Stability analysis of pendulum Test Rig as per ECE R29 Using Modal Analysis

Mr. Sharad D. Sakharekar[1], Dr. D.P. Tambuskar[2], Mr. Sagar D. Patil[3]
[1]Dept of Mechanical Engineering
[2]Professor, Dept of Mechanical Engineering
[3]Assistant Professor, Dept of Mechanical Engineering
[1, 2, 3] PHCET, Rasayani

Abstract- *The study includes design of pendulum test rig to improve stability of the structure. The purpose of this study was to meet ECE R29 regulation; hence the test rig was designed according to the norms. A 3D model of proposed pendulum test rig was structural stability and the setup was again analysed. Modified structure was found to be stable up to 10.91Hz which is well within acceptable criterion. This modified test setup showed a significant improvement in stability and met the required regulations. Since the test setup was modelled using hyper mesh 14.0 software. The test rig was further analysed for structural stability by modal analysis. After modal analysis it was observed that the designed baseline structure was not stable at the frequency of 10 Hz, hence design modifications were suggested to improve the designed with better stability this test rig can be recommended for further frontal impact test of HCV's.*

Keywords- ECE R29, Hyper mesh, LS-DYNA, Modal Analysis.

I. INTRODUCTION

Heavy Commercial Vehicles (HCV's) involve in large number of accidents. This results in severe injuries or even fatalities. Various vehicle regulatory bodies all over the world along with vehicle manufacturers are trying to make HCV designs safe. Various regulations, according to regions or countries exist like FMVSS (US), ECE (Europe), and AIS (India) etc. focus on the aspect typically to improve crash safety for HCV's. The regulations like ECE R29, AIS -29 etc. governfrontal crash safety for HCV's. In the last few years, safety concerns in truck cabin design have become more and more imperative. Among different types of truck collisions, frontal impact is one of the severest and causing several deaths to the drivers. Because of these accidents, a significant number of injuries and deaths occurred to the truck occupants. In order to ensure the safety of the occupant, certain design measures are necessary. These design measures need to be evaluated in terms for crash and overall strength of structure. Impact testing by a pendulum is important device that can be used to estimate vehicle sub-systems and predict how they will behave during a crash. Impact testing by a pendulum proposes flexibility and control in the design of the vehicle.

To understand severity of these crashes or accidents, there is need to look at data over the years regarding crashes. In 2015, 4067 people killed in crashes involving large trucks; there is a 4 per cent increment in the value from 2014. 116,000 people were injured in crashes in same year; an increase of 4% from a value of 111,000 in 2014. In 2015, 74% of people who were occupants of the other vehicles got killed in large-truck crashes. 78% of the severe crashes involving large trucks in 2015 occurred on weekdays. 2% of the large-truck drivers involved in fatal crashes had blood alcohol concentrations (BACs) of 0.08 g/ dL or higher which is much lower than other vehicle types drivers (21%, 20% and 27% for passenger cars, light trucks, and motorcycles respectively). Large truck drivers involved in lethal crashes in 2015 had the most noteworthy rate (20.1%) of beforehand recorded accidents contrasted with drivers of other vehicle types (motorcycles, 18.9%; passenger cars, 18.3%; and light trucks, 16.7%). In 2015 large trucks will probably be included in fatal multiple vehicle crashes instead of lethal single-vehicle crashes than were passenger vehicles (82% of lethal accidents including large trucks are multiple vehicle crashes, compared with 61% for fatal accidents including passenger vehicles). [HYPERLINK \l "Tra17" **1**]

A. Need

Analysis of truck driver fatality and injury in crashes provides a better understanding of how injury occurs. Regulatory bodies focus on reducing the number of truck occupant fatalities and the severity of injuries. The high kinetic energy content associated with severe truck crashes results in dissipation of forces. These forces exceed the crashworthiness of cab structures to resist deformation thereby compromising survival space which increases occupant injury risk. There are practical limits in the ability of cab structures to manage these high energy events. These impact force needs to be analysed and subsequently cab crashworthiness should be checked against these forces. Hence sufficient measures need to be taken for the safety of the occupants which will be accomplished when the crash analysis test rig is reputed as per standard regulations and norms.

II. LITERATURE SURVEY

The safety analysis contained in NHTSA report is based on the University of Michigan Transportation Research Institute's (UMTRI's), Trucks Involved in Fatal Accidents (TIFA) survey file, NHTSA's General Estimates System (GES) file and the Large Truck Crash Causation Study (LTCCS). TIFA and GES data years 2006 – 2010 were used in the categorical analysis while the LTCCS data was used for an additional clinical review of cabin behaviour in frontal and rollover crash. Results showed that the cabin strength has improved significantly because of improvements in structural analysis, design development and durability analysis. This study did not include calculation of forces or accelerations imparted on truck cabins or occupants; however the research noticeably recognizes crash scenarios and injury mechanisms that can be tied to probable counter measures including cab crashworthiness. [2]

Mirzaamiri et al [3] found during the design and development of truck cabins, the safety of the driver and the front seat passenger in an accident is an important task and should be considered. The study aims at investigating the behaviour of Iran Khodro (IKCO) 2624 truck. An FE model was developed to simulate ECER29 regulation for Iran Khodro 2624. The 3D cad model of different parts of the truck was made and assembled together to construct the whole structure of the truck cabin. The verification of FE model was carried out by an indirect method. A similar FE model was prepared for C1500 Pick-up Truck, and it underwent similar Tests. The obtained results were compared with experimental data available in literature. The good agreement between the measured and computed results indicates that the developed model of Iran Khodro 2624 is somehow accurately made.

Hitchings et al[4]conduct tests on three different vehicle subsystems and presented as examples of impact testing that has been performed using a pendulum. This paper discuss about the use of a pendulum as a repeatable method for impact testing vehicle subsystems and components. Impact testing on two different roll bar designs to compare effectiveness, on various van side cargo doors to compare door latch integrity, and on various fuel tanks to compare the integrity of the fuel tank with and without shields installed. These tests were not supposed to calculate the forces, energy, or impact pulse on the vehicle during actual collision. It evaluates vehicle subsystem's reaction to impacts that were similar to the collision which occurred.

DevendraGendar[5] suggested a numerical solution using non-linear explicit methods to reduce the number of tests during the design process. Driver and front seat passenger safety is given a prime importance for design of truck cabin. The cabin must be designed in such a way that adequate survival space is provided for the occupants in event of crash. A numerical simulation using non-linear explicit code helps in predicting deformation of driver cabin and load of individual components within the elastic and plastic range of the material. In this paper, application of this numerical simulation method with example of one of the cabin is presented and compared to the results from the physical test carried out as per ECE R-29 regulation.

Sharma et al.[6] have designed and developed FE model of HCV cabin and validated it with tests according to AIS029. Their research also includes the evaluation of the energy absorption capabilities of the HCV cabin during the pendulum impact test. Author suggested that the safety of the HCV occupants in an accident is an important task and during the design and development of cabins it should be considered. In this study, firstly a detailed 3D FE model of HCV cabin was developed. Then pendulum impact test simulation was carried out using LS-Dyna explicit solver. FE model with proper non-linear material properties were used to simulate the test scenario. Required survival space was provided in the FE model and in the test after the impact. The load transfer path was described at the time of pendulum impact. The largest amount of impact energy which was more than 70% was absorbed by the frontal region of the cabin. Simulation results were compared with the test results. Adequate correlation was found between test and CAE results in terms of structural deformation and survival space. Results show that the front cabin suspensions were under tensile load while rear cabin suspensions sustained compressive load during frontal pendulum impact. Peak acceleration was found to be less than 30g which was within the safe limit at the time of real accidents.

Cerit et.al[7]performed frontal crash analysis of the structure of a bus front body as per ECE-R29 regulation norms and the strength of the structure was checked whether the safety requirements are satisfied. For this analysis, nonlinear explicit finite element code LS-DYNA was used. Frontal crash analysis of the baseline structure without any modifications was studied in the first stage and weak parts of the front body were

From the extensive literature survey it has been observed that most of the work has been done to evaluate the effect of impact test on vehicle cabin and subsystems. There is no substantial data regarding design and development of the pendulum impact test setup. The simulation and analysis work of pendulum test rig has not been found in literature. Hence crash analysis topic can be taking further with designing and simulating with the help of software. This work consist development and simulation of FE model of pendulum impact test rig.

The frontal crash scenarios in HCV are tested in laboratory environment using a suspended load case of pendulum impact on front side of vehicle. The pendulum impactor in this case needs to be very large in size to generate equivalent forces on the vehicle that are generated during crash. The equivalent force should be 45 KJ as per ECE R29 regulation[8]. Thus the test device has to be specifically designed to generate accurate & variable impact forces for each test conducted. Hence test setup should be tested under impact analysis to evaluate variable impact forces. It is necessary that the pendulum impactor to be dimensionally very stable even after repetitive test loads. As the impactor is heavy various safety considerations need to be addressed. Hence pendulum test setup should be tested under modal analysis for better stability. Further it should meet the test setup requirements specified in regulation ECE R29.

The current paper deals with development of pendulum impact test rig model and further simulations of design using FEA techniques such as modal analysis. The aim of the work is to develop and analyse pendulum test rig as per ECE R29 regulation. The above aim is fulfilled considering the following objective.

1. To develop 3D model of pendulum test according to ECE R29 specifications.
2. To simulate the model by use of FEA techniques.
3. To test the setup under modal analysis for stability of the structure.

III. DESIGN OF PENDULUM IMPACT TEST RIG

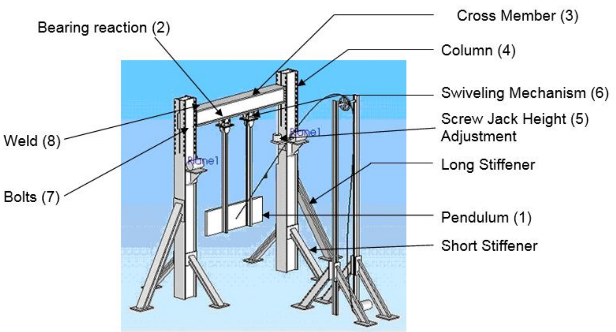


Figure 1 Schematic diagram of Pendulum test rig

Selection of material for structural components of the rig: Ductile materials have property of deformation to greater values without failure. Thus, they have greater ability to resist impact loads. Therefore mild steel (M.S.) is selected as material for the structural components of the rig. Mild steel has following material properties:

Yield strength: 245 MPa

Modulus of elasticity: 210 GPa

Density: 7850 N/mm2

The only data available from ECE- R29 was:

(a) Mass & dimensions of the pendulum plate.
(b) Length, web height and spacing of the I-beams.
(c) Energy to be dissipated by the plate.

From this data, the impact force was to be calculated for the design of all components. Hence, it was calculated from a) the Principle of Kinetics, b) Vibration Analysis, and c) Collision Theory.

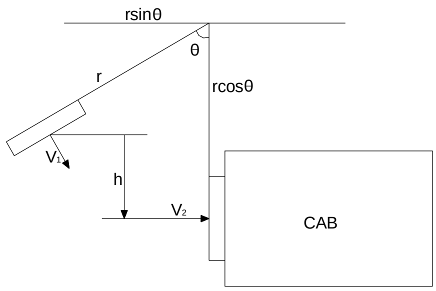


Figure 1Pendulum impact parameters

A. Evaluation of important parameters

E1 = PE1 = mgh

h = 3.058 m

E = Impact Energy = 45000 J.

H = Height of Release = 3.058 m.

θ = Angle of Impact = 82^0.

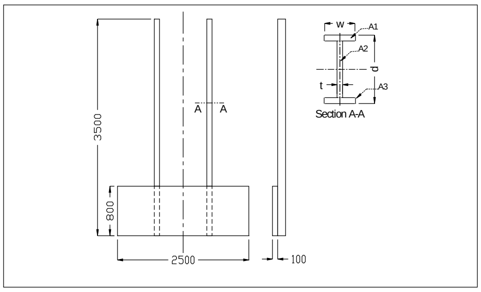


Figure 2schematic of pendulum assembly

B. Angular velocity of Pendulum

Maximum energy that the pendulum must possess as specified by standard, at the time of impact is 45 kJ. This energy is stored in pendulum in the form of Potential energy (P.E.) when it is in the lifted position. It goes on converting into Kinetic energy (K.E.) as it swings down due to gravity. When pendulum is in its vertical position all P.E. gets converted into K.E.The Kinetic energy in the terms of Mass moment of inertia and angular velocity,

C. FE MODEL PREPERATION

Vertical Beam Assembly

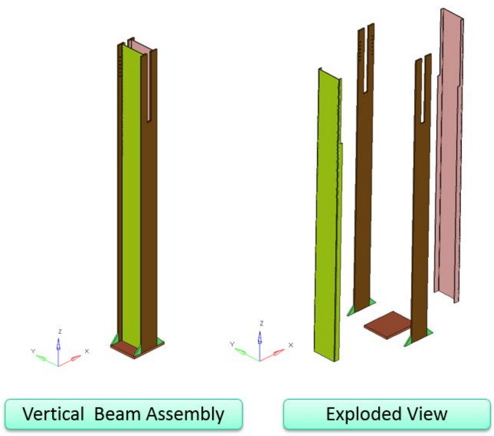


Figure 3 Vertical beam Assembly

Figure4 shows Vertical Beam assembly which is made with various plates & gussets. The attachments are done by welding.

Horizontal Beam Assembly

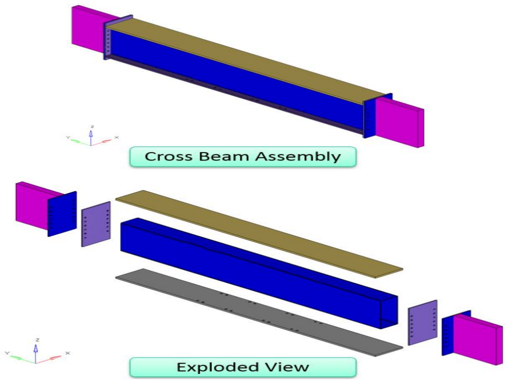


Figure 4Horizontal Beam Assembly

Figure 5 shows Horizontal Beam assembly which is made with various plates & gussets. The attachments are done by welding.

Pendulum Assembly

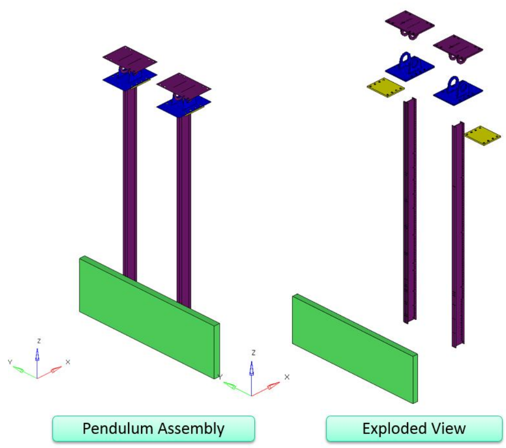


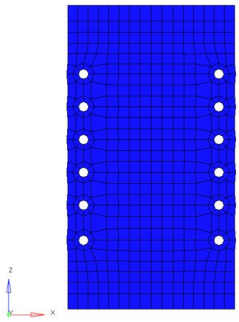
Figure 5Pendulum Assembly

The Figure 6 shows pendulum assembly which is designed so as to meet the impact requirement and to provide articulation at the top for swinging action.

IV. FE MODELLING OF TEST SETUP

A. Meshing

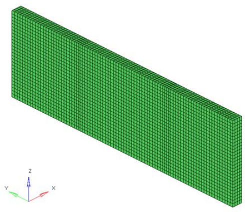
The flat plates are typically modelled with 2D shell elements in FE discretization. A typical mesh part is as shown in Figure 6.



2D Shell Element Quality Parameters		
Parameter	Acceptable Limit	Achieved
Min. Length	> 5 mm	8.8 mm
Max. Length	< 50 mm	49.84 mm
Warpage	< 5 deg	0.006 deg
Min. Angle Tria	> 25 deg	29.77 deg
Max. Angle Tria	< 120 deg	96.12 deg
Min. Angle Quad	> 45 deg	45.01 deg
Max. Angle Quad	< 135 deg	134.7 deg
Jacobian	> 0.6	0.6
Aspect	< 5	3.22

Figure 62D shell element meshing

The Pendulum mass is only modelled with 3D Solid elements. The meshing & quality parameters are as shown in Figure 7.



3D Solid Element Quality Parameters		
Parameter	Acceptable Limit	Achieved
Min. Length	> 5 mm	21.25 mm
Max. Length	< 50 mm	31.94 mm
Warpage	< 5 deg	4.84 deg
Min. Angle Tria	> 25 deg	NA
Max. Angle Tria	< 120 deg	NA
Min. Angle Quad	> 45 deg	86.31 deg
Max. Angle Quad	< 135 deg	94.81 deg
Jacobian	> 0.6	0.95
Aspect	< 5	1.47

Figure 73D solid element meshing

B. MATERIAL SELECTION

Materials used by other components:

Materials play a vital role in designing a pendulum test rig; the materials are imparted to the components according to its importance and function from the LS-Dyna material library. The

Table 1Materials used by other components

Sr. No.	Material Name	Card Image	ρtonns/mm³	EMPa	σ MPa	% Elongation
1	MAT1-Steel	MATL1	7.85e-9	210000	-	-
2	Bob Material	MATL1	7.24e-9	210000	-	-
3	Bob null	MATL9	7.85e-9	210000	-	-
4	Mat 20	MATL20	8.57e-9	210000	-	-
5	Rigid wall	MATL20	7.85e-9	210000	-	-
7	Mat MS	MATL24	7.85e-9	210000		
9	340 XF	MATL24	7.85e-9	210000	340.680	22
10	Spring	SDMAT1	7.85e-9	210000	-	-

Error! Reference source not found.shows the list of materials used by different components of the pendulum test rig.

The materials used for the critical components of the pendulum test rig play a vital role in the safety of the occupant; hence according to the test, the materials were selected from the LS-Dyna material library as shown inTable 1.

Table 1Materials used for critical components

Sr. No.	Material Name	Card Image	ρtonns/mm³	EMPa	σ MPa	% Elongation
1	420 XF CRS	MATL24	7.85e-9	210000	420.840	19
2	830 R CRS	MATL24	7.85e-9	210000	830	2

C. Connections

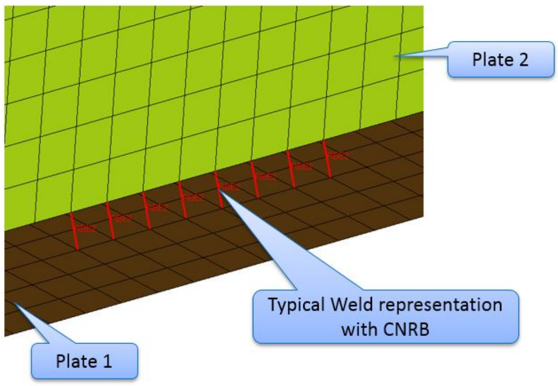


Figure 8 Typical Welded joint FE modelling

The typical connections used for assembly are welded joints & bolted joints. The FE representation of weld is as shown in Figure 8. Spot weld connection are normally used between relatively thin sheet metal components. Rigid elements are used to model spot weld.

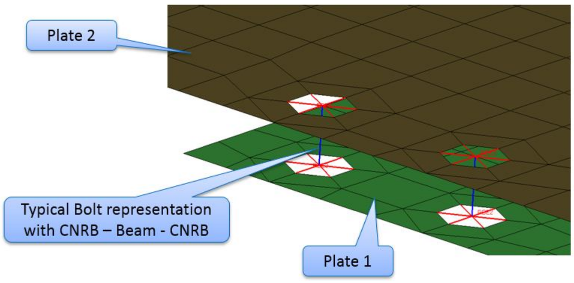


Figure 9Typical bolted joint FE modelling

Figure 9shows the representation of bolt assembly with constraints nodal rigid bodies. The FE representation of bolted joints is as shown in Figure 9. Centre of bolt is connected to inner and outer layers via beam elements of diameter equal to ‘d’ and 2 times ‘d’ (d=core diameter) as shown in Figure 9.

V. MODAL ANALYSIS RESULTS

The base line model is designed using Finite Element Method and further iterations are done taking the basic boundry condition. In first iteration only the baseline model is considered and analysed for its structural stability. The results from the first iteration illustrates that the structure is not sufficiently stable. Further second iteration is carried out in which the baseline model is designed with two stiffeners giving the structure more stability as compared to baseline model but again it fails to meet the required stability and further two more iteration are carried out considering more number of stiffeners in the design and the increasing the thickness of the column structure of the rig. Finally the results from the fourth iteration show that the structure is stable.

A. FE Model Set-up

A Constrained Modal (Frequency) Analysis is done to observe the mode shapes of the structure.

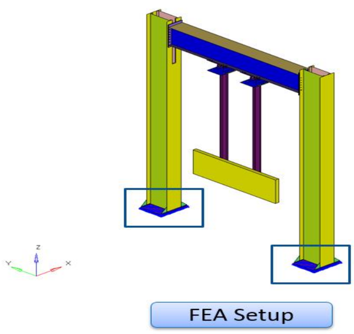


Figure 10 FE model setup

The basic test rig is designed as per ECE R29 regulation is illustrated in Figure 10consists of two vertical column, horizontal column and pendulum assembly.

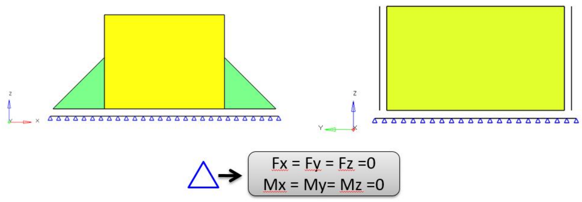


Figure 11 Boundry condition of baseline model

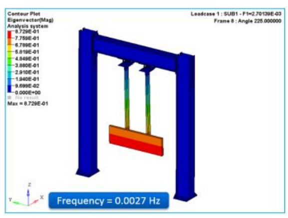
Figure 11shows the boundary condition of baseline model where force and moment in all degree of freedom is constrained. Forces and moment in all direction i.e. Fx, Fy, Fz and Mx, My, Mz are zero.

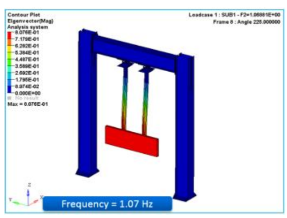
Baseline Design Results

B. FE Model

The basic structure has been designed as per the ECE R29 regulation and it has been shown in the Figure 12. The basic structure of pendulum beam assembly consists of vertical and horizontal plates and gussets which are attached by welding. In this assembly the base of vertical plates are constratined as shown in Figure 12

Figure 12Baseline model





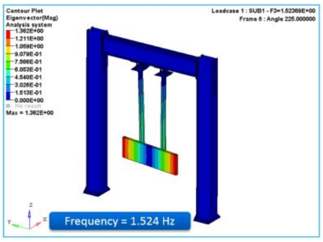


Figure 13 Mode 1,2,3 of baseline model

Figure 13shows mode 1, mode 2 and mode 3 where the frequency values are 0.0027 Hz, 1.07 Hz and 1.524 Hz respectively. In first three modes of analysis, which are local modes, the impact force comes on pendulum assembly, but as stability of whole structure is to be validated, therefore further test on the whole structure taken into consideration. The stability of structure is depending upon the frequency of a structure and the pendulum is supposed to be in swing motion. During the impact the frequency of the whole structure is to be measured. As first three modes are on pendulum assembly they are treated as local modes.

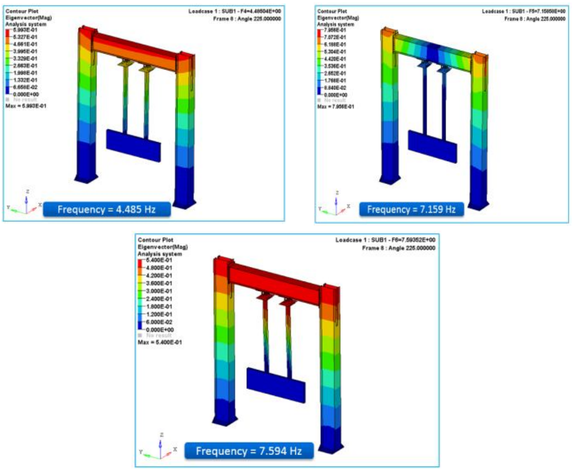


Figure 14 Mode 4, 5, 6 of baseline model

Figure 14shows mode 4, mode 5 and mode 6 where the frequency values are 4.485 Hz, 7.159 Hz and 7.594 Hz respectively. Fourth mode shows the first structural mode. As per recommendation mentioned in ECE R29 frequency on first structural mode should be greater than 10Hz. However, the frequency on fourth mode is 4.485Hz which is less than 10Hz hence to achieve structural stability modification in the design is being done by incorporating long stiffener to column.

C. Iteration 1 Design Results

The first iteration design structure has been designed as per the ECE R29 regulation and it has been shown in the Figure 15. The first iteration design structure of pendulum beam assembly consists of vertical and horizontal plates and gussets which are attached by welding. In this assembly, long stiffener attached to its end of vertical column as shown in Figure 15.

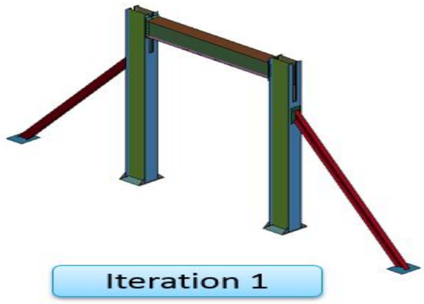


Figure 15 First iteration design

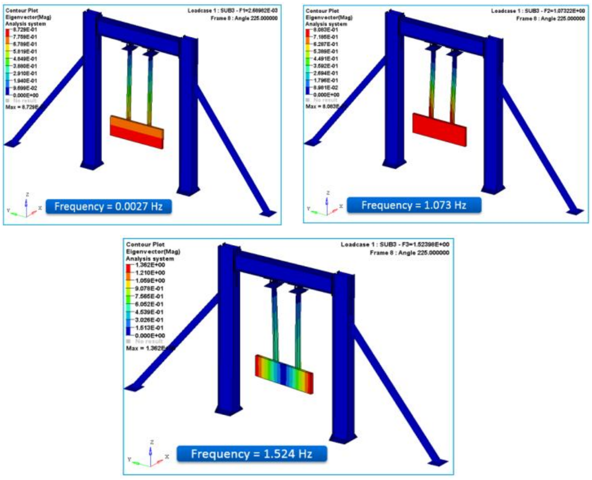
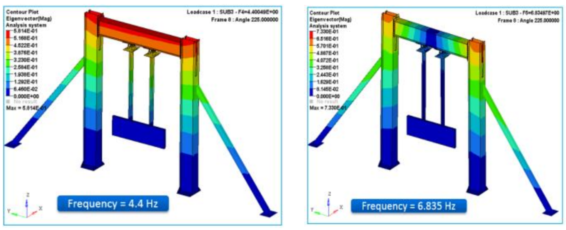


Figure 16 Mode 1,2,3 of first iteration design

Figure 16shows mode 1, mode 2 and mode 3 where the frequency values are 0.0027 Hz, 1.073 Hz and 1.524 Hz respectively. In first three modes of analysis, which are local modes, the impact force comes on pendulum assembly, but as stability of whole structure is to be validated therefore further test on the whole structure taken into consideration. The stability of structure is depending upon the frequency of a structure and the pendulum is supposed to be in swing motion. During the impact the frequency of the whole structure is to be measured. As first three modes are on pendulum assembly they are treated as local modes.



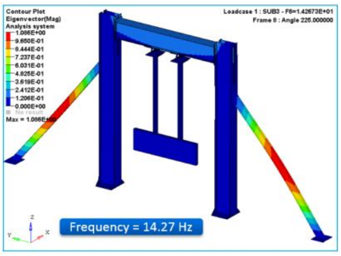


Figure 17 Mode 4, 5, 6 of first iteration design

Figure 17shows mode 4, mode 5 and mode 6 where the frequency values are 4.4 Hz, 6.835 Hz and 14.27 Hz respectively. Fourth mode shows the first structural mode. As per recommendation mentioned in ECE R29 frequency on first structural mode should be greater than 10Hz. However, the frequency on fourth mode after adding stiffener is 4.4Hz which is less than 10Hz hence to achieve structural stability modification in the design is being done by incorporating short stiffener to front side of column.

D. Iteration 2 Design Results

The second iteration design structure has been designed as per the ECE R29 regulation and it has been shown in the Figure 18. The second iteration design structure of pendulum beam assembly consists of vertical and horizontal plates and gussets which are attached by welding. In this assembly a short stiffener attached to its front middle span of vertical column as shown in Figure 18.

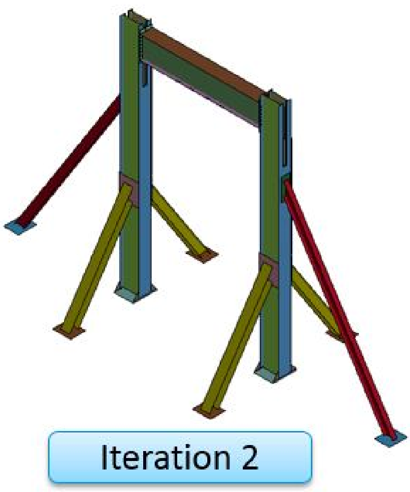


Figure 18 Second iteration design

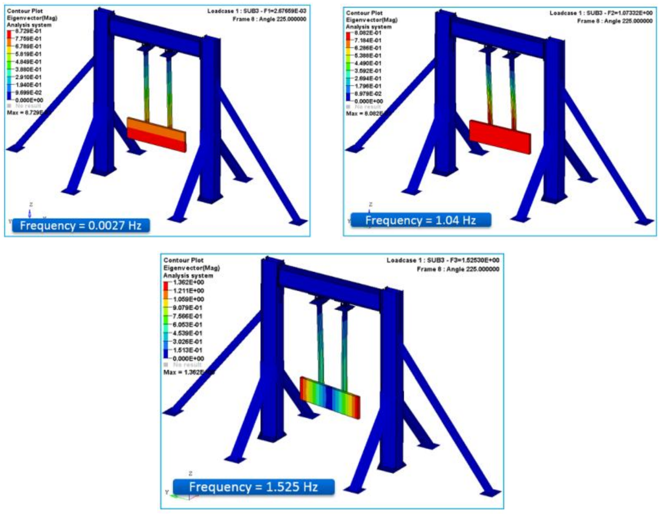


Figure 19 Mode 1, 2, 3 of second iteration design

Figure 19shows mode 1, mode 2 and mode 3 where the frequency values are 0.0027 Hz, 1.04Hz and 1.525Hz respectively. In first three modes of analysis, which are local modes, the impact force comes on pendulum assembly, but as stability of whole structure is to be validated therefore further test on the whole structure taken into consideration. The stability of structure is depending upon the frequency of a structure and the pendulum is supposed to be in swing motion. During the impact the frequency of the whole structure is to be measured. As first three modes are on pendulum assembly they are treated as local modes.

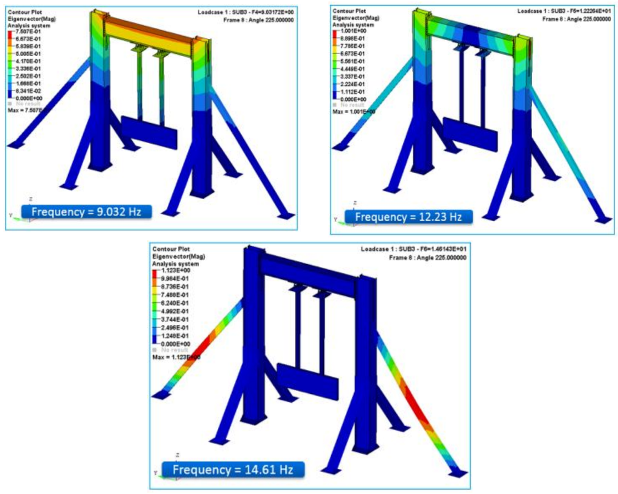


Figure 20 Mode 4,5,6 of second iteration design

Figure 20 shows mode 4, mode 5 and mode 6 where the frequency values are 4.4 Hz, 12.23 Hz and 14.61 Hz respectively. Fourth mode shows the first structural mode. As per ECE R29 frequency on first structural mode after adding short stiffener to column should be greater than 10Hz. However, the frequency on fourth mode after adding stiffener

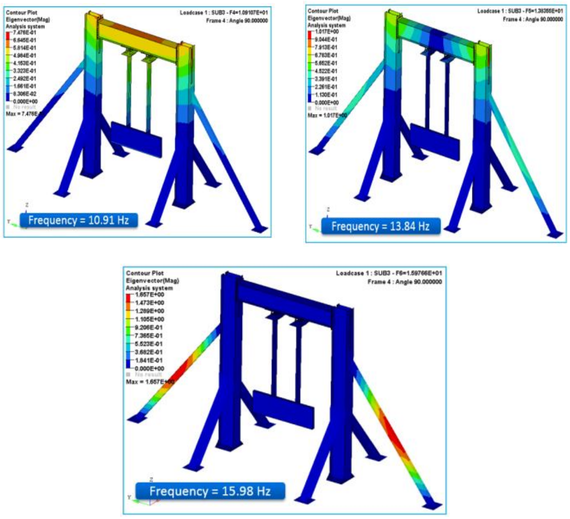


Figure 23 Mode 4,5,6 of modified design model

is 4.4Hz which is less than 10Hz hence to achieve structural stability modification in the design is being done by adding a plate on column.

E. Iteration 3 Design Results

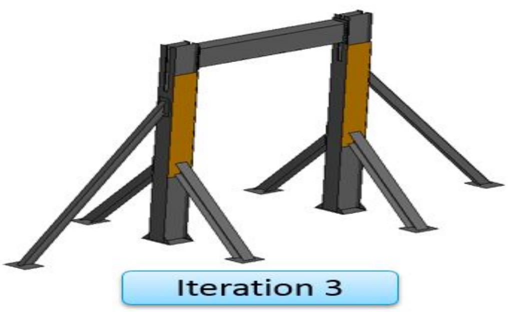


Figure 21 Modified design model

The third iteration design structure has been designed as per the ECE R29 regulation and it has been shown in the Figure 20. The third iteration design structure of pendulum beam assembly consists of vertical and horizontal plates and gussets which are attached by welding. In this assembly material plates are attached to its front middle span of vertical column for enhanced stability of structure as shown in Figure 20.

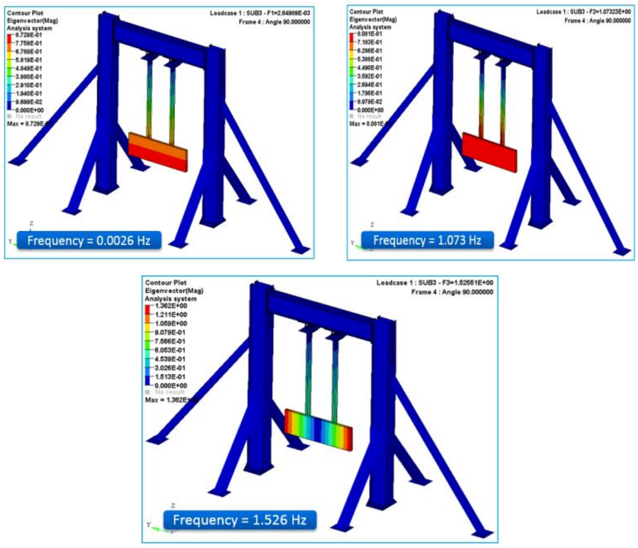


Figure 22 Mode 1, 2, 3 of modified design model

Figure 22shows mode 1, mode 2 and mode 3 where the frequency values are 0.0026 Hz, 1.073Hz and 1.526Hz respectively. In first three modes of analysis, which are local modes, the impact force comes on pendulum assembly, but as stability of whole structure is to be validated therefore further test on the whole structure taken into consideration. The stability of structure is depending upon the frequency of a structure and the pendulum is supposed to be in swing motion. During the impact the frequency of the whole structure is to be measured. As first three modes are on pendulum assembly they are treated as local modes.

Figure 24 shows mode 4, mode 5 and mode 6 where the frequency values are 10.91 Hz, 13.84 Hz and 15.98 Hz respectively. Fourth mode shows the first structural mode. As per ECE R29 frequency on first structural mode after adding plate to column meet the requirement of 10Hz frequency.

F. Observation

The results for baseline design as well as all the iterations are tabulated as given in the table below.

Table 2 Modal Analysis Results

Mode	Baseline	Mode Shape	Iteration 1	Mode Shape	Iteration 2	Mode Shape	Iteration 3
1	0.0027	Local	0.0027	Local	0.0027	Local	0.0027
2	1.07	Local	1.073	Local	1.04	Local	1.073
3	1.524	Local	1.524	Local	1.525	Local	1.526
4	4.485	Fore & Aft	4.4	Fore & Aft	9.032	Fore & Aft	10.91
5	7.159	Twist	6.835	Twist	12.23	Twist	13.84
6	7.594	Lateral	14.27	Mixed	14.61	Mixed	15.98

It is observed that:

- The first three modes are the local modes of the pendulum assembly as it is free pivot from the Horizontal Beam.
- The fourth mode represents the first structural mode.
- It is desirable to have first structural frequency greater than 10 Hz. to have more stability during the impact loading.
- Iteration 3 shows the first structural mode as Fore and Aft of 10.91 Hz which is greater than requirement of 10 Hz.
- Hence the Iteration 3 design meets design requirements and will be further evaluated for impact loading scenario.

VI. CONCLUSION

The design of the ECE R29 test rig started with classical approach. It gives the basic design which can withstand the test load of 45 kJ. Based on the Modal analysis it was observed that the baseline design shows very low frequency for first structural mode. This may induce vibrations & instability during the impact testing. Hence to avoid this design was improved by adding stiffeners. Thus the Iteration 3 design gave us the desired first structural mode of greater than 10 Hz. It was observed that the pendulum arm was losing dimensional stability in impact which was ascertained by the permanent strains induced in it. The current test rig is designed to sustain only 45 kJ of impact energy. The design can be improved to meet energy requirement for various vehicles which need to be tested for higher energies than 45 kJ. The design of Horizontal Beam & Vertical Beams is based on classical design approach and there is a scope to optimize the design. The impact analysis is carried on the rigid plate; further study is required to evaluate the test rig behaviour on actual truck cabins. Effect of fatigue and durability was not considered in the current study and may be necessary as the test device has to repeatedly use for testing. Use of alternate materials like Cast Iron, Aluminium etc. can be evaluated to design the test rig.

REFERENCES

[1] "Traffic Safety Facts," National Highway Traffic Safety Administration, February 2017.

[2] J. Woodrooffe, and D. Blower, "Heavy truck crashworthiness: injury mechanisms and countermeasures to improve occupant safety.," National Highway Traffic Safety Administration.(NHTSA), May 2015.

[3] M.Esfahanian, S. Ziaeni-Rad R. Mirzaamiri, "Crash Test Simulation and Structure Improvement of IKCO 2624 Truck According to ECE R29 regulation," International Journal of Automotive Engineering, pp. 180-192, 2012.

[4] Clarence R. Hitchings and Jerry G. Wallingford, "A Discussion On Using A Pendulum as a Method for Impact Testing Vehicle Sub-systems," SAE Standard, SAE 01-0687, March 2002.

[5] Devendra Gendar, "Numerical Simulations for Testing of commercial Vehicle as per ECE- R29 Regulation," SAE International, 2007.

[6] S. Sharma, S. Tiwari, and U. Gupta, "Finite Element Simulation and Validation of Fully Suspended Heavy Duty Commercial Vehicle (HCV) as per AIS029 Pendulum Impact Test," SAE International, SAE Technical Paper 2015-01-2873.

[7] Muhammed E. Cerit , Mehmet A. Guler, and Bertan Bayra, "Improvement of the Energy Absorption Improvement of the Energy Absorption," in 11th International LS-DYNA Users Conference, Turkey, 2010, pp. 15-24.

[8] "ECE R29 Regulation uniform provisions concerning the approval of vehicles with regards to the protection of the occupants of the cab of a commercial vehicle,".