IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 1 Issue 3, May 2014.

www.ijiset.com

ISSN 2348 - 7968

Structural Design and Analysis of Impact Test Rig for Road Transport Vehicles

B.SUDHA BINDU

ABSTRACT

Road transport vehicles experience impact loads due to speed breakers, pot holes, etc. on roads. When the vehicle passes over these the critical sub systems like transmission system, which in turn consists of gearbox, engine, etc will be subjected to impacts. These impacts will in turn cause excessive displacement of the sub systems, which may lead to permanent structural failure. However once the sub systems are realized their design adequacy is required to be ascertained against these impact environments by experimental means. Hence a test rig is essentially required to simulate those impacts, which will be experienced by the road transports during their course of application. Structural design of road transport impact test rig is taken up in this project to meet the requirement. It is understood that many of test facilities which are being used to simulate impact environments makes use of either hydraulic or pneumatic actuation means which makes their realization and operation relatively complex. In principle the proposed design makes use of gravity and hence realization of such test rig will be very much feasible. After evolving the design structural analysis of impact test rig is carried out using Finite Element Method (FEM) in ANSYS software. Structural analysis comprises of static, modal and harmonic analyses. Analysis is carried out against the specified loads. The objective of the structural analysis is to assess the design adequacy against the functional loads and to validate the design calculations. Apart from this modal analysis reveals the first natural frequency of the rig, which needs to be ascertained that it is away from operational frequency.

1.1 INTRODUCTION

Road transport vehicles will be subjected to impact loads during their course of application. These impact loads will be experienced due to speed breakers, pot holes,etc. on roads. When the vehicle passes over these the critical sub systems like transmission system, which in turn consists of gearbox, engine, etc will be subjected to impacts. These impacts will in turn cause excessive displacement of the sub systems, which may lead to permanent structural failure.

Sl. No.	Parameter	Specification
1.	Maximum impact magnitude	300 m/sec^2
2.	Time to attain acceleration	15 milliseconds
3.	Deceleration	39.24 m/sec^2
4.	Maximum weight of unit under test	180 Kg

Table 1.1 Specifications of the proposed design

2. LITERATURE SURVEY

Literature survey is broadly classified into.

- Impact
- Impact testing

2.1.1 Impact

A mechanical or physical impact is a sudden acceleration or deceleration caused, for example, by impact, drop, kick, earthquake, or explosion. Impact is a transient physical excitation.

Figure 2.1 Free fall impact test machine

Fig 2.2 Drop machine

Limitations of Free fall impact testing machine

- Oil leakage problems
- Lack of repeatability

Pneumatic impact test machine: A typical pneumatic impact-testing machine is shown in Figure 2.10.

Figure 2.3 Pneumatic impact test machine

These machines will produce a impact pulse in the vertical direction using compressed air to force the carriage to impact on the impact machine base. Elastomer pads are used between the carriage and the base in the impact area to produce half-sine pulses, lead pellets for sawtooth pulses and pneumatic cylinders for square pulses. The design of the programmers affect the time duration of the pulse while the air pressure and drop height are used to accelerate the carriage to determine the magnitude of the impact pulse.

Limitation of Pneumatic impact testing machine

- High-pressure air supply set up is needed

3. STRUCTURAL DESIGN OF ROAD TRANSPORT IMPACT TEST RIG

3.1 DESIGN PHILOSOPHY

As discussed earlier the present design concept is evolved based on the principle of simple pendulum. A typical simple pendulum is shown in Figure 3.1.

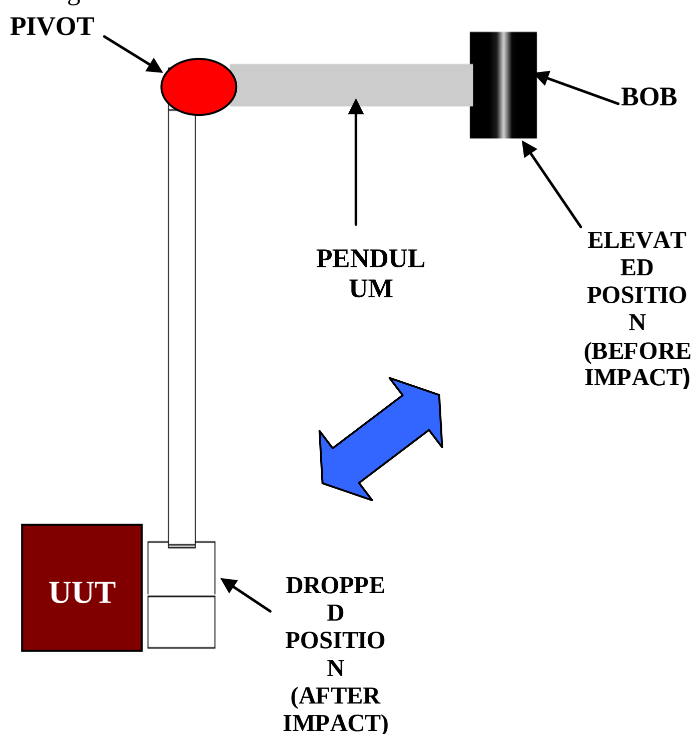


Figure 3.1 Typical simple pendulum

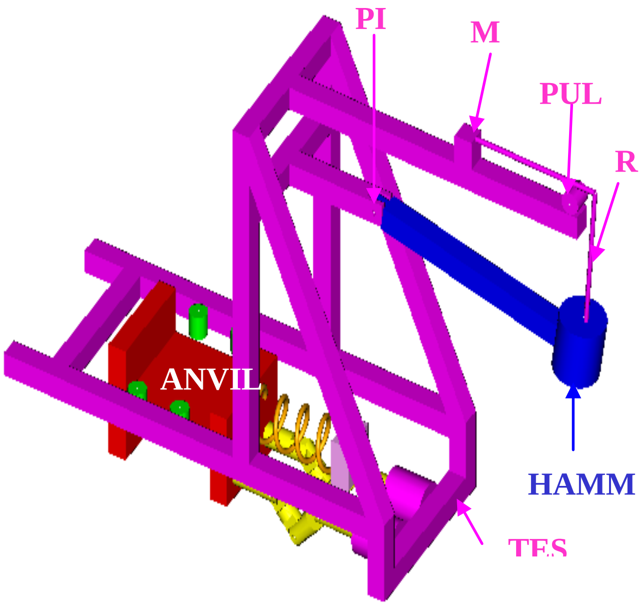
When it is raised to a certain height and dropped it will come back to equilibrium position due to gravitational torque. When it is at elevated position its velocity will be minimum and when it reaches dropped position its velocity will be maximum. If any component is held at the dropped position in such a way that pendulum bob hits when it reaches that position this velocity will be imparted to the component in form of impact energy. This principle

ISSN 2348 - 7968

is exploited for evolving the design of impact test rig for road transport systems

3.1 DESIGN CONFIGURATION

The proposed design configuration is shown in Figure 3.2.



These critical elements in exploded view are shown in Figure 3.3 for better understanding.

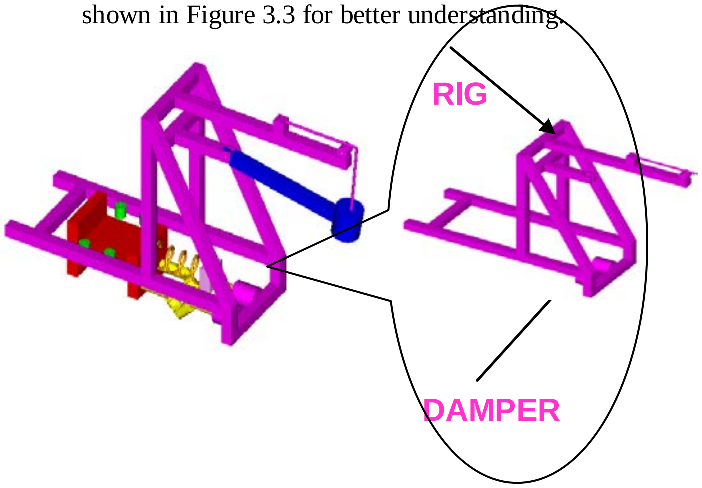


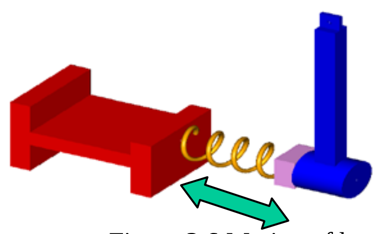
Figure 3.4 Exploded view

Rig serves as basic load supporting platform and it houses all the critical elements. Rigidity of the test rig is essentially required in order to ensure effective transfer of intended motion between hammer and the unit under test. Bottom surface of the rig will be anchored to the ground for providing rigidity to the test rig.

Summary of construction details is given below.

- Hammer is attached through a pivot to the rig
- Unit to be tested will be placed on anvil
- Anvil is attached to bottom of rig through rollers
- Spring is held between anvil and block
- Linkage is held between anvil and damper

There are four basic motions involved in the rig. They are



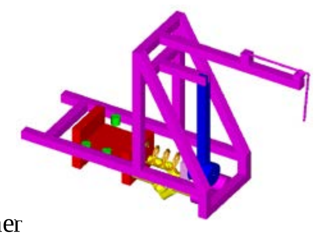
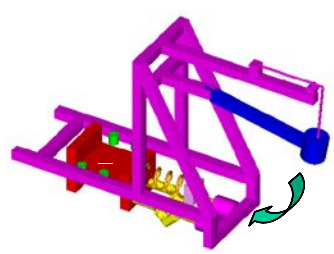


Figure 3.6 Motion of hammer



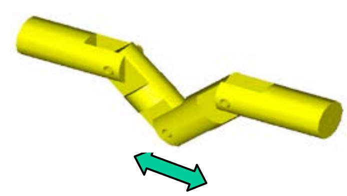


Figure 3.7 Motion of spring

Figure 3.8 Motion of Linkage

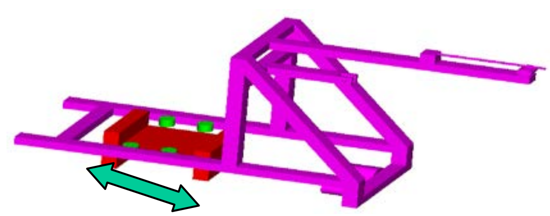


Figure 3.9 Motion of Anvil

Steps in operating sequence are given below.

- Motor ON
- Hammer retracts back and held in normal position
- Hammer will be released
- Hammer impacts block and in turn spring
- Spring converts impact to pure sinusoidal motion
- Anvil and unit under test moves with help of roller
- Roller permits specified motion of anvil
- At the end anvil attains desired final velocity
- Retardation of anvil
- Motion will be cushioned by damper through linkage
- Rope gets attached with hammer
- Motor ON
- Hammer retracts back and held in normal position
- Motor OFF

4. STRUCTURAL ANALYSIS OF TEST RIG

As the rig is the basic supporting platform and as it experiences the loads, analysis is carried out for rig with out other subsystems.

The following analyses are carried out for the test rig.

- Static analysis
- Modal analysis
- Harmonic analysis

4.1 Design constraints

- Minimum desired factor of safety against static and harmonic loads is 1.5.
- First natural frequency of the test rig must be $\geq$ operational frequency (Frequency corresponding to impact pulse i.e. 1/0.015 sec = 66.67 Hz).

Load

Maximum impact amplitude is 300 m/sec^2. In the commercial software package i.e. ANSYS impact amplitude load is applied as inertia load. FE model is shown in Figure 4.3.

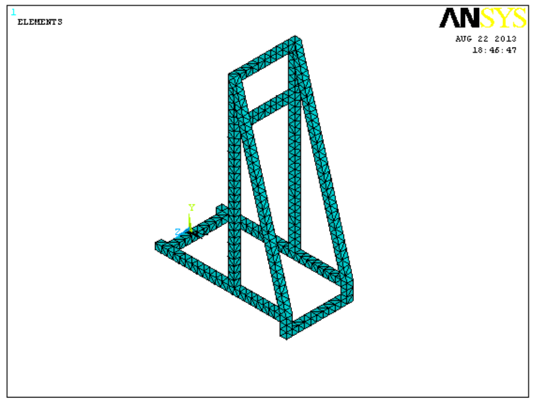


Figure 4.1 FE model of test rig

The FE model is then solved for Von Misses stress and displacement using ANSYS software. Maximum stress plot is shown in Figure 4.2 in which maximum stress location is visible in red color.

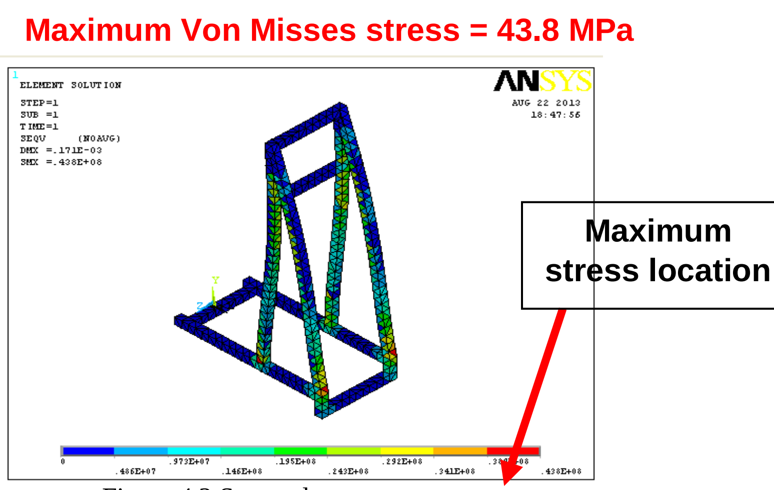


Figure 4.2 Stress plot

Maximum displacement plot is shown in Figure 4.3.

ISSN 2348 - 7968

Maximum Displacement = 0.17 mm

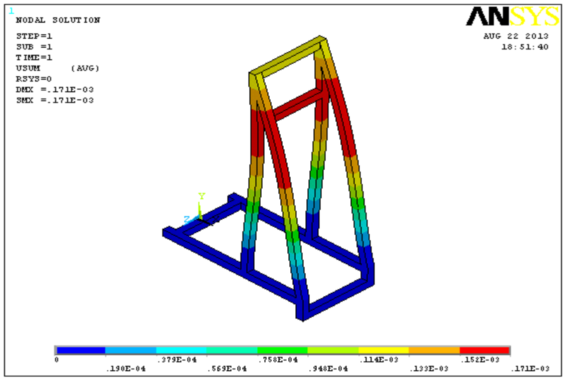
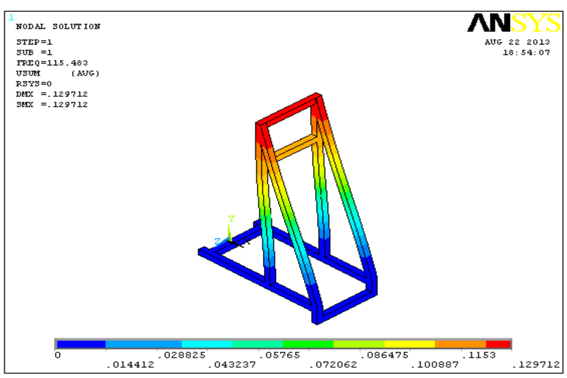


Figure 4.3 Displacement plot

Modal Analysis: The primary objective of modal analysis is to obtain the natural frequencies and mode shapes. Modal analysis of the test rig is carried out to in order to check the interference of first natural frequency of the test rig with that of operational frequency in order to avoid resonance situation which leads to vibration with higher amplitude and can effect the intended functioning of the test rig. Same FE model used for carrying out static analysis is extended for modal analysis. Then the model is solved for natural frequencies and mode shapes using Block Lanczos algorithm of ANSYS software. Mode shape corresponding to first natural frequency is shown in Figure 4.4.

Mode 1: Frequency = 115 Hz

Figure 4.4 Mode shape at first natural frequency



Maximum Von Misses stress = 133 MPa

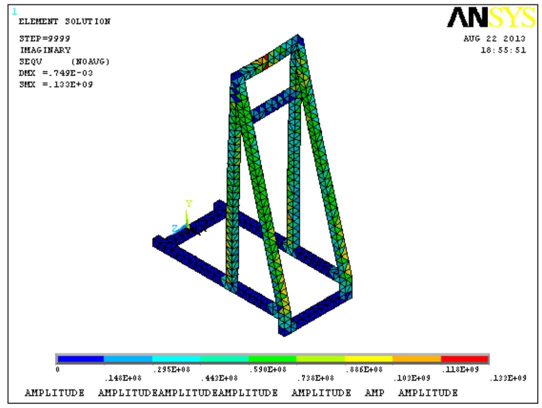
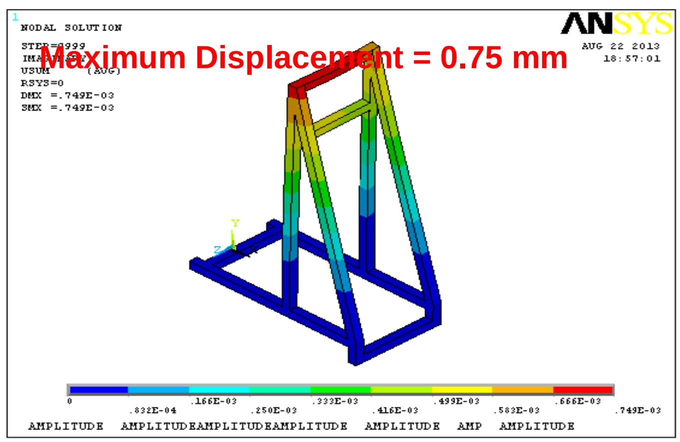


Figure 4.4 Mode shape at first natural frequency

Harmonic Analysis: After ensuring that the design is safe against static load, it is essential to ascertain the design adequacy against harmonic loading condition. For meeting this requirement harmonic analysis is carried out for test rig against harmonic load of 75000 N. As an outcome of the harmonic analysis maximum stress and displacement are obtained. Same FE model used for carrying out modal analysis is extended for harmonic analysis. Stress plot is



shown in Figure 4.5.

Figure 4.5 Stress plot Maximum displacement plot is shown in Figure 4.6.

5. SUMMARY OF RESULTS

The results are segregated into the following two categories for summarizing.

- Design parameters
- FE analysis results

The outcome of design calculations are summarized in Table 5.1.

Sl. No.	Description	Value
1.	Spring stiffness	4569 N/mm
2.	Maximum velocity of anvil	4.5 m/sec
3.	Velocity of hammer	5.4 m/sec
4.	Maximum Deflection of spring	26 mm
5.	Diameter of spring wire	10 mm
6.	Outer diameter of spring	60 mm
7.	Number of turns of spring	8
8.	Free length of spring	130 mm
9.	Drop height of hammer	1.5 m

Table 5.1 Summary of design parameters

FE analysis results are summarized in Table 5.2.

Sl. No.	Description	Static analysis	Modal analysis	Harmonic analysis
1.	Maximum Stress	43.8 MPa	----	133 MPa
2.	Maximum displacement	0.17 mm	----	0. 75 mm
3.	Factor of safety	7	----	2.4
4.	First natural frequency	----	115 Hz	----

Table 5.2 Summary of FE analysis results

5.1 DISCUSSIONS

Static analysis

- Maximum Von Misses stress is observed to be 43.8 MPa.
- Available factor of safety is observed to be 7 by comparing the maximum stress with that of allowable stress (Yield) of steel material i.e. 330 MPa.
- As the available factor of safety (7) is more than minimum desired factor of safety (1.5) design is safe.
- Maximum displacement is observed to be 0.17 mm.

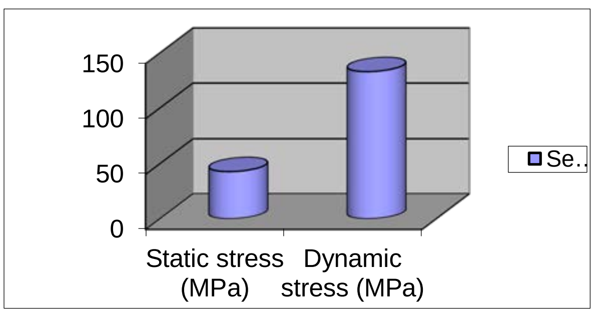
Modal analysis

- First natural frequency of the test rig is observed to be 115 Hz which is far beyond the operational frequency i.e. 66.6 Hz.

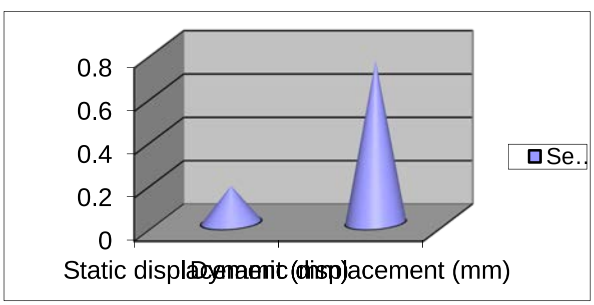
Harmonic analysis

- Maximum Von Misses stress is observed to be 133 MPa.
- Available factor of safety is observed to be 2.4 by comparing the maximum stress with that of allowable stress (Yield) of steel material i.e. 330 MPa.
- As the available factor of safety (2.4) is more than minimum desired factor of safety (1.5) design is safe against dynamic loading condition also.
- Maximum displacement is observed to be 0.75 mm.

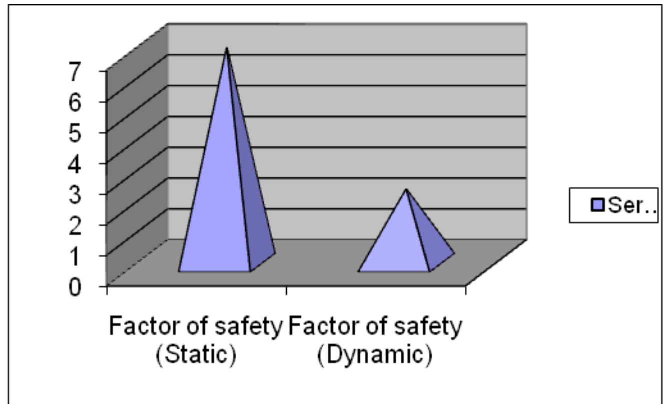
Graphs for Harmonic and Static analysis:



For static and dynamic stress



For static and dynamic displacement



For static and dynamic of Factor of safety

CONCLUSION

A simple and well-known concept is exploited in generating impact testing philosophy for road transport systems. The proposed design overcomes all the limitations associated with present systems in use. Mathematical model of the test rig is established and design calculations are performed. Dimensional models of all critical systems are arrived. Structural analysis carried out against functional loads revealed that the deign adequacy is as per the requirement which clears the design for fabrication.

REFERENCES

1. Mechanical shock theory, jpkj05.sust.edu.cn.
2. Selecting Accelerometers for and Assessing Data from Mechanical Shock Measurements, Patrick L. Walter, Ph. D.
3. Mechanical Shock Test - Mil-Std-883 Method 2002
4. http://www.mradcorp.com
6. ANSYS user manual