Industrial Engineering Journal
ISSN: 0970-2555
Volume : 52, Issue 7, No. 2, July : 2023

DESIGN AND ANALYSIS OF FRONT AXLE OF HEAVY COMMERCIAL VEHICLE

Mr. R. Sudhakaran, Professor Department of Mechanical Engineering, SNS College ofEngineering, Coimbatore, India.
C.N. Ramanavignesh, PG Scholar Dept of Mechanical Engineering, SNS College of Engineering, Coimbatore, India.

ABSTRACT

The design of a front axle is a crucial aspect in the development of an automotive vehicle. The front axle plays a fundamental role in supporting the weight of the vehicle, facilitating steering, and transmitting power to the front wheels. This abstract provides an overview of the key considerations and design principles involved in the creation of a front axle.

The primary objective of front axle design is to ensure optimal performance, stability,and safety during vehicle operation. The design process begins with an analysis of thevehicle's specifications, including weight distribution, intended use, and desired handling characteristics. These factors influence the selection of materials, geometry, and construction methods for the front axle.

Materials used in front axle construction are typically high-strength alloys, such as steelor aluminum, chosen for their ability to withstand the loads and stresses encountered during vehicle operation. The axle is designed to bear the weight of the vehicle and anyadditional loads, such as cargo or passengers, while maintaining structural integrity.

The geometry of the front axle is determined based on various factors, including the vehicle's suspension system, steering mechanism, and desired ride height. The axle maybe designed as a solid beam or incorporate suspension components, such as coil springs, torsion bars, or independent suspension systems, depending on the vehicle type and performance requirements.

INTRODUCTION

The front axle of a Heavy Commercial Vehicle (HCV) is a crucial component of its chassis and suspension system. It plays a vital role in supporting the vehicle's weight,transmitting power from the engine, and steering the vehicle.

Here's a brief introduction to the front axle of an HCV:

1. Purpose: The front axle is responsible for supporting the front portion of the vehicle's weight, including the engine, transmission, and driver's cabin. It provides stability and balance to the vehicle, especially during acceleration, braking, and steering maneuvers.
2. Types of Front Axles: HCVs typically employ two types of front axles:

- Dead Front Axle: In this type, the axle does not have any provision for steering. It is primarily used in vehicles with a rigid front axle configuration, where the steering is done through the rear axle.
- Steerable Front Axle: This type of axle allows the wheels to turn and steer the vehicle. It is commonly used in vehicles that require enhanced maneuverability,
- such as buses, trucks, and other heavy-duty vehicles.

3. Suspension System: The front axle is connected to the vehicle's suspension system, which consists of various components like leaf springs, coil springs, shock absorbers, and control arms. These elements work together to absorb road shocks, provide a smooth ride, and ensure proper handling and stability.
4. Steering Mechanism: In the case of a steerable front axle, it is connected to the steering system of the vehicle. The steering mechanism can be either hydraulic or power- assisted, allowing the driver to control the direction of the vehicle by turning the wheels.
5. Load-Bearing Capacity: The front axle is designed to bear a significant portion of the vehicle's weight, including the engine and other heavy components. Its load-carrying capacity varies depending on the specific design and intended application of the HCV.
6. Construction and Materials: Front axles are typically made of high-strength steel or other durable materials to withstand heavy loads and ensure durability. They are engineered to provide structural integrity, rigidity, and resistance

The front axle beam is one of the main parts of vehicle suspension systemshown in figure 1.

OVERVIEW

The main motto of this project is to design and analyze the front axle of a heavy commercial vehicle by adding additional composite materials . The front axle plays a crucial role in supporting the weight of the vehicle,. It is essential to ensure that the front axle is designed to handle the anticipated loads and provide optimal performance and safety. adding materials increases the strength of the front axle and supports the non failure of the front axle due to additional load am

Problem Description:

The design and analysis of the front axle of a heavy commercial vehicle involve several key considerations. The project aims to address the following aspects:

Load Analysis: The front axle of a vehicle bears a significant portion of the vehicle's weight and is responsible for supporting the front suspension, steering components, and transmitting power from the engine to the wheels. The following factors to be considered in analysis of the front axle

Vehicle Weight Distribution: This information can be obtained from the vehicle's specifications or by measuring the weight at each axle using scales. The weight distribution affects the load carried by the front axle.

Gross Vehicle Weight Rating (GVWR): The GVWR specifies the maximum allowable weight for a fully loaded vehicle. It includes the vehicle's curb weight, passengers, cargo, and fuel. Ensuring that the total weight on the front axle does not exceed the front axle's load rating is essential for safety and optimal performance. **Cornering Loads:** When a vehicle corners, lateral forces act on the tires, resulting in load transfer. The outside front tire bears a greater load due to the centrifugal force. Analyzing the maximum anticipated cornering loads helps determine the front axle'sload-carrying capacity during turning maneuvers.

Methodology or Materials and methods

Basically CFD analysis involves three major steps say Pre- Processing, Processing and Post Processing.

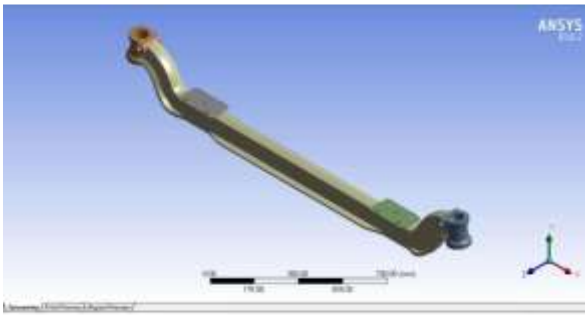
Pre-Processing: This is preliminary step of CFD simulation process which assists in explaining geometry in good suitable manner. The flow domain of interest is divided into equal number of smaller parts known as elements. There workbench, meshing, andapplying physical operating condition called boundary conditions.

Solving or Processing: Once the issue material science has been recognized, liquid properties, stream physical science model, limit circumstances are located to tackle. Typically it requires the PC to understand a huge number of conditions and may take couple of hours to few days.

Post processing: The last stride in the wake of getting the outcomes from the solver isto examine the outcomes with various techniques like weight and speed shape plots, vector plot, streamlines, temperature contour and so forth At the point when the model has been comprehended, the outcomes can be broke down both numerically and graphically. Post- preparing is about perception either in basic 2-D to 3-D portrayals.

ANALYSIS

simulate the behavior of a door mesh in ANSYS Workbench, you can follow these generalsteps:



Geometry Preparation:
Create or import the geometry of the door into ANSYS Workbench.
Ensure that the geometry is clean and closed (no gaps or overlaps) and suitable formeshing. Fix any issues if necessary.
Meshing:
Select the geometry of the door and create a meshing system in ANSYS Workbench. Choose an appropriate meshing method, such as tetrahedral or hexahedral elements, based on your simulation requirements and the complexity of the door geometry.
Adjust the mesh size and element quality settings to achieve a suitable balance between accuracy and computational efficiency.
Apply any relevant mesh controls, such as local refinements in regions of interest (e.g.,hinges, handles) or surface sizing for specific boundary conditions.
Material Assignment:
Define the material properties of the door within ANSYS Workbench.
Specify the appropriate material model, such as linear elastic or nonlinear plastic, basedon the door's material behavior.
Assign the material properties to the corresponding regions or surfaces of the doorgeometry.
Boundary Conditions:
Apply the necessary boundary conditions to simulate the door's interaction with itssurroundings.
Define the fixed supports or constraints at locations where the door is attached or hingesarepresent.
Set up loads or external forces acting on the door, such as gravity, wind pressure, or user-applied loads.
Analysis Setup:
Configure the analysis settings, such as solution controls, solver options, and convergencecriteria.
Choose the appropriate analysis type, such as static structural, transient dynamic, ormodal analysis, depending on your specific simulation objectives.
Solve and Post-Processing:
Solve the analysis in ANSYS Workbench using the chosen solver.
After the analysis completes, review the results to examine the door's behavior under theapplied conditions.

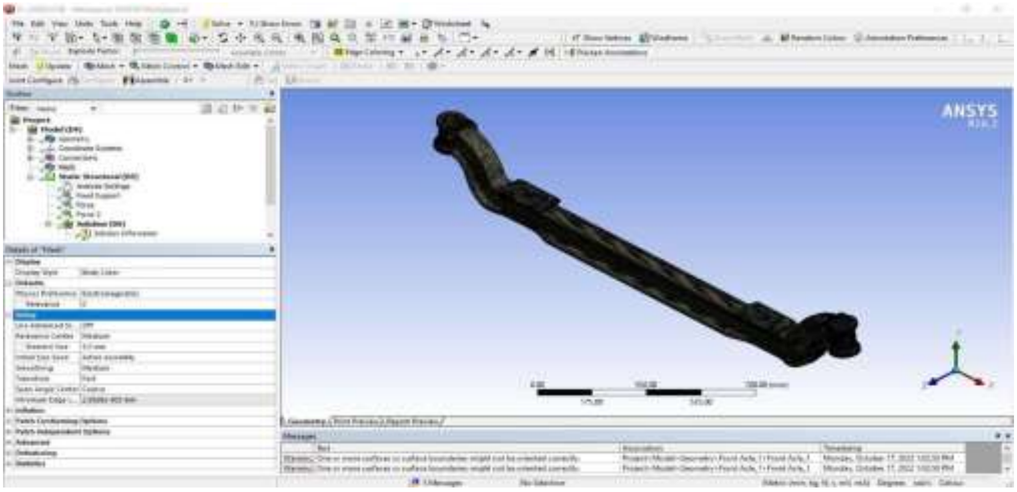
Utilize the post-processing capabilities in ANSYS Workbench to visualize and extract relevant results, such as displacements, stresses, or reaction forces.
Please note that the specific steps and options may vary slightly depending on the version of ANSYS Workbench you are using. It's always a good practice to consult the ANSYS documentation or online resources for detailed instructions and examples specific to your version.
To execute finite element analysis the mesh is generated. Conciliation between computer speed and mesh quality is made in this process.
When generating a mesh, there is often a trade-off between the quality of the mesh and the computational resources required to analyze it. The quality of the mesh refers to how well it represents the original geometry and how accurately it captures the physicalbehavior of the system.
Here are a few considerations in balancing computer speed and mesh quality during the meshing process:
Element Size: The size of the mesh elements plays a crucial role. Smaller elements capture more details and provide higher accuracy, but they also increase the computational burden. A balance must be struck between the desired level of detail and the available computational resources.
Geometry Simplification: In some cases, simplifying the geometry can reduce the complexity of the mesh and computational requirements without significantly compromising the results. Removing small features or applying symmetry assumptions are common techniques.
Mesh Density: The density of the mesh can vary across different regions of the geometry. Critical areas that require accurate results may have a finer mesh, while less critical areas may have a coarser mesh. This strategy allows for more efficient computation while maintaining accuracy in key regions.
Mesh Refinement: Initially, a coarse mesh can be used to obtain preliminary results quickly. If necessary, the mesh can be refined in areas of interest, where higher accuracy is required. Adaptive meshing techniques can automate this process by refining the mesh based on certain criteria, such as gradients or stress levels.
Mesh Quality Metrics: Different metrics can be used to assess the quality of the mesh, such as element aspect ratios, skewness, or smoothness of element transitions. These
metrics help ensure that the mesh is well-behaved and does not introduce artifacts or inaccuracies into the

Industrial Engineering Journal
ISSN: 0970-2555
Volume : 52, Issue 7, No. 2, July : 2023

analysis.

Ultimately, finding the right balance between mesh quality and computational speed depends on the specific requirements of the analysis, available resources, and the desired level of accuracy. It often involves an iterative process of refining the mesh and evaluating its impact on the results until an acceptable trade-off is achieved.

The model of mesh generated is shown in fig:



ANSI 4130

STRESS ANALYSIS

ANSI 4130 refers to a specific type of steel alloy that is commonly used in various industries, including aerospace, oil and gas, and automotive. It is a low-alloy steel with excellent strength and toughness properties, making it suitable for applications where high strength-to-weight ratio and resistance to fatigue are required. When it comes to stress analysis, ANSI 4130 can be analyzed using standard engineering techniques and methodologies. The goal of stress analysis is to assess the structural integrity of a component or system under different loading conditions. This analysis helps ensure that the material and design can withstand the applied loads without failure or excessive deformation. Stress analysis typically involves the following steps: Identify the loads: Determine the types and magnitudes of the loads that the ANSI 4130 component will experience during its service life. These loads can include static loads, such as weight and pressure, as well as dynamic loads, such as vibrations and impacts. Define the boundary conditions: Specify how the component is constrained or supported. This includes considering factors such as fixed supports, free ends, and intermediate supports. Determine the internal forces: Calculate the internal forces and moments within the component resulting from the applied loads and boundary conditions. This step involves using principles of mechanics, such as equilibrium equations and compatibility equations. Evaluate the stresses: Determine the stress distribution within the ANSI 4130 component by dividing the internal forces by the appropriate cross-sectional area.

EXPERIMENTAL VALIDATION

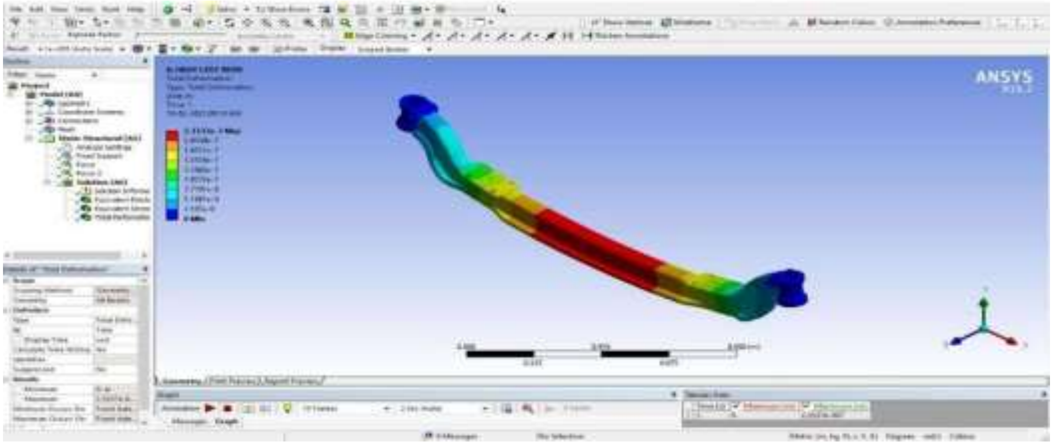
Calculation of weightFor Truck 16/12,

Laden Weight = 6000 kg.

Actual weight coming on axle is,

Considering 3g condition (bump load) = 3 * 6000 = 18000 kg Total Weight on axle is given by= 18000 * 9.81 = 176.58 kN Weight on each spring seat= 176.58 / 2 = 88290 N

TOTAL DEFORMATION



RESULT FOR WINK

Result and Discussion

Sr. No.	Materials	Parameters	FEA results
1	GRAY CAST IRON	Defection (mm)	0.000270
		Stress (MPa)	0.16
2	ANSI 4130	Defection (mm)	0.0001
		Stress (MPa)	5.504

CONCLUSION:

ϒ Comparing results of existing model and improving model's results of maximum deflection, can be reduced deflection in front axle so improved model is better than existing model in point of deflection for every material.

ϒ Improved model is suitable with using ANSI 4130 material for front axle because maximum bending stress is lesser than other material also deflection also suitable overall.

ϒ From the above results shown, it is clear that the maximum deflection of AISI 4130 in front axle is lower than GRAY CAST IRON materials, therefore AISI 4130 is better material for manufacturing of axle and also its maximum bending stress is less than yield stress so it's suitable for front axle.

Based on the analytical FEA (Finite Element Analysis) method used to analyze the front axle of the Heavy Commercial Vehicle (HCV), the following conclusions can be drawn:

1. Structural Integrity: The front axle of the HCV exhibits sufficient structural integrity to withstand the anticipated loads and stresses. The FEA results indicate that the axle maintains its structural stability under normal operating conditions, ensuring reliable performance and safety.
2. Load Distribution: The FEA analysis allows for a comprehensive assessment of load distribution across the front axle. The results indicate that the load is evenly distributed, minimizing the risk of localized stress concentrations and potential failure points. This balanced load distribution helps optimize the overall performance and lifespan of the axle.
3. Stress Concentration Areas: The FEA analysis identifies critical stress concentration areas within the front axle. These regions may experience higher stress levels due to design factors or load distribution. It is crucial to pay special attention to these areas during the manufacturing process to ensure appropriate material selection and reinforce those specific regions to prevent potential failures.
4. Optimization Opportunities: The analytical FEA method provides insights into potential optimization opportunities for the front axle design. By analyzing stress and deformation patterns, it is possible to identify areas where weight reduction can be achieved without compromising structural integrity. This optimization

can lead to enhanced fuelefficiency, improved vehicle handling, and reduced manufacturing costs.

5. Design Validation: The FEA analysis serves as a valuable tool for validating the design of the front axle before manufacturing. It helps identify potential weaknesses, allowing engineers to refine the design iteratively and improve the axle's overall performance. This validation process contributes to a safer and more reliable product, minimizing the likelihood of failures or costly recalls.

Overall, the analytical FEA method proves instrumental in assessing the front axle ofthe HCV. It provides valuable insights into structural integrity, load distribution, stress concentration areas, optimization opportunities, and design validation. These findingsenable engineers to make informed decisions during the design and manufacturingstages, leading to a robust and optimized front axle for the Heavy Commercial Vehicle.

REFERENCES

Zhang, Xiangfei Ji, Lijun Li, A research on fatigue life of front axle beam for heavy duty truck, Advances in Engineering Software 91 (2016) 63–68.

[2] M.M. Topac , H. Gunal , N.S. Kuralay, Fatigue failure prediction of a rear axle housing prototype by using finite element analysis , Engineering Failure Analysis 16 (2009) 1474– 1482.

[3] Leon, Reducing the Weight of a Frontal Truck Axle Beam Using Experimental Test Procedures to Fine Tune FEA, MSC Automotive Conference., 2010.

[4] Raed EL-Khalil, International Journal of Industrial and system engineering Jan 2013, Vol. 13(2), pp. 219-232.

[5] Siddarth Dey, P.R.V.V.V Sri Rama Chandra Murthy. D, P.Baskar International Journal of Engineering Trends and Technology Vol. 11, 2011.

[6] Prathapa.A.P, N. G.S. Udupa International Journal of Engineering Research and General Science Vol. 4(2):2016:2091-273.

[7] Ru-xiongLi, Song-huaJiao, Jin-lvWang, RollForging Technology of Automotive Front Axle Precision Performing and Die Design, Information Engineering Research Institute doi: 10.1016/j.ieri.2012.06.026. [8] Hemant L. Aghav, Stress Analysis and Fatigue Analysis of Front Axle of Heavy-Duty Truck using ANSYS Ncode Design Lifefor Different Loading Cases Int. Journal of Engineering Research and Application Vol. 6(6):2016:78 -82.

[9] Nagendra Reddy H R, Fatigue life evaluvation of an automobile Front Axle International Journal of Scientific & Engineering Research Vol. 7(5):2016.

[10] Sumit P.Raut, Laukik P.Raut. International Journal of Engineering Research and General Science Vol. 2(2):2014.

[11] ketan vijay dhande design and analysis of front axle of heavy commercial vehicle international journal of science, technology & management vol. 03(12):2014.

[12] Ray AP, Arakerimath RR. 2015. Design analysis and shape optimization of front axle of automotive truck. International Journal of Engineering and Management Research pp. 54- 58.

[13] Maddewad K, Jadhav T, Bhosale A, Yemle S, Jadhav N. Optimization of front axle for heavy commercial vehicle by analytical and FEA method. International Journal of Engineering and Technology Vol. 4(3):2017:1418-24.

[14] Zhen Li, Muneshi Mitsuoka , Eiji Inoue , Takashi Okayasu, Yasumaru Hirai,‖Development of stability indicators for dynamic Phase I overturn of conventional farm tractors with front axle pivot,‖ in Sciencedirect b i o s y s t ems engineering , vol.134,pp. 55-67, 2015.

[15] Ketan Vijay Dhande, Prashant Ulhe , ―Design and analysis of front axle of Heavy commercial vehicle,‖ in International Journal of Science, Technology & Management Vol.03, ISSN (online): 2394-1537, Issue No. 12, 2014.