اونيۏرسيتي مليسيا ڤهڠ
UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL AND AUTOMOTIVE ENGINEERING TECHNOLOGY

BHA4102 FINITE ELEMENT METHOD 2

LAB TITLE:

Project 1 – Bumper Crash Impact Analysis

Name : Desmond Ling Ze Yew (HB19024)
Hasif Farhan bin Hanisofian (HB19025)
Date : 9th July 2023

ASSESSMENT CRITERIA	MARKS
Objective	/5
CAD model	/10
Methodology a) Describe boundary conditions and DOFs by hand drawing b) State all the units used for this simulation c) State the material properties, material model, contact type and velocity used. d) Mesh convergence analysis.	/25
Result a) Plot graph for force vs displacement using Ls-dyna	/30

b) Plot graph for kinetic energy and internal energy vs time using Ls-dyna. c) Table for maximum displacement, stress and internal energy. d) Figures for the deformation pattern.	
Lesson Learned a) What is the relationship between kinetic energy and internal energy? b) How does velocity affect the deformation? c) Describes two main challenges in modelling the impact and how you handle it?	/15
Recommendations	/10
References	/5
	/100

1.0 Introduction

With the rapid development of the automobile industry, traffic accidents are frequent and frontal impacts occur most frequently in all traffic accidents. Automotive crashworthiness and lightweight design have attracted the attention of automotive manufacturers and researchers. In most collision accidents, the bumper system is the first vehicle component involved in the impact and protects the car body and passengers. It is expected to be sufficiently deformable to absorb impact energy and thereby should have enough stiffness to protect the other vehicle components. The bumper system comprises of bumper beam and crash box as the energy absorber. Bumper beam is a structural component that absorbs low-impact energy and dissipates the high-impact energy by collapse. Therefore, it is necessary to study the optimization of bumper system to reduce the weight and improve the crashworthiness of the vehicle.

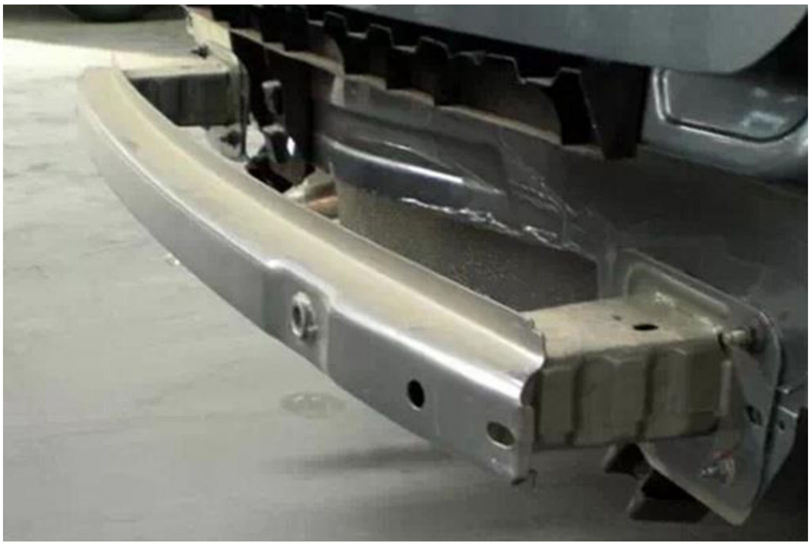

Figure 1: The bumper system consists of a bumper beam and the crash box (Source: made-in-china.com)

In this lab session, we are going to design and analyse the impact of a crash between the bumper system and a rigid wall to observe and study the deformation and energy absorbed by the bumper during an impact. We will be using the ANSYS LS-Dyna software to perform and run the analysis.

2.0 Objective

By the end of this lab session, students will achieve the following objectives:

- To introduce students on bumper crash analysis tube analysis using LS-Dyna.
- To simulate the impact between the bumper system and the rigid wall.
- To study the deformation and the energy absorbed by the bumper system.
- To extract and analyse the data from the results of the analysis using LS-Dyna.
- To familiarise with the keyword functions available in LS-Dyna.

3.0 Methodology

For this lab session, we will design the bumper system model by using SolidWorks while the rigid wall model will be designed in LS-Dyna. Later, we will define the bumper system as shell and will be using aluminium properties as its material while the rigid wall is made up of solid steel. Finally, we will define the boundary conditions, contacts, termination and the database of the analysis.

We will also be using the following consistent units throughout the analysis:

Mass	Length	Time	Force	Stress	Energy
kg	mm	ms	kN	GPa	kN.mm

- **CAD Modelling and Meshing – Bumper System**

  As stated above, we designed a simple model of the bumper system using the SolidWorks software as seen in Figure 2, while Figure 3 show the dimension of the model. The model is then saved as IGES file to be imported to LS-Dyna.

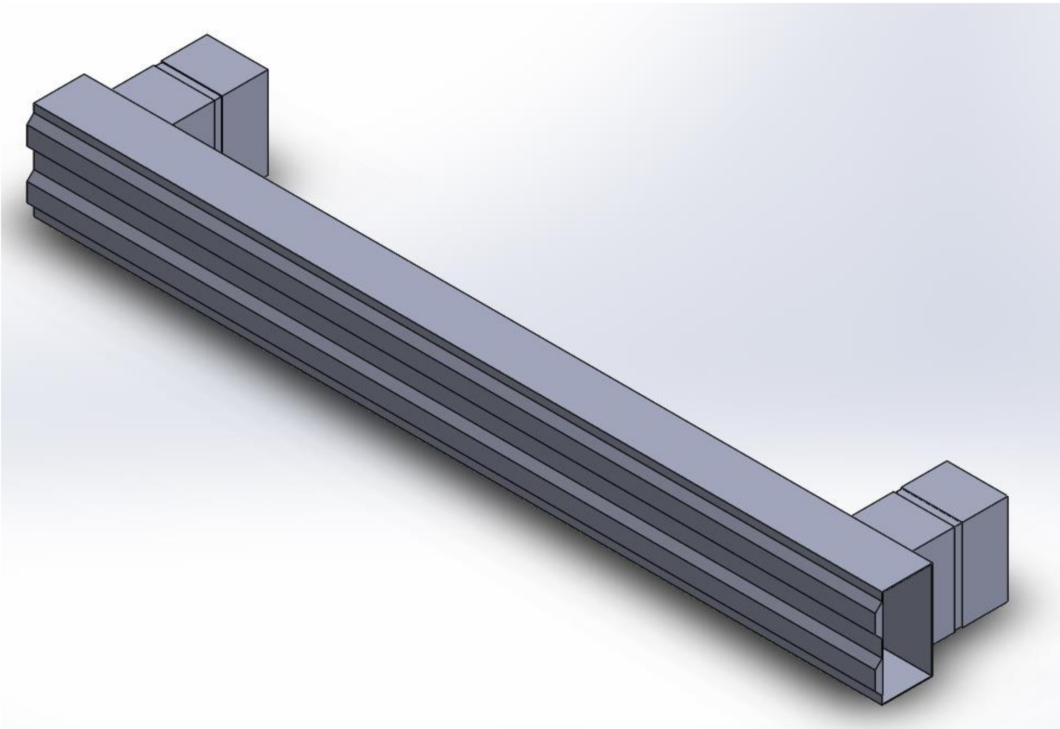

Figure 2: The 3D model of the bumper system.

As we open and import the IGES file on LS-Dyna, we will be greeted by the *IGES Read Options* and for this lab session, we are going to use the default setting and specify the unit to mm following the consistent units that we have decided earlier. We discover that due to the model is drawn facing the z-axis plane, the model will be facing upward following the z-axis direction in LS-Dyna and later we will rotate the model after the we mesh it.

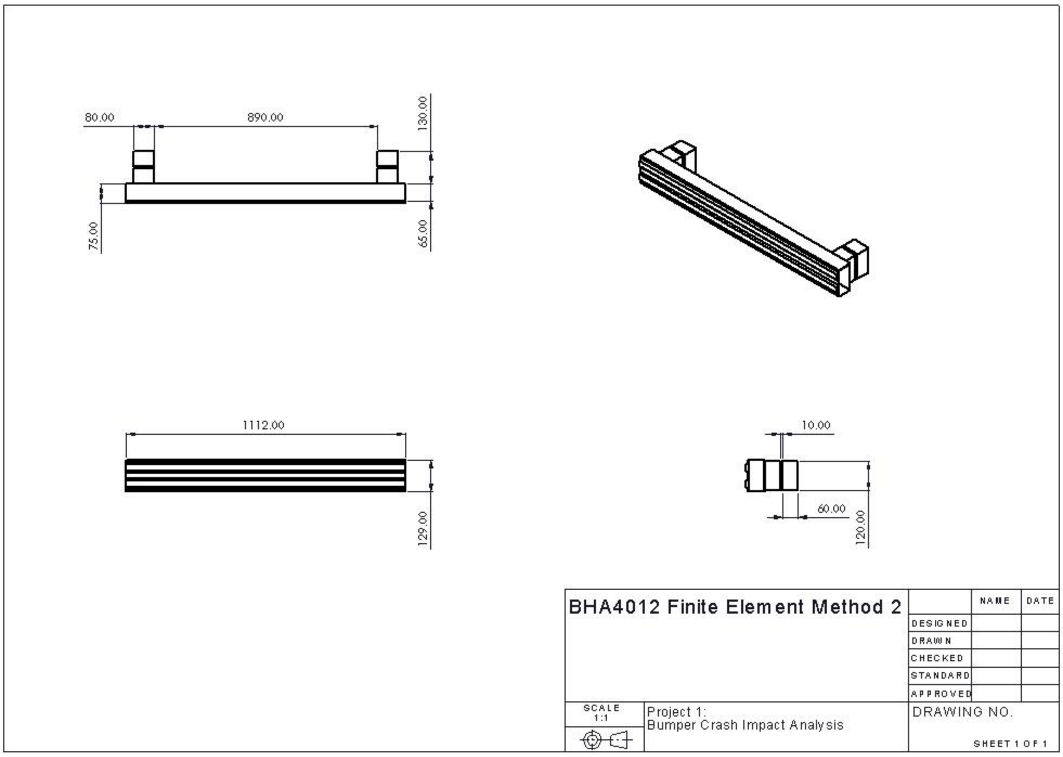

Figure 3: The orthographic drawing of the bumper system with the dimension.

To mesh the model, we will be using the *Auto Mesher* function and mesh the bumper beam and the crash box separately to create 2 different parts for the analysis. The element size of the mesh is set as 22mm for both part (as the recommended computed size is 22.024mm). After the model is meshed, we will delete the IGES model and thus leaving us with only the meshing of the model.

Next, we will rotate the model by using the *Transform* function in the *Element Tools* and set the origin of rotation as the coordinate of (0, 0, 0) with the rotation angle of 90 along the z-rotation axis. After selecting all the model, we will then click *Rotate-* and the model will now be facing the y-axis plane, making it easier to be viewed in isometric on LS-Dyna.

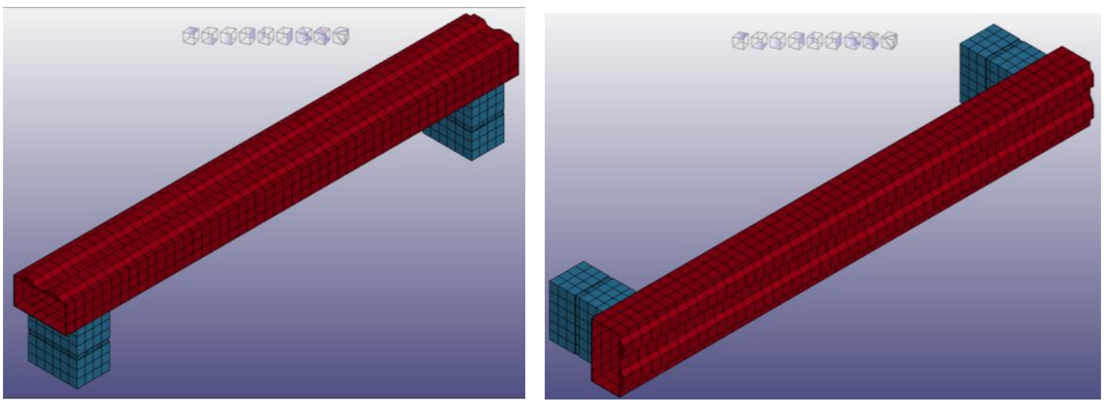

Figure 4: The geometry of the bumper system model. (Left: Before Rotation; Right: After Rotation)

- **Rigid Wall Model**

  The rigid wall is first designed using the *Shape Mesher* function to create a solid box entity with the element size of 30mm in every direction and input the coordinate of the x-axis (min: -800; max: 800), y-axis (min: 50; max: 100), and z-axis (min: -100; max: 100).

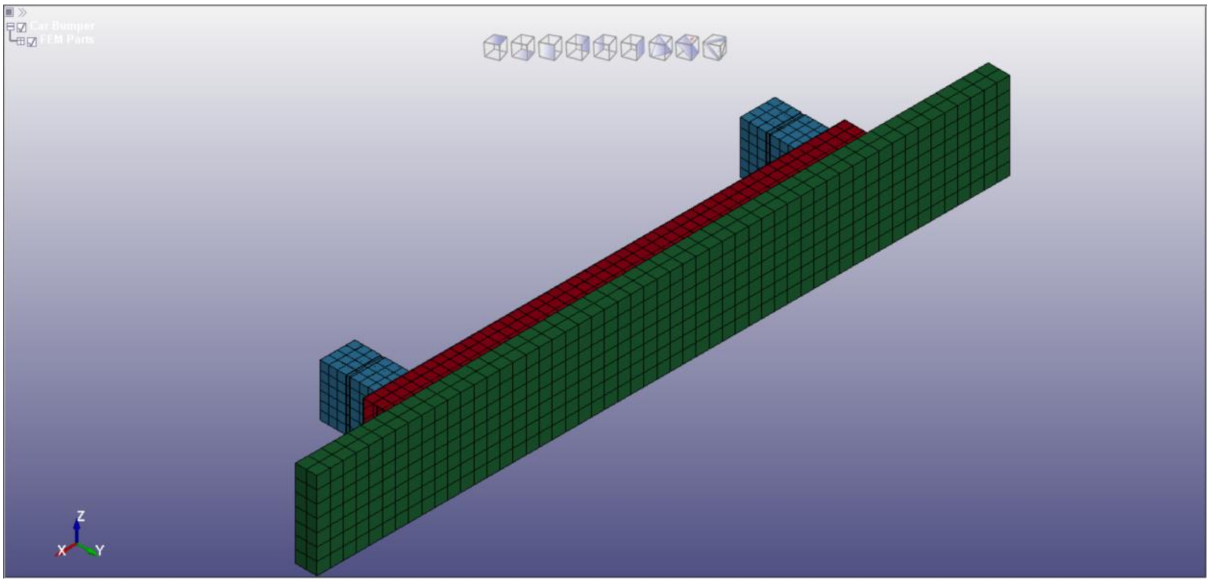

Figure 5: The initial geometry of the rigid wall model.

However, we discovered upon running the simulation that it will take an estimated time as long as 42 days to finish the analysis. By troubleshooting the problem and referring to the available resources on the internet related to the analysis, we have decided to define the rigid wall by creating a moving planar rigid wall entity. By using the *Create Entity* function, we will create a rigid wall entity and set it as *Planar – Moving* with a mass of 800kg and a velocity of 10m/s.

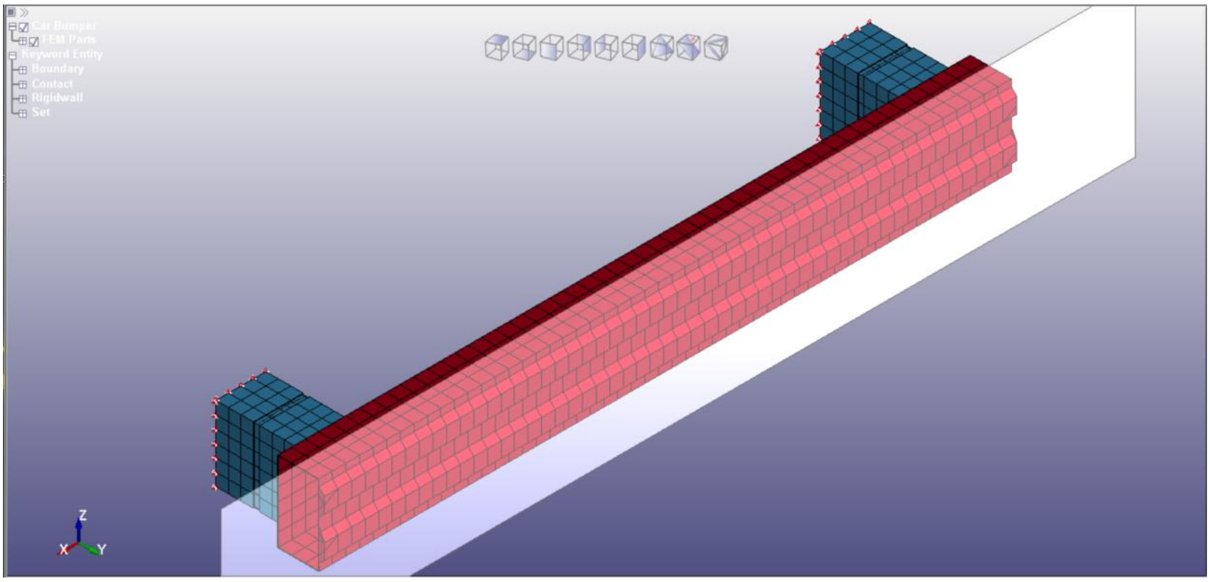

Figure 6: The final geometry of the rigid wall model.

- **Material and Section**

  The material for both bumper beam and the crash box will be the same, which is aluminium and defined by using *MAT_024_PIECEWISE_LINEAR_PLASTICITY* in the *Keyword Manager*. The properties are as follows:

Density	Young Modulus	Poisson's Ratio	Yield Stress
7.830x10^{-6} kg/mm^3	200 GPa	0.3	0.366 GPa

  Meanwhile, the section will be defined as *Shell* for both parts with default setting but have a different thickness, with the thickness of the bumper beam is 1.0mm while the crash box has a thickness of 1.6mm.

- **Boundary Condition**

  The nodes at the end of the crash box will be combined as a set node entity and later be constraint to be fixed. By using the *Set* Data option in the Create *Entity* function, we can select all the related nodes and create a set node named *Fixed_Nodes*. Afterwards, we will set the constraint using the *Boundary* option and select *Spc*. Here, we will select the node set ID that we have just created and select *All Fix [1 1 1 1 1 1]* to ensure the nodes in fixed in all directions and rotations.

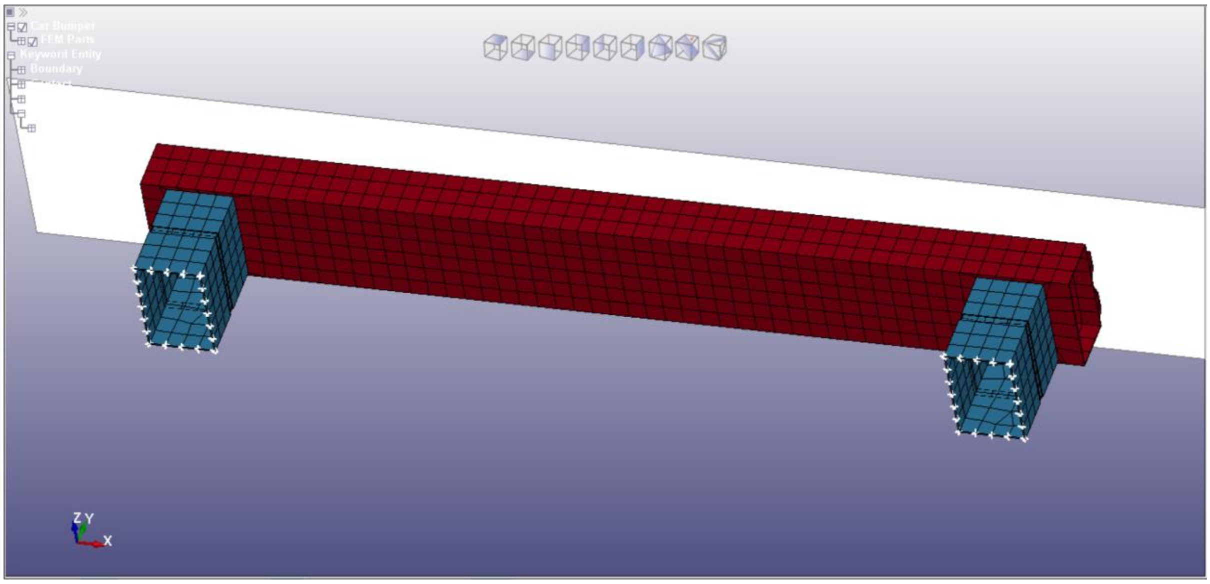

Figure 7: The fixed nodes is shown with the white 'x' marks.

- **Contacts**

  There are 3 main contacts that we need to define, with all the contacts has the value of static coefficient and dynamic coefficient of 0.08. The first one is the contact of the rigid wall hitting the bumper. Due to the wall is created as a rigid wall entity, it is best to define the contact using *AUTOMATIC_SINGLE_SURFACE* with the slave and master ID will be left undefined.

Next, the contact between the bumper crushing the crash box is define using the contact of *AUTOMATIC_SURFACE_TO_SURFACE_MORTAR* due to the contact is between the surface of the bumper beam and the edge of the crash box. The slave will be assigned to the crushed part, which is the crash box, while the bumper beam will be assigned as the master. Finally, the contact between the crash box with itself is define using the contact *AUTOMATIC_SINGLE_SURFACE_MORTAR* with the slave is assign to the crash box.

- **Termination**
  The end time setup means that we need to define the time interval in which the analysis will take place. This can be done by selecting *TERMINATION* in the *Keyword Manager* function under the *Control* option. The termination time will be defined as 20ms.

- **Binary D3Plot**
  The D3Plot is important to be defined to get the results for the stresses, displacements and many more. We will select *BINARY_D3PLOT* in the *Keyword Manager* function under the *Database* option and set the time interval to be 1.0ms.

- **ASCII Option**
  We will set the default time interval as 0.1s and select on *GLSTAT (Global Statistics)*, *MATSUM (Material Energies)*, *BNDOUT (Boundary Conditions Forces and Energy)*, *NODOUT (Nodal Displacement/Velocity/Acceleration Data)*, *NODFOR (Nodal Force Groups)*, *RCFORC (Resultant Interface Forces)*, and *RWFORC (Rigid Wall Forces)*.

- **Model Checking**
  Lastly, the model needs to be checked before running the simulation under the *Keyword Check* option in the *Model Checking* function as below:

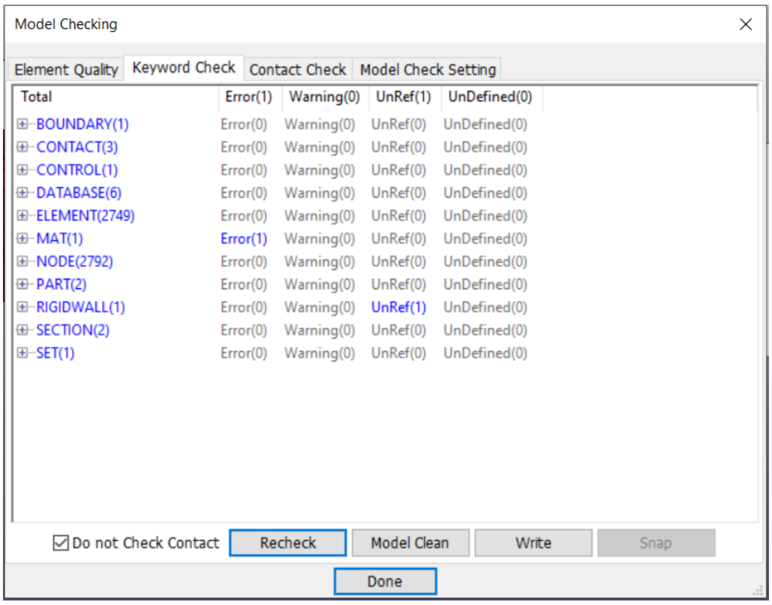

Figure 8: The Model Checking window for the Keyword Check.

4.0 Results

The keyword file can be run by using LS-Run and using the default setting. After the status *Finished (Normal Termination)* shown, it means that the simulation has been run properly and the result is ready to be analysed.

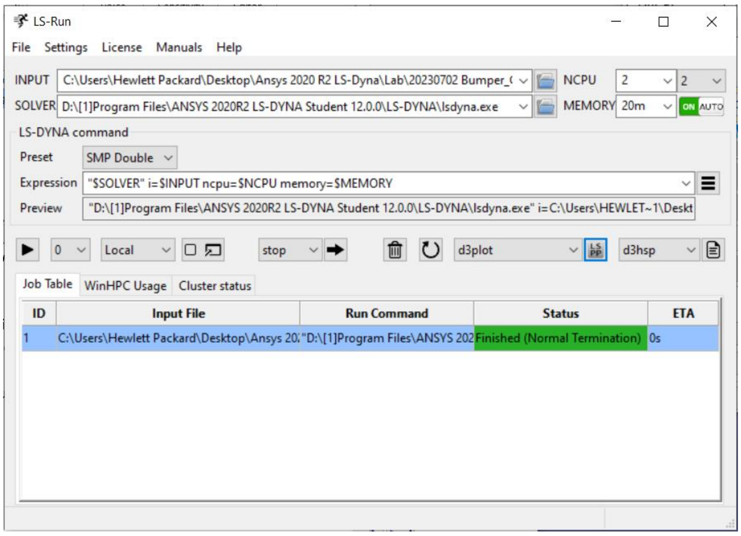

Figure 9: The LS-Run window with the finished simulation status.

- **Force vs Displacement Graph**
  We can plot the graph for the force vs displacement by using the *ASCII* function to save the force-time and the displacement-time plot before crossing it using *XY Plot* function. The force-time graph can be plot using *rcforc* option in the *ASCII* function. Then, the displacement-time graph will be plotted using the *History* function. Both plots need to be saved as CSV file. Next, by using the *XY Plot* function, we can select the *Cross* option and assign the displacement-time plot as the x-axis while the force-time plot as the y-axis.

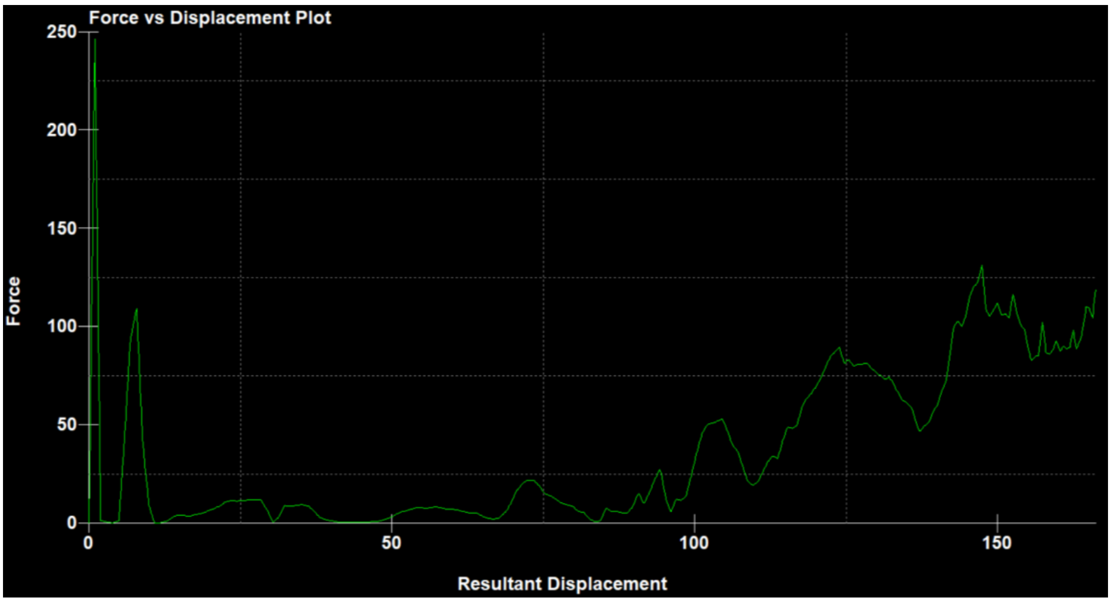

Figure 10: The force vs displacement graph plot.

- **Kinetic Energy and Internal Energy vs Time Graph**

  By using the *ASCII* function, we can also plot the kinetic energy vs time graph, as well as internal energy vs time graph. To plot both graph in one window, we need to select both *Kinetic Energy* and *Internal Energy* in the *GLSTAT* option and click *Plot*.

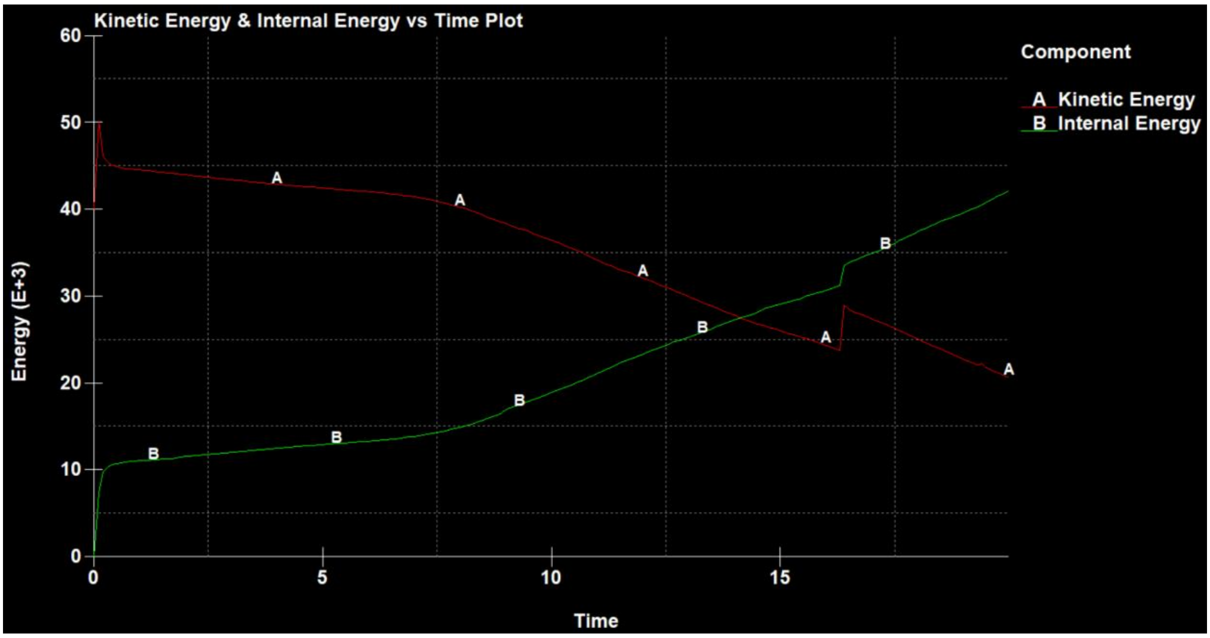

Figure 11: The kinetic energy and internal energy vs displacement graph plot.

- **Deformation Pattern**

  After running the simulation, we can observe how the wall crush the bumper system and as the result deformed the bumper beam and the crash box with the pattern as seen in Figure 12 and 13 below. We observe that the centre of the bumper beam will have the least deformed part while the crash box is totally crushed. This is accurate to the real-world application where the crash box function as a ‘cushion’ and absorbed the impact before the forces reach to the passenger.

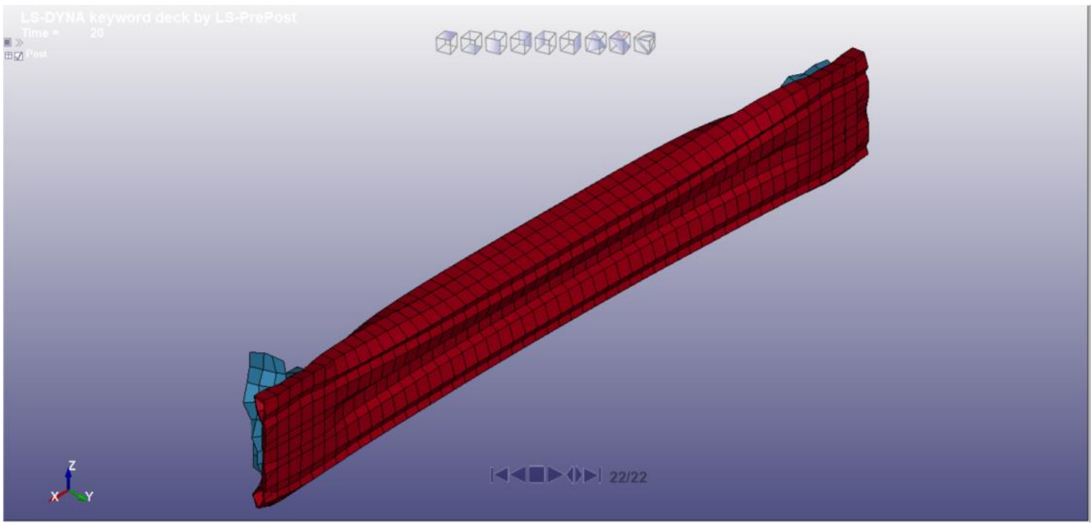

Figure 12: The isometric view of the deformed model.

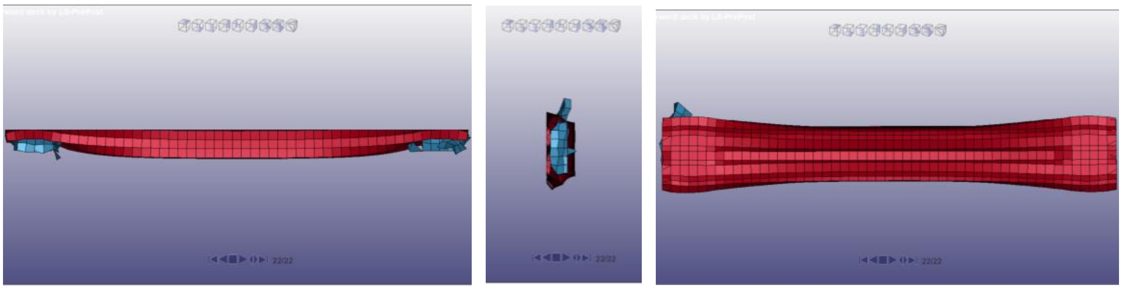

Figure 13: The top, front and right view of the deformed model.

- **Maximum Displacement, Stress and Internal Energy**

  LS-Dyna can calculate the values of the maximum displacement, stress and energy absorbed as shown in the table below:

Result	Maximum Displacement	Maximum von Mises Stress	Internal Energy
Value	166.283 mm	0.366002 GPa	31759.8 kN.mm

5.0 Discussion and Recommendations

Based on the results, we have achieved the main objectives of the lab session. Hence, there are several takeaways from this lab that needs to be addressed in this discussion.

- **Lesson Learned**

  a) <u>What is the relationship between kinetic energy and internal energy?</u>
  - From the result achieved, we can observe that as the kinetic energy decreases, the internal energy will increase. This show that as energy cannot be destroyed as the kinetic energy is absorbed by the plate and transform into internal energy.

  b) <u>How does velocity affect the deformation?</u>
  - Effectively, as the impact velocity increase, more deformation can be observed due the increase in stress and reaction forces on the bodies.

  c) <u>Describes two main challenges in modelling the impact and how you handle it?</u>
  - **Rigid Wall Modelling:** As stated before, we initially modelled the rigid wall by using the using the *Shape Mesher* function to create a box solid mesh for the wall. However, we find out that when running the simulation, it takes days to complete the analysis. Upon troubleshooting by changing the size of the element and the mesh, we discover that the common practice used for the impact simulation is by using the *Rigid Wall* entity and thus what we did for the simulation.

  - **Material Selection**: Based on our research paper reference, we have defined a different material property for both the bumper beam *(MAT 54 as CFRP* *composite)* and the crash box *(MAT 24 as aluminium alloy)*. However, the simulation show that the crash box will be crushed into pieces and does not simulate accurately. To ease the practice, we use the same aluminium properties for both bumper beam and crash box.

- **Recommendation**

  For future reference on using LS-Dyna, we recommend that:

  a) Student to understand the definition of the selected keyword with referring to the LS-Dyna *Keyword User's Manual* provided.

  b) Student to explore another option in solving the long computational time for the simulation with the rigid wall part included.

  c) Student to understand and explore different keyword to define the input or achieve the output, and understand how the size of the mesh affects the computational time.

  d) The process can be recorded for future reference especially during the writing of the lab report.

6.0 Conclusion

The conclusion is that the LS-Dyna is a great way to study the bumper crash impact analysis for beginner as it was a quite simple and straightforward analysis with the step-by-step reference available to ease the process of learning. As the results, we have achieved the main objectives set for the lab session with better understanding of the analysis. We also learned how to solve problem faced when running the simulation, for example the way we solved the long simulation time by using different method to represent the rigid wall. Furthermore, we discovered that the results can be varied based on the input given, observed when we changed the material properties of the bumper system. Lastly, we have determined on things to be improved and recommended as reference for future lab session.

References:

[1] Livermore Software Technology (LST). (2020, July 17). LS-DYNA Keyword User's Manual: Vol. II. LS-DYNA R12 (r:13113).

[2] Li, M., Sang, L., Xiong, Z., & Hou, W. (2021, October 27). Design and optimisation of vehicle frontal bumper beam with alternative carbon fibre-reinforced plastics in corrugated sandwich structure. *International Journal of Crashworthiness*, *27*(6), 1635–1647. https://doi.org/10.1080/13588265.2021.1981189

[3] Acar, E., Yilmaz, B., Güler, M. A., & Altin, M. (2020, August 31). Multi-fidelity crashworthiness optimization of a bus bumper system under frontal impact. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, *42*(9). https://doi.org/10.1007/s40430-020-02572-3

[4] *LS DYNA Tutorial | Crash analysis procedure | Step By Step*. (2021, August 8). [Video]. YouTube. Retrieved July 9, 2023, from https://youtu.be/7gmuYSD4e_k

[5] *LS-DYNA Tutorial | Collapse Analysis of Crush Box Assembly | 1809*. (2018, March 19). [Video]. YouTube. Retrieved July 9, 2023, from https://youtu.be/vgm3c_knXS8