# DESIGN AND ANALYSIS OF CAR CHASSI ALIGNER

N.Chandra kanth[1], Mohd Zainulabodin Khan[2], V.Sainath [2], P.Mano Sri Saiesh[2], N.Rahul Chowdary [2] , Mohd. Fahath[2]

[1] *Assistant Professor, Department of Mechanical Engg, NNRG, Hyderabad, India*
[2]*Student, Department of Mechanical Engg, NNRG, Hyderabad, India*

## ABSTRACT

*In Automobile sector, Chassis is an important load bearing member and intern protects the internal parts. Usually, the members of chassi will get replaced when major accidents occur and chassi members undergo bending action. In this present work, an attempt is made to resolve this replacing action by designing hydraulic Car chassi Aligner. Design had been carried out with the maximum of 10 Tone hydraulic capacity consideration. To verify the safe conditions, simulations has been performed for the same design. The 3D geometrical was modeled using Pro-E tool and the same model had been imported to ANSYS-16.0 solver. The structural Parameters like stress, strain, strain-energy, deformation was derived analytically by considering the maximum bending resistance of 100MPa. Numerical and analytical results were compared and found in close agreement. Thus, it can be concluding that the design is safe under the maximum loading condition.*

**Keywords**: - *ANSYS 16.0, Stress, Car aligner, Chassis, and Hydraulic Pump*

## 1. INTRODUCTION

A chassis consists of an internal vehicle frame that supports an artificial object in its constructions and use can also provide protection for some internal parts.[1] The chassis or frame is the most important safety aspect of a modern vehicle, aside from the internal components such as the engine and transmission as well as being a point for all components to attach to the frame is essential during a crash, when the car is at risk of collapsing from pressure due to either rolling or having a weight pressing from above.

Chassis alignments is important to the health of a car. When a car happens to be in a major accident, the chassis experiences the most of the impact of the damage. In such cases the only solution happens to be replacing of the chassis member. Usually the replacement causes heavy investments. To resolve this replacement action, Car chassi Aligner is designed. The Car Aligner is an equipment which helps in eliminating the replacing operation with repairing of the member. The car chassi aligner helps in aligning the deformed chassi member which has been damaged due to the impact of the accident by means of hydraulic pressure. This may help in reducing the expenditure made on replacing the chassis member. By replacing the chassis member, the resale value of the car will get reduced. To resolve this problem repair is made using car aligner which helps in maintaining the resale value of the car.

In this present work, an attempt is made to develop the car chassi aligner keeping the maximum bending resistance of 100MPa into consideration [2]. To confirm the safe condition of the design structural analysis is carried for the same and results are compared.

## 2. MATERIAL SELECTION AND DESIGN

The main design of the hydraulic chassi aligner was carried with the consideration of maximum bending resistance offered by the chassi member and suitable material.

### 2.1 Material

Mild steel with grade of IS4923 is selected to build the main member of the aligner which bears both the hydraulic and bend resisting load. The table:1 gives the properties of the Mild steel IS 4923. The same material is considered to design the supporting member of the hydraulic equipment.

**Table-1.** Properties of IS4923 [4]

| | |
|---|---|
| Ultimate Tensile Strength | 450 MPa |
| Density | 7850 |
| Yeild Strength | 310 GPa |
| Young's Modulus | 200 GPa |
| Poisson Ratio | 0.3 |
| Bulk Modulus | 1.666E+11 |
| Shear Modulus | 7.692E+10 |

### 2.2 Design

The was design is carried out taking the maximum bending resistance offered by the chassi member during the alignment process into consideration. For this maximum resistance, the sustainable cross-sectional dimensions for the vertical load bearing member was derived. The other dimensions of the aligner were decided with the reference of maximum size of a car. The table:2 shows the various design parameters and dimensions representation. Having the factor of safety of 2.

**Table-2** Design Specification

| | |
|---|---|
| Max Hydraulic Capacity | 10 Tones |
| Max Bending Resistance | 100MPa |
| Cross-sectional Dimensions [3] | 120×60×5 (mm) |
| Vertical Height | 1524 mm |
| Horizontal Length | 4527 mm |

The angular force acting on the vertical member is resolved into horizontal and its vertical component and resultant force has been derived. Due to these forces the stress developed in the member was calculated and compared with the maximum allowable stress. Figure (1) shows the member under angular loading and its components.

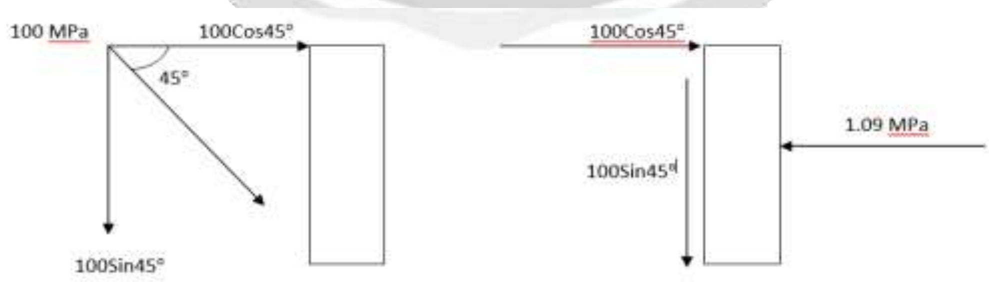


**Fig. -1** Resolution of Forces

Principal stress is given by,

$$\sigma_{1,2} = \frac{\sigma x + \sigma y}{2} \pm \sqrt{(\frac{\sigma x + \sigma y}{2})^2 + \tau x y^2}$$

Where $\sigma_x$ & $\sigma_y$ are directional stresses in x & y direction. In this case $\sigma_y = 0$.

And shear stress is the vertical component of the angular load which is represented as $\tau_{xy}$.

$$\sigma_x = \sigma_{x_1} - \sigma_{x_2}$$

$$\sigma_x = 69.62\, MPa$$

$$\tau_{xy} = 70.71\, MPa$$

$$\sigma 1 - 2 = 34.81 + \sqrt{1211.7361 + 4999.9041}$$

$$\sigma 1 - 2 = 34.81 + 78.8139$$

$$\sigma_{1,2} = 113.6\, MPa$$

The equivalent stress existing is less than the ultimate allowable stress of the member. Thus, the design is under safe condition.

## 3. MODELLING AND ANAYLSIS

To know the safe condition, the derived design is converted to 3D geometry and analyzed using the ANSYS static structural tool.

### 3.1 Modelling

The 3D geometry is developed for the same design using Pro-E tool where, the modelling was carried out for individual parts and assembled upto fully constrained by applying degree of freedom. The 3D geometry is shown in the figure (2).

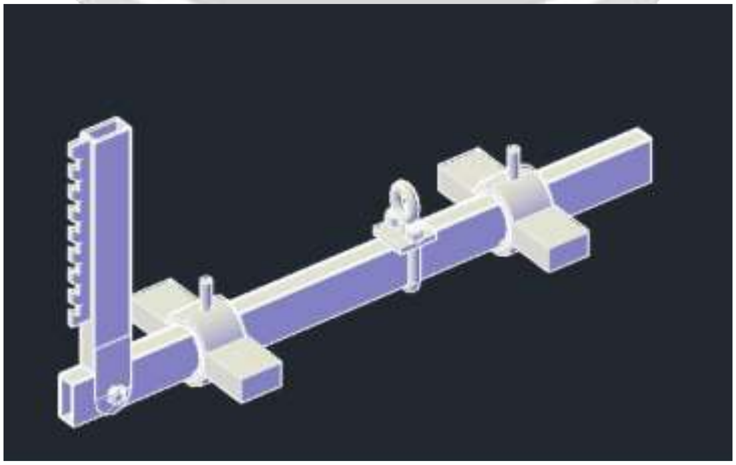

**Fig:2** 3D Geometry of the Design

### 3.2 Simulation

The 3D model has been imported into ANSYS-16 for simulation. The geometry is meshed using auto mesh and boundary conditions like fixed support, the remote displacement along three mutually perpendicular directions and axis, hydraulic force, opposite resisting force were applied. Then, the simulation was under gone for various result convergens like equivalent stress, respective strain and deformation. Three iterations where carried by varying the location of force applications with the same standard procedure. Figure (3) and (4) shows the meshed body and applied boundary conditions respectively.

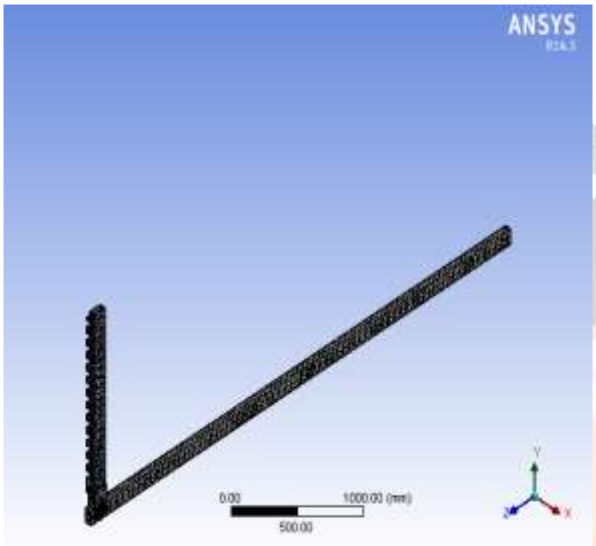


**Fig-3** Meshing

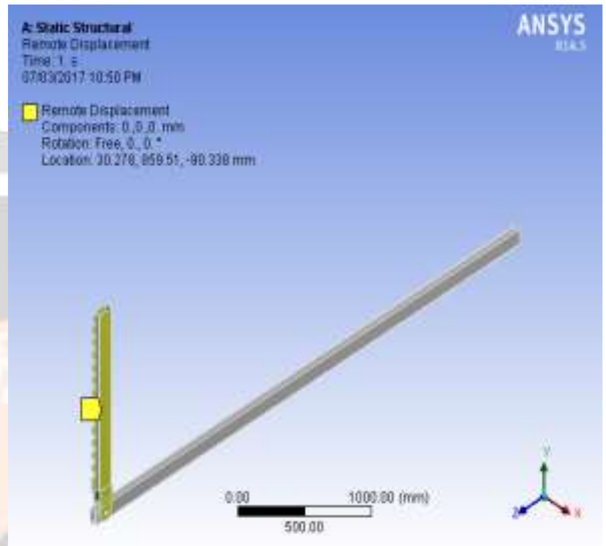


**Fig-4** Boundary Conditions

## 4. RESULTS AND DISCUSSION

As per the ANSYS results the vertical member of the hydraulic equipment is under maximum stressing condition when the load is acted on the top with the maximum angle of load application with respect to horizontal member. The existing stress for this condition is 179.52MPa and the actor of safety is above 2. For the same condition the value of strain is 0.00093. Figure (5) and (6) shows the equivalent stress and strain existence for medium angle of load application i.e. for 45° respectively.

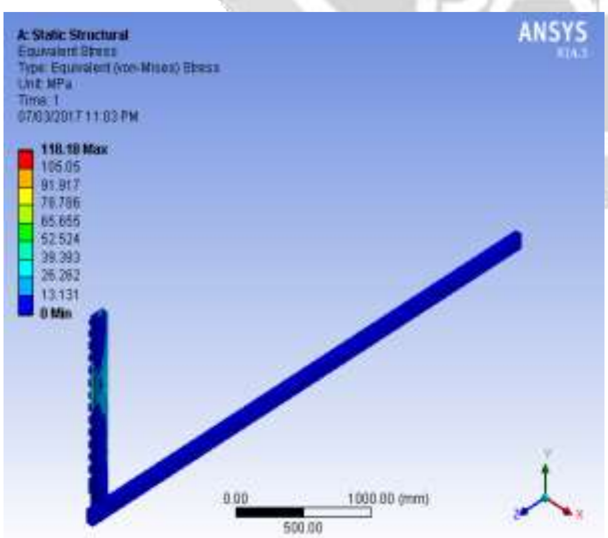


**Fig-5** Equivalent Stress

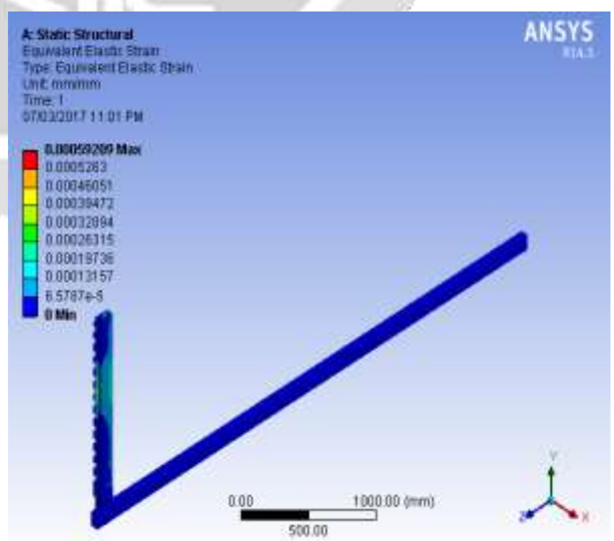


**Fig-6** Equivalent Strain

The total deformation and directional deformation is represented in figure (7) and (8) respectively. The value for total deformation and directional deformation is 1.2207 and 0.03557 mm respectively for and angle of 45º load application.

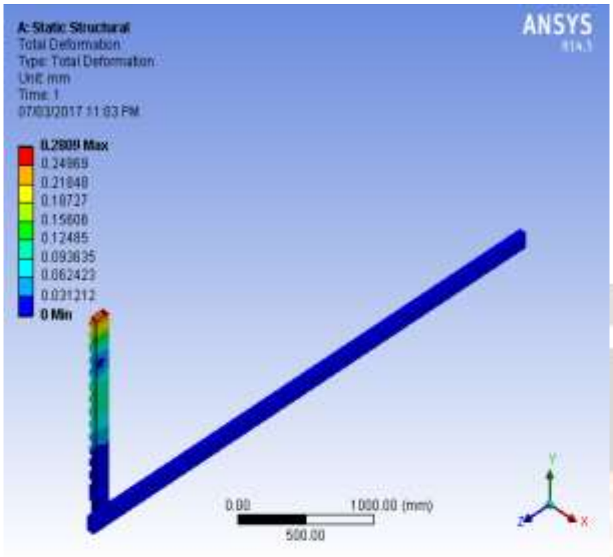


**Fig-7** Total Deformation

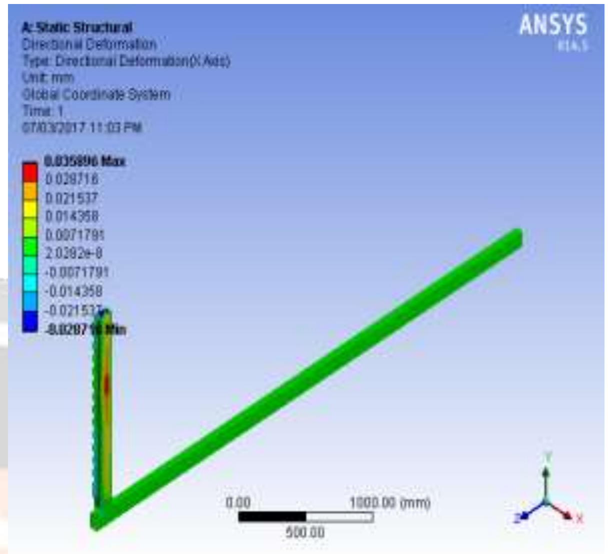


**Fig-8** Directional Deformation

The same procedure was followed to carry out the simulation for other two different angle of load application and results were analyzed. Table- (3) reflects the comparison between the theoretical and analytical results, error and safe condition of a design.

**Table- 3** Theoretical and Analytical Result Comparison

| S.No | Angle of Load Application | ANSYS Result (STRESS) in MPa | THEORETICAL result (STRESS) in MPa | ERROR | Max. Limit Allowable STRESS | FACTOR OF SAFETY |
|---|---|---|---|---|---|---|
| 1 | $75^o$ | 179.56 | 172.12 | 4.14% | 310 | 1.8 |
| 2 | $45^o$ | 118.18 | 113.6 | 3.875% | 310 | 2.7 |
| 3 | $15^o$ | 144.72 | 139.77 | 3.42% | 310 | 2.2 |

from table, it can be observed that as the angle of load application on the vertical member decreases, the member is under low stress condition. But, for lower angle of load application the stress value found to be increased again. Also, for all the three cases the factor of safety is around 2 and the design is safe.

## 5. CONCLUSIONS

The hydraulic equipment is designed, modeled and simulated using the respective numerical, geometrical and analytical approach. The design is evaluated for three different loading location using an analytical tool and compared with that of the theoretical results. The comparison concludes that the results are in close agreement with the error difference 3.87%. Also, it has found that the functioning member under the loading condition is stressed within its sustainable limit. Thus, the member is under safe condition with the factor of safety of around 2.

## 6. ACKNOWLEDGEMENT

We Gratefully Acknowledge Mr. Raghavendra Savannavar and Mr. Vinayak S Uppin for their Help and Technical Support.

## 7. REFERENCES

[1]. Association of Licensed Automobile Manufacturers (U.S.) (1922). Official Handbook of Automobiles. National Automobile Association. p. 180. OCLC 6360726.

[2]. Birajdar, M., & Mule, J. Y. Design Modification of Ladder Chassis Frame.

[3]. Patil, K. Y., & Deore, E. R. Stress Analysis of Ladder Chassis with Various Cross Sections.

[4]. Indian Standard HOLLOW STEEL SECTIONS FOR STRUCTURAL USE-SPECIFICATION (Second Revision) IS : 4923 - 1997 (Reaffirmed 2009)

[5]. Patel, V. V., & Patel, R. I. (2012). Structural analysis of a ladder chassis frame. *World Journal of science and Technology*, *2*(4), 05-08.)

[6]. Nagaraju, N. S., Kumar, M. S., & Koteswarao, U. (2013, December). Modeling And Analysis of An Innova Car Chassis Frame by Varying Cross Section. In *International Journal of Engineering Research and Technology* (Vol. 2, No. 12 (December-2013)). IJERT.

[7]. B. Ramya, Sridhar Reddy, Dr. C. Udaya Kiran, 2014 Design and Structural Analysis of Chassis ISSN: 2321-9939. In *International Journal of Engineering Research and Development* (Volume 2, Issue 4)