Redesigning and Analysis of Assembly Line Transfer Dolly for Truck

By

Vijay R. Murav
(10MMCC20)

DEPARTMENT OF MECHANICAL ENGINEERING
INSTITUTE OF TECHNOLOGY
AHMEDABAD-382481

MAY 2012

Redesigning and Analysis of Assembly Line Transfer Dolly for Truck

Major Project

Submitted in partial fulfillment of the requirements

For the degree of

Master of Technology in Mechanical Engineering(CAD/CAM)

By

Vijay R. Murav
(10MMCC20)

Guided By
Prof. N.D. Ghetiya

DEPARTMENT OF MECHANICAL ENGINEERING

INSTITUTE OF TECHNOLOGY

AHMEDABAD-382481

MAY 2012

Declaration

This is to certify that

I. The thesis comprises my original work towards the degree of Master of Technology in Mechanical Engineering(CAD/CAM) at Nirma University and has not been submitted elsewhere for Degree.

II. Due Acknowledgment has been made in the text to all other material used.

-Vijay R. Murav

(10MMCC20)

Undertaking for Originality of the Work

I, **Murav Vijay Rajesh**, Roll. No.**10MMCC20** , give undertaking that the Major Project entitled "**REDESIGNING AND ANALYSIS OF ASSEMBLY LINE TRANSFER DOLLY FOR TRUCK**" submitted by me, towards the partial fulfillment of the requirements for the degree of Master of Technology in **CAD/CAM** of Nirma University, Ahmedabad, is the original work carried out by me and I give assurance that no attempt of plagiarism has been made. I understand that in the event of any similarity found subsequently with any published work or any dissertation work elsewhere; it will result in severe disciplinary action.

____

Signature of Student

Date:_________________

Place: NU, Ahmedabad

Endorsed by

(Signature of Guide)

Certificate

This is to certify that the Major Project entitled **"Redesigning and Analysis of Assembly Line Transfer Dolly for Truck"** submitted by **Vijay R. Murav (Roll No: 10MMCC20)**, towards the partial fulfillment of the requirement for the degree of Master of Technology in mechanical Engineering (CAD/CAM) of Instutite of Nirma university, Ahmedabad is the record of work carried out by him under my supervision and guidance. In my opinion, the submitted work has reached a level required for being accepted for examination. The result embodied in this major project, to the best of my knowledge, haven't been submitted to any other university or institution for award of any degree or diploma.

N.D.Ghetiya,
Guide and Assistant Professor,
Department of Mechanical Engineering,
Institute of Technology,
Nirma University, Ahmedabad

Dr. R.N.Patel
Head and Professor
Department of Mechanical Engineering
Institute of Technology
Nirma University, Ahmedabad

Dr K Kotecha
Director,
Institute of Technology,
Nirma University, Ahmedabad

Acknowledgements

With immense pleasure, I would like to present this report on dissertation work related to "Redesigning And Analyzing of Assembly Line Transfer Dollies For Truck". I am very thankful to all those who helped me for the successful completion of the first phase of the dissertation and for providing valuable guidance throughout the project work.

I would first of all like to offer thanks to **Mr.GaneshSharma**, Head , H.R. Dept, AMW for allowing me to carry out the project work in company. I would like to thanks to **Mr.VilasPetvule** , **Mr.NeerajRawat** and **Mr.GauravChoudhary** for their keen interest and excellent knowledge base helped me to finalize the topic of the dissertation work. Their constant support, encouragement, and constructive criticism has been invaluable assets through my project work. They have shown keen interest in this dissertation work right from beginning and has been a great motivating factor in outlining the flow of my work.

My sincere thanks and gratitude to **N.D.Ghetiya**, Assistant Professor, Mechanical Engineering Department, Institute of Technology, Nirma University,for their continual kind words of encouragement and motivation throughout the Dissertation work.

I am thankful to all the employees , for their kind help and encouragement in my project work. Also I would like to thanks to my friends for their continuous support during my dissertation work.

- Vijay Murav
10MMCC20

Abstract

Continuous improvement in the design of any component or machine is useful in the field of manufacturing and in the mass production of the component. It is also benefited to the person who is connected to the field of manufacturing. One can use the component in easy way by improvement in design.

In the field of the automobile industries the time reduction in the automobile assembling is an important parameter and to achieve this continuous improvement on the assembly line is need. Here in this dissertation one of those component called Assembly Line Transfer Dolly redesign is carried out. Dolly is mobile platform which is moving continuously on the assembly line at constant speed with the help of chain conveyor and the truck are assembly on it. Sometime due to the improper balance of the truck chassis on it lift up on the truck and it is one of the safety issues.Here we are taking the help of CAD to do redesign and analysis of the dolly.

List of Figures

List of Tables

Contents

Chapter 1

Introduction

1.1 About The Company

Asia Motor Works Ltd

Established in the year 2002, AMW specializes in design and manufacture of high tonnage vehicles on a hybrid platform. AMW has entered into collaborative partnerships with pioneering aggregate suppliers like Cummins, ZF, Valeo and Meritor, and has produced India's first Global Truck series: 4923 TR and 2523 TP. After extensive research, AMW has designed and developed a modern truck on a truly global platform. AMW employs a stringent testing and quality control processes, to deliver to its customers, vehicles that can withstand the diverse andoften demanding operating conditions. AMW has a state-of-the-art fully integrated manufacturing facility spread over 600 acres in Bhuj. The facility houses advanced production units built in collaboration with world leaders like Schuler, etc. The company has a vision to be recognized and respected as the market leader delivering value to its customers, employees and society.

1.2 About The Dolly

Definition of Dolly: A low mobile platform that rolls on casters used for transporting heavy loads.

Dollies are simple platform on which the special kind of fixture type of arrange are made for transferring the material from one place to another on the rolling casters basically it is used as material handling equipment in storage department of different industries and on the assembly line of the different industries. For e.g. Automotive, food industries, etc.

In automobile industries where the automobile are assembled it is necessary to transfer the automobile from one stage to another so that at different stage different components are assemble and whole automobile is made. For this they used the Assembly line, AGV or Conveyor System. On the conveyor line a dolly is used to transfer the automobile from one stage to another for assembly it runs on conveyor line.

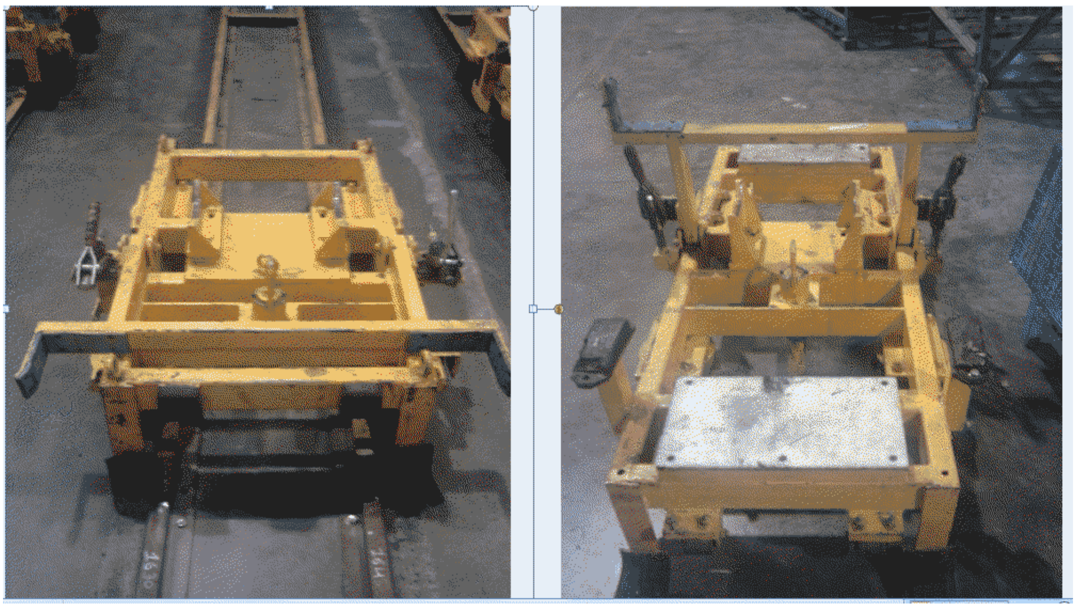


Figure 1.1: Front dolly and Rear dolly

1.3 Application of Dollies

(a) Dolly for Volvo Truck Assembly

(b) Dolly for Motor Cycle Assembly

(c) Dolly for Material Handling

Figure 1.2: Application of Dollies in Industries

1.4 Aim of the Project

1. To do the study of the old dollies.
2. From study identifying the problems of old dolly.
3. Design new dolly to over come the problems.

4. Do the stress analysis of dolly.
5. Contact stress analysis between dolly wheel and slope of the track.

1.5 Methodology

1. Study of the old dolly.
2. Force calculation on dolly
3. Modeling of dolly using NX 7.5
4. Carry out stress analysis in the ANSYS.
5. Calculation of contact stress and wear of the track.

Chapter 2

Literature Review

2.1 Introduction

Designing and analysing of the assembly line transfer dolly is basically problem of improvement in the design to over com the issue on the assembly line. For this first all study and analysis of the present dolly is carried out then the new design is prepared from the result of the analysis. Listed below are some of the literatures surveyed that are deemed to be significant to this research that is conducted on analysis of body structure of a dolly.

Roland Siegwart et al [1].In this paper they have presented an innovative, wheeled rover which provides excellent climbing and steering capabilities. Based on a parallel architecture allowing for high ground clearance and excellent stability, the vehicle is able to passively overcome steps of twice its wheel diameter, to climb stairs or to move in very rough terrain. These capabilities are mainly provided by the parallel architecture of the front fork and the bogie in combination with non-hyperstatic contact of all its wheels with the ground.This robot is therefore a perfect candidate for planetary exploration or terrestrial applications in the field of mining, construction, agriculture, post-earthquake assistance or demining.

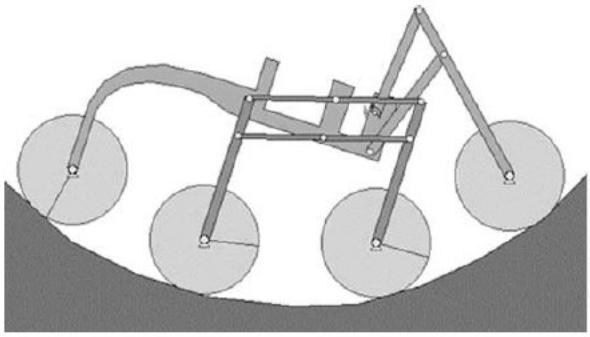


Figure 2.1: Robot on Concave ground

J. Pijuan et al.[2] In this study two main active control strategies have been analyzed with the objective of improving the capacity of a wheeled vehicle to overcome obstacles. In particular, a 4-axle vehicle with double bogie facing an upward slope is the situation that has been analyzed.The kinematics and dynamics of the multi-body system has been described and the model has been parametrically solved, so that the proposed active strategies could be studied as a function of actuation inputs. Different results have shown the improvement in obstacle surpassing ability obtained by the use of active bogies and height regulation.

Ashutosh et al.[3] work contains the load cases and boundary conditions for the stress analysis of chassis using finite element analysis over ANSYS. Finite element model of the vehicle chassis is made. Shell elements have been used for the longitudinal members cross members of the chassis. The advantage of using shell element is that the stress details can be obtained over the subsections of the chassis as well as over the complete section of the chassis. Beam elements have been used to simulate various attachments over the chassis, like fuel tank mountings, engine mountings, etc. Spring elements have been used for suspension and wheel stiffness of the vehicle. The results of finite element analysis have been checked by experimental methods.

X. Potau et al.[4] Three vehicle configurations with 2-, 3- and 4-axles have been modeled in order to study their ability to overcome obstacles when climbing an upward slope. In all models the kinematics and dynamics of the multibody system have been accurately described mathematically. The models were defined in a parametric way, so that they are suited both for a comprehensive design sensitivity analysis and multi-objective optimization.

Comparison between the conventional, 3-axle and 4-axle vehicle has shown the improvement in obstacle surpassing ability obtained by the use of passive bogies. Next, the influence of obstacle traversing speed and vertical position of the center of gravity has also been investigated. Optimization of these parameters allows to have the best performance for a given vehicle geometry.

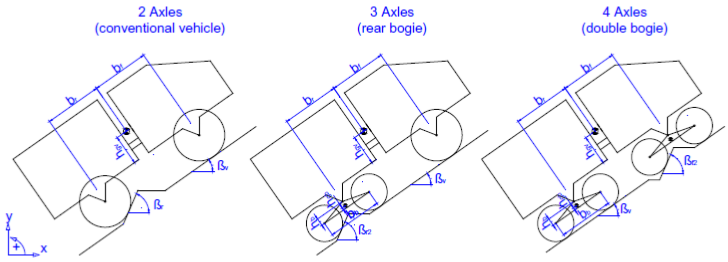


Figure 2.2: Sketch of the different vehicle configurations that have been considered.

M A Arslan,[5] In this study, numerical analysis of stressstrain characteristics of 3D RailWheel contact is successfully carried out using FEA. Applications of realistic FE loading and boundary conditions have played an important role in terms of preserving the originality of the contact load transfer. From the mechanic of materials point of view this contact is highly nonlinear in its nature. As it is obvious, representing the real loading conditions (force and moment) would directly affect the true behavior of the contact mechanism.Once we get the right loading on the 3D FE model, the out coming behavior and stress/strain results are going to be more accurate than 2D axi-symmetric FEA results. Presented 3D FE analysis results look very promising and give us a good insight about the simulating contact characteristics. Finally, 3D analysis results are observed as very helpful in determining/visualizing the wear characteristics of rail top surface as well. But it should also be noted that there is always room for additional interpretation of the present results from experienced railway people.

Masahiro et al.[6] kinematical analysis method for vehicle is carried out by conventional method in ADAMS and NASTRAN and found out that elastic characteristic of frame can be used to predict the dynamic stress of frames when a vehicle travels.

Valda et al.[7] The dynamics of an overhead crane with moving hoist with FEA is considered in the Paper.

Jia-Jang Wu et al.[8] Here the dynamic response of the moving crane is carried out.First the prototype of the crane structure is made of 1/10 size in the lab. Then they divided the structure in two part the static part and dynamic part to find out the static and dynamic load of the structure .The calculation of the equivalent nodal forces to represent the moving loads has been performed by three approximate methods. In the first "full" method, equivalent nodal forces and moments were calculated. This required the shape functions for the element.

The second method simply ignored the moment calculated using method 1.
The third, "simple", method ignored any moment applied at the nodes at the outset and therefore did not require knowledge of the shape functions.
Then carry out the stress analysis in I-DEAS software.

Harish et al. [9] Redesigning of tractor trolley axle by optimizing it and doing stress analysis using ANSYS.

Alkin et al. [10] The crane design was model using both solids and surfaces.Finite element meshes with 4-node tetrahedral and 4-node quadrilateral shell elements were generated from the solid and shell models. After a comparison of the finite element analyses, the conventional calculations and performance of the existing crane, the analysis with quadratic shell elements was found to give the most realistic results.

Mohammad, [11] The paper investigates into the vibrational characteristics of the truck chassis including the natural frequency and mode shapes with the help of ANSYS.

Jaju et al. [12] In this paper the FEM analysis of manually operated tilting mechanism of three furrows reversible plough is presented. The main objective is to optimized the plough for that the cad model is prepared and the analysis is carried out in ANSYS.

N S Kuralay, [13] In this study, the finite element analysis of a truck chassis was carried out. The analysis showed that increasing the side member thickness can reduce stresses on the joint areas, but it is important to realise that the overall weight of the chassis frame increases. Using local plates only in the joint area can also increase side member thickness. Therefore, excessive weight of the chassis frame is prevented. In this case, changing the side member thickness using the local plates seems to be suitable. In both cases, stresses in the connection plate rise slightly. Increasing the connection plate thickness can reduce stresses in the connection plate. When the length of the connection plate is increased, stresses in both side member and connection plate decrease. If the change of the side member thickness using local plates is not possible, choosing an optimum connection plate length (L) seems to be practical solutions for decreasing the stress values..

M. Ereke et al. [14] In this paper the failure of the rear suspension spring is analyzed in detail. The rear axle suspension system of the truck and fractured flat spring is investigated. Fracture surface, mechanical and chemical properties and microstructure of the spring material is analyzed. Forces acting on the spring is determined and strength calculations are carried out. Later, failure behaviour and cause of fracture is revealed after carefully analysis of microstructure and results of calculations. At the end precautions to be taken to prevent a similar failure is recommended.

Tushar et al.[15] Static analysis of the gear box casing is carried out to reduce the cost of the gear box casing by optimizing the design with the help of the software.The optimization of the gear box help in cost reduction and help for selecting the material. It also help in reduce time in preparing prototype model and testing machine.

Rangaswamy et al. [16] Analysis of the composite drive shaft for power transmission is carried out. The design variable is continues throughout the analysis and study of the composite material is carried out.

Chapter 3

Problem Description and Trail Design of Dolly

3.1 Problem Statement

In the present work the redesign and analysis of assembly line transfer dolly is carried. Dolly is a low mobile platform used to transfer from one stage to another on the assembly line. But at one place on the track there is a slope on which when dolly climbs due to the unbalance force on the dolly the back wheels of front dolly is lift up so, that we have to design a new dolly to resolve this problem.

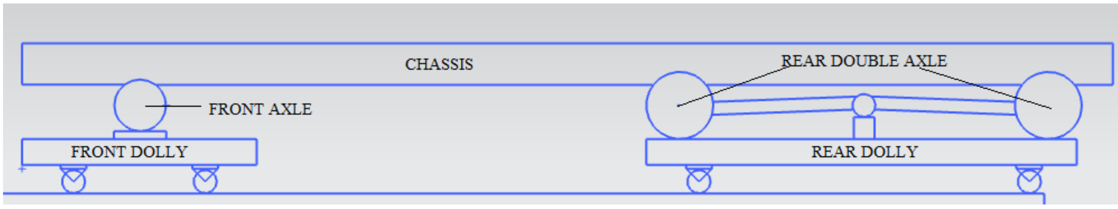


Figure 3.1: Arrangement of the Axles on the Front and Rear Dolly

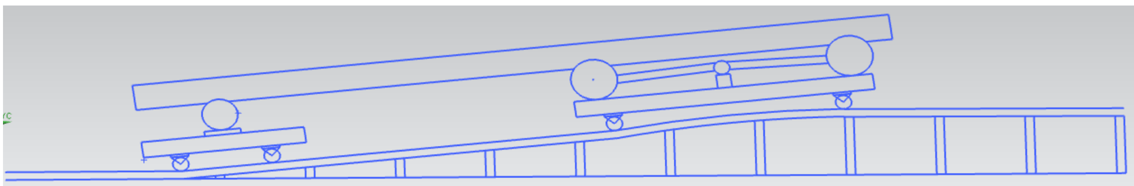


Figure 3.2: Arrangement of the dollies climbing on the slope

Fig 3.1 and Fig3.2 show the arrangement of how the axles of the truck are placed on the front and rear dolly and the situation when the dollies are climbing the slope of the track.

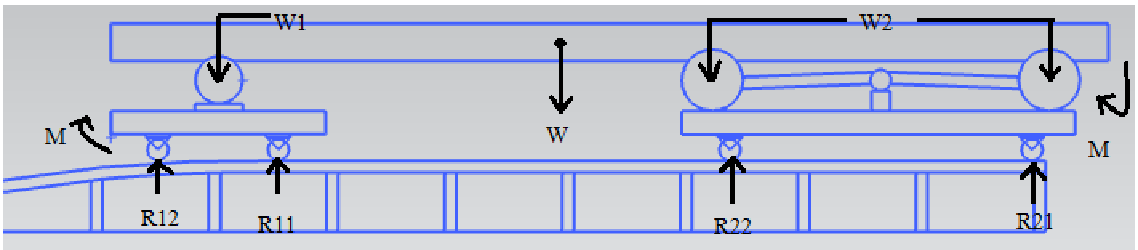


Figure 3.3: Force Acting on the Dollies

Where,

W = Total weight of the truck chassis and axles.

W_2 = Weight of the rear double axle acting on rear dolly.

W_1 = Weight of the front axle acting on front dolly.

R_{11} and R_{12} = Reactions at front and rear wheels of front dolly.

R_{21} and R_{22} = Reactions at front and rear wheels of rear dolly.

Stepwise problem Description.

1)Fig 3.1 show the arrangement when both the dollies are moving on the flat track and are in stable positions.

2)Fig 3.2 show the stating arrangement when the dollies are climbing the slope on the track.

3)Fig 3.3 show that the rear dolly is completely climb the slope of the track and front dolly is climbing the slope.

4)At a particular instant of time due to the higher weight of the rear double axles (W_2)which try to maintain the chassis in the straight position. Due to this the front dolly W_2 slightly lift up and that causes rear wheels of front dolly to lift up.

5)When the rear wheels lift up the whole weight of the front dolly is maintain on the front wheels of the dolly for a moment of time.

6)At that moment of the time when the whole weight is on front wheels which produce the higher contact stress at that point between wheels and track which causes wear of the track.

7)Our problem is to design new dolly while reduce contact stress and wear of the track on the slope.

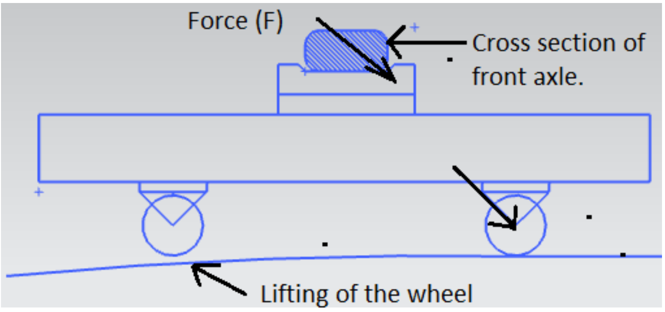


Figure 3.4: Lifting of the back wheel of front dolly

3.2 Trail Design of the Dolly

3.2.1 Design 1

Design 1 is the first trail dolly for solving the problem. In this dolly the links are connected with pin joints as shown in fig 3.5. The concept of joints used in the dolly to give the dolly more flexibility so, that while climbing the slope dolly can change its shape for force transfers. But due to more number of joints design is failed due to joints each link is try to move itself which increases the stress at the joint. Also the more number of joint is not stable when the load is applied on it produce high stress at pin joints and get fail. It will caused the higher maintenance.

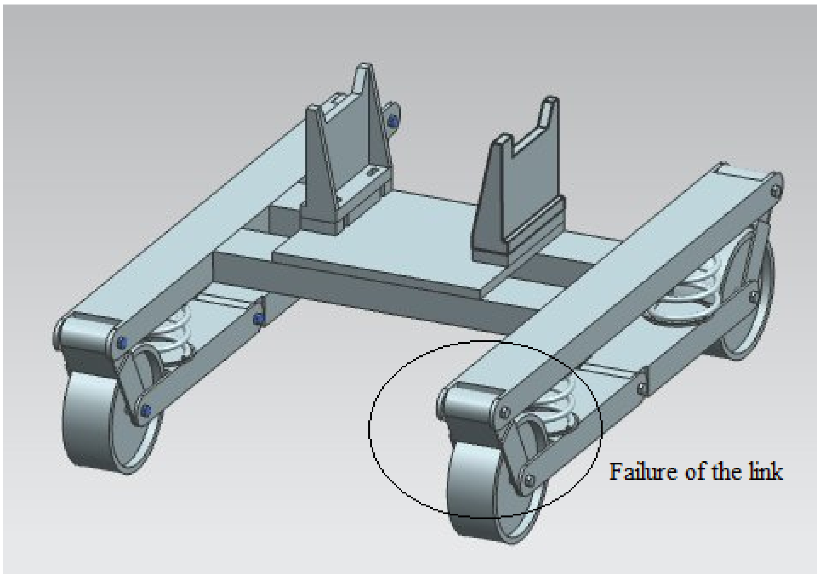


Figure 3.5: Dolly with the spring suspension and flexible link

3.2.2 Design 2

In design 2 the spring dampers are used to solve the problem but damper will absorb the load or shock which is acting on the dolly and the damper work as rigid link. While climbing the slope the spring on the front wheel of the dolly will remain compress and spring get deflected and on the back wheel spring will try to expand and make the contact between the wheel and the track. But it will not be stable and stresses at the front wheel spring increase and it will fail and problem remain same.
In this design due to the damper attachment the wheel are going to come in contact with the track but, it will due to the expansion of the spring damper. It will not come in actual contact with the track so that it can transfer the forces from wheels to track. It also produce the higher stress with the flexible link which is connected with the damper between the rigid and flexible link. The FEA analysis of the dolly is show in the fig 3.7 which is showing the higher concentration of the stress is at the flexible link and its maximum stress is 60 MPa.

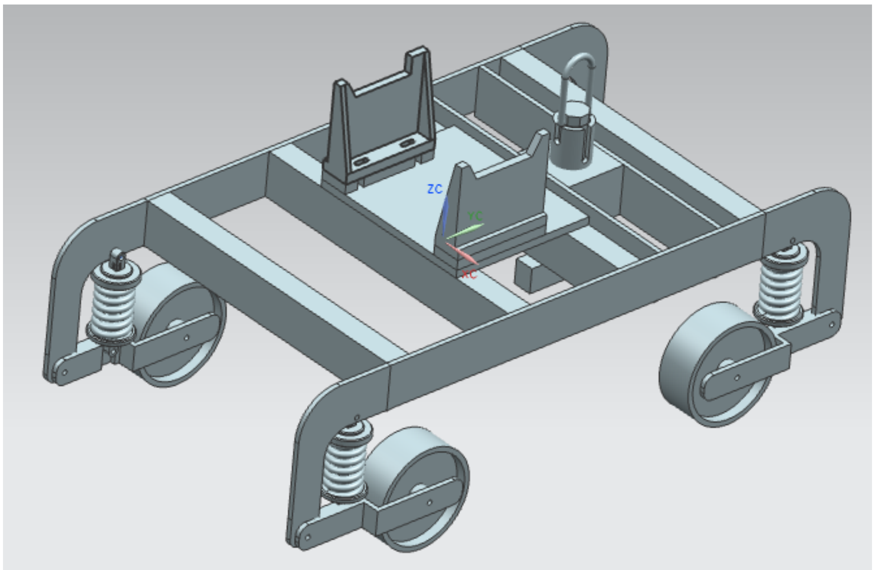


Figure 3.6: Dolly with Spring Damper

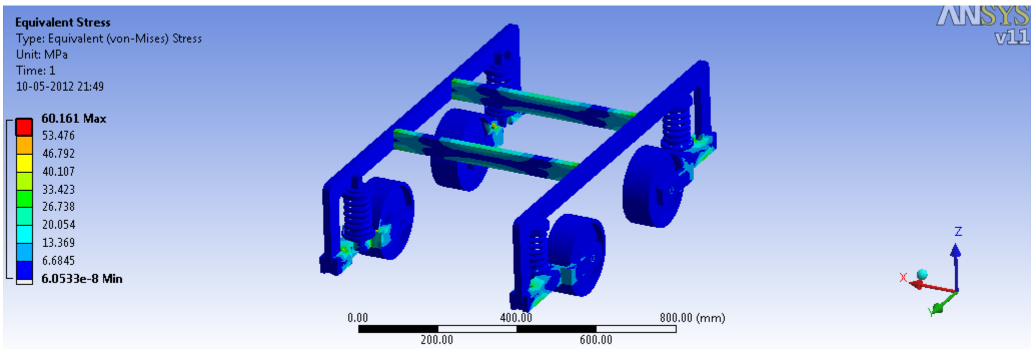


Figure 3.7: Stress Analysis of Damper Dolly

Chapter 4

Design and Analysis of New Dolly

4.1 Design of the new dolly

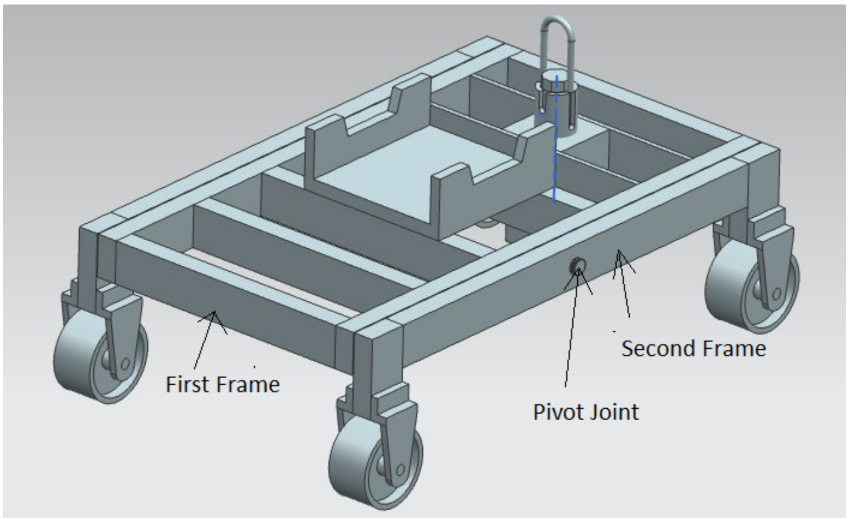
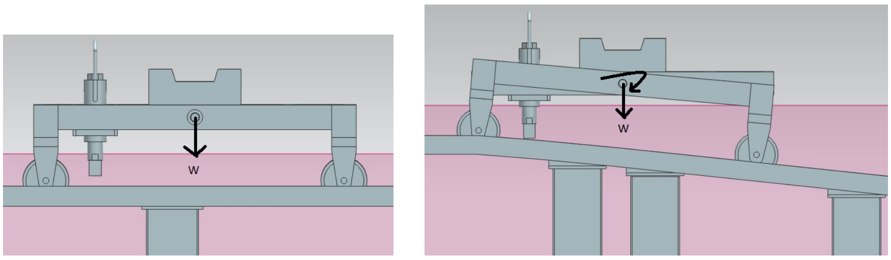


Figure 4.1: Front dolly and Rear dolly

Design of new dolly consists of mainly two frames which is show in the figure above. One the first frame the front axle of the chassis of the truck rest on it and on another frame the wheel of the dolly is connected. The second frame is connected to the first dolly with the help of the help of the pivot joint which is shown in the figure. When the dolly is climbing on the slope the first frame remain straight and the second frame move according the shape of the track curvature.



(a) Dolly on the Straight Track (b) Dolly on the Slope of Track

Figure 4.2: Positions of Dolly

As shown in the above figures the positions of the dolly. In Fig.4.2(a)show that the dolly is on the flat or straight track and the weight W is the weight of axle acting on it while in Fig.4.2(b)shows that dolly at a slope in which the first frame remain straight due to the weight of the axle and the second frame move along the pivot according to the slope of the track.

4.2 Locking mechanism between to frame

In order to lock the first and second frame of the dolly magnetic lock mechanism is used.In this the two attractive magnet are placed between the frames so that the first frame remain straight with respect to the second frame of the dolly. It is need because of the pivot joint the first frame try to rotate about the second frame. We have to maintain this frame in straight position until the load of the axle place on it. Here magnetic force equation is used for selecting the appropriate magnet.

$$\mathrm{F} = \mu_0 \frac{H^2 A}{2} = \frac{B^2\ A}{2\mu_0}$$

where,

A is the area of each surface, in m^2

H is their magnetizing field, in A/m.

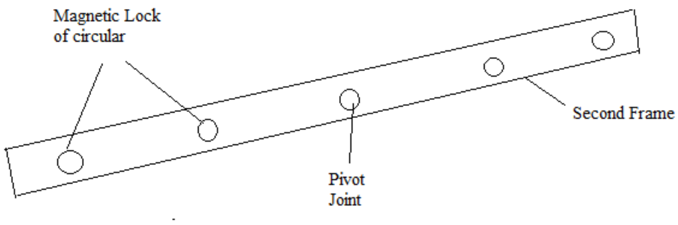


Figure 4.3: Arrangement of magnet on second frame

μ_0 is the permeability of space, which equals 4p10-7 Tm/A

B is the flux density, in T

4.3 Stability of the Dolly on the slope

Center of Gravity of Dolly

Dolly is Axis Symmetry and the center of gravity is always near the higher value of

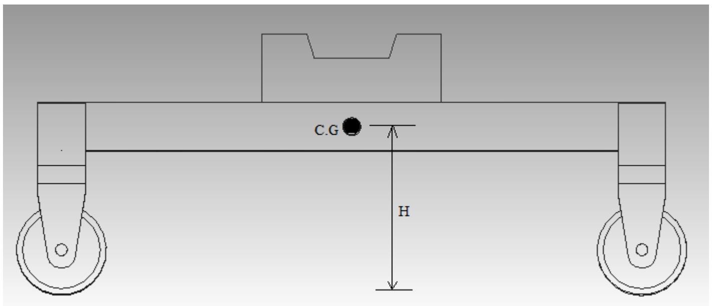


Figure 4.4: Forces acting on dolly while climbing slope

the weight. Here the higher weight is of frame so that the center of gravity will be on the upper side of dolly.

Now the frame section is rectangular center of gravity of it is given by.

C.G of rectangle = Height/2

C.G of rectangle = 80/2

C.G of rectangle = 40 mm

Height of C.G from ground = H + h

where,

h=240 mm

Height of C.G = 40 + 240

Height of C.G = 280 mm

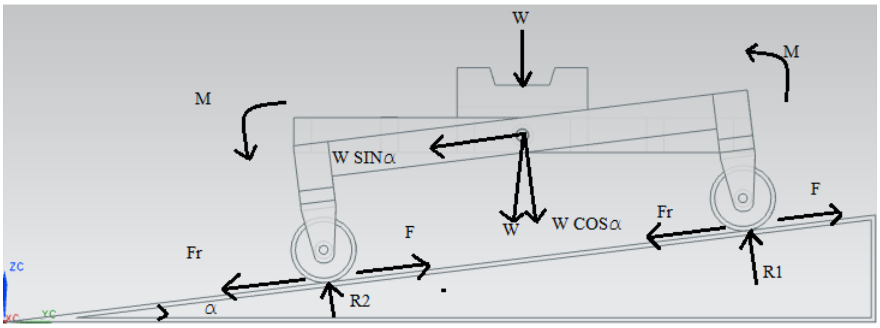


Figure 4.5: Forces acting on dolly while climbing slope

Here weight on the first frame is W because that frame remain straight and reactions forces on the wheels is W COSα/2 because that frame is making an angle with the track.

For dolly to remain in stable position on slope weight acting on the dolly should be greater then reaction forces acting on the wheels of the dolly.

$$W > R1 + R2$$

$$W > W\ Cos\alpha/2 + W\ Cos\alpha/2$$

$$W > W\ Cos\alpha$$

$$7500 > 7500\ Cos7$$

$$7500\ N > 7444\ N$$

From the above calculation the it can be said that the dolly is stable on slope.

4.4 Theoretical Calculation of the deflection

One can understand that the maximum deflection of the dolly occurs on the part where the axle is resting. On the dolly that part is the shown in the fig 4.6. Consider that part as the simply supported beam on which the load acting at the center and two ends are fixed.

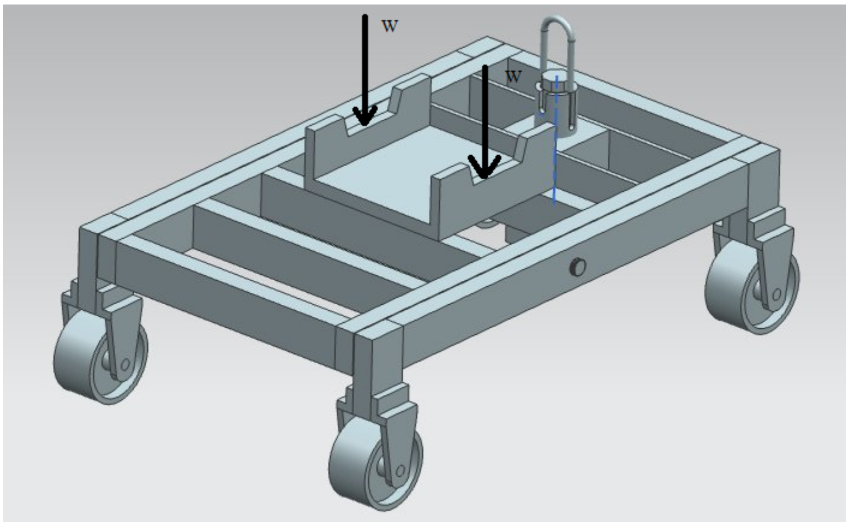


Figure 4.6: Axle weight acting on the dolly

Maximum deflection of simply supported beam is given by:

$$\delta = \frac{W\ l^3}{48\ EI}$$

Given Data:

Sr. no	Parameters	Values
1	Load(P)	22500 N
2	Length(l)	300 mm
3	Young's Modulus(E)	2x10^5 N/mm^2
4	Moment of Inertia(I)	1.744x10^6 mm^4

$$\delta = \frac{22500 \quad 300^3}{48 \quad 2x10^5 \quad 1.744x\ 10^6}$$

$$\delta = 0.0406 \text{ mm}$$

4.5 Calculation of Pivot Joint

Here on the pivot pin two stresses are acting shear and bending. We have to calculate the diameter according to both shear and bending stress.Considerin the pivot as circular section beam.

1.Shear Stress

$\tau_{shear} = \frac{QP}{Ib}$

Where,

$I = \frac{\pi r^4}{4}$

$Q = Ay = \left(\frac{\pi r^4}{4}\right)\left(\frac{4r}{3\pi}\right) = \frac{2r^3}{3}$

b= 2r

I = Moment of Inertia

Q = Second moment of Inertia

$\tau_{shear} = \frac{WQ}{Ib} = \frac{W(\frac{2r^3}{3})}{(\frac{\pi r^4}{4})(2r)} = \frac{4W}{3\pi r^2} = \frac{4W}{3A}$

Therefore,

$\tau_{shear} = \frac{4W}{3A}$

Calculating the diameter according to shear stress.

$d^2 = \frac{16\ W}{3\ \pi\ \tau_{max}}$

$\tau_{shear} = 0.6\sigma_t$

$\tau_{shear} = 192$ N/mm^2

$d^2 = \frac{16\ (10)(7500)}{3\ \pi\ (192)}$

d =25.75 mm

2.Calculating the diameter according to bending stress.

$\sigma_b = \frac{M}{Z}$ Where, $M = xW$

$Z = \frac{\pi d^3}{32}$

M = Bending Moment

Z = Section Modulus

$\sigma_b = \frac{32\ M}{\pi d^3}$

$d^3 = \frac{32\ M}{\pi \sigma_b}$

$d^3 = \frac{32\ (20)(75000)}{\pi(320)}$

d = 36.3 mm

Here the diameter of pivot pin is greater in bending stress so we will consider as the diameter of pin.

3.Stress in hole of pivot joint

Cross-sectional area of plate

A = (b – d)t

A = (80 – 36.3)40

A = 1748 mm^2

$\sigma = \frac{P}{A}$

$\sigma = \frac{75000}{1748}$

σ =42.9 N/mm^2

$\sigma_{max} = k_t\ \sigma$

$\sigma_{max} = 2.17\ (42.9)$

k_t = 2.17 is take from the ratio of b/d.

$\sigma_{max} = 93.093\ N/mm^2$

Chapter 5

Finite Elements Analysis of Dolly

5.1 Introduction to FEA

Finite Element Analysis (FEA) has become commonplace in recent years, and is now the basis of a multi billion dollar per year industry. Numerical solutions to even very complicated stress problems can now be obtained routinely using FEA, and the method is so important that even introductory treatments of Mechanics of Materials such as these modules should outline its principal features.
Finite Element Analysis (FEA) is a numerical method for solving a differential or integral equation. It has been applied to a number of physical problems, where the governing differential equations are available. The method essentially consists of assuming the piecewise continuous function for the solution and obtaining the parameters of the functions in a manner that reduces the error in the solution.

5.2 Introduction of ANSYS

ANSYS is the original (and commonly used) name for a Multi physics, general-purpose finite element analysis software. ANSYS Multi physics are self contained analysis tools incorporating pre-processing (geometry creation, meshing), solver and post process- ing modules in a unified graphical user interface.At ANSYS,the possibilities for most complex design challenges through fast, accurate and reliable simulation.

The technology enables organizations to predict with confidence that their products will thrive in the real world. Figure 5.1 shows the flowchart for finite element analysis using this software.

Now, we come to the dolly simulation using Finite Element Method with the simula-

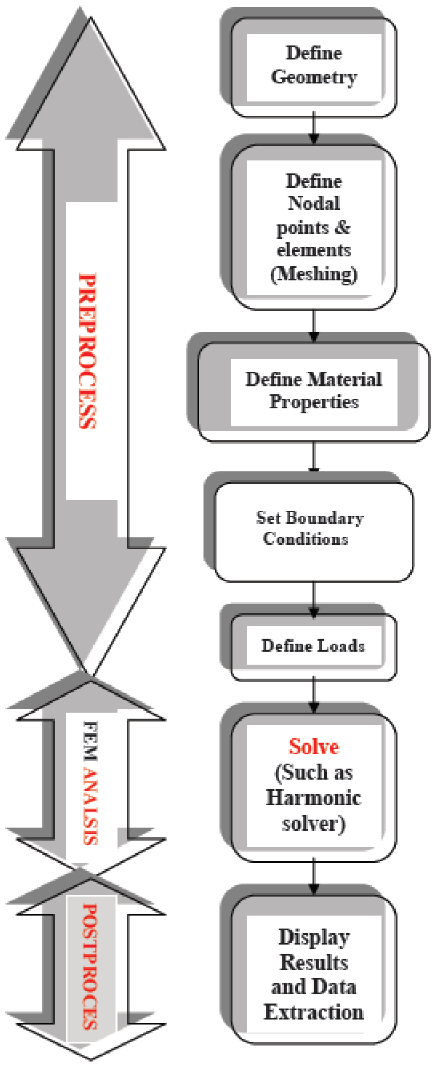


Figure 5.1: Flow chart of FEA analysis using ANSYS

tion tool ANSYS. As shown in figure 5.1, the task sequence in an ANSYS simulation has some main parts which are essentially the same no matter how your simulation project looks like:

a. Geometry Modeling

b. Setting up material properties(such as permeability,resistivity,conductivity etc)

c. Meshing

d. Application of loads and degrees of freedom. Deciding what boundary conditions have to be full filled.

e. Numerical solving: This gives us the solution for every nodes or elements(discrete!)

f. Postprocessing: Visualization of element solution and data export

5.3 Geometry and Material properties

ANSYS gives the facility to create the geometry whose analysis has to carry out. But it is always easy to create geometry in any modeling software like NX , Pro-E, Solid Works etc. Here the modeling is carried out using the NX and then that solid model is exported in IGES file then this .igs file is called in the ANSYS simulation wizard by importing it. ANSYS also gives the facility to define the material to each and every part to calculate the accurate stress and deformation of the geometry. Here the elements of the geometry are made up of structural steel.

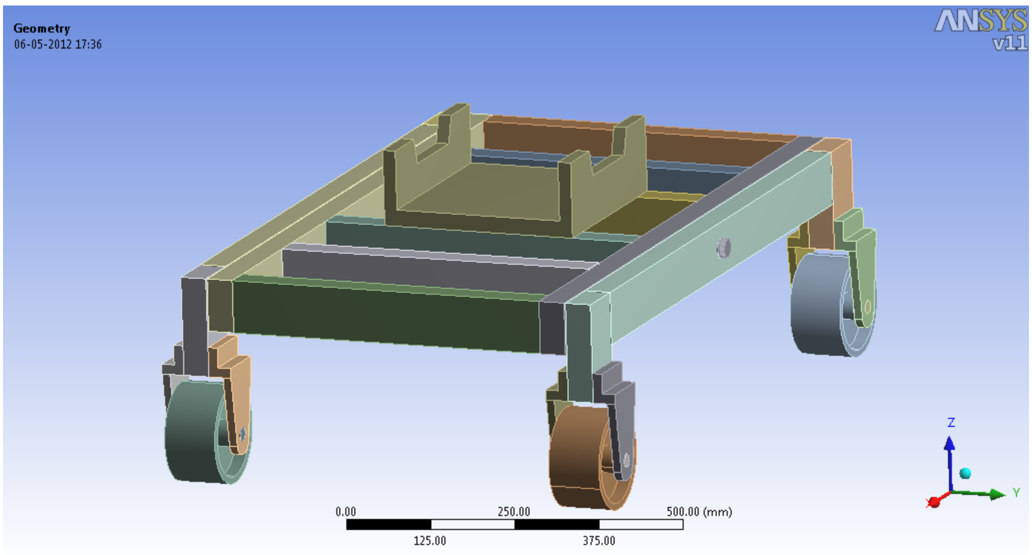


Figure 5.2: Geometry of Dolly Import from NX

5.4 Meshing and element types

Meshing is the process in which geometry is spatially discretized into elements and nodes. This mesh along with material properties is used to mathematically represent the stiffness and mass distribution of structure. Here, we have applied Patch conforming meshing. This is a meshing technique in which all faces and their boundaries (edges and vertices) [patches] within a very small tolerance are respected for a given part. Patch conforming meshing is invariant to loads, boundary conditions, Named Selections, results or any scoped object. That is, when change the scope of an object, there is no need to re-mesh. The Patch Conforming Tetra mesh method provides Support for 3D inflation, Built-in pyramid layer for conformal quad-tet transition and Built-in growth and smoothness control. We can try to create a smooth size variation based on the specified growth factor.A solid Tetrahedron elements are generated because the method used for meshing is Tet method on the wheel and its frame.

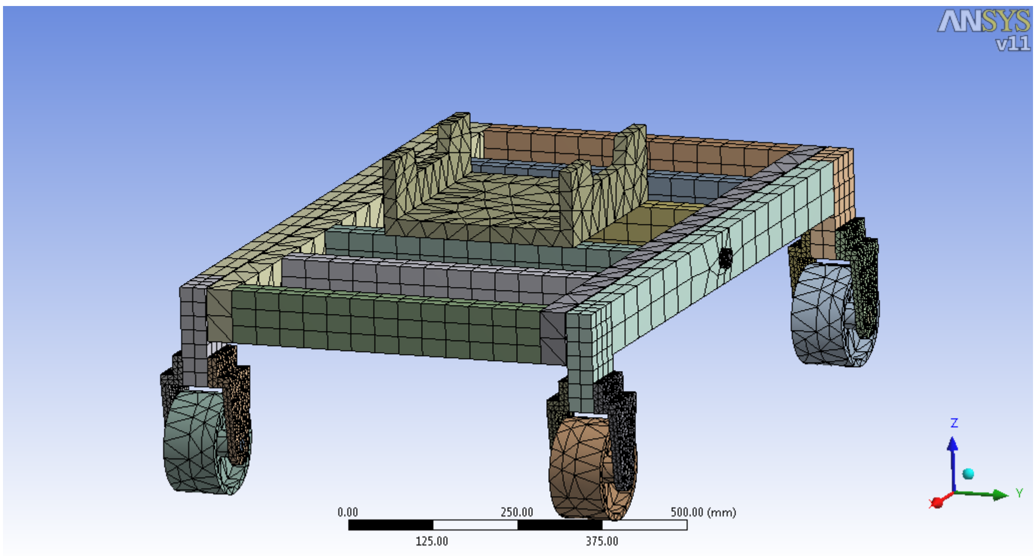


Figure 5.3: Meshing of the Dolly

5.5 The Load

After having defined the simulation geometry along with the material properties and having defined the finite element mesh along with the boundary conditions, we have to define the loads.On the dolly the load is the weight of the axle resting on it. As shown in fig 5.4 the force acting where the axle is place.

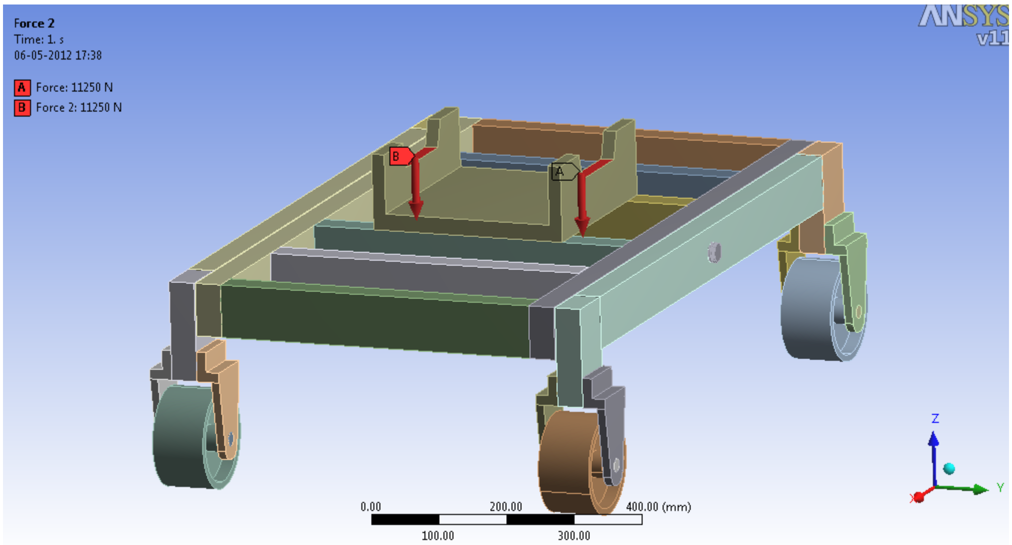


Figure 5.4: Forces Acting on the Dolly

5.6 Analysis Type and Solving

ANSYS has different analysis solving like Static, Dynamic , Transient, Modal , Harmonic etc. One has to define the analysis type and define which type of result they want. For e.g. here we have define analysis type as static and result desire are total deformation and stress on the dolly. Then solve the model with solve command.

5.7 Post processing

After building the model and obtaining the solution, it is need to show solution which defined before solving. To shoe it, we should perform post processing. Post processing means reviewing the results of an analysis. It is probably the most important step in the analysis, because one is trying to understand how the applied loads affect the design, how good the finite element mesh is, and so on. In ANSYS, two post processors are available to review the results: POST1, the general post processor, and POST2, the time-history postprocessor. Via the post processor of ANSYS, it is possible to plot or visualize the results at Nodal as well as element based solution.

Table 5.1: Comparing Deflection by Theoretical and FEA

Sr No	Parameters	Theoretical	FEA
1	Deflection	0.0406 mm	0.0306 mm

Fig 5.5 and Fig 5.6 are post processing result of ANSYS which is showing the solution of deflection and Stress in the Dolly.

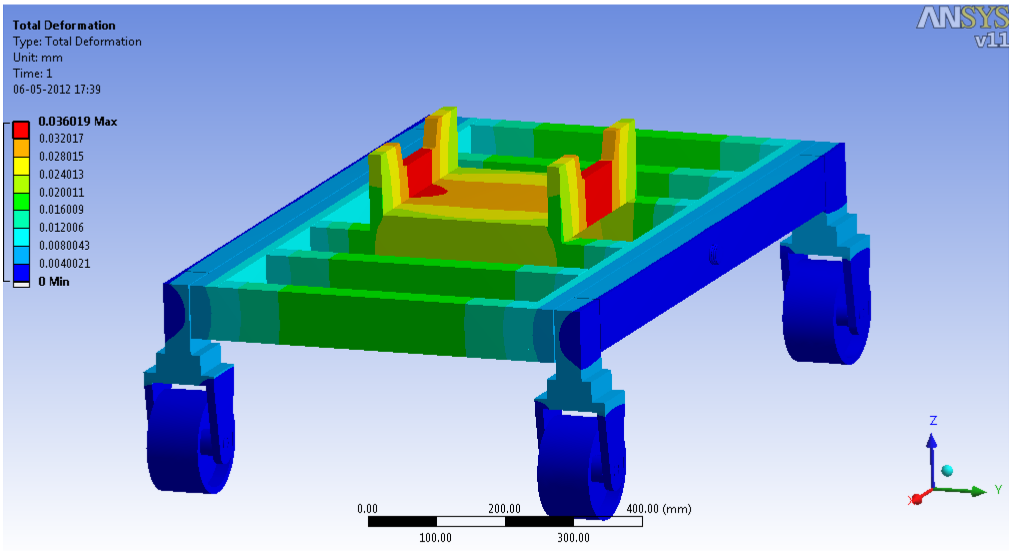


Figure 5.5: Total Deformation of the Dolly

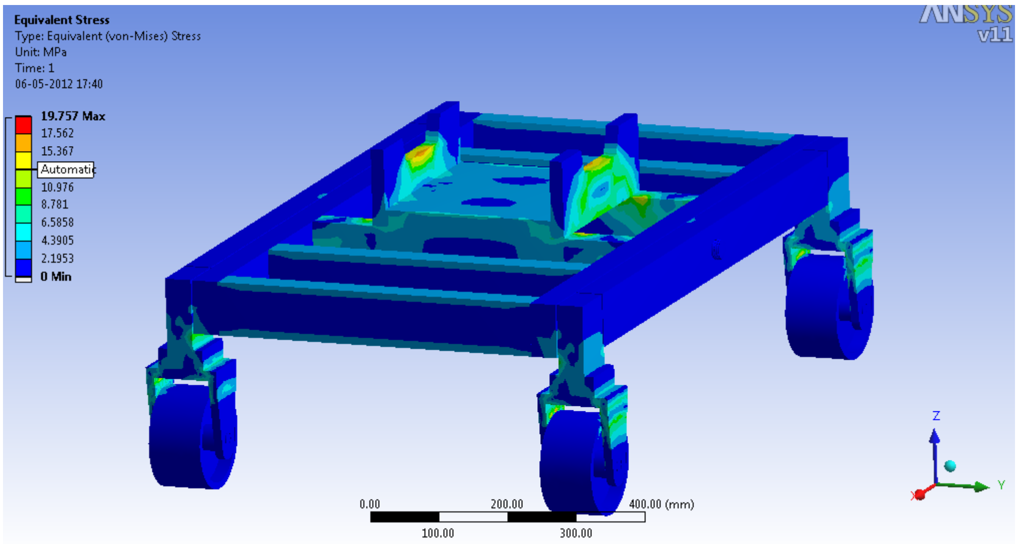


Figure 5.6: Stress acting on the Dolly

5.8 Shear stress at pivot joint

Pivot point is one of the critical component of new dolly. From this point the load will transfer from one frame to second frame. Due to the vertical load acting on the pivot pin it can be consider as circular beam and shear stress analysis is carried out.The maximum shear stress value is 22.673MPa. Fig 5.7 and 5.8 shows the meshing and shear solution in ANSYS.

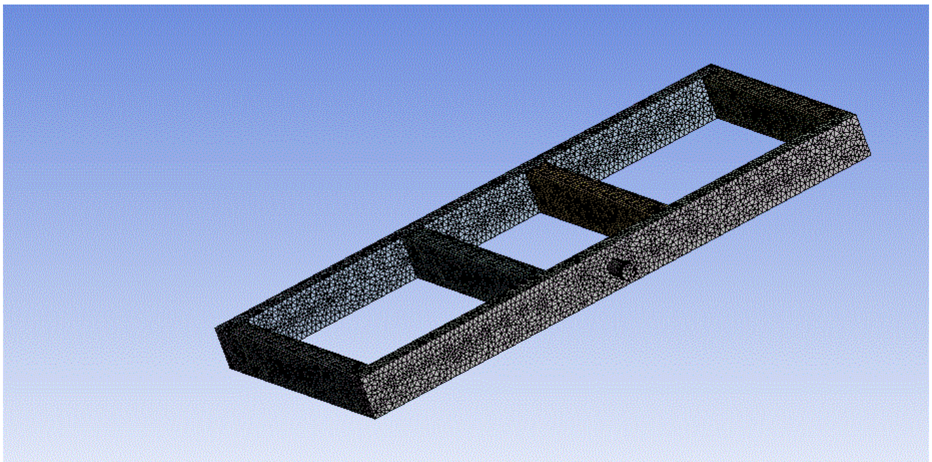


Figure 5.7: Stress acting on the Dolly

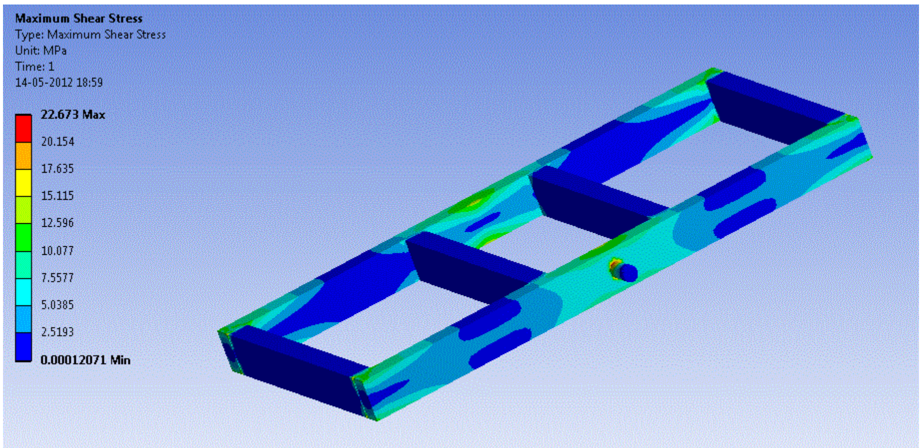


Figure 5.8: Stress acting on the Dolly

Chapter 6

Analysis of Contact Stress

6.1 Contact Stress

Contact stress occur when a part of the body presses against the hard or sharp edges. Many mechanical devices have sliding or rolling contacts. Generally contacting surfaces are non-conforming so that the area through which the load is transmitted is very small, even after some surface deformation, and the pressures and local stresses are very high. These stresses reduce the life of the component and due to the fatigue stress the component may fail. The failure of the components depend type of the materials and the intensity of the applied load as well as the surface finish, lubrication and relative motion.

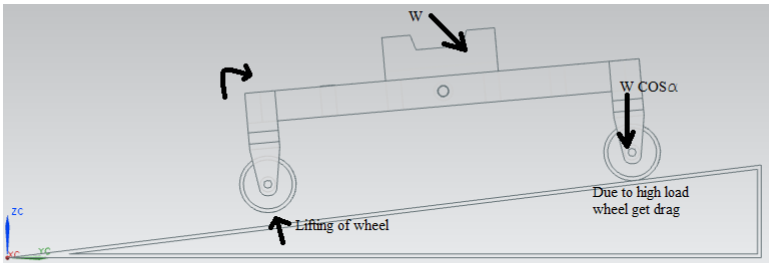
The contact are of two types:
1) Concave or Conforming: If the surfaces of the two elements fit exactly or even closely together without deformation are called concave or Conforming.
2) Convex or Non-Conforming : Elements that have dissimilar profiles are considered to be Nonconforming or Convex.

Contact problem analysis are based on the Hertz theory, which is an approximation on two counts. First, the geometry of general curved surfaces is described by quadratic terms only and second, the two bodies, at least one of which must have a curved surface, are taken to deform as though they were elastic half-spaces. The accuracy of Hertz theory is in doubt if the ratio a/R (a is the radius of the contact area and R is the radius of curvature of contacting elements) becomes too large.With metallic elements this restriction is ensured by the small strains at which the elastic limit is reached. However, a different situation arises with compliant elastic solids like rubber. A different problem is encountered with conforming (concave) surfaces in contact, for example, a pin in a closely fitting hole or by a ball and socket joint. Here, the arc of contact may be large compared with the radius of the hole or socket without incurring large strains.

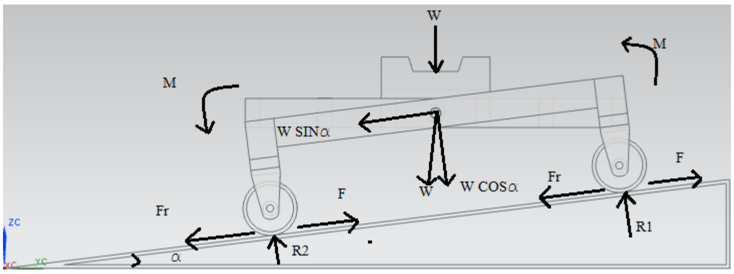
Here the contact stress occurs between the track of conveyor of assembly line and the wheel of dolly. Three cases are discussed and the result between them is compared.

1) Old design dolly wheel is moving on the inclined track. Fig 6.1(a)
2) New design dolly wheel moving on the inclined track. Fig 6.2(b)

As shown in the above fig 6.1(a) when the old dolly is climbing the slope a point come when the chassis of the truck will became completely straight when the rear dolly completely straight and at the same time the front dolly is at the slope So, it will generate the moment at the front wheel of dolly and the back wheels of the dolly will get lifted at that point the whole weight acting on the dolly is directly come on the front wheels. As the dolly gets lift up the maximum load will transfer to the front wheels as result of this contact stress increase at the front wheels and dolly get drag. Fig 6.1(b) show the new design of dolly climbing the slope here as we have discuss the frame with wheel move individually so, that it maintain the proper contact between both the wheels and the track. Due to the proper contact of the wheels the weight transfer through each wheels are same and as result the equal contact stress will generate.



(a) Old dolly on the slope



(b) New dolly on the slope

Figure 6.1: Positions of Wheel on Track

6.2 Analytical Stress calculation

Contact stress calculated by the Hertz contact stress equation used for two cylinders are in contact. We can use this equation because there is a line contact is between wheel and track.

The resulting pressure causes the line of contact to become a rectangular contact zone of half-width b given as:

$$b = k_b(F)^{1/2} \tag{6.1}$$

$$k_b = [2/\pi l]^{1/2}\left[\frac{\left(\frac{1-\ v_1{}^2}{E_1}+\frac{1-\ v_2{}^2}{E_2}\right)}{\frac{1}{d_1}+\frac{1}{d_2}}\right]^{1/2} \tag{6.2}$$

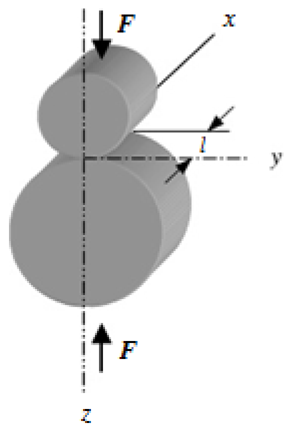


Figure 6.2: Two right circular cylinders held in contact by forces F uniformly distributed along cylinder length l

Where,

F= Applied Force

v_1,v_2 = Poission's ratios for cylinders 1 and 2

E_1,E_2 = Elastic Modulus for cylinders 1 and 2

d_1,d_2 = diameter of the cylinders 1 and 2

l= length of the cylinders

The maximum contact pressure between the cylinders acts along a longitudinal line at the center of the rectangular contact area, and is computed as:

$$P_{max} = \frac{F}{\pi bl} \tag{6.3}$$

Calculation of Principal Stresses and Maximum Shear Stress for the first case where the old dolly climbing the slope and weight acting on dolly directly coming on front wheel. There is the lift of 10 to 15 mm at the back wheel that means it form a angle between 10 degree.

$$\sigma_x \ = -2vPmax[\sqrt{1+\ {\zeta_b}^2} - \zeta_b] \tag{6.4}$$

$$\sigma_y = -Pmax[\left(\frac{1+2{\zeta_b}^2}{\sqrt{1+\ {\zeta_b}^2}}\right) - 2\zeta_b] \tag{6.5}$$

$$\sigma_z = -Pmax \frac{1}{\sqrt{1 + \ \zeta_b{}^2}} \tag{6.6}$$

$$\tau max = [\sigma_y - \sigma_z \ ]/2 \tag{6.7}$$

where, σ_x , σ_y and σ_z = Principal Stresses

τmax = Maximum Shear Stress

Now Calculating the stress for both the cases:

A) For the old dolly.

B) For the new dolly. Calculation of contact stress for Case A.

Table 6.1: Input Data for Case A

Sr. no	Parameters	Values
1	E_1	100000 N/mm^2
2	E_2	200000 N/mm^2
3	d_1	200 mm
4	d_2	0
5	v_1	0.265
6	v_2	0.3
7	F	3722 N

1. Compute contact half-width,b

$$K_b = \sqrt{\left[\frac{2\ R_1}{\pi\ l}\ \frac{(1-\ v_1{}^2)}{E_1} + \frac{(1-\ v_2{}^2)}{E_2}\right]}$$

$$K_b = \sqrt{\left[\frac{2\ 100}{\pi\ 75}\ \frac{\left(1-\ 0.265^2\right)}{10^5} + \frac{\left(1-\ 0.3^2\right)}{2\ 10^5}\right]}$$

$K_b = 3.43^{-03}$ mm/$N^{1/2}$

b = $K_b \sqrt{F}$

b = $3.43^{-03}\ \sqrt{3722}$

b = 0.209 mm

$\zeta_b = \frac{0.1}{0.209}$

$\zeta_b =$ 0.478

2. Maximum Pressure, Pmax

$P_{max}= \frac{2\ (3722)}{\pi\ (0.209)(75)}$

P_{max}=151.16 N/mm^2

3. Hertz Contact Stresses

$\sigma_x= -2\,(0.3)\,(151.16)\ \left(\sqrt{(1+0.478^2)}-0.478\right)$

σ_x=-57.17 N/mm^2

$\sigma_y = \ -(151.16)\left[\left(\frac{1+2(0.478)^2}{\sqrt{1+\ (0.478)^2}}\right)-2(0.478)\ \right]$

$\sigma_y = \ -54.19\ N/mm^2$

$\sigma_z = \ -(151.16)\ \frac{1}{\sqrt{1+(0.478)^2}}$

$\sigma_z = -136.38\ N/mm^2$

4. Maximum Shear Stress

$\tau_{max} \ = \ \frac{\sigma_y - \ \sigma_z}{2}$

$\tau_{max} \ = \ \frac{-136.38+54.19}{2}$

$\tau_{max} = -41.1\ N/mm^2$

Table 6.2: Output Data of Case A

Sr. no	Parameters	Values
1	k_b	3.43 10^{-3} mm/$F^{1/2}$
2	b	0.209 mm
3	Pmax	151.16 N/mm^2
4	σ_x	-57.17 N/mm^2
5	σ_y	-54.19 N/mm^2
6	σ_z	-136.38 N/mm^2
7	τmax	-41.1 N/mm^2

Calculation of contact stress for Case B.

1. Compute contact half-width,b

$K_b = \sqrt{\left[\frac{2\ R_1}{\pi\ l}\ \frac{(1-\ v_1{}^2)}{E_1} + \ \frac{(1-\ v_2{}^2)}{E_2}\right]}$

Table 6.3: Input Data for Case B

Sr. no	Parameters	Values
1	E_1	100000 N/mm^2
2	E_2	200000 N/mm^2
3	d_1	200 mm
4	d_2	0
5	v_1	0.265
6	v_2	0.3
7	F	1875 N

$$K_b = \sqrt{\left[\frac{2}{\pi}\frac{100}{75}\ \frac{\left(1-\ 0.265^2\right)}{10^5} + \frac{\left(1-\ 0.3^2\right)}{2\ 10^5}\right]}$$

$K_b = 3.43^{-03}$ mm/$N^{1/2}$

b = $K_b\ \sqrt{F}$

b = $3.43^{-03}\ \sqrt{1875}$

b = 0.149 mm

$\zeta_b = \ \frac{0.1}{0.148}$

$\zeta_b = \ 0.675$

2. Maximum Pressure, Pmax

$P_{max} = \ \frac{2\ (1875)}{\pi\ (0.149)(75)}$

$P_{max} = 106.8 \ \ N/mm^2$

3. Hertz Contact Stresses

$\sigma_x = \ -2\,(0.3)\,(106.8)\ (\sqrt{\left(1 + 0.675^2\right)} - 0.675)$

$\sigma_x = -34.1\ N/mm^2$

$\sigma_y = \ -(106.8)\left[\left(\frac{1{+}2(0.675)^2}{\sqrt{1{+}\ (0.675)^2}}\right) - 2(0.675)\ \right]$

$\sigma_y = \ -25 \ \ N/mm^2$

$\sigma_z = \ -\,(106.8)\,\frac{1}{\sqrt{1{+}(0.675)^2}}$

$\sigma_z = -88.52\ N/mm^2$

4. Maximum Shear Stress

$\tau_{max} = \frac{\sigma_y - \sigma_z}{2}$

$\tau_{max} = \frac{-88.52+25}{2}$

$\tau_{max} = -31.1\ N/mm^2$

Table 6.4: Output Data of Case B

Sr. no	Parameters	Values
1	k_b	3.43 10^{-3} mm/$F^{1/2}$
2	b	0.149 mm
3	Pmax	106.8 N/mm^2
4	σ_x	-34.1 N/mm^2
5	σ_y	-25 N/mm^2
6	σ_z	-88.52 N/mm^2
7	τmax	-31.1 N/mm^2

6.3 Contact Stress Analysis by FEA

Procedure to carry out the finite element analysis.

A. Geometry Modeling

Geometry Modeling is done in the ANSYS itself by using Modeling command from Preprocessor. Elements Type used is Solid 185 which is having 8 nodes.

B. Setting up material properties(such as permeability,resistivity,conductivity etc)

Here two material are define for wheel and track.

Table 6.5: For wheel Material

Sr. no	Parameters	Values
1	E_1	100000 N/mm^2
2	v_1	0.265

Table 6.6: For track Material

Sr. no	Parameters	Values
1	E_2	200000 N/mm^2
2	v_2	0.3

C. Meshing Mesh tool is used for meshing the elements in which the volume sweep mesh is used. Fig 6.3 show the meshing along with boundary condition on the model.

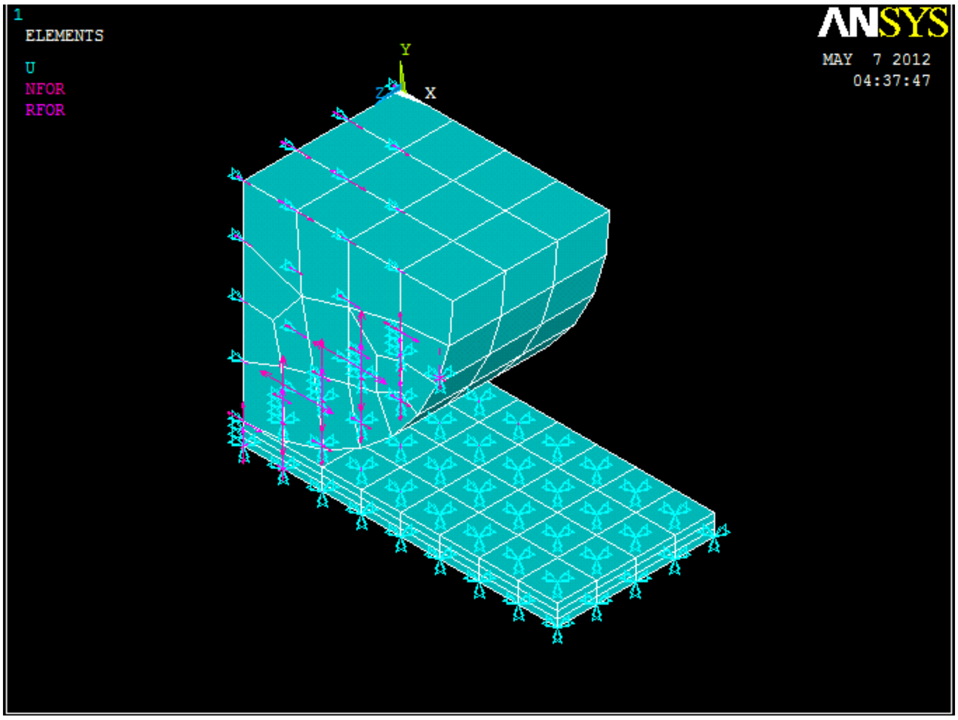


Figure 6.3: Meshing and Boundary Condition of Contact Elements

D. Application of loads and degrees of freedom. Deciding what boundary conditions have to be full filled.

E. Numerical solving: This gives us the solution for every nodes or elements(discrete!)

F. Postprocessing: Visualization of element solution and data export

Table 6.7: Comparing Theoretical and FEA Contact Stress

Sr no	Dolly	Theoretical	FEA
1	Old Dolly	-41.1 N/mm^2	-45.606 N/mm^2
2	New Dolly	-31.1 N/mm^2	-32.88 N/mm^2

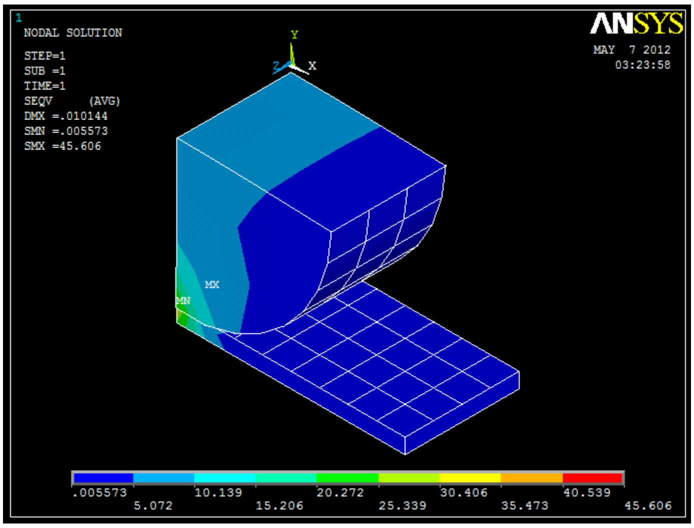


Figure 6.4: Contact Stress due to old dolly on slope

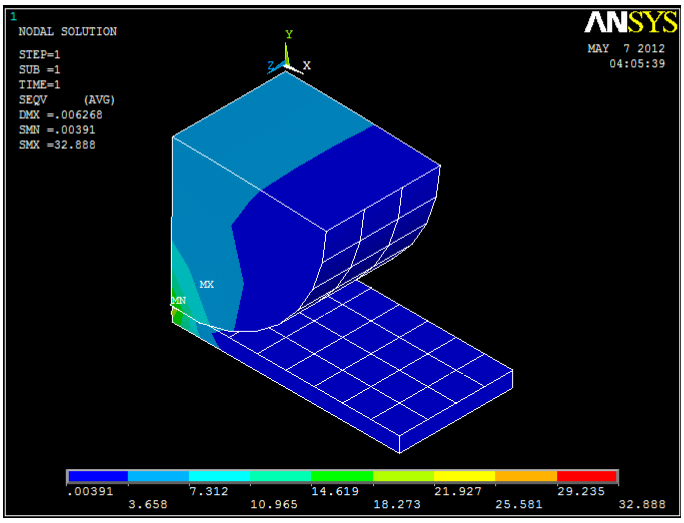


Figure 6.5: Contact Stress due to new dolly on slope

6.4 Surface Wear of the Track

The flow Diagram for a systems analysis of wear failure is indicated in Fig 6.6. The methodology comprises an analysis of operating variable and a structural analysis to determine the nature of the wear process.

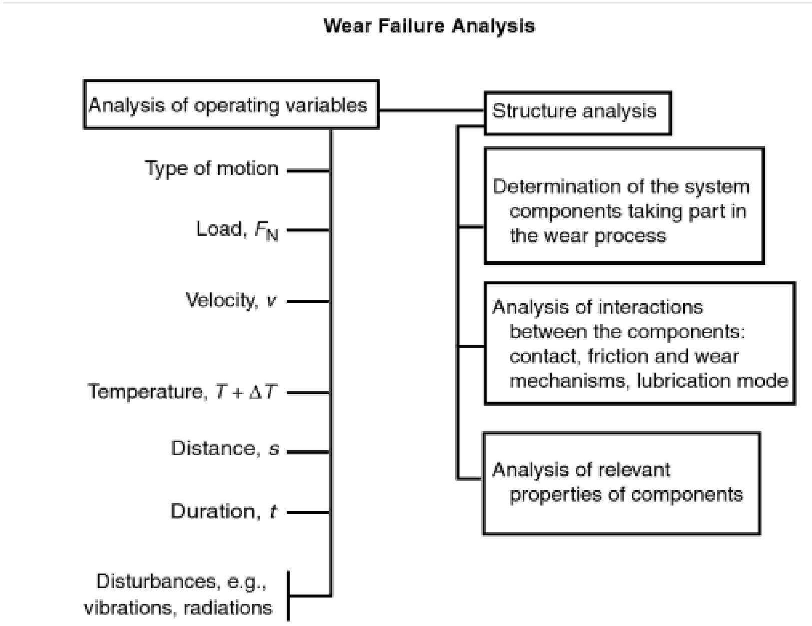


Figure 6.6: Flow Diagram for a systems analysis of Wear Failure

6.4.1 Calculation of Wear

A common used equation to compute the wear rate is

$$Vi = ki\ F\ s$$

where,

F = the normal load

s = the sliding distance

Vi = wear volume

ki = specific wear rate coefficient.

The k-value is given in m3/Nm or m2/N, sometimes in mm3/Nm. From design view the wear displacement h is more convenient than V. With hi =Vi /A, the contact pressure p=F/A where A is the area subjected to wear then:

$$hi = ki\ p\ s$$

The sliding distance s can be replaced by s=v.t where v is the mean value for the slide rate and t the running time.

Calculating the wear for Case A.

Table 6.8: Input Data for Case A

Sr. no	Parameters	Values
1	ki	10^{-6} mm^2/N
2	p	3 N/mm^2
3	s	20 mm/s

hi = (10^{-6})(3)(20)

hi = 6 10^{-5} mm

Calculating the wear for Case B.

Table 6.9: Input Data for Case B

Sr. no	Parameters	Values
1	ki	10^{-6} mm^2/N
2	p	2.14 N/mm^2
3	s	20 mm/s

hi = (10^{-6})(2.14)(20)

hi = 4.28 10^{-5} mm

Table 6.10: Output Data

Sr. no	Case	Values
1	A	6 10^{-5} mm
2	B	4.28 10^{-5} mm

Chapter 7

Conclusions

In the present work of redesigning and analysis the following conclusions are made.

1. Calculation of the forces acting on dolly and proving the stress with in the limit.
2. Design of new dolly and calculation of its stability on the slope of the track.
3. Calculation of contact stress analysis is carried out between the dolly wheel and slope on the track of assembly line and compare the result with finite element analysis.
4. Contact stress reduce on the slope of track by 24.33 % from new design of dolly.
5. Wear on the slope of the track is reduce by 28.66 % from new design of dolly.

References

[1] R.Siegwart, P. Lamon , T. Estierand ,M. Lauria, R. Piguet, Innovative design for wheeled locomotion in rough terrain,Robotics and Autonomous Systems 40,p.151-162,2002.

[2] J. Pijuan, M. Comellas, M. Nogues, J. Roca, X. Potau, Active bogies and chassis levelling for a vehicle operating in rough terrain,Journal of Terramechanics,Vol 49,issue 3,2012

[3] Aushutosh Dubey and Vivek Dwivedi, Vehicle Chassis Analysis:Load Cases and Boundary Condition for Stress Analysis, TATA Research and Development Centre,Pune.

[4] X. Potau, M. Comellas, M. Nogues, J. Roca, Comparison of different bogie configurations for a vehicle operating in rough terrain,Journal of Terramechanics 48,p.7584,2011.

[5] Mehmet Ali Arslan,Oyus Kayabasi, 3-D RailWheel contact analysis using FEA,Advances in Engineering Software 45,p.325331,2012.

[6] Masahiro Kioke,Sanshirou Shimoda,Toshihide Shibuya,Hirofumi Miwa,Developement of Kinematical Analysis Method for Vehicle,KOMATSU Technical Report,VOL.50 No.153,2004

[7] Valda Gasic,Milorad Milovancevic and Zoran Petkovic,FEA Implementation in Moving Load Problem at Bridge Crane,50th Anniversary of the Faculty of Technical Science, 2010.

[8] Jia-Jang Wu, A.R. Whittaker, M.P. Cartmell,The use of finite element techniques for calculating the dynamic response of structures to moving load,Computers and Structures,VOL. 20,p.789-799,2000.

[9] Harish V. Katore, Santhosh B. Jaju,Redesigning of Tractor Trolley Axle using ANSYS,International Journal of Engineering Science and Technology, VOL. 42, NO. 4,2006.

[10] C.Alkin, C.E. Imrak, H.Kocabas,Solid Modeling and Finite Element Analysis of an Overhead Crane Bridge, Acta Polytechnica,vol. 45, No.3,2005.

[11] Mohammad Reza Forouzan,Rouhollah Hoseini,Dynamic Analysis of Modified Truck Chassis,July 2010.

[12] Dr. S.B. Jaju,N.K. Mandavgade,Anil R. Sahu,FEA Analysis of Tilting Mechanism of three Furrows Reversible Plough, G.H. Raisoni College of Engineering,Nagpur.

[13] N. Sefa Kuralay,Cicek Karaoglu,Stress analysis of a truck chassis with riveted joints,Finite Elements in Analysis and Design 38,p.1115-1130,2002.

[14] I.B. Eryurek, M. Ereke, A. Goksenli, Failure analysis of the suspension spring of a light duty truck,Engineering Failure Analysis 14,p.170178,2007.

[15] Tushar. N. Khobragrade, P. Priadarshni, Static Analysis of Gear Casing, Technplogy Centre Greaves Cotton Limited",Aurangabad

[16] T.Rangaswamy,S.Vijayarangan,R.A.Chandrashekar,T.K. Venkatesh,K.Anantharaman, Optimal Design and Analysis of Automotive Composite Drive Shaft, International Symposium of Research Students on Materials Science and Engineering, 2004