency still lie within highly nonlinear transient dynamic finite element analysis (FEA) using explicit time integration. LS-DYNA has been widely used in many cases such as automobile, aerospace, construction, military, manufacturing as well as bioengineering industries.

Dr. John O. Hallquist, the developer of the origin 3D FEA program for LS-DYNA, namely DYNA3D was created at Lawrence Livermore National Laboratory (LLNL). In the year of 1976, DYNA3D was invented in order to simulate the impact of the Full Fusing Option (FUFO) or nuclear bomb for low altitude release since at that time, none of the 3D or 2D software is capable of dealing with such simulating impact. DYNA3D used explicit time integration to study any nonlinear analysis problems, by concentrating the applications of being mostly stress analysis of the structures when undergoing different impact scenarios. Unfortunately during that time, the software is very limited in terms of its functionality due to the insufficient of computational resources. Since 1978, DYNA3D has been widely used by the public and many huge companies without restriction on various kinds of complex 3D simulations.

Over the years DYNA3D had been optimally improved and programmed in terms of processing, functionality, as well as sustainability. CRAT-1 supercomputers had been introduced to run and process the simulation and this breakthrough has successfully improve the time integration of the software simulation. Additional material models such as explosive-structure and soil-structure interactions had also been released and permitted the analysis of structural response due to penetrating projectiles. Hall quiest was the sole developer of DYNA3D until the joint venture of Dr. David J. Benson in the year of 1984. Since then, many features and capabilities were added including beams, shells, rigid bodies, single surface contact, interface friction, discrete springs and dampers, optional hourglass treatments, optional exact volume integration, and other operating system compatibility like UNIX, COS and so on.

By the end of the year 1988, Livermore Software Technology Corporation (LSTC) was founded to continue the development of DTNA3D in a much more focused manner. This is when the name of LS-DYNA been introduced after being shortened from LS-DYNA3D. LSTC has greatly expanded the capabilities of LS-DYNA in an attempt to create a universal tool for most simulation needs.

The typical uses of LS-DYNA can be expressed in terms of nonlinear and transient dynamics manner.

i. Nonlinear means at least one of the following complications:

- Changing boundary conditions
- Large deformations
- Nonlinear materials that do not exhibit ideally elastic behavior

ii. Transient dynamic means analyzing high speed, short duration events where inertial forces are important.

- Automotive crash
- Explosions
- Manufacturing

LS-DYNA works in a simpler way than the time of DYNA3D now. It consists of a single executable file and is entirely command-line driven. And thus, the required conditions to run LS-DYNA is a command shell, the executable, and input file, and enough free disk space to run the calculations as output files. The input files are actually in simple ASCII format and can be easily prepared using any text editor. However, input files are usually prepared using the aid of a graphical preprocessor. Many third-party software products are available for preprocessing LS-DYNA input files now including TrueGrid. LSTC on the other hand has also developed its own preprocessor, LS-PrePost which is freely distributed and runs without the necessity of a valid license. LS-DYNA however, will require a valid license which will then allow the users to access to all of the program's capabilities, from simple linear static mechanical analysis up to advanced thermal and flow solving methods.

LS-DYNA has many potential applications and can be tailored to many fields. It is not limited to any particular type of simulation. LS-DYNA's various features can be combined to model a wide variety of physical events. One of the widely knows example of simulation processed which involves a unique combination of features is the NASA JPL Mars Pathfinder landing which simulated the space probe's use of airbags to aid in its landing. Below are some other capabilities of LS-DYNA:

- Full 2D & 3D capabilities
- Rigid body dynamics
- Nonlinear dynamics
- Linear statics

ii. LS-PrePost

LS-PrePost is an advanced pre and post-processor and model editor from LSTC that is delivered free with LS-DYNA, preparing input data and processing results from and in concurrence with LS-DYNA analyses. The user interface is designed to be both efficient and intuitive. LS-PrePost runs on Windows, Linux, and Unix utilizing OpenGL graphics to achieve fast rendering and XY plotting.

LS-PrePost is particularly capable of importing, editing and exporting LS-DYNA keyword files for generating LS-DYNA input files. This software has been constantly being further developed and new development trends can be incorporated quickly. The software can be installed and implemented on every computer since free license is offered by DYNAmore.

LS-PrePost offers the following features:

- Pre-Processing Features
  - ✓ Import of Nastran-, IGES-, VDA0, I-DEAS-Universal- and Step-files possible
  - ✓ Mesh generation for 2D mesh Sketchboard, nLine Meshing, Tet-Meshing, Automatic surface meshing, and meshing of simple geometric objects
  - ✓ Special applications like metal forming, ALE, and model check
  - ✓ LS-DYNA entity creation such as coordinate systems, sets, parts, masses, CNRBs, SPC's, initial velocity, and accelerometers.
- Post-Processing Features
  - ✓ 3D animation
  - ✓ Eigenmoden animation
  - ✓ BINOUT processing
  - ✓ X-Y plots
  - ✓ Vector Plots
  - ✓ Fringe plots
  - ✓ ASCII plotting
  - ✓ Section analysis

2. CLASSIFICATION OF IMPACT

According to Miyamoto et al (1991), impact can generally be classified into 2 parts, soft impact and hard impact. Soft impact is defined as an impact where the kinetic energy from the impact is absorbed by the plastic deformation in the striking body. In this case, the velocity of the impact will be lower and the propagation of stress waves after the impact can be neglected. As for the failure mechanism of the soft impact, it is as similar as those associated with static loading. On the other hand, hard impact occurs when the kinetic energy from the impact is completely absorbed by the stuck body and hence, the striking body will suffer lesser deformation. During this event, the velocity is generally high and complicated stress waves propagate through the struck body and will eventually lead to severe failure.

	Impact Phenomenon	Type of Impact
Single Impulsive Blow	Vehicular collisions onto handrails of expressways or freeways	Soft
	Ship or vehicular collision onto bridge piers	Soft
	Ship collision onto offshore structures or gravity platforms for oil extraction	Soft
	Aircraft collision onto nuclear power plants	Soft
	Slow speed vehicle collision with columns in multistory car parking garages	Soft
	Explosions on concrete structures	Hard
Repeated Impulsive Blows	Blows from car tires across expansion joints	Soft
	Rocks falling onto roof of protection shelters in mountainous regions	Soft
	Blows on concrete piles during hydraulic piling	Soft/hard
	Ship or iceberg brushing against offshore structures or gravity platforms	Soft
	Meteorites falling onto concrete lunar structures (in future)	Soft

TABLE 1: Classification of Impact

Based on the definition in Table 1 by Miyamoto et al (1991), vehicular collision on to reinforced concrete bridge piers can be classified as either soft or hard impact since in this crashing, both bodies will show the sign of taking damage and deform.

3. VEHICLE MODELS USED IN IMPACT SIMULATION

Apparently, there have been many experiments and investigations regarding automobile impact simulation. Models of heavy vehicles have been widely developed and used in many countries. Some of the investigations including impact between vehicle and rigid structures (Mahmood et al 1996 and Zaouk et al 1996), vehicle and flexible structures (Rierra 1982, Brandes 1982, King and Miyamoto 1994, and Miyamoto et al 1984), as well as vehicle on other vehicles (Bedewi et al 1995 and Nicholson and Moraes 2001) have been conducted in the past. Types of vehicle models used nowadays can be classified into two categories:

- Mass, Spring, and Dashpot (MSD) Models
- Finite Element (FE) Models

In general, FE models are much more widely used nowadays compared to MSD models for several valid reasons. MSD models are a system of lumped masses interconnected by nonlinear visco-elasto-plastic axial and rotational springs. Technically, these models are much simpler to simulate and formulate. However, extensive calibration is required in order to yield a more realistic result. These on the other hand, can be done by using FE models for simulation since the technology today has become more powerful and affordable. FE models have more flexibility in terms of texture and joints as well as contact surfaces. FE simulation is capable of computing the in depth details of the whole crashing scenario, although it is a lot more time consuming as compared to using the MSD models.

iii. Ford F800 Series Truck (8-ton)

Vehicle model Ford F800 Series Truck is selected as representative of Malaysia's 8 ton heavy truck vehicle category. This vehicle is chosen because majority of the heavy vehicle in Malaysia has more similarities to F800 than differences. The chassis are all parallel-rail frame types with front and rear leaf spring suspension. V-8 diesel engines are attached including the dual-wheel rear axles. F800 also has mounted cargo on a series of lateral C- or I-Beams which are welded to C-Channels that run parallel and directly atop the chassis main frames. The Cargo-body C-Channels are fasten with large U-Bolts to the chassis main rails.

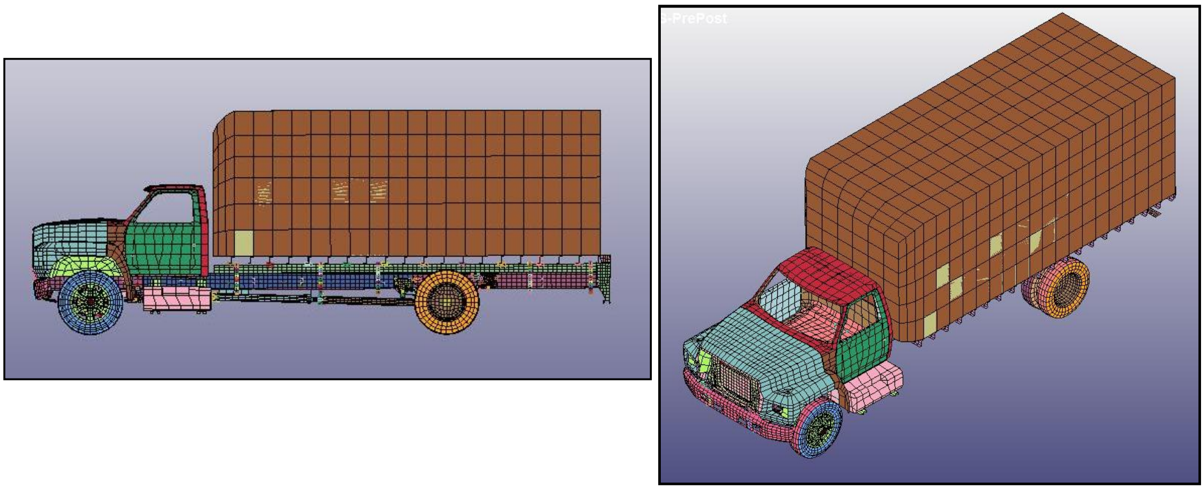


FIGURE 1: Ford F800 Series Truck

This model of truck has been validated and updated from time to time by the National Crash Analysis Center (NCAC). The elements and materials used in the model are broken into many types and behavior similar to a real truck. Material properties of the truck are vital to produce a realistic result. Even though a lot of elements or parts being applied to the truck, without a realistically modeled material behavior, the model will not perform closely to a real truck.

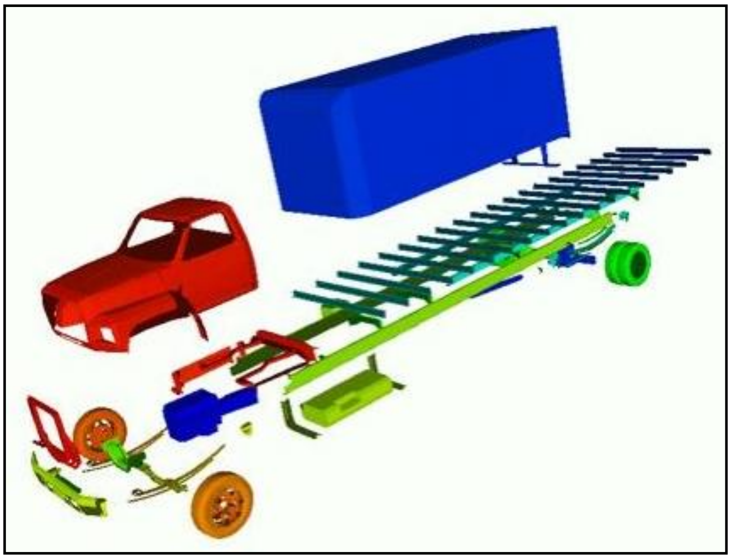


FIGURE 2: Breakdown of F800 Truck in Accordance to respective Material Behavior

The model is built mainly on a longitudinal rail structure that acts as its backbone. Thus, the rail has to be accurately modeled in terms of geometry and material. The material used is specified in the Service Manual as the high Strength Low Alloy steel of yield point 350 MPa. The material model includes the effects of strain rate sensitivity especially for steel. Mild low-carbon steel is primarily used in body structure and by volume, constitutes the largest part of the vehicle. Other parts of the model has been designed as elastic and plastic material of each elements. Detailed material behavior has been included in even the joint of every elements. This is although very tedious but is especially essential in order to create a realistic model.

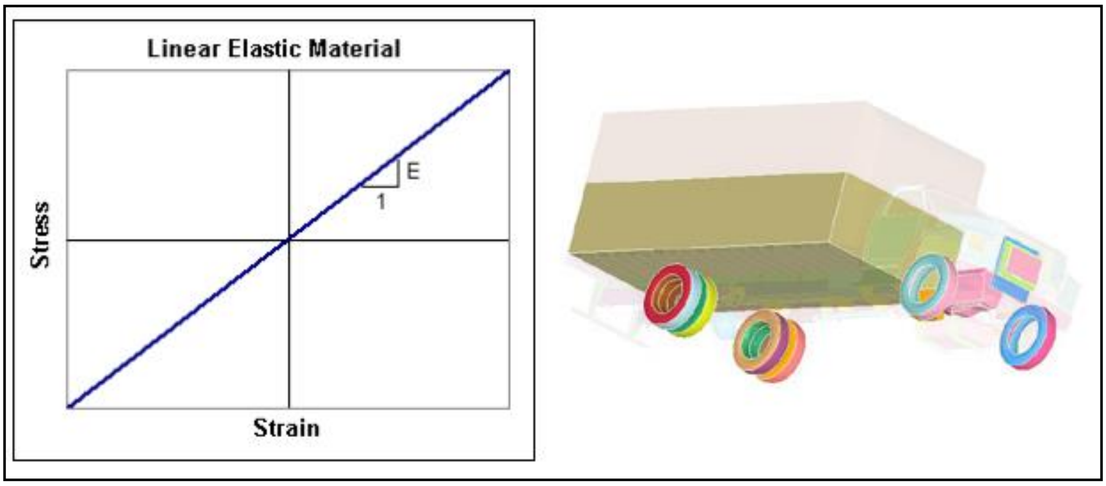


FIGURE 3: Elastic Material Behavior of F800 Truck

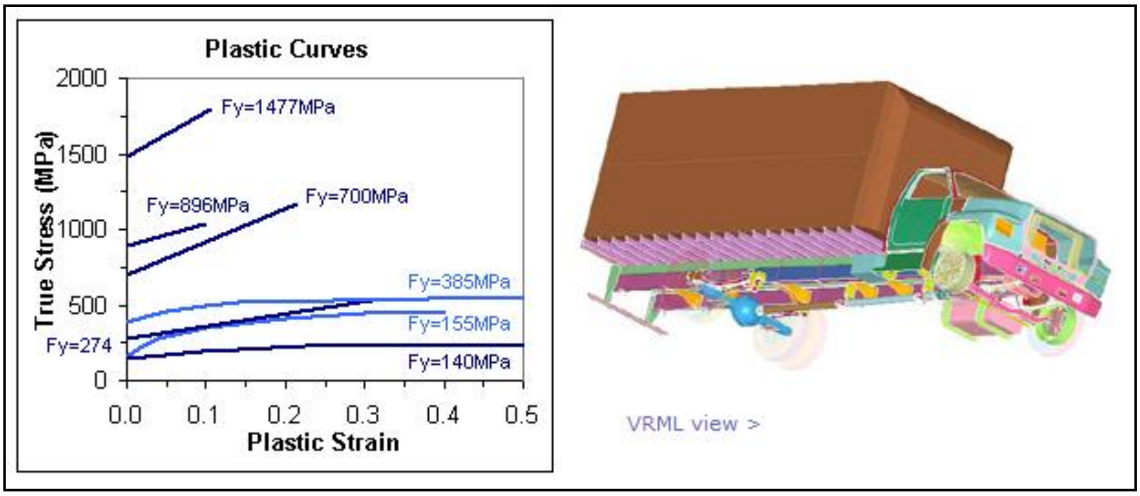


FIGURE 4: Plastic Material Behavior of F800 Truck

Additional and detailed material properties as well as behaviors of other parts and elements of the truck are presented in the Appendix (1).

iv. HGV 16-ton Truck

The second truck chosen for the simulation is the Heavy Good Vehicle (HGV) 16-ton Truck. This truck is modeled according to the requirements of the European standard EN 1317 with regard to both dimensions as well as weight. HGV 16-ton Truck is chosen because it resembles and complies the category of the heavy truck widely used in Malaysia. Similar to Ford F800 Series Truck, HGV 16-ton Truck is also a pre-modeled finite element vehicle retrieved from NCAC library. As such, this vehicle is almost reliable for most of the impact simulations conducted in this research.

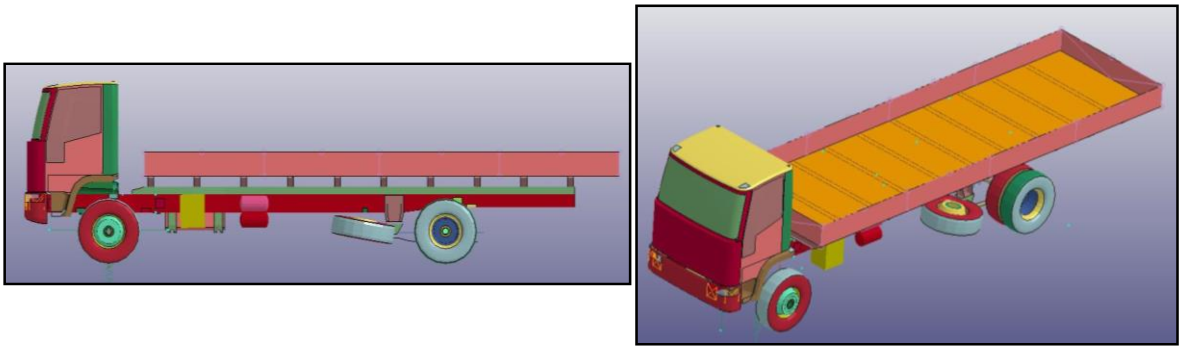


FIGURE 5: HGV 16-ton Truck

As illustrated in the figure above, the colors of the vehicle represents different parts being assembled together. These parts are classified into many different types of material properties used. With each and every one of the parts assigned to specific characteristics, the whole truck will react and perform as a realistic visualized truck which is closed to the actual one. According to NCAC, all the parts of the truck have been experimented and verified closest to the actual truck. Although most of the parts are not yet being validated from the main source, this model is still sufficient for the purpose of simulating in this research.

Massive parts of the truck like engine and wheel hub were modelled using solid elements, while most of the parts were modeled using shell elements. In order to avoid unrealistic zero-energy modes of deformation, hourglass control on specific parts was defined. The other different parts of the model were connected using merging coincident nodes, spot-weld elements, and defining joint constraints or nodal rigid bodies. Most parts of the truck are made of steel. This material was characterized using Material #24 in LS-DYNA which is the elastic piecewise linear plasticity with the definition of a stress-strain curve for the plastic field. Two different steel alloys with specific stress-strain curve and yield stress were considered. The yield stress for the first steel alloy which is characterized as non-structural parts of the truck is 450 Mpa while the second alloy which was used for structural parts has yield stress of 610 Mpa. For more realistic behavior of the truck, the windshield and windows were modelled using as elastic piecewise linear plasticity material model. Material #1 was defined for the tires as a linear elastic material model. Parts of the truck that undergo negligible deformations like wheel hubs or the engine, were defined as rigid, Material #20 in LS-DYNA definition.

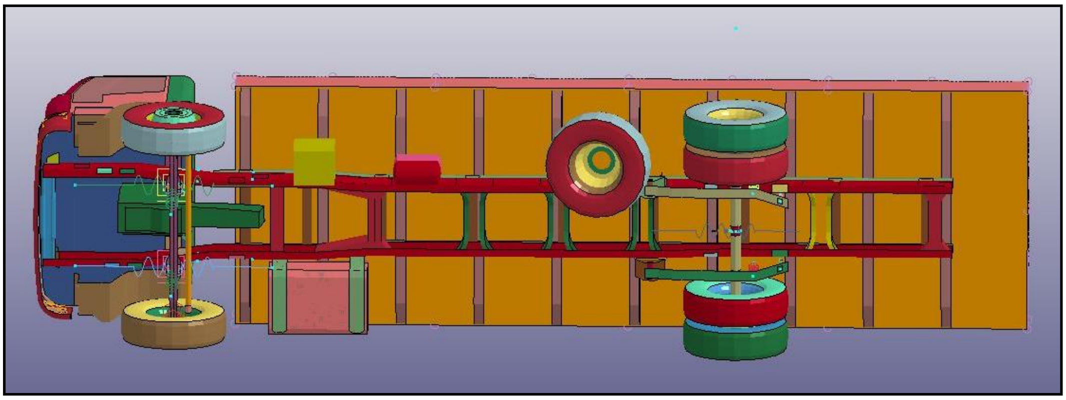


FIGURE 6: Bottom View of HGV 16-ton Truck

To make the truck more realistic in the simulation, some major and explicit parts of the HGV 16-ton truck are modelled in much deeper detail.

- Frame:

  This truck contains the frame of two C-cross-section side members linked one to the other by transverse members. The connection used to replace the rivets or bolts for the members is a spot-weld elements with no failure criteria. This frame is crucial as it acts as a backbone for the truck which transfer the forces throughout the whole truck.

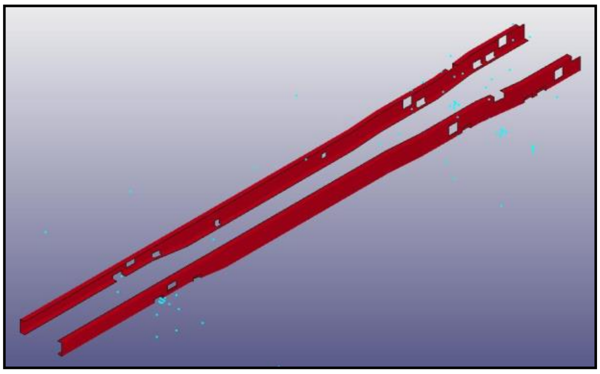


FIGURE 7: Frame of HGV 16-ton Truck

- Tyres:

  The tyres are modelled with shell elements of different thickness with regard to the tread and the rubber walls. Elements having the airbag properties were used to control the volume contained between tyre and rim. Tyres must be modelled properly so that the truck will exert the exact force proportional to the speed assigned.

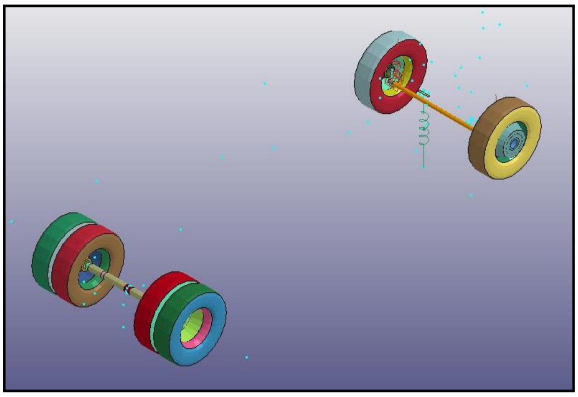


FIGURE 8: Tyres of HGV 16-ton Truck

- Driving Cabin:

The modelled cabin which is assigned above the engine is chosen. During previous studies, it is found during the preliminary simulations, the lack of glasses caused excessive deformations of the driving cabin, also windshield and lateral windows were modelled. The interior of the cabin is less of interest and as such, rough modeling and sufficient information for the impact collision simulation is assigned to the part mentioned.

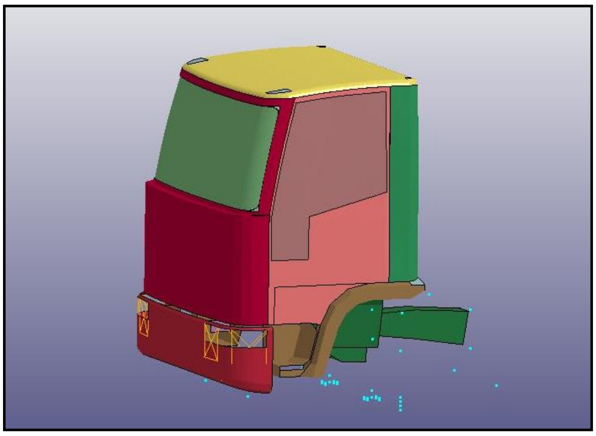


FIGURE 9: Driving Cabin of HGV 16-ton Truck

- Ballast and Inertial Properties:

The ballast is modelled as discrete mass-elements in different locations of the FE truck model. These locations consists of rear axle (differential), front axle (steering equipment), frame (fuel and air tanks), and driving cabin.

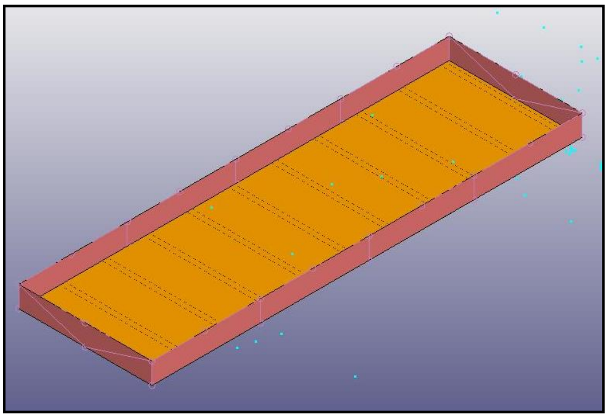


FIGURE 10: Ballast Section of HGV 16-ton Truck

4. STRENGTH OF BRIDGE PIERS

According to C. Eugene Buth et al (2010) research, when bridge piers are subjected to impact by a heavy vehicle, the piers are typically subjected to large shear and bending forces. These are one of the factors that cause the major structural failure of the piers. In most of the investigations done by their research, they found that most of the structural failure in the bridge columns occurred as a result of the impact. Generally, when collision occurred between a heavy truck with the pier, the force is relatively close to the ground surface. Although a large bending force is applied to the pier, the truck collision caused large shear force which exceeds the shear capacity of the pier, resulting in shear failure mechanism of the pier.

C. Eugene Buth et al (2010) has also shown the structural analyses on several piers impacted during a collision. It was informed that the nominal shear strength of each pier was calculated using the AASHTO LRFD Bridge Design Specifications, Fourth Edition, 2007. Throughout their investigation, they found that the failure mechanism was based on two failure planes resisting the force. These plans radiate approximately 45 degrees from the applied impact force as shown in Figure 1 below.

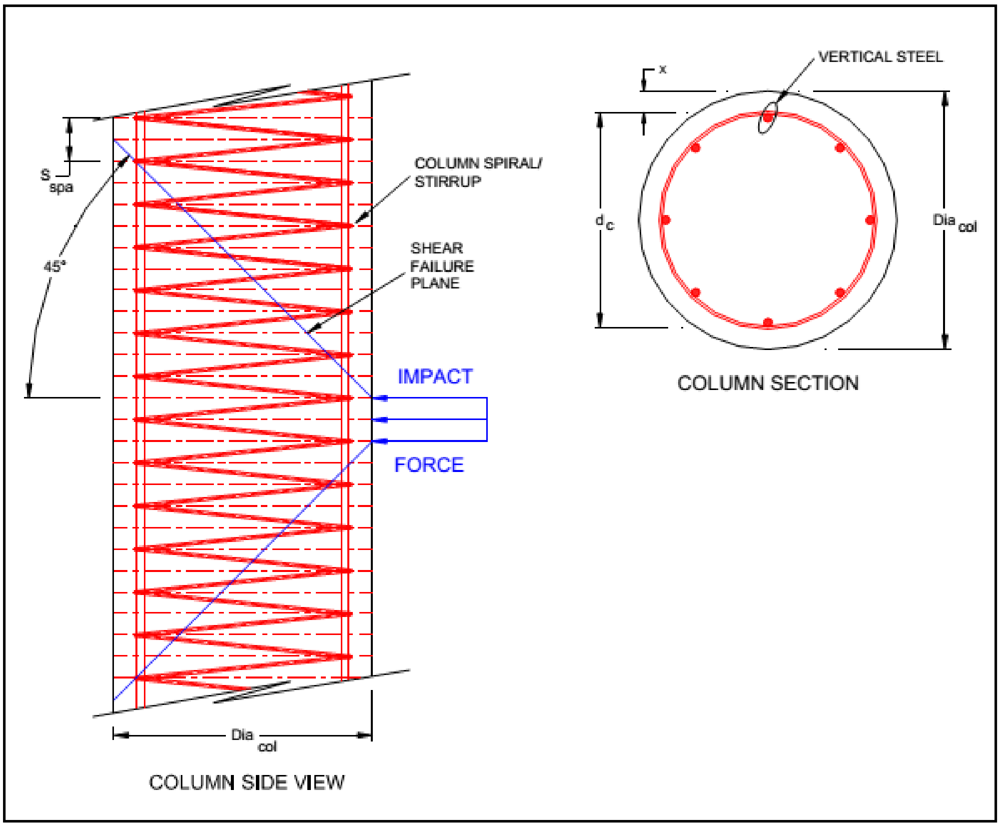


FIGURE 11: Failure Mechanism from Impact Force on Pier

The failure mechanism shown was observed in many piers which were impacted by large trucks. Thus, it was concluded that this research is to be concentrated on these planes of failure in order to reduce the chance of bridge being collapse in a collision. Simulation studies will proceed with expected failure mechanism as shown in the research done above.

5. ADDITIONAL IMPACT RELATED RESEARCH

Many researches have been conducted regarding collision between vehicle and structure. One of the researches done by El-Tawil et al (2004) is regarding the vehicle collision with different specification of bridge piers as stated in the AASHTO-LRFD design. The research was conducted using two truck models, a 14-kN Chevy truck and a 66-kN Ford truck. These truck models were crashed at various speed approaches into finite element of pier models with various structural properties. From the result of the simulation, parameters including stress and strain at key location, pier, foundation and superstructure deformations, and transient impact forces. Besides these important parameters, this study also suggested that accuracy of the project should be increased by carrying out some exercises on the simulations. Such exercises include extensive sensitivity studies, mesh refinement studies, momentum conservative checks and monitoring hourglass control energy during the crash simulation.

Another research done by Y. Sha and H. Hao (2012) on the impact between a barge with the bridge pier has shown that the current code of design used to construct that pier has limitation and needs to be amended immediately. The simulation of the impact scenario was presented using LS-DYNA where finite element was adopted as the pier and barge models. According to the research, previous studies of impact simulations were not complete or realistic because those studies used imperfect models. Thus, the paper suggested of using finite element orientated software like LS-DYNA for a proper and more idealized simulation. From the barge-pier impact, force time history, barge crush depth and pier displacements are the focus. These results were then computed to generate a reliable design of bridge pier.

CHAPTER 3

METHODOLOGY

This research will be done in two phases. Stage 1 basically covers the collection of the data and information required for the simulation. Since LS-DYNA is fresh and heavy software, effort will be spent on it to familiarize the limit and function of LS-DYNA simulation. After completing the first stage, simulation will be initiated in the second stage where the impact scenario will be assessed and result will be studied. In either stage, planning ahead is vital since the time consumption for the LS-DYNA to process the simulation is very long.

In phase 1, LS-PrePost will be used to modify and model the 8-tons nonlinear finite element heavy vehicle and the circular bridge pier. LS-Prepost is part of the modeling sub-software of LS-DYNA. The original model of the 8-tons vehicle will be acquired from National Crash Analysis Center (NCAC) and modification on the model will be done to cope with Malaysia's standard of heavy vehicle classification. As for the crashing structure, a rigid circular pier will be modeled. This pier will be submitted into the vehicle model to create a proper and realistic impact scenario taking into consideration on the speed limit and angle of collision.

Once the crashing scenario has been tested and completed, stage 2 will be initiated. In this phase, the crashing simulation will be run at full scale. Bridge pier will now be modeled with every necessary detail in it including the reinforcements. Upon running the simulation in different desired conditions, the result of the impact forces, time strain, stress level and other factors are assessed. These results will then be compared with the current design code in the country for further discussion.

Theoretical calculation of the impact force will be determined. This result will compare with the impact force acquired from the simulation and analyze whether the result is verified or otherwise. Such methodology is to assure that the sole result obtained from the simulation is reliable and close to the actual theoretical assumption. And thus, to be able to compare the results, the simulation must be modelled as close to the actual scenario as possible for the best outcome.

1. MODEL DEVELOPMENT OF HEAVY VEHICLE

Heavy truck vehicles are very complex in finite element detailing. Every details of the vehicle must be close to perfection in order to build a realistic simulation during the impact. Materials used in constructing the model have to be verified so that the properties are close to the actual truck. Due to time constraint on modeling the detailed visual vehicle, some of the models which are close to the classification of Malaysia's standard vehicle are acquired for this research.

Both the trucks have been further improved and modified in terms of suitability of the realistic scenario creation. From the genuine truck file, addition details regarding the route of velocity have been added as shown in the figure below. The route is essential and will act as a restraint to prevent and keep all the vehicle parts in places as they will be throughout the simulation.

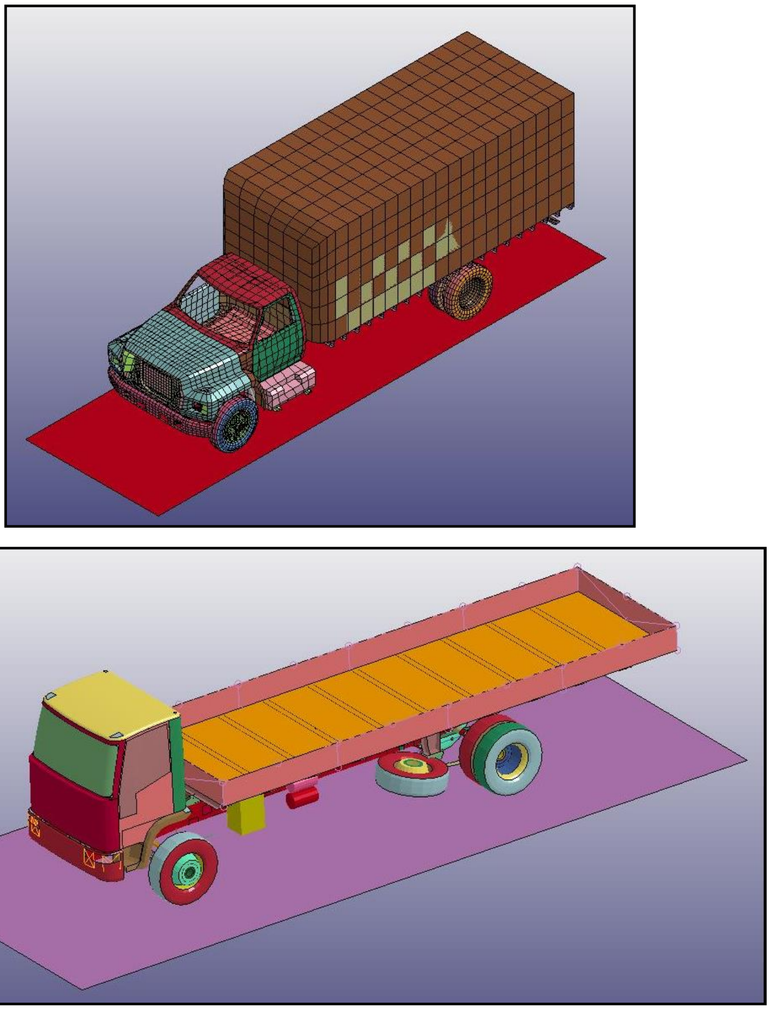


FIGURE 12: Route of Truck Movement is added

In LS-DYNA, the contact of collision must be identify in order to record the details or forces required. In this simulation, the whole truck will be highlighted and be included into the simulating process for the best outcome. This is because although selecting only certain crucial parts will save up some time in processing, the outcome or the output result is not so satisfying considering some parts of the truck is not included for force transmitting. Thus, including the whole truck model into the contact will assure a better result as well as more realistic impact simulation.

Figure below shows the parts selected for both the trucks in this research simulation processing.

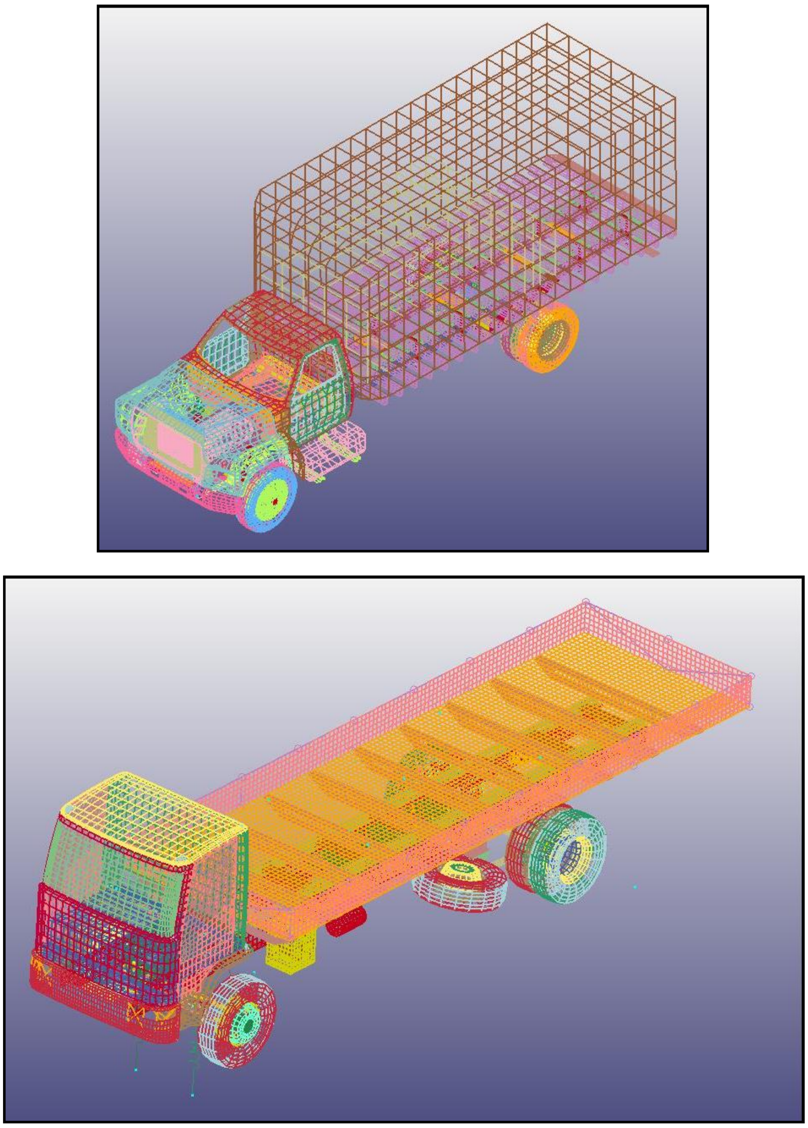


FIGURE 13: Parts of the Truck Model Considered in Collision Impact

2. MODEL DEVELOPMENT OF BRIDGE PIER

The pier used in this research will be circular in shape. Before replacing the rigid pier with reinforced concrete pier, the scenario must be first be determined. By applying rigid pier which is totally solid and will not deform, including rigid material property, simulations are tested. The rigid pier is placed right in front of the truck model so that the time of processing can be minimized as possible.

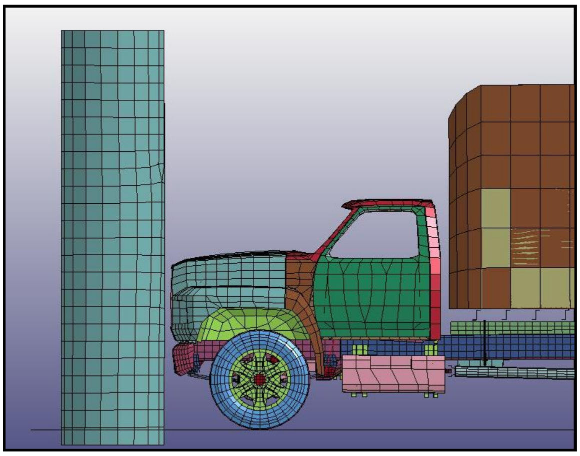


FIGURE 14: Position of Rigid Pier

The pier boundary is fixed at both end, on top and bottom, so that it will not be displaced during collision. Throughout the simulation, it is expected that the bridge pier will not deform or whatsoever and the detail regarding the forces onto the pier might not be accurate of realistic due to the rigid material property of the pier. This is when the pier will be replaced once the scenario has been determined, in order to acquire the forces and other information necessary.

Since the experimental structure in this research is the pier, it will have to be manually designed and modeled out. The cross-sectional of the circular pier will be 1000mm in diameter and approximately 7m in height in accordance to the height of the 8-tons truck. One end of the pier will be fixed at the bottom and the other will be pin supported at the top. Below is the figure showing the detail of the pier model to idealize the real scenario.

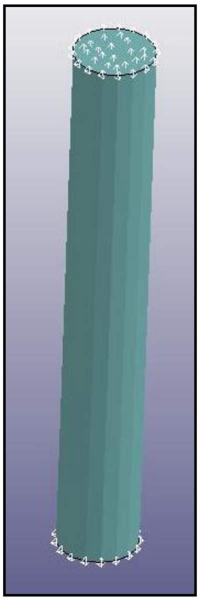


FIGURE 15: Pier with White Dotted Representing Restrain Assigned

3. CONTACT KEYWORD ASSIGNED

After assigning the parts for the contact, both the pier and the truck will then be assigned with respective experimental contact keyword to determine the behavior of collision. Three contact types will be tested:

- CONTACT_AUTOMATIC_GENERAL
- CONTACT_AUTOMATIC_SINGLE_SURFACE
- CONTACT_AUTOMATIC_SURFACE_TO_SURFACE

All these three contact types are often used for crash analysis. According to previous researches, assigning ideal contact keyword to the simulation is very crucial in order to obtain the optimum result. In addition, contact definitions also define whether the simulation will react as a more realistic scenario and also help to prevent errors in running the simulation.

Both the CONTACT_AUTOMATIC_GENERAL and CONTACT_AUTOMATIC_ SINGLE_SURFACE are defined as one single-surface-type contact that includes all parts which may interact during the crash event. With these types of contact, the slave surface which is one of the parameter to be defined in the keyword, is typically defined as a list of part ID's. No master surface needs to be defined. Contact is considered between all the parts in the slave list, including self-contact of each part. If the model is accurately defined, these contact types are very reliable and accurate. However, if there is a lot of interpenetrations in the initial configuration, energy balances may show either a growth or decay of energy as the calculation proceeds.

CONTACT_AUTOMATIC_SINGLE_SURFACE is generally much more recommended to be used in crash analysis since it is the most popular contact option. This contact option has been widely used in many complicated projects and being improved from time to time for its suitability of a proper and realistic crash analysis. However, limitation will not be settled here since the contact options in LS-DYNA have more and each of them has unique and reliable characteristics.

CONTACT_AUTOMATIC_GENERAL is basically a contact treatment with option similar to type 13 (in LS-DYNA). The main difference was that three possible contact segments, rather than just two, were stored for each slave node. The main feature of the GENERAL option is that shell edge-to-edge and beam-to-beam contact is treated automatically. All free edges of the shells and all beam elements are checked for contact with other free edges and beams.

CONTACT_AUTOMATIC_SURFACE_TO_SURFACE is a total different contact entity from that of the two mentioned. This contact option considered both the slave part and master part for simulation instead of just one sided. Thus, both the slave and the master parameter need to be assigned to any parts regardless of which part to which parameter. For this research, the whole truck part set will be assigned to master part ID while the pier will be assigned to the slave part ID for uniformity. In addition, this contact option has the ability of making the simulation more reliable and processing with consideration of more aspects as compared to the two single surface contact entities mentioned. Thus, this contact option will also be taken into consideration and to be tested whether it is suitable for the simulation in this research.

4. MATERIAL KEYWORD ASSIGNED

Since the material properties of the trucks have already been pre-defined, this focus will be put on the properties and behavior of the pier. Initially, the pier is designed as a rigid structure in order to create a better understanding with the simulation and produce an actual impact collision scenario between the pier and the trucks. After the scenario has been confirmed, the pier can then be designed as a concrete pier by assigning a proper material property to it. There are generally two types of material keyword to be tested.

- MAT_ELASTIC (#001)
- MAT_CSCM_CONCRETE (#159)

Both of these material entities have similar behavior close to a concrete bridge pier if and only if the parameters in each of these entities are assigned correctly. Among all the keywords in material for LS-DYNA, these two have been chosen because they are more closely related to the concrete pier property that is going to be used in this simulation. According to researches, material entities used must be correctly assigned in order to achieve optimum forces and stresses distribution.

MAT_ELASTIC is an isotropic hypo elastic material and is available for beam, shell and solid elements in LS-DYNA. Although the specialization of this material allows the modeling of fluids in general, with the parameters option provided, this material will enable this research to be assigned as a concrete material as well. Material keyword as such having parameter similar to concrete property like mass density, Young’s modulus and also Poisson’s ratio can be manipulated to the suitability of the impact collision scenario. It is however, according to the manual of LS-DYNA, this hypo elastic material model may not be stable for finite strains. If large strains are expected, a hyper elastic material model should be used instead.

MAT_CSCM_CONCRETE on the other hand is another material entity that is closely related to a concrete pier property as well. This material is a smooth or continuous surface cap model and is available for solid elements in LS-DYNA. Since it is a preset of concrete keyword, only strength of the concrete is needed to be input into the parameter in order to process it. Although it seems simpler than that of using MAT_ELASTIC, MAT_CSCM_CONCRETE has actually more complex parameters to be included into the pier property. This has been one of the factor why MAT_CSCM_CONCRETE has not been widely used in most simulations due to its complexity. Despite such condition, this material keyword is also taken into consideration in this research for better outcome.

5. KEY PROJECT MILESTONE

No.	Detail / Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Finalize Simulation Scenario														
2	Define Keywords														
3	Improve Simulation Progress														
4	Identify Parameters														
5	Run Simulation with Different Velocity														
6	Run Simulation with Different Trucks (Force)														
7	Run Simulation with Different Pier Strength														
8	Analyze and Compare all the Results														
9	Finalize Results														

Key Milestone

6. GANTT CHART AND KEY MILESTONE (FYP TIMELINE)

No.	Detail / Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Selection of Project Topic														
2	Preliminary Research Work														
3	Submission of Extended Proposal														
4	Proposal Defense														
5	Project Work Continues														
6	Submission of Interim Draft Report														
7	Submission of Interim Report														

Key Milestone

Process

No.	Detail / Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
9	Project Work Continues														
10	Submission of Progress Report														
11	Project Work Continues														
12	Pre-SEDEX														
13	Submission of Draft Final Report														
14	Submission of Dissertation (soft bound)														
15	Submission of Technical Paper														
16	Viva														
17	Submission of Project Dissertation (Hard Bound)														

Key Milestone

Process

CHAPTER 4

RESULTS AND DISCUSSION

1. GENERATING SIMULATION SCENARIO

To create the most realistic impact simulation, focus is emphasized on using the Ford F800 Series Truck and HGV to be collided with the bridge pier.

The simulation has been successfully conducted using the Ford F800 Series Truck and a pre-designed rigid pier. Simulation has progression almost similar to the actual impact between the pier and the truck. As expected earlier, the bridge pier will not record any damage or deformation since it is entirely rigid body material. As such, forces acting on the pier will not be as accurate at this stage.

The simulation is using CONTACT_AUTOMATIC_SURFACE_TO_SURFACE after several trials of testing with other contact entities, CONTACT_AUTOMATIC _SINGLE_SURFACE and CONTACT_AUTOMATIC_GENERAL. It is found that this contact will give the most closely related result and realistic impact scenario for this research. And thus, by applying the slave and master part set respectively, manipulating other parameter to required standard with proper units assigned, simulation is successfully conducted and results are recorded by using rigid material for the pier.

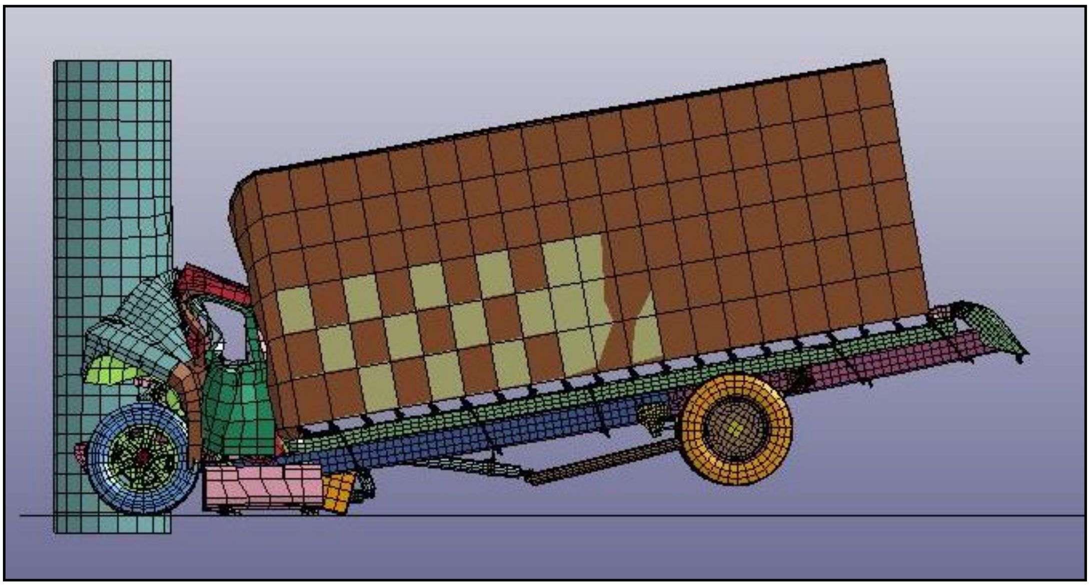


FIGURE 16: Impact Collision between Bridge Pier and F800 Truck

The truck on the other hand, contains the requested computed information according to the input by user. Stress and strain of the materials are recorded throughout the simulation and these detailed results are recorded in terms of finite element analysis. Energy and forces transferred in the whole truck model is also recorded and plotted in the graphs.

This simulation is conducted in a timeframe of 1 second. In other words, the truck takes less than 1 second to hit head on with the rigid pier. This however, takes the server a minimum of 2 days to compute and simulate the whole event due to various in-depth details of analysis being recorded during the impact, element by element.

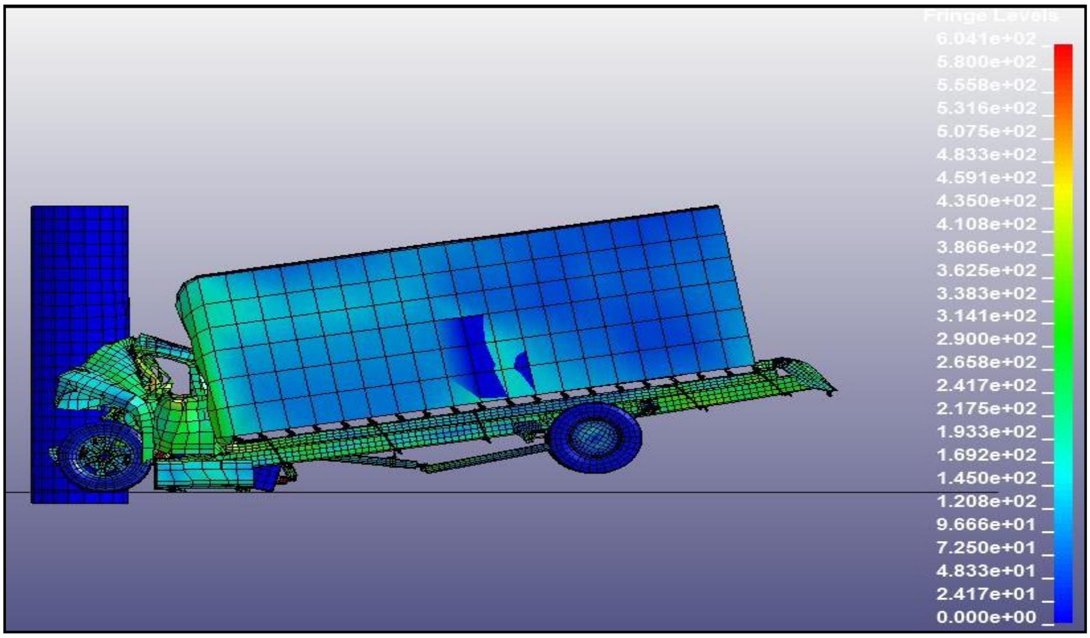


FIGURE 17: Von Mises Stress Analysis of the Truck

Figure above shows the stresses experienced by the truck model and the rigid pier. It is obviously shown that the rigid pier does not record any energy changes at all. As for the truck model, the stresses undergone by all the different parts are recorded accordingly. The highest stress recorded in the simulation is within the rail or the backbone of the truck as well as some components in the front part. The rail experienced the highest stress due to the material property given which is 385 MPa, higher than most of the major parts in the model.

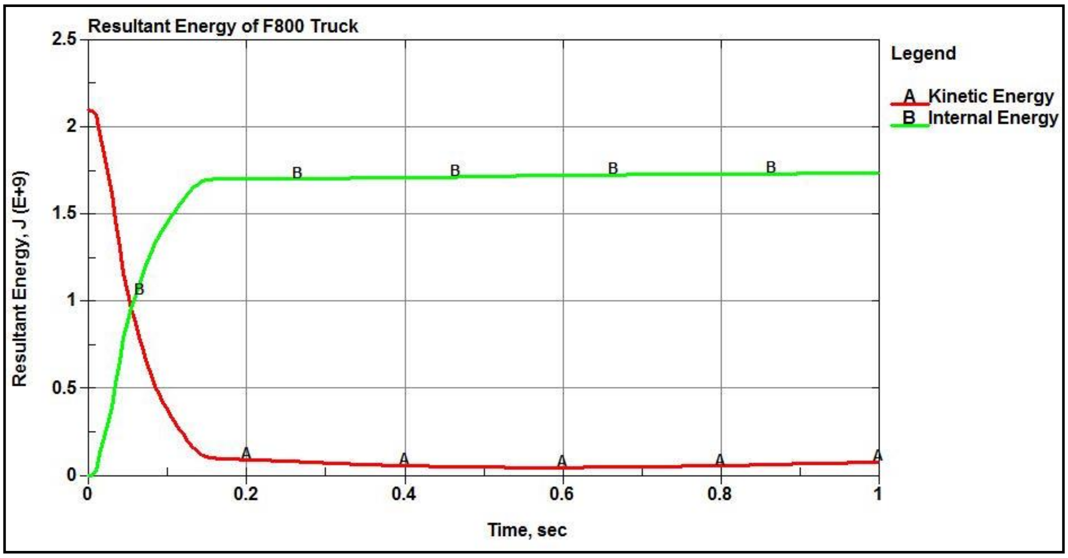


FIGURE 18: Resultant Energy Graph of F800 Truck

FIGURE 12 indicates the energy conversation of the truck during the collision. The kinetic energy from the truck is converted into internal energy, which means the energy being transferred is valid throughout the simulation. However, due to the rigid pier which will not record any detail during the collision, the kinetic energy is reflected fully back to the truck itself instead of onto the pier. This is why the internal energy of the truck is almost similar to the initial kinetic energy. Nevertheless, the simulation is a success knowing that the energy transferred through the truck is verified.

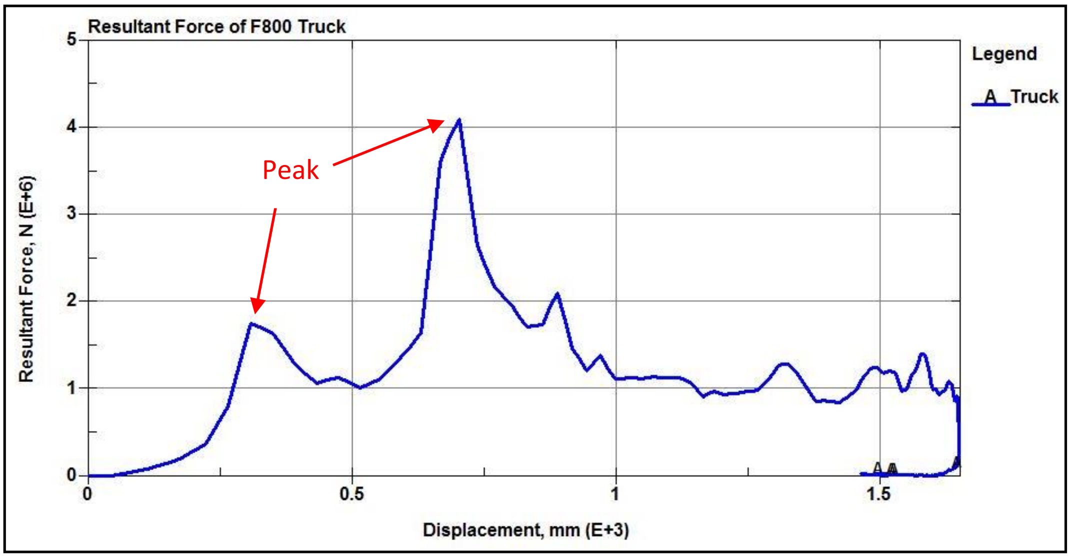


FIGURE 19: Resultant Force Graph of F800 Truck

As indicated in the graph above, the global force in the simulation is shown. There are two peaks, although not significant, showing the impact of the truck during the collision. First peak indicates that the front part of the truck is being hit head on at the rigid pier. Second peak shows the lagging of second impact which comes from the load or container of the truck carried.

Material of the bridge pier is then converted from rigid to MAT_ELASTIC and MAT_CSCM_CONCRETE for further decision on producing an actual impact simulation. These two materials had been tested and processed for many criteria and conditions by adjusting the parameters and units. After many trials and errors, it is decided that using the MAT_ELASTIC entity will be the best solution for now. This is due to the complexity of the MAT_CSCM_CONCRETE entity that will need to be experimented even more to get a better and clear understanding on how to utilize it.

As mentioned before the MAT_CSCM_CONCRETE is actually more suitable for the crash simulation with a concrete bridge pier. However, MAT_ELASTIC has been chosen instead mainly due to time constraint of the research. And thus, simpler material is used which is also closed to the actual concrete property and almost similar result from the processing simulation.

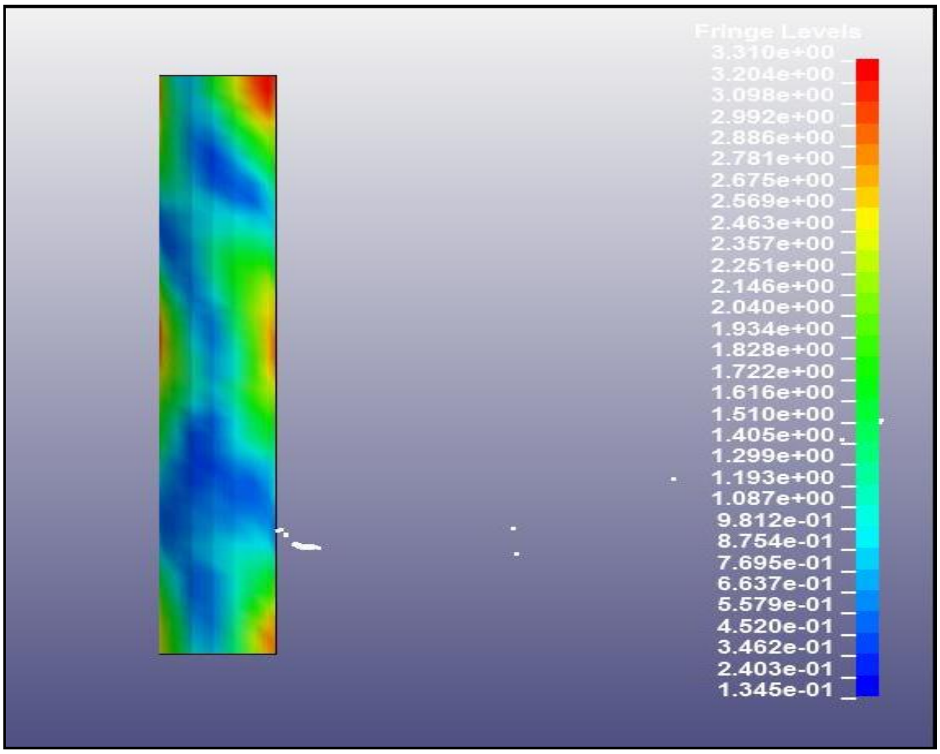


FIGURE 20: Von Mises Stress of MAT_ELASTIC Pier

2. RESULTANT FORCE

i. Ford F800 Series Truck

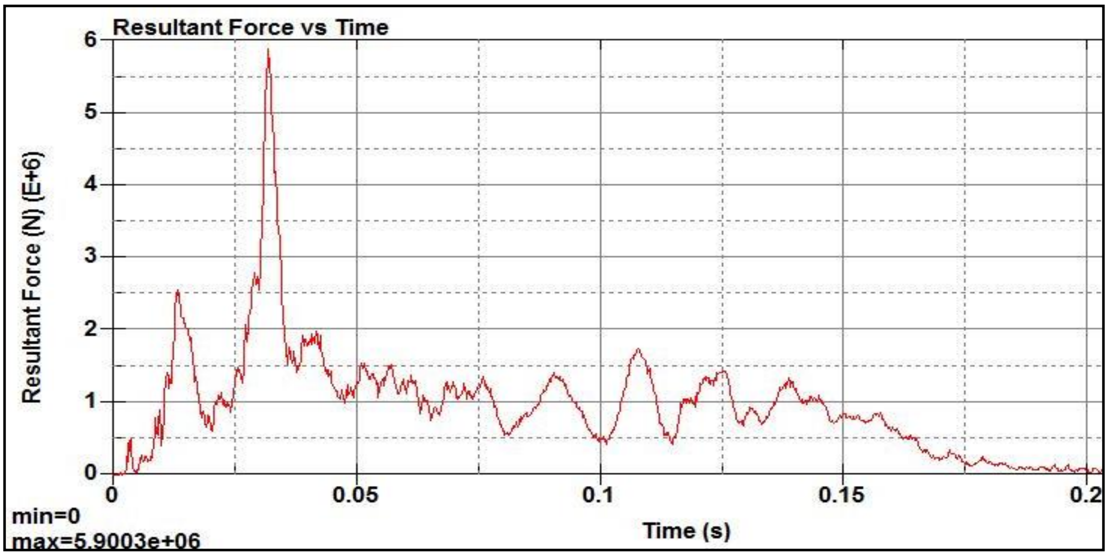


FIGURE 21: Resultant Force versus Time

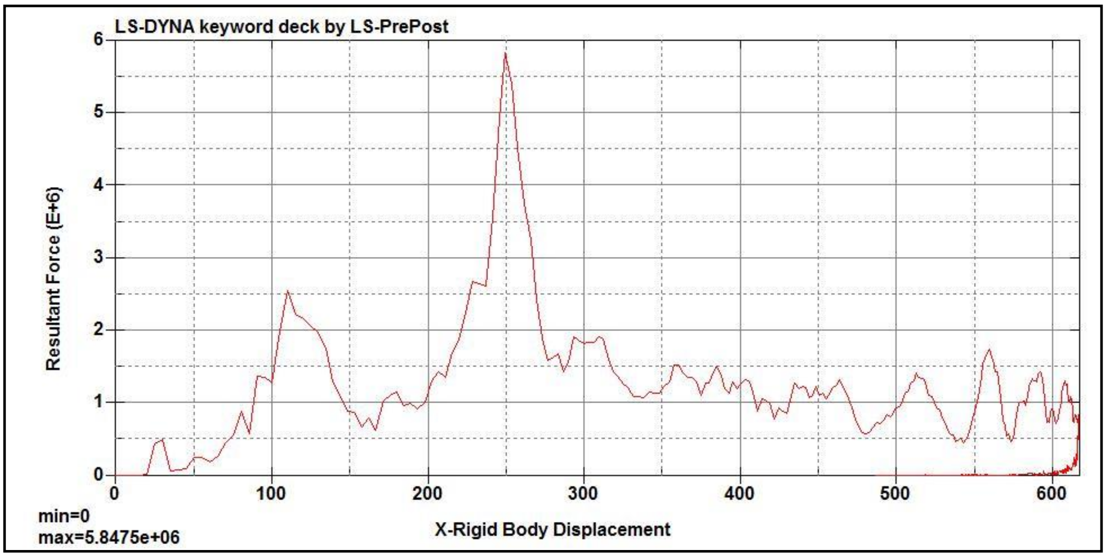


FIGURE 22: Resultant Force versus Displacement

The graphs above show the total force experienced by the 8-ton truck and transmitted onto the concrete pier. The velocity assigned for the truck is 80 km/h with direct impact straight to the pier. The force is considerably high for this type of truck and velocity and the pier shows a little cracking as well. There are two high peaks of the force indicating the initial head one collision for the cabin drive and second peak representing the collision of the cargo.

ii. HGV 16-ton Truck

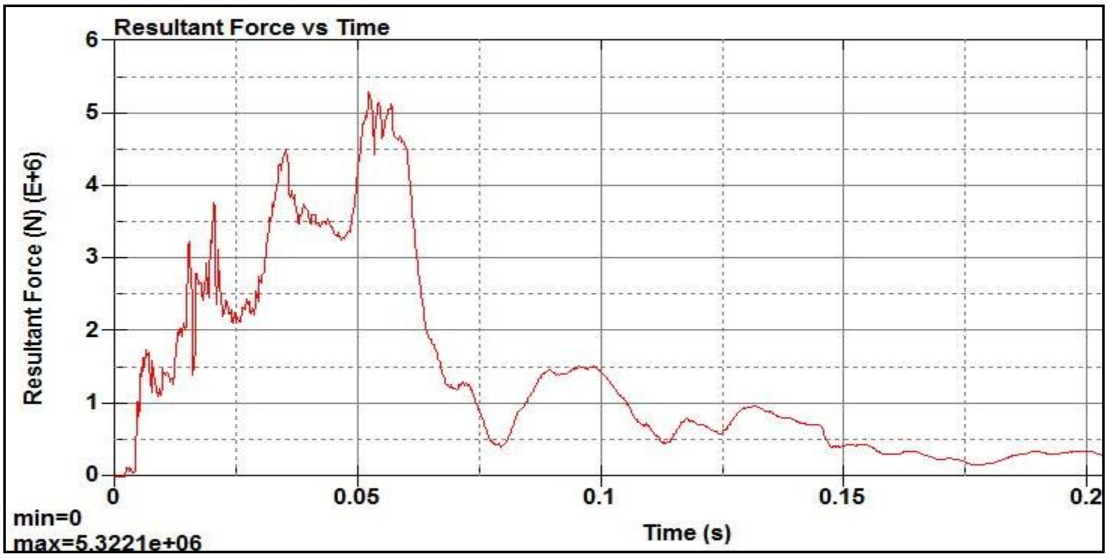


FIGURE 23: Resultant Force versus Time

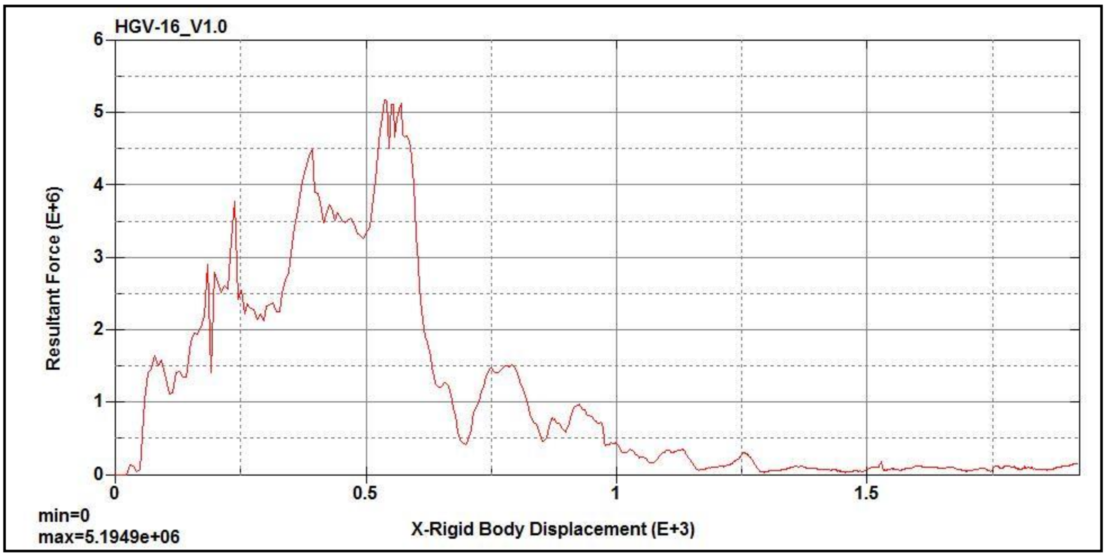


FIGURE 24: Resultant Force versus Displacement

The velocity of the truck is set to be 80 km/h. It can be seen that the resultant force is a bit lower than that of the 8-ton truck. Result for 16-ton truck might not be as accurate as theoretically one and thus, further investigation might be needed to find out the actual cause of reduction in resultant force. As shown in the graph, the ballast from the truck does not seems to have any effect on force increment as compared to 8-ton's truck which has two high peaks of resultant force.

Impact force for HGV 16-ton Truck shows a lower value as compared to Ford F800 Series Truck. Theoretically, since the velocity of the trucks and dimension of the pier are the same, HGV 16-ton Truck should give a higher value than Ford F800 Series Truck. However, this shows otherwise due to some error during the processing progress of the simulation. Apart from that, the impact force pattern should shows the same for HGV 16-ton Truck.

3. ENERGY

i. Ford F800 Series Truck

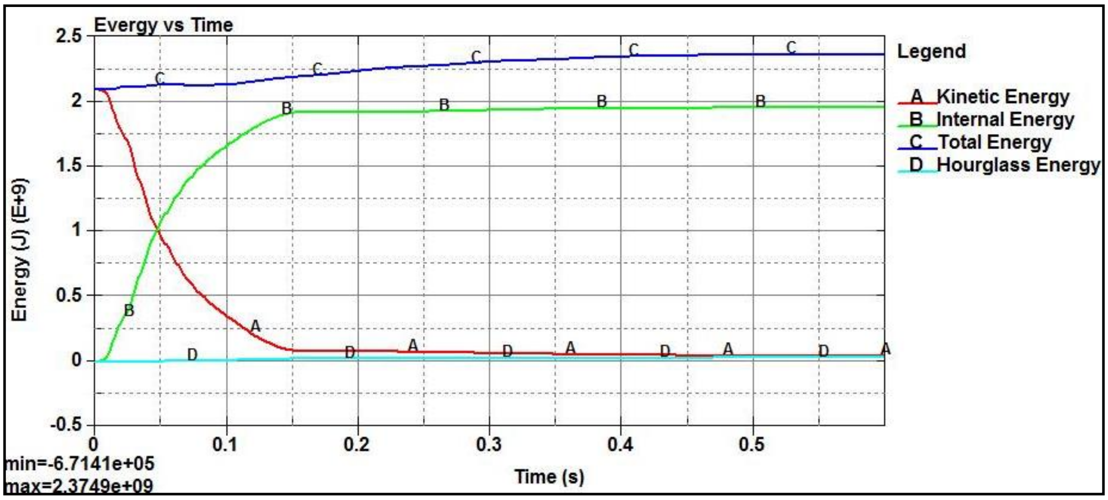


FIGURE 25: Energy versus Time

ii. HGV 16-ton Truck

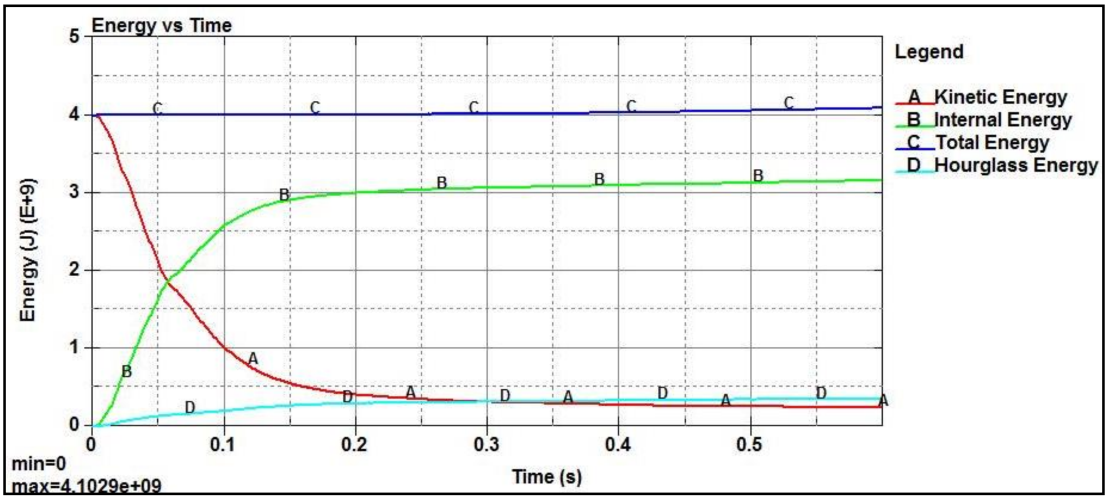


FIGURE 26: Energy versus Time

The energy shown in 8-ton's truck seems to be lower as compared to that of 16-ton's truck. The hourglass energy of both the trucks stay within the safe limit which is below 15% of the total energy. Hourglass energy is the wasted energy during simulation processing. This energy is generated mainly because of the limitation of LS-DYNA processing complicated situation. Thus, monitoring this energy level is vital in order to control the reliability of the result. As shown in both the graphs, the conversion of kinetic energy to internal energy shows that the trucks are transferring the impact forces onto the pier upon collision impact. Thus, resulting the cracking of the pier due to the internal energy transmitted.

4. DEFLECTION

i. Ford F800 Series Truck

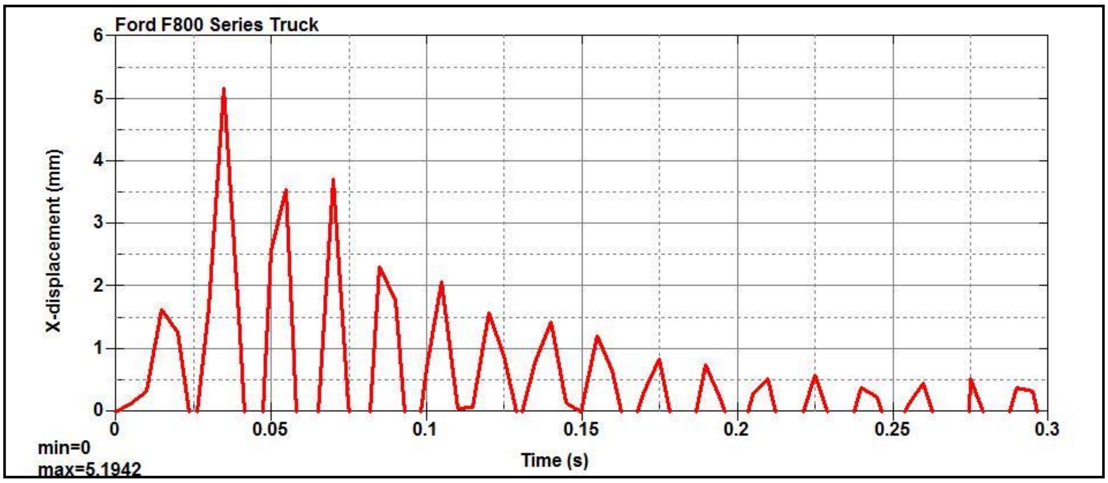


FIGURE 27: Deflection versus Time

The deflection of the pier occurred when the driving cabin and the cargo collide on the pier. Initially, the collision from the cabin drive deflects the pier but due to lower impact force. It then deforms even more when the cabin strikes in generating higher impact force on the pier. Since the material property of the pier is defined as elastic, deflection will tend return to its original position after some deflection stages from the heavy vehicle.

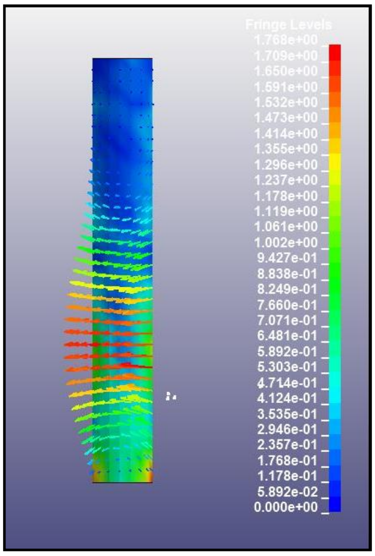


FIGURE 28: Vector Illustration of Pier Deflection

The deflection of the pier is illustrated as shown in figure above. The pier deflects in such manner due to the impact imparted from the truck to that position. Notice the red colored vector shows that the deflection is maximum at that spot and cracks will first generate from there.

ii. HGV 16-ton Truck

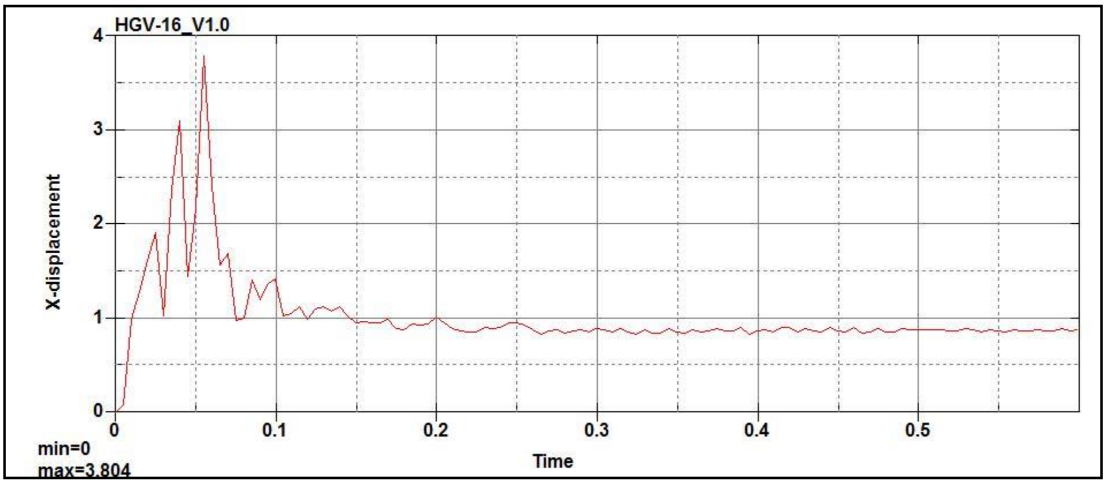


FIGURE 29: Deflection versus Time

Deflection is obvious in the beginning due to the high impact force from the overall truck especially from the cabin drive. Forces of impact reside gradually since the impact forces and energy are transferred to the pier. The graph shows that the deflection is quite high after the impact. Deflection is illustrated as shown below.

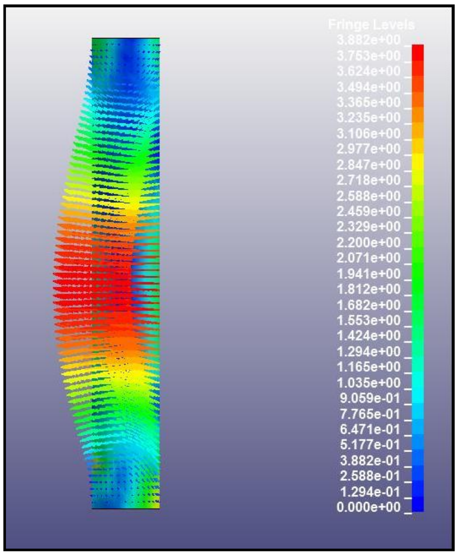


FIGURE 30: Vector Illustration of Pier Deflection

Deflection is more obvious in this case. HGV 16-ton Truck generates a higher impact force and will cause the pier to deflect more than the other truck mentioned. Deflection occurs severely in the middle of the pier as indicated in red region of the vector shown.

Deflection of both the cases shown by these trucks indicating that the pier has possibility of cracking in such a failure pattern. The graph delivers the value of the deflection, whether the maximum value has exceed the allowable safety value or otherwise, and will have to be taken into consideration to prevent further casualty. Nevertheless, the actual deflection cannot be fully verified since the cracking pattern can only be assumed.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

Researches have shown many bridge piers designs might not be suitable for current traffic condition especially when the collision of the piers involve heavy vehicles. Piers constructed in Malaysia might be highly vulnerable in terms of heavy vehicle impact and give a great threat to the safety of road users. Previous research shows that the AASHTO-LRFD design guidelines are not very appropriate anymore. It has been proven in that research regarding the unreliability of the old guidelines. Thus, this research is essential in order to review current design code used by Malaysia for the safety concern of everyone.

This research is recommended to be done using a realistic simulation, taking advantages of the software LS-DYNA which focuses on the finite element modeling. Parameters include impact forces, characteristics of structure and vehicle, impact scenario, stress and strain value and so on should be computed and compared with the current design code.

Forces, energies and other parameters of the simulation are successfully being acquired from the progress. From the analysis of the result, although results seems reliable as compared to some recognized sources, it seems that sufficient details are still needed to support the objective of this research. Forces and energies from both the scenarios need to be further access to find out the missing data for the simulation. It is noted here that the investigation should be focus on the material used for modeling the pier and also parameters in manipulating the conditions of impact to obtain realistic result.

At this point, the impact forces shown by the both trucks have exceeded the requirement of the code of design which only withstand a force of 1000 kN. Thus, this result shall be further verified in the future to ensure the eligibility of the current code of design to be used for bridge pier construction.

Nevertheless, the research on getting a simulation close to the actual impact collision between the bridge pier and heavy vehicle has been successfully produced. This simulation might be suitable to be used for the further investigation on the impact between any piers of material carefully defined with a heavy vehicle.

REFERENCES

A. S. Joshi and L. M. Gupta, 2012, "A Simulation Study on Quantifying damage in Bridge Piers Subjected to Vehicle Collisions," *International Journal of Advanced Structural Engineering*, 4:8, 2012.

C. Eugene Buth, W. F. Williams, M. S. Brackin, D. Lord, S. R. Geedipally, and A. Y. Abu-Odeh., 2010, "Analysis of Large Truck Collisions with Bridge Piers: Phase 1. Report of Guidelines for Designing Bridge Piers and Abutments for Vehicle Collisions," *Texas Transportation Institute*, 2009-2010.

El-Tawil, S., Severino, E. and Fonseca, P., 2004, "Vehicle Collision with Bridge Piers," *Accepted for Publication in the Journal of Bridge Engineering*, ASCE.

F800 Single Unit Truck FEM Model for Crash Simulation, *National Transportation Research Center*. Retrieved on 12 April 2014 from http://single-unit-truck.model.ntrci.org/description/

LS-DYNA General Overview, *DYNAmore Finite Element Solutions*. Retrieved on 20 July 2014 from http://www.dynamore.de/en/products/dyna

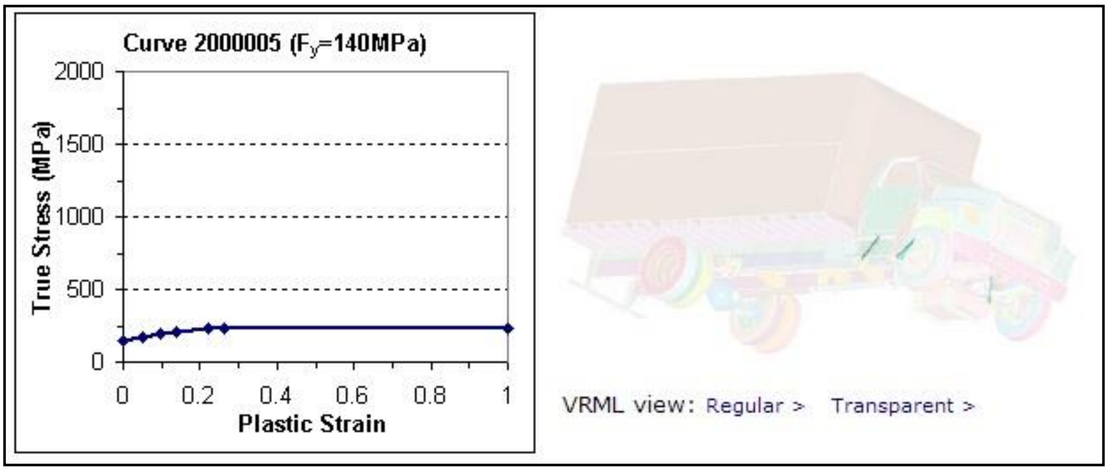
LS-PrePost Documentation, *Livermore Software Technology Corp.* Retrieved on 20 July 2014 from http://www.lstc.com/lspp/content/overview.shtml

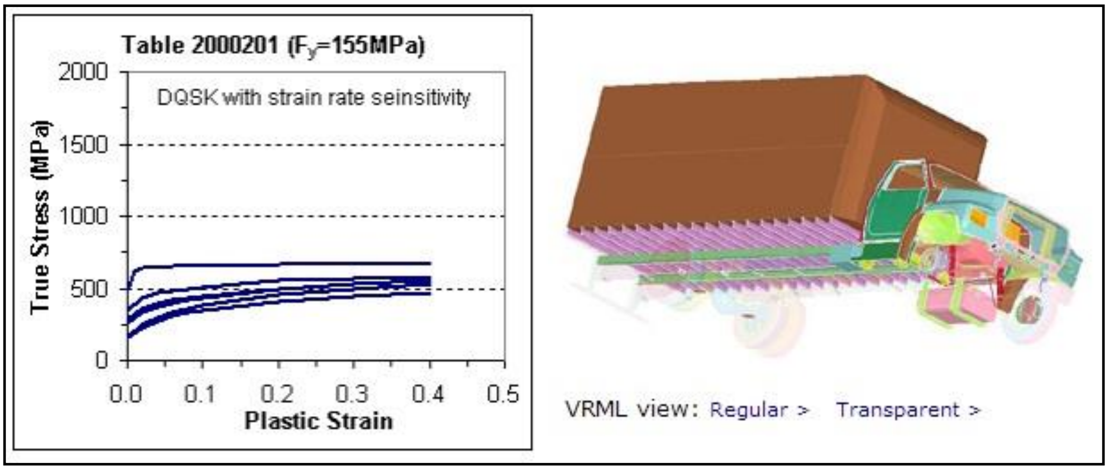
W. Teo, H. Yin, M. Maharun, S. Karuppanan, M. S. Liew, A. R. Salleh and T. C. Wong, 2014, "Research on HGV Collisions with Concrete Bridge Piers," Universiti Teknologi PETRONAS, Malaysia.

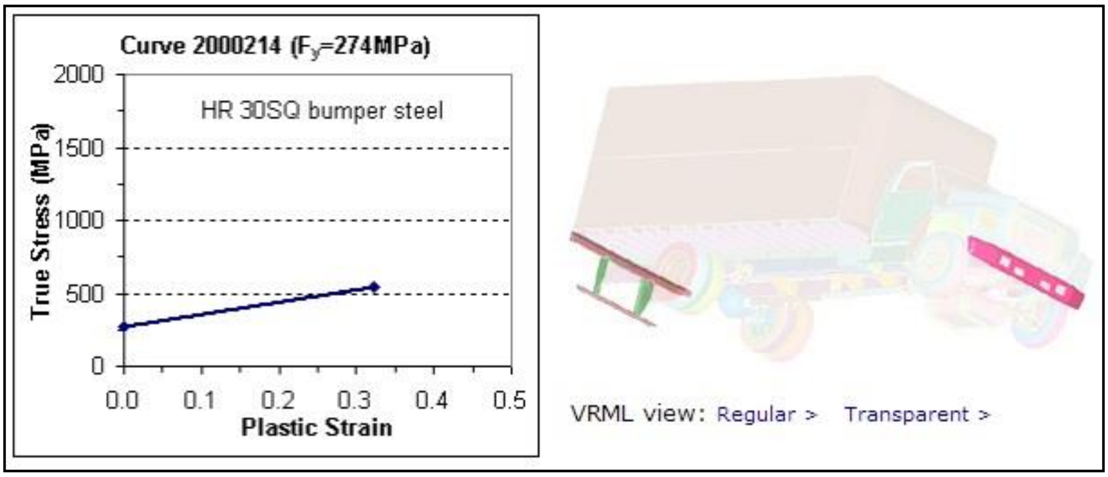
Y. Sha and H. Hao, 2012, "Nonlinear Finite Element Analysis of Barge Collision with a Single Bridge Pier," *SciVerse Science Direct*, 2012.

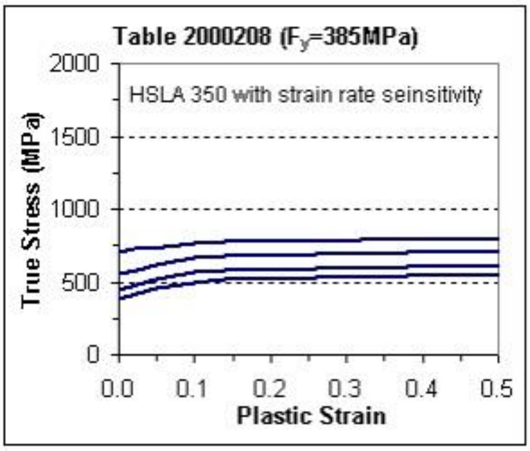
APPENDIXES

1. MATERIAL PROPERTIES OF FORD F800 SERIES TRUCK

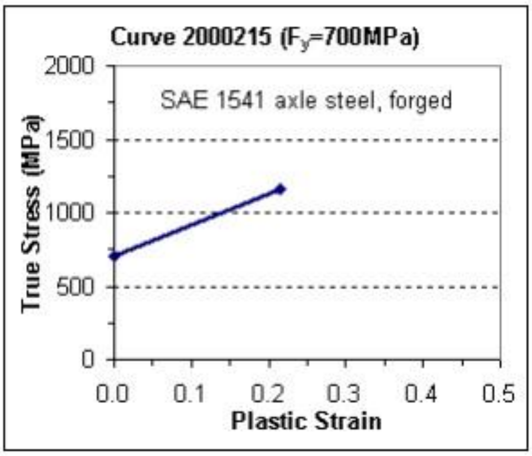


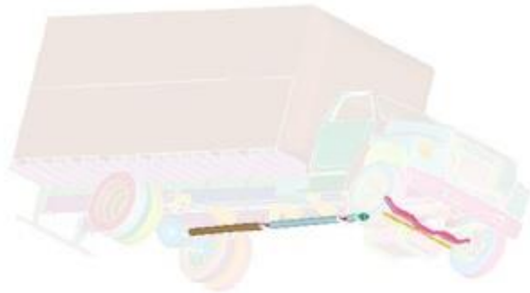




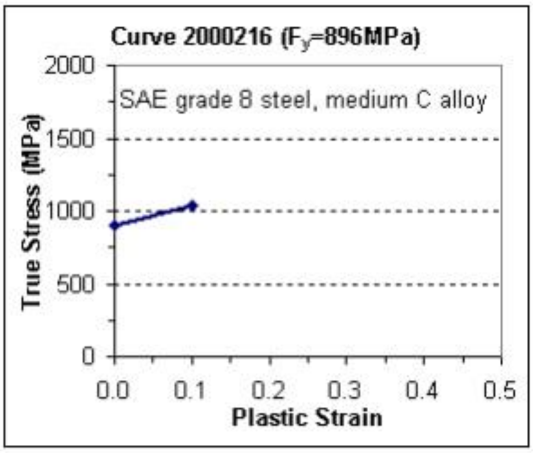


VRML view: Regular > Transparent >

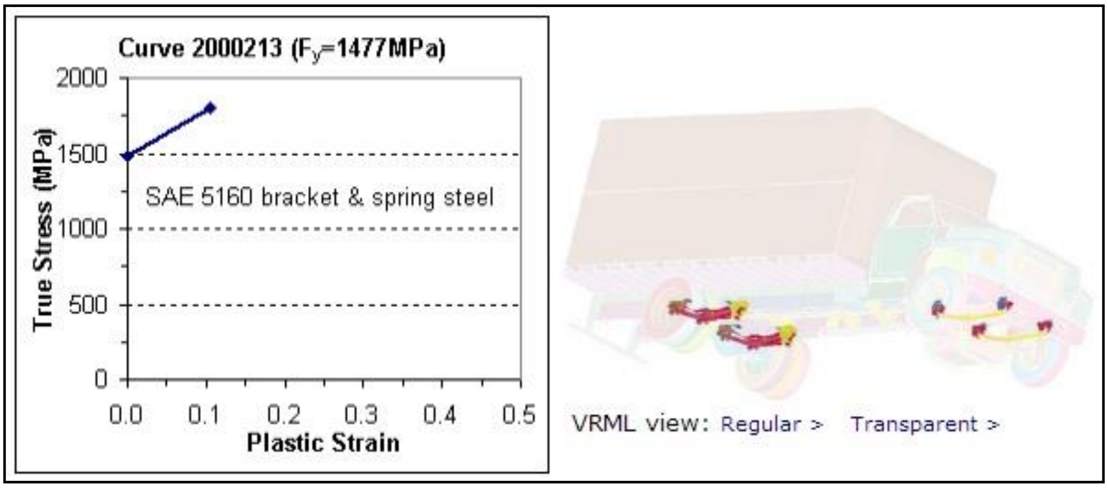




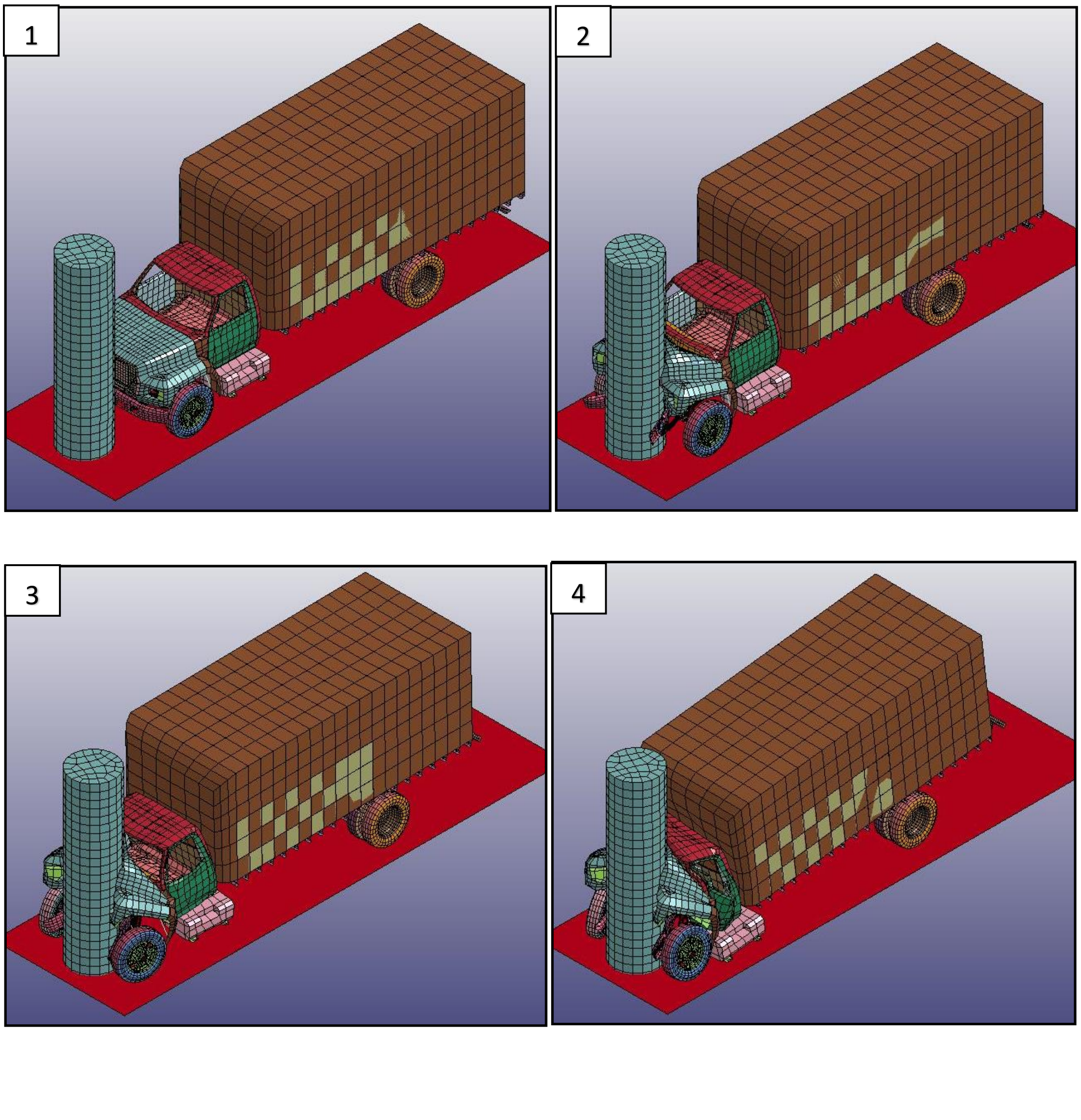
VRML view: Regular > Transparent >

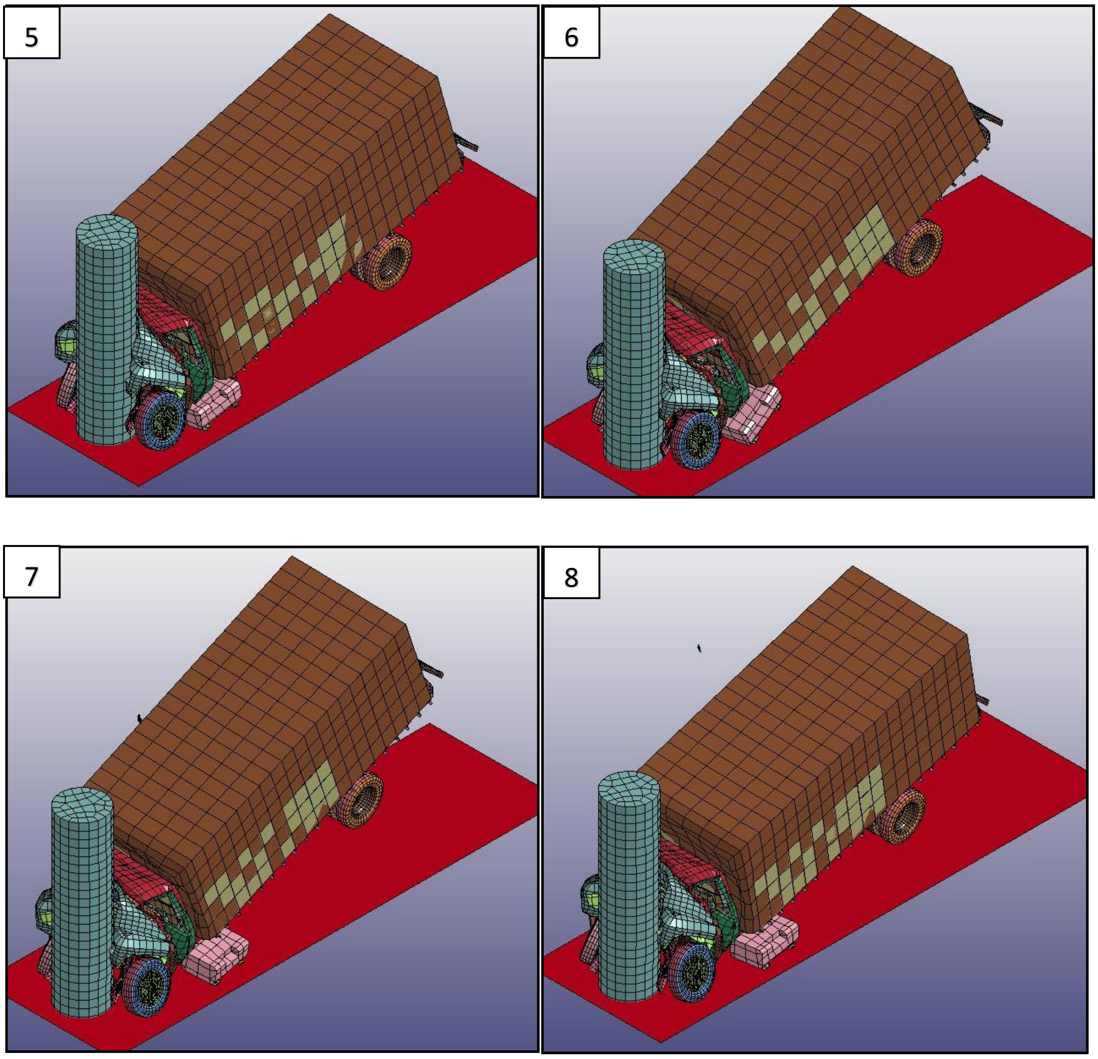


VRML view: Regular > Transparent >

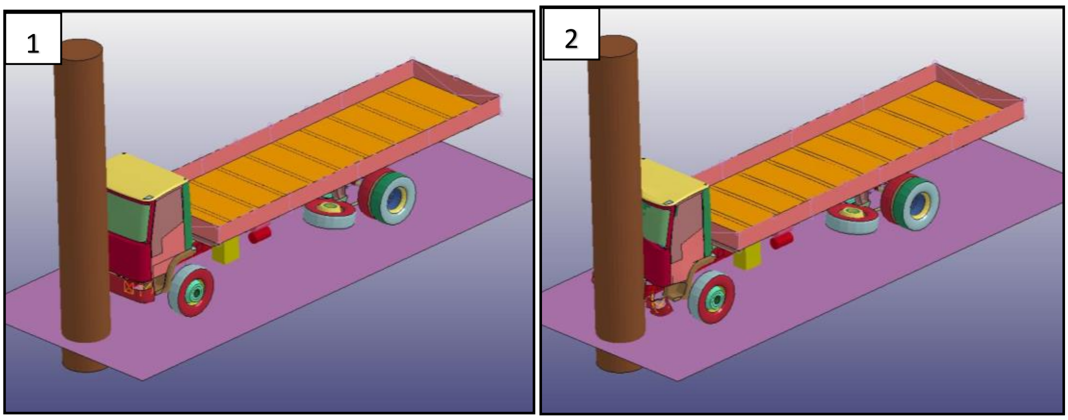


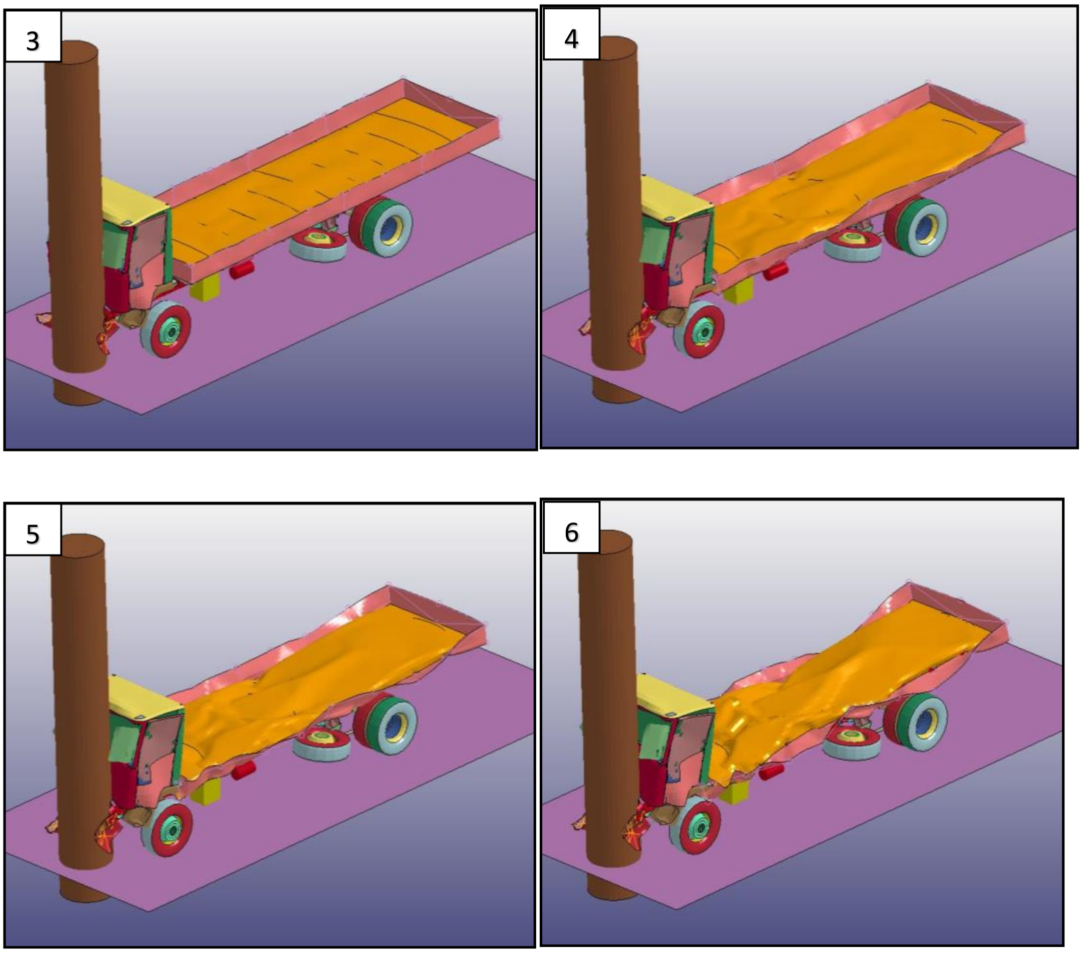
2. SIMULATION PROGRESS OF FORD F800 SERIES TRUCK





3. SIMULATION PROGRESS OF HGV 16-TON TRUCK




3
4
5
6