Design of Heavy Vehicle Chassis Frame By using Ansys

Kalluri Raviteja, A.Udhya Bhaskar

Abstract— Chassis is main part of the vehicle it is also called back bone of the vehicle. the total load of the vehicle will occur on the chassis frame only. In this paper we did structural analysis on the chassis frame by the fem approached software i.e. ansys14.5. the main aim of the project is reducing the weight of the chassis by using the different types of composite materials. by this weight reducing we can reduce the carbon emission of from the vehicle.

Keywords: composite materials, reducing, structural analysis , ansys14.5.

1. INTRODUCTION

Auto mobile chassis is the back bone of the vehicle. In this chassis only we will mount all the parts of vehicle. so the chassis frame should be strong to restrict the load results, vibrations and other loads. In the design of chassis the main consideration is shape of the channel in this project we are using c shape channel to design the chassis frame. Coming heavy vehicles they are using chassis frame till now but coming four wheel automobiles they are using unibody

Fig 1 Chassis Frme

2. DESIGN CONSIDERATION OF CHASSIS

In this project we designed a chassis with shape of c cannel with the specifications of total length of the chassis is 6750mm, wedith of the chassis frame is 2750mm locatation of whell base at 3400 form the both sides it is over hanging type on both sides with the distance of 1675 from the both sides the ground clearance is 300mm we selected ISMC -400 for the longitudinal members and the ISMC -350 for the lateral members

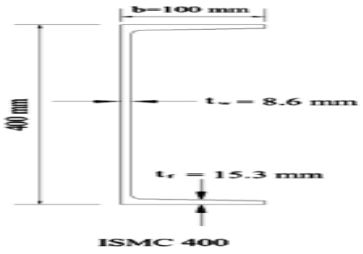


Fig 2: ISME 400

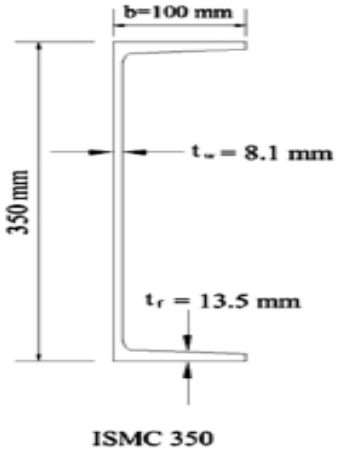


Fig 3: ISMC 350

3. LITERATURE REVIEW

1. In the year of 2018 bhushan, sanjay from Maharashtra they did structural analysis on the chassis of 30 tons with the different type materials in this project they concluded they by using the composites the 73% of overall weight reduced
2. in the year of 2017 divyansh Sharma from utharapradesh the they designed the various types of chassis frmes and did structural analysis they were concluded that they reduced weight of chassis with the same stiffness .
3. In the year of 2014 R.L.PATEL FROM C.U.SHAH university ,Gujarat they did analysis on tata2516tc chassis frame they calculate the factor of safety they concluding that their design have five times to factor of safety.

Revised Manuscript Received on July 10, 2019.

Kalluri Raviteja, M.TECH (cad/cam), Dept. of Mech, Newton's Institute of Science And Technology Alugurajupally, Macherla, Guntur, Andhra Pradesh India.

A.Udhya Bhaskar, Assistant Professor Dept. of Mech,Newton's Institute of Science And Technology Alugurajupally, Macherla, Guntur , Andhra Pradesh India.

4. In the year of 2017 prashanth, suresh from the Hyderabad they worked on the design and analysis of the chassis in that analysis they used e-glass composite materials to reduce the weight of the chassis in that they concluded that by using e-glass they reduce 75-85 % of the weight .

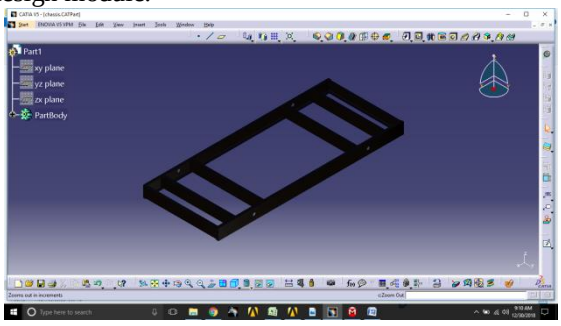
3. INTRODUCTION TO CATIA

CATIA is simply know as computer –aided – design which invented by the dassult systems in the year of the 1977 as parent company with the participate company as IBM in our technical stream we have so many 3d cad modeling software like pro-e, solid works, uni-grphics, solid work etc …but coming catia it is very easy and clear then all the software's we we mentioned above it has a unique feature like all the software's are parametric based software's but coming to catia it is an feature based software with using one option we can perform only one function. catia is came with nearly 17 different modules which used in different fields .

in aerospace design they will use catia tool for the surface design why because this one is flexible tool for the surface design by using catia we can perform 2d, 3d modeling, drafting, assembly, wireframe, sheet metal design for civil people we have plant design for aerspace we have generative shaper design now a days in medical field also they are using catia for their tool designs.

in recent day the dassult system stared the cam and cae tool in catia so now we doing manufacturing as well as analysis also in catia it self.

in this project we design the chassis frame by using the catia v5 r20 with given dimensions we used part design for the design of assembled all the parts of the chassis frame after completing the part design we went assembly module for the assembly we did assembly of the parts which we designed in part design module.



CHASSIS MODEL IN CATIA V5R20

4. FINITE –ELEMENT-ANALYSIS

The Finite-Element –Method is a numerical technique to find approximate solutions with using partial differential equations it was started with need of solving complex problems in aerospace, mechanical and civil engineering in this structural analysis we will calculate the strength , stiffness of the structure and weight of the structure.

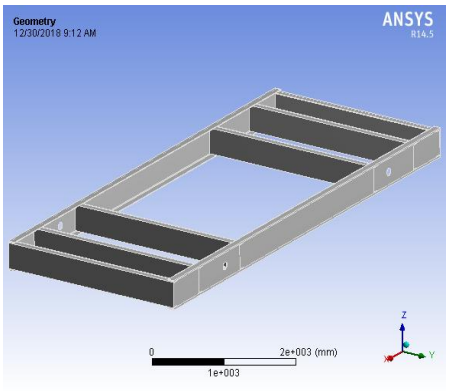
latter on it came with the heat equations vibration analysis flow analysis by using this fem analysis we have different types of fem tools which originated in 1970's i.e. Ansys, hyper mesh, ABAQUS, ls-dyna, nastran etc…….

In this project we used Ansys as a analysis tool. Ansys was invented by the john Swanson in the year of 1970.it made numerous acquisitions for the engineering companies. in this Ansys we have nearly 24 types of analysis. it used for structural, thermal, fluid dynamics electronic design and other analysis.

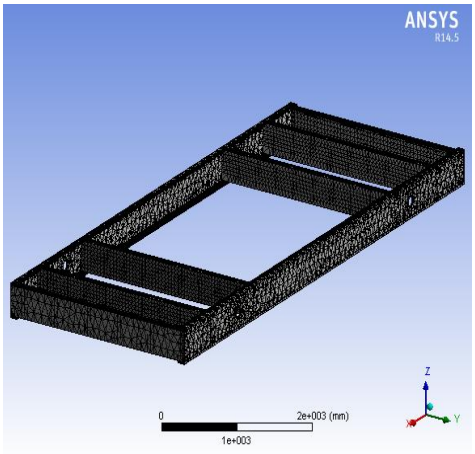
5. STRUCTURAL -ANALYSIS & RESULTS

In this paper we have a tendency to style chassis frame in catia v5r20 once completion of the look we have a tendency to stared a method in Ansys here we have a tendency to did study static structural analysis during this beginning is engineering information during this engineering date we have a tendency to gave the martial that we have a tendency to mentioned in our abstract i.e. AISI1015 and AM162 .after the completion of engineering information we want to to the pure mathematics during this ster we have a tendency to foreign the look that we have a tendency to did in catia. before import we have a tendency to saved that half design in .igs format then we have a tendency to wiil get that model in to the ansys once completion of the commercialism we have a tendency to did meshing then we have a tendency to gave the hundreds and boundary conditions within the last stage we have a tendency to calculate the deformation ,equivalent stress and strain over the suspension systems for the all 2 materials.

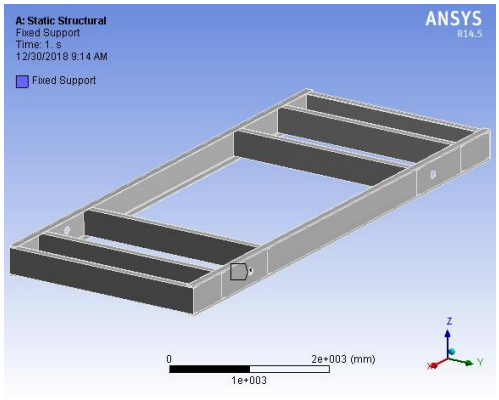
6. ANALYSIS OF CHASSIS IN ANSYS



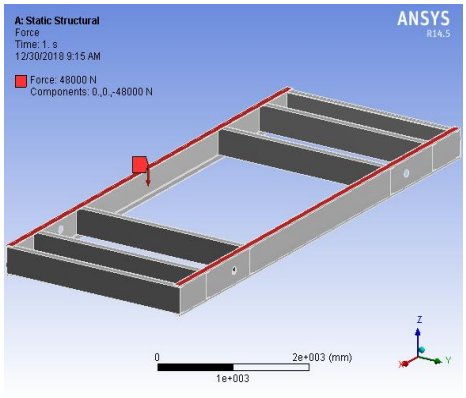
Chassis model in Ansys



Mesh Model in Ansys

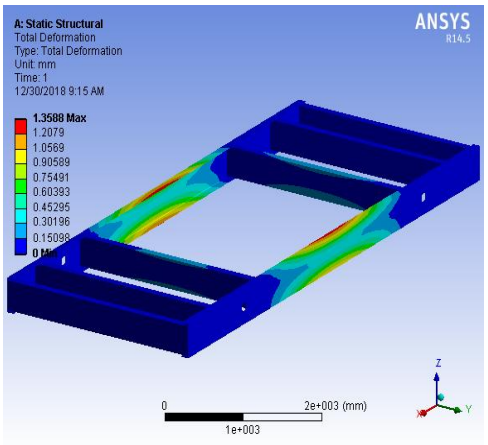


Fixed Support

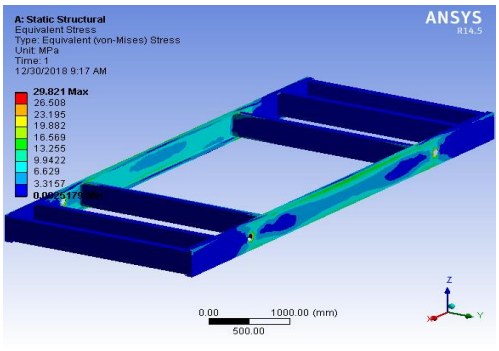


Load

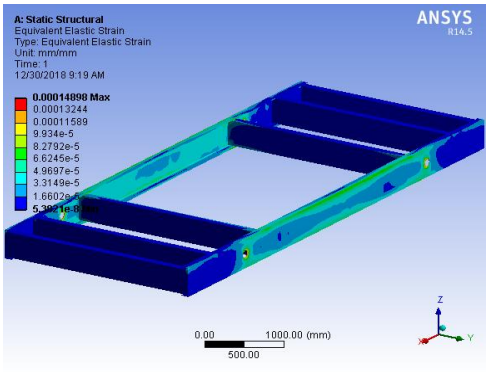
AISI 1015



Total Deformation

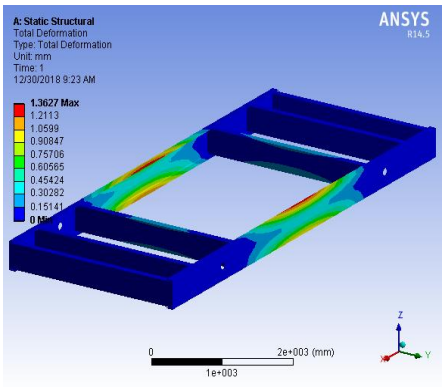


Equivalent Stress

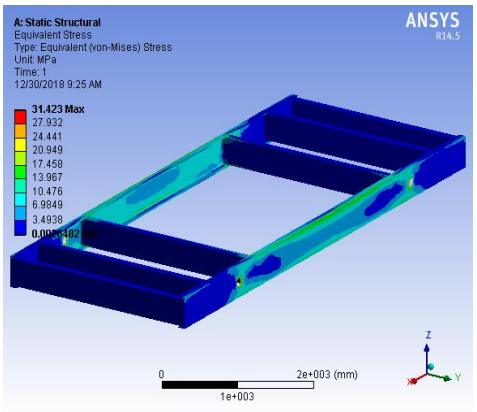


Equivalent Elastic Strain

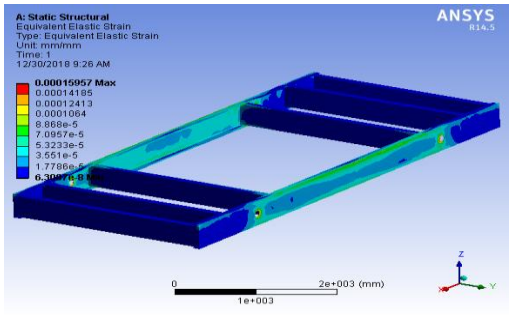
AM162



Total Deformation



Equivalent Stress



Equivalent Elastic Strain

Values observed during static analysis of chassis frame

MATERIAL NAME	TOTAL DEFORMATION	EQUIVALENT STRESS	EQUIVALENT STRAIN	WEIGHT
AISI1015	1.3588 mm	29.821 MPa	1.4898e-004 mm/mm	1125.8 kg
AM162	1.3627 mm	31.423 MPa	1.5957e-004mm/mm	297.63 kg

Table.1

7. CONCLUSION

From the outcomes, an obvious weight decrease of 73.61% has been seen in the trailer structure utilizing a metal network composite of Aluminum-Beryllium as cutting edge lightweight material, i.e. AlBeMet AM162 Extruded Bar for 'C'- Channels ISMC 400 as longitudinal individuals and ISMC 350 as cross individuals having adequate solidarity withstand every the loads with a greatest avoidance of 5.307 mm and prompts the reduction in fuel utilization up to 40% and decrease in CO2 discharge up to 100 g/km according to NHTSA report which keeps the earth from vehicular contamination.