Design and Analysis of Door Lifting Mechanism Frame Structure, Rope & Hook

A Project Report

Submitted by

A.Dorai Swamy - 71205402001

in partial fulfillment for the award of the degree
of

Master of Engineering
in
CAD/CAM

DEPARTMENT OF MECHANICAL ENGINEERING

KUMARAGURU COLLEGE OF TECHNOLOGY
COIMBATORE – 641 006

ANNA UNIVERSITY:: CHENNAI 600 025

JUNE – 2007

ANNA UNIVERSITY :: CHENNAI 600 025

BONAFIDE CERTIFICATE

Certified that this project report entitled **"Design and Analysis of Door Lifting Mechanism Frame Structure, Rope and Hook"** is the bonafide work of

Mr. A. Dorai Swamy - Register No. 71205402001

Who carried out the project work under my supervision.

Signature of the Head of the Department

Signature of the supervisor 13/6/2007
Mr. S.KALIDASS

Internal Examiner

4/7/07
External Examiner

DEPARTMENT OF MECHANICAL ENGINEERING

KUMARAGURU COLLEGE OF TECHNOLOGY

COIMBATORE 641 006

MAHENDRA ENGINEERING COLLEGE

Mahendhirapuri, Mallasamudram (w), Vadugapalayam (Po),
Namakkal Dt., - 637 503.

FS68172

Certificate

This is to certify that

Mr./ Ms.

has presented a paper entitled

in the National Conference on

Emerging Trends in Civil and Mechanical Engineering - ETCM 2007

Organized by the Departments of Civil & Mechanical Engineering,

Mahendra Engineering College, Mahendhirapuri on 20th April, 2007

Prof. C. Prabhu Rama Krishnan
Convenor

Prof. T. M. Murali
Convenor

Dr. K. Sukumaran
Patron

ABSTRACT

A coke oven plant targets at converting coal into coke. In this process the coal is heated for a period of around 24-36 hours. Burning transforms coal into coke and this is done in a series of furnaces. To avoid heat loss during burning the ovens are closed by a door. The door weighs 8 tons in weight. There exists a mechanism which is meant to lift the door called "Door Lifting Mechanism". The function of this mechanism is to lift the door and lower the door. There exists only one door lifting mechanism to lift and lower all 256 doors. This is done in an interesting way where a rope is actuated by the door lifting mechanism wound around a pulley. This pulley is actuated by a hydraulic cylinder to lift and lower. The rope end has a hook and a continuous rope goes across the entire stretch of series of ovens. Which ever door needs to be lifted is hooked on to this common rope. A series of problems have been anticipated, first the structure is getting skewed, a form of buckling phenomenon taking place. Second is the failure of the hook and the last problem is that of the damage caused on the rope. The problems discussed above are tried to solve using the analysis software ANSYS.

ஆய்வுச்சுருக்கம்

பழுப்பு நிலக்கரி ஆலையானது நிலக்கரியை பழுப்பு நிலக்கரியாக மாற்றுவதை இலக்காக கொண்டு செயல்படுகிறது. இச்செயலில் நிலக்கரி 24 மணி முதல் 36 மணி நேரம் வரை சூடு படுத்தப்படுகிறது. எரியுத் பொழுது நிலக்கரி ஆனது பழுப்பு நிலக்கரியாக மாற்றப்படுகிறது. இது தொடர்முறை அடுப்புகளில் நடக்கிறது. வெப்ப இழப்பை தவிர்க்க அடுப்புகள் கதவுகளால் அடைக்கப்படுகின்றன. கதவானாது 8டன் எடை கொண்டது. இதில் Door Lifting Mechanism – எனும் முறை பயன்படுத்தப்படுகிறது. இந்த முறையானது கதவை திறந்து மூட உதவுகிறது. மொத்தம் உள்ள 256 கதவுகளை திறந்து மூடவும் ஒரேயொரு Door Lifting Mechanism – தான் உள்ளது. Pulley –ல் இணைக்கப்பட்டுள்ள இரும்பு கயிறு மூலமாக door Lifting mechanism செயல்படுகிறது. இரும்புக் கயிறின் இறுதிப் பகுதியானது Hook –ஐ கொண்டதாகும். இம்முறையில் தொடர் பிரச்சினைகள் காணப்பட்டன. முதலாவதாக அழுத்தத்தினால் ஏற்படும் மாற்றம், இரண்டாவதாக Hook-ன் செயல் இழப்பும், மூன்றாவதாக இரும்புக் கயிறில் ஏற்படும் சேதம் போன்றவையாகும். மேலே கண்ட மூன்று குறைகளும் 'ANSYS' என்ற மென்பொருள் கொண்டு களையப்பட முயற்சி / ஆய்வு மேற்கொள்ளப்பட்டுள்ளது.

ACKNOWLEDGEMENT

The author grateful to his guide **Mr. S.Kalidass.**, Lecturer, Department of Mechanical Engineering, Kumaraguru College of Technology, Coimbatore for his excellent, utmost motivation, valuable advice, untiring support, timely suggestion, constant encouragement, enthusiasm, relentless patience, and inspiration throughout the study, holding in all the places.

The author expresses humble gratitude to **Dr.N.Gunasekaran**, Professor of the Mechanical Engineering, Kumaraguru College of Technology, Coimbatore for facilitating conditions for carrying out the work smoothly.

The author wishes to express his deep sense of reverential gratitude to **Dr.Joseph V. Thanikal**, Principal, Kumaraguru College of Technology, Coimbatore, for providing the facilities to conduct this study.

The author owes his immense gratitude and thanks to **Dr.P.Palanisamy**, Assistant Professor, Department of Mechanical Engineering, Kumaraguru College of Technology, Coimbatore for his valuable guidance, encouragement and support during the entire course of this project.

The author also expresses his sincere thanks to all the faculty and staff members, those who helped in carrying out the project work.

CONTENTS

LIST OF SYMBOLS & ABBREVATIONS

A, B - Respectively longer and shorter leg length of angle section.

B - Flange width of beam, column or channel sections.

D - Depth of beam, column or channel sections.

R1 - Radius at fillet or root.

R2 - Radius at toe.

t - Thickness of web of beam, column or channel section.

T - Thickness of flange of beam, column or channel section.

K - Element Stiffness Matrix.

V - Nodal Point Displacement

DLM - Door Lifting Mechanism.

FEM - Finite Element Method.

FEA - Finite Element Analysis.

DOF - Degrees of Freedom

LIST OF FIGURES

Chapter 1

Objective

Problem 1

Skewing of the structure is predictably due to the criteria. The critical backlog load of the structure is 8.2T which is close to the total load of 8T. The problem will be resolved by introduction of extra diagonal members of structure according to the behavior of the structure.

Problem 2

The second problem is that of the rope failure. The main reason for the failure is the rope portion around the pulley. The after stroke comes to pulley. The compression & tension fibre direction changes. Due to this, the design needs to be changed for ropes will be reverse cycled rather than repeated cycles. But the rope cannot be changed as it is costly process. Instead, pulley realignment / repositioning will be considered as an alternate solution.

Problem 3

The third problem is that of the hook which will be overcome by realigning the hook direction & changing the hook itself by going for a standard hook.

Chapter 2

Literature Survey

Shigley and Mischke et al. (1989) has discussed design analysis and review if stresses at a point in detail. He says safety is an important consideration in any Product Design and design safety can be checked by finite element analysis.In the brief discussion he makes the readers to understand some of the terms that frequently occur in design analysis.

Dean Q.Lewis et al.(1991) gives the guidelines and equations that can predict the performance of features which are needed for efficient design and also gives the guidelines for designing features.

Bret Pettichord et al. (1998) provides practical suggestions that will inspire teams to make their software products more testable. He argues that all automation requires some instrumentation and that all and that automated testing must differ, to some degree, from actual customer usage.

Y.S. Choo et al. (1997) highlights the force split ratio for double sling arrangements. He presents three analytical models namely, the free slipping model, the fixed model and the friction models which analyze the tensions of double slings. He develops a corresponding computational model to include the effects of structural stiffness.

Toshihiro SAITO et al. (2002) has developed a simulation method which allows simultaneous evaluation of dynamic behavior and ring stress for estimation of durability of metals pushing V-belt for CVT. He focuses on dynamic behavior of pulley behavior and the newly improved simulation method which allows the pulley stiffness to be reflected.

Benjamin Brooks et al. (2002) aims to describe how people make decisions about health and safety risks. He attempts to combine several factors that influence the decision process – cultural, historical, social, and cognitive in the analysis. He considers this model with respect to decisions made by fishermen and wood product workers.

Stephen W. Attaway et al. (1996) presents an analysis of the loads in a typical climbing rope system subjected to a dynamic loading from a fall. He also considers the effects of friction, dynamic rope modulus, and rope condition. He uses the energy method to solve for the maximum strain energy in the rope.

Tytko et al. (1999) reviews different sets of equipment which are available for use for the non-destructive wire ropes on a world-wide basis. He lists twenty six companies who either design, manufacture or market NDT equipment. He also describes the main principles of operation magnetic NDT equipment.

R. Drake et al. states, for maximum safety the crane has two rope systems in order that a load can be safely supported in the event of a single rope failure. He discusses rope failure in 9 DOCK RAH crane, hoist block between pulleys, and head pulley assembly.

Mohammed Raoof et al. (1996) reviews the recent theoretical work on the 2*2 stiffness matrix describing the axial/torsional coupling of large diameter wire ropes. He also reviews the salient features for both approaches with an emphasis on the characteristics of various wire rope constructions.

Chapter 3

Finite Element Method

3.1 INTRODUCTION

The Finite Element Method (FEM) is a numerical technique to obtain approximate solutions to a wide variety of engineering problems where the variables are related by means of algebraic, differential and integral equations. Finite Element Analysis is a way to simulate loading conditions on a design and determine the design's response to those conditions.

The design is modelled using discrete building blocks called elements. Each element has exact equations that describe how it responds to a certain load. The "sum" of the response of all elements in the model gives the total response of the design. The elements have a finite number of unknowns, hence the name finite elements.

Finite elements has become the industry standard for solving multi-disciplinary engineering problems that can be described by equations of calculus. Applications cut across several industries by virtue of the applications – solid mechanics (civil, aerospace, automotive, mechanical, biomedical, electronic), fluid mechanics (geotechnical, aerospace, electronic, environmental, hydraulics, biomedical, chemical), heat transfer (automotive, aerospace, electronic, chemical), acoustics (automotive, mechanical, aerospace), electromagnetic (electronic, aerospace) and many, many more.

3.2 THEORETICAL APPROACH TO FEM

The analysis of a structure by the Finite Element Method can be divided into several distinctive steps. These steps are to a large extent similar to the steps defined for the matrix method.

Various Steps involved in theoretical approach are

3.2.1 Discretization

Discretization is the process of dividing a problem into several small elements, connected with nodes. All elements and nodes must be numbered so that we can set up a matrix of connectivity. It is important to remember that the order the nodes and elements are numbered greatly affects the computing time. This is because we get a symmetrical, banded stiffness matrix, which bandwidth is dependent on the difference in the node numbers for each element, and this bandwidth is directly connected, with the number of calculations the computer has to do. Computer FEM-programs have internal numbering that optimises this bandwidth to a minimum by doing some internal renumbering of nodes if they are not optimal.

Rules for Discretization

- Two distinct elements can have common points only on their common boundaries if such boundaries exist. No overlapping is allowed. Common boundaries can be points, lines or surfaces.
- The assembled element should leave no holes within the two elements and approximate the geometry of the real body or structure as closely as possible to do.
- When the boundary of a structure or body cannot be exactly represented by the elements selected, an error cannot be avoided. Such error is called Geometric Discretization Error and it can be decreased by reducing the size of the elements or by using elements allowing boundaries to become curved.

3.2.2 Element Analysis

The element analysis has two key components, expressing the displacements within the elements, and maintaining equilibrium of the elements. In addition, stress-strain relationships are needed to maintain compatibility.

The final result is the element stiffness relationship. S = kV. For beam elements this relationship was obtained using the exact relationships between forces and moments and the corresponding displacements. These results could therefore be interpreted as being obtained by the governing differential equation and boundary condition of the beam elements.

For e.g. a plane stress problem it is not possible to use an exact solution. The displacements within the elements are expressed in terms of shape functions scaled by the node displacements. Hence, by assuming expressions for the shape functions, the displacements in an arbitrary point within the element are determined by the nodal point displacement.

The section of the structure that the element is representing is kept in place by the stresses along the edges. In the finite element analysis it is convenient to work with nodal point forces. Equivalent nodal point forces may in the general case replace the edge stresses by demanding the element to be in an integrated equilibrium using work or energy considerations. This technique is often referred to as to "lump" the edge forces to nodal forces.

This requirement results in a relationship between the nodal point displacements and forces to be given as:

$$\mathbf{S = KV + S^0}$$

Where,

S - Generalized nodal point forces

K - Element stiffness matrix

V - Nodal point displacements

S^0 - nodal point forces for external loads

Computer programs usually have many options for types of elements to choose among. The most usual elements are:

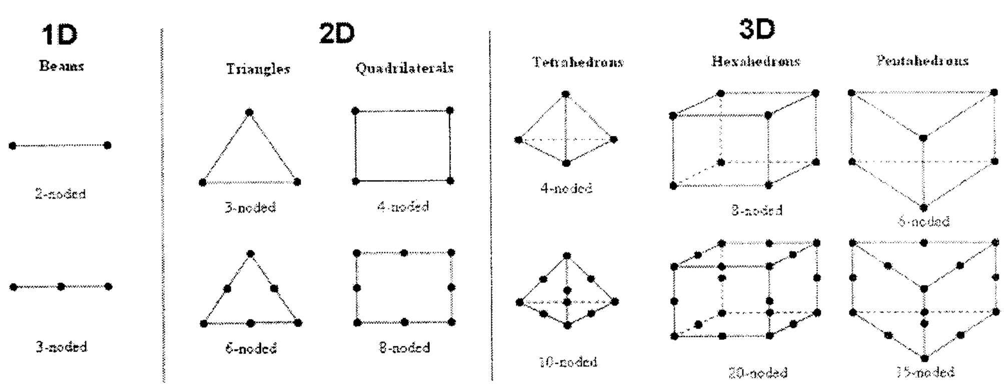


FIG 3.1TYPES OF ELEMENTS

3.2.3 System Analysis

A relationship between the load and the nodal point displacements is established by demanding equilibrium for all nodal points in the structure:

$$R = \mathbf{Kr} + \mathbf{R^0}, \text{ where}$$

$$K = \sum_j a^T{}_j k_j a_j$$

$$R^0 = \sum_j a^T{}_j S^0{}_j$$

The stiffness matrix is established by directly adding the contributions from the element stiffness matrices. Similarly the load vector R is obtained from the known nodal forces.

3.2.4 Boundary Conditions

Boundary conditions are introduced by setting nodal displacements to known values or spring stiffness are added.

3.2.5 Finding Global Displacements

The global displacements are found by solving the linear set of equations stated above:

$$\mathbf{r = K^{-1}(R - R^0)}$$

3.2.6 Calculation of Stresses

The stresses are determined from the strains by Hook's law. Strains are derived from the displacement functions within the element combined with Hook's law. They may be expressed generally by:

$$\boldsymbol{\sigma}(x, y, z) = \mathbf{D}\ \mathbf{B}(x, y, z)\ \mathbf{v}$$

Where,

v = a r

D - Hooke's law on matrix form

B - Derived from u (x, y, z)

Output interpretation programs, called postprocessors, help the user sort out the output and display it in graphical form.

3.3 STAGES OF FINITE ELEMENT ANALYSIS

A Finite Element analysis consist of three separated stages; Pre-processing, processing, and post processing. A complete finite element analysis is a logical interaction of these three stages.

A) Pre-processing
B) Processing/ Solution
C) Post processing

3.3.1 Pre-processing

As the name indicates, pre-processing is something you do before processing your analysis. The Pre-processing involves the preparations of data, such as nodal coordinates, connectivity, boundary conditions and loading and material information.

The preparation of data require considerable effort if all data are to be handled manually. If the model is small, the user can often just write a textile and feed it into the processor, but as the complexity of the model grows and the number of elements increase, writing the data manually can be very time consuming and error-prone. It's therefore necessary with a computer pre-processor, which help with mesh plotting and boundary conditions plotting.

3.3.2 Processing/ Solution

The processing stage involves stiffness generation, stiffness modification, and solution of equations, resulting in the evaluation of nodal variables. This is a typical "black box"-operation, as the user will see little of what's going on. You feed data from the pre-processor, and you get data out.

3.3.3 Post Processing

The post processing stage deals with the representation of results. Post-processing means reviewing the results of an analysis. It is probably the most important step in the analysis, because you are trying to understand how the applied loads affect your design, how good your finite element mesh is, and so on.

Typically, the deformed configuration, mode shapes, temperature, and stress distribution are computed and displayed at this stage. POST1, the general postprocessor, allows you to review the results over the entire model at specific load steps and sub steps (or at specific time-points or frequencies). In a static structural analysis, for example, you can display the stress distribution for any specific load step.

3.4 ANSYS INTRODUCTION

ANSYS is a general-purpose finite element-modelling package for numerically solving a wide variety of mechanical problems. The ANSYS program has a comprehensive graphical user interface that gives users easy, interactive access to program functions, commands, documentation, and reference material.

ANSYS can be used for the following

- Build computer models or transfer CAD models of structures, products, components, or systems.
- Apply operating loads or other design performance conditions.
- Study the physical responses, such as stress levels, temperature distributions, or the impact of electromagnetic fields.
- Optimize a design early in the development process to reduce production costs.
- Do prototype testing in environments where it otherwise would be undesirable or impossible (for example, biomedical applications).

3.5 TYPES OF ANALYSIS IN ANSYS

3.5.1 Structural Analysis

Structural analysis is probably the most common application of the finite element method. Examples of some structures are ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts, and tools. The types of structural analysis are

a) Static Analysis

Static analysis is used to determine the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. A static analysis can also include steady inertia loads (such as gravity and rotational velocity), and time varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes).

The kinds of loading that can be applied in a static analysis include:

- Externally applied forces and pressures
- Steady-state inertial forces (such as gravity or rotational velocity)
- Imposed (nonzero) displacements
- Temperatures (for thermal strain)
- Fluences (for nuclear swelling)

b) Modal Analysis:

You use modal analysis to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It also can be a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic response analysis, or a spectrum analysis

c) Harmonic Analysis

Any sustained cyclic load will produce a sustained cyclic response (a harmonic response) in a structural system. Harmonic response analysis gives you the ability to predict the sustained dynamic behaviour of your structures, thus enabling you to verify whether or not your designs will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations.

d) Transient Dynamic Analysis

Transient dynamic analysis (sometimes called time history analysis) is a technique used to determine the dynamic response of a structure under the action of any general time-dependent loads. You can use this type of analysis to determine the time-varying displacements, strains, stresses, and forces in a structure as it responds to any combination of static, transient, and harmonic loads. The time scale of the loading is such that the inertia or damping effects are considered to be important. If the inertia and damping effects are not important, you might be able to use a static analysis instead.

e) Spectrum Analysis

A spectrum analysis is one in which the results of a modal analysis are used with a known spectrum to calculate displacements and stresses in the model. It is mainly used in place of a time-history analysis to determine the response of structures to random or time-dependent loading conditions such as earthquakes, wind loads, ocean wave loads, jet engine thrust, rocket motor vibrations, and so on.

f) Buckling Analysis

Buckling analysis is a technique used to determine buckling loads - critical loads at which a structure becomes unstable – and buckled mode shapes - the characteristic shape associated with a structure's buckled response.

g) Non-linear Analysis

If a structure experiences large deformations, its changing geometric configuration can cause the structure to respond nonlinearly. An example would be the fishing rod. Geometric non-linearity is characterized by "large" displacements and/or rotations.

3.5.2 Thermal Analysis

ANSYS supports two types of thermal analysis

a) Steady-State Thermal Analysis

It determines the temperature distribution and other thermal quantities under steady-state loading conditions. A steady-state thermal analysis calculates the effects of steady thermal loads on a system or component. Engineer/analysts often perform a steady-state analysis before doing a transient thermal analysis, to help establish initial conditions.

A steady-state analysis can also be the last step of a transient thermal analysis performed after all transient effects have diminished. A steady-state loading condition is a situation where heat storage effects varying over a period of time can be ignored.

b) Transient Thermal Analysis

It determines the temperature distribution and other thermal quantities under conditions that vary over a period of time. A transient thermal analysis follows basically the same procedures as a steady-state thermal analysis. The main difference is that most applied loads in a transient analysis are functions of time. To specify time-dependent loads, you first divide the load-versus-time curve into load steps. Each "corner" on the load-time curve can be one load step, as shown in the following sketches.

3.5.3 Magnetic Analysis

Magnetic potentials, magnetic flux, magnetic current segments, source current density, infinite surface.

3.5.4 Electric Analysis

Electric potentials (voltage), electric current, electric charges, charge densities, and infinite surface.

3.5.5 Fluid Analysis

Velocities& pressures.

3.6 TYPES OF LOADS

The word loads in ANSYS terminology includes boundary conditions and externally or internally applied forcing functions, as illustrated in Figure 3.2. Examples of loads in different disciplines are:

Loads include boundary conditions as well as other types of loading.

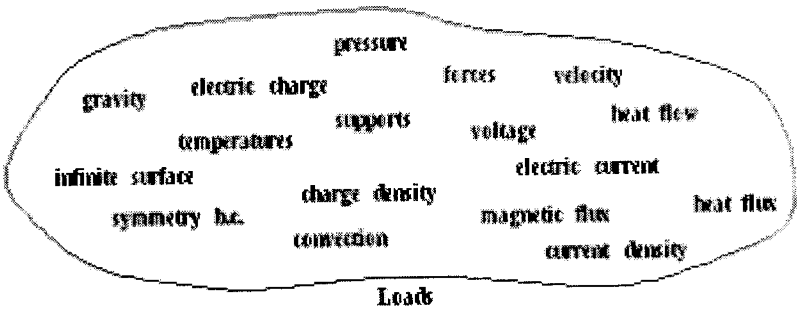


FIG 3.2 TYPES OF LOADS

Loads are divided into following categories:

a) DOF Constraints

A DOF constraint fixes a degree of freedom (DOF) to a known value. Examples of constraints are specified displacements and symmetry boundary conditions in a structural analysis, prescribed temperatures in a thermal analysis, and flux-parallel boundary conditions.

b) Forces (concentrated loads)

A force is a concentrated load applied at a node in the model. Examples are forces and moments in a structural analysis, heat flow rates in a thermal analysis, and current segments in a magnetic field analysis.

c) Surface Loads

A surface load is a distributed load applied over a surface. Examples are pressures in a structural analysis and convections and heat fluxes in a thermal analysis.

d) Body Loads

A body load is a volumetric or field load. Examples are temperatures and fluences in a structural analysis, heat generation rates in a thermal analysis, and current densities in a magnetic field analysis.

e) Inertia Loads

Inertia Loads are those attributable to the inertia (mass matrix) of a body, such as gravitational acceleration, angular velocity, and angular acceleration. You use them mainly in a structural analysis.

f) Coupled-field Loads

Coupled field loads are simply a special case of one of the above loads, where results from one analysis are used as loads in another analysis. For example, you can apply magnetic forces calculated in a magnetic field analysis as force loads in a structural analysis.

3.7 IGES TRANSFER

The car wheel modelled in CATIA is imported to ANSYS using the IGES import option. IGES is a neutral file format, which provides a way to transfer files between different CAD systems and to other applications that utilize CAD geometry. The IGES Import Options are

a) Defeature or No Defeature Model

This option is used if some small features hinder the transfer. ANSYS sometimes faces hardships while handling these small features and graphical artifacts. In order to overcome such complicated situations ANSYS eliminates such small features. The transfer that allows eliminating small features is called Defeature Model. NO Defeaturing is performed without any elimination of features.

b) Merge Coincident Key Points

Many solid modelling operations create coincident key points, lines, and areas. This option is used to merge such coincident key points. For example, suppose that you have two separate but coincident key points. If you use the command merge coincident key points to merge the key points, the higher numbered key point will be deleted and will be replaced with the lower numbered coincident key point. A single key point will thus replace two coincident key points.

c) Delete Small Areas

In most cases, deletion of small areas can speed up the processing time and import models successfully.

The following options were chosen for during the IGES Transfer of the wheel from CATIA.

File – Import IGES – Import option (No-Defeaturing, Merge coincident key points, create solid if applicable and delete small areas).

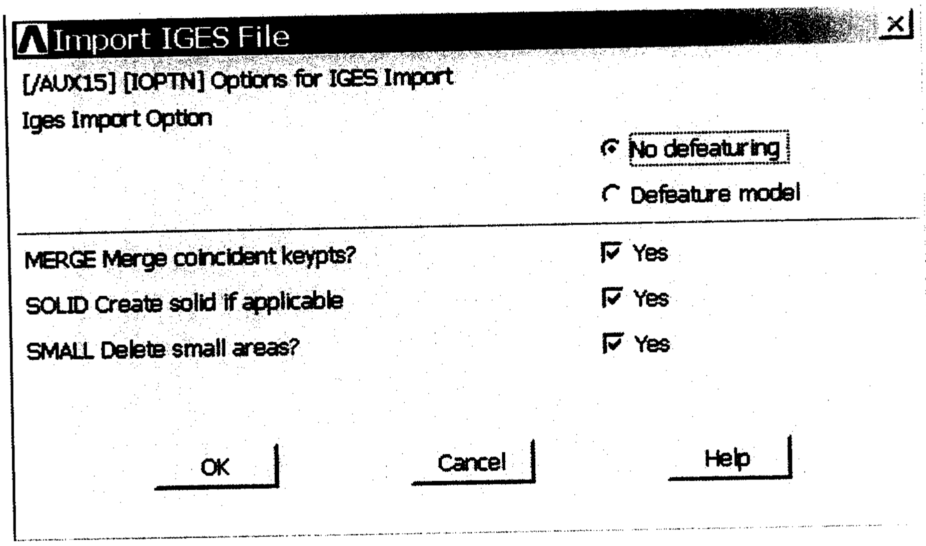


FIG 3.3 IGES IMPORT OPTION

3.8 PREFERENCES SELECTION

The structural analysis has two methods namely the H-method and the P-method.

a) H-Method

The h-method, a powerful but demanding analysis tool is well suited for analysts. They are represented by a large number of well-established software tools like, for example, ANSYS, NASTRAN or ABAQUS h-method of FEA uses elements with field variable (e.g. displacement) described by first or second order polynomials and the polynomials do not change during solution.

Since h-method uses lower order polynomials, the shapes are limited to simple geometric primitives and typically require meshes with large number of small elements. The h-method uses elements of simple shapes like tetrahedral, wedge, and hexahedral as shown in Fig3.4 and Fig 3.5. Because it is not always possible to mesh geometry with nicely shaped elements and keep their number reasonably low, models often end up with degenerated elements, a huge number of elements, or both.

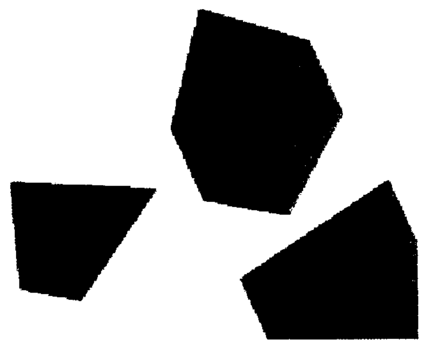


FIG 3.4 SIMPLE ELEMENT SHAPES IN H-METHOD

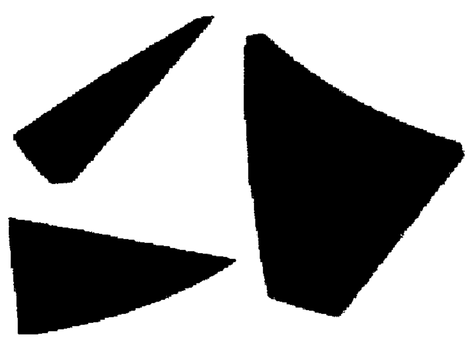


FIG 3.5 COMPLEX ELEMENT SHAPES IN P-METHOD

5.7 ROPE ANALYSIS

In the door lifting mechanism system the rope was tending to fail, the reason for failure was understandable after a detailed study. For a clear explanation let us consider figure shown below.In the figure, we see there is a rope which goes around two pulleys. The sector which is encompassing the pulley 1 will have two sides. Side A as it is on the outer side is subjected to tension and side B as it is inside is subjected to compression. When this sector is moved to the next pulley during the stroke, we observed that the side A falls inside the pulley which means side A is subjected to compression. Similarly side B is outside the pulley hence subjected to tension.

From these observations, it is clear that both the sides are subjected to reverse bending. But the rope was designed for a repeated bending and not for reversed bending. To avoid this problem the bottom pulley is shifted by 290mm.Hence decreasing the angle of bend from 90 deg to 60 deg on either pulley will increase the life of the rope.

CALCULATION

TO FIND THE SHIFTING LENGTH OF THE PULLEY

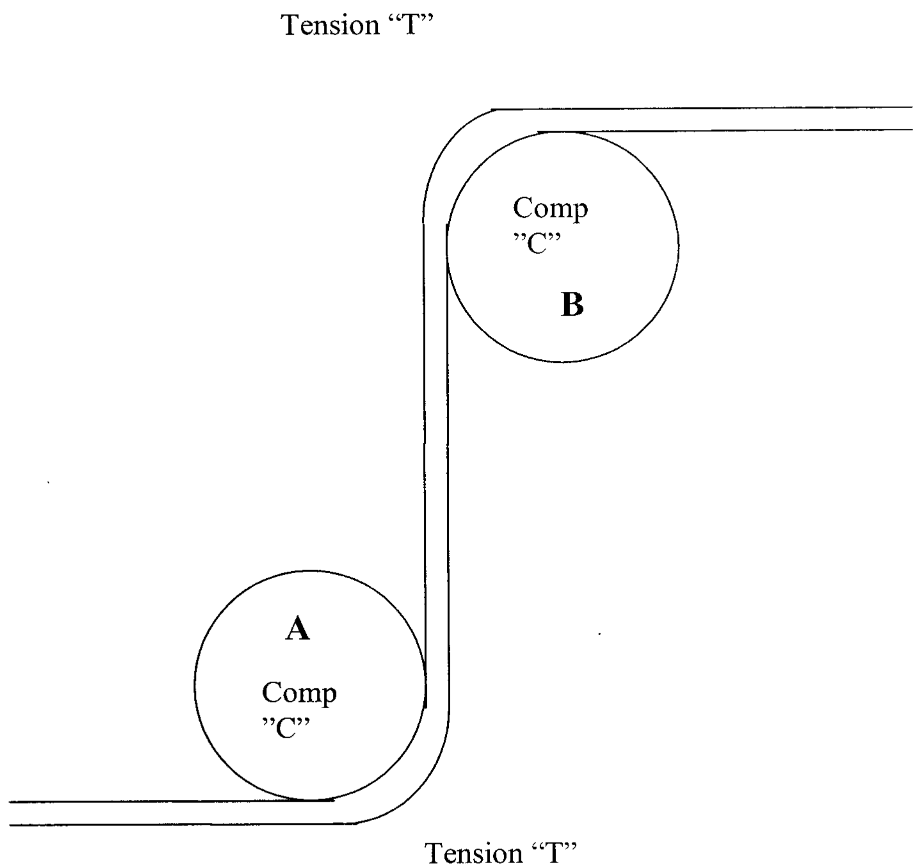
Reducing the angle of bend from 90 to 60 degrees,

(500/Distance “d”) = tan 60

Therefore

Distance (d) = 500/tan 60

Distance (d) = **290mm**

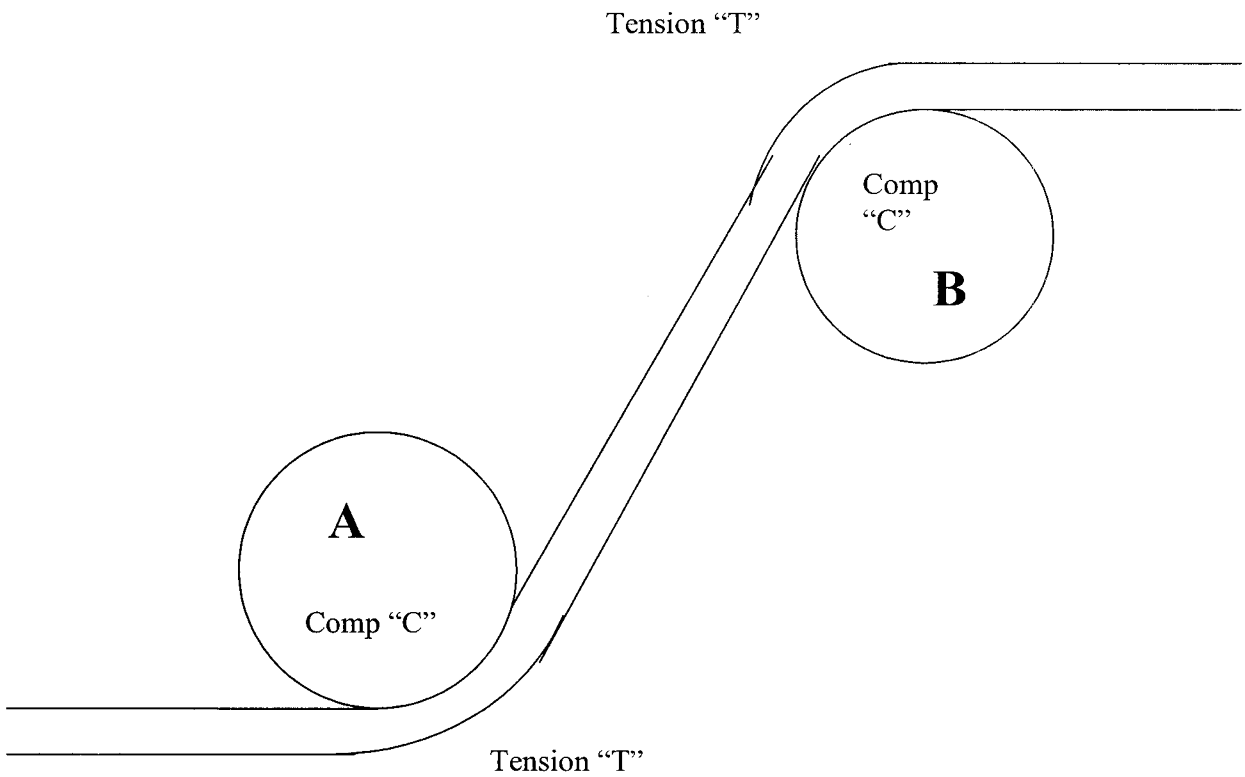


T - Tension

C - Compression

A - Pulley A

B - Pulley B



T - Tension
C - Compression
A - Pulley A
B - Pulley B

Chapter 6

Hook Analysis

6.1 HOOK – INDRODUCTION

On general-purpose cranes which are carry loads of various shapes the load is handled by means of chain or rope slings attached to hooks. Standard (single) and ramshorn hooks are the most popular designs used for this purpose.

Sometimes use is made of solid and build-up triangular hooks.standard and ramshorn hooks may be flat-die or closed-die forged or else made of a serious of shaped plates. One-piece forged hooks are used for lifting loads weighing up to 100 tons(single hooks-up to 50 tons ,ramshorn hooks beginning from 25 tons and upwards), while triangular and laminated hooks can be employed to carry over 100 tons.

All types of hooks are made of steel 20.After the forging and machining operations hooks are carefully annealed and cleaned from scale. The inner diameter of hooks should be sufficient to accommodate two strands of chain or rope which carry the load.

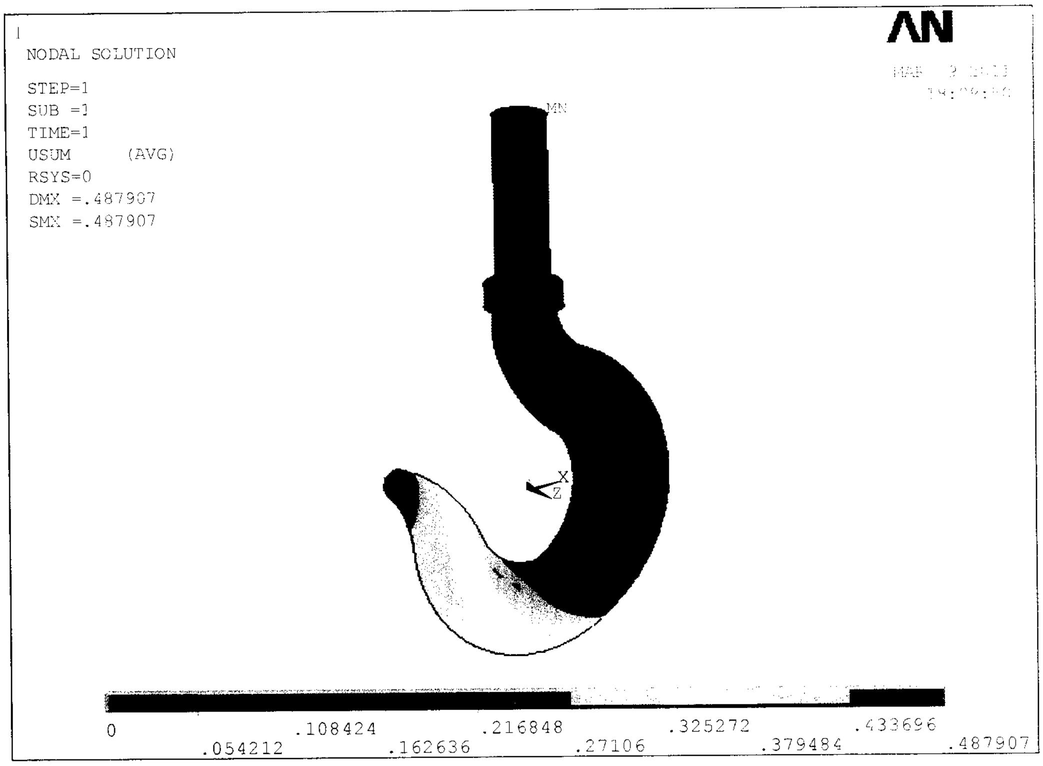
More often than not hooks have a trapezoidal section, made wider on the inside. A trapezoidal section makes for a better utilization of the material and less complicated design. On top, the hook ends in a round shank operating only in tension. The upper part of die-forged hooks is threaded for suspension from crosspieces of load carrying devices.

6.2 HOOK ANALYSIS

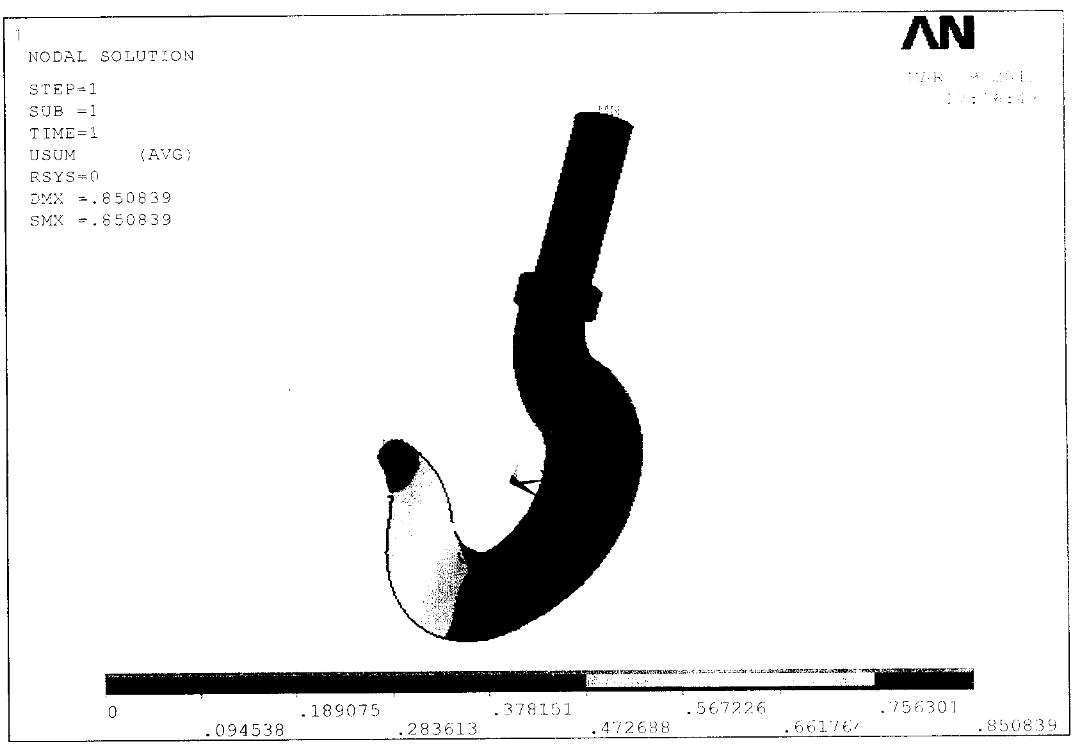
The problem of the hook which will be overcome by realigning the hook direction & changing the hook itself by going for a standard hook.

The previous design of the hook had a functionality failure. Hence to avoid this, a new hook is being introduced whose holding direction will be in line with rope direction. This will avoid slipping of the hook.

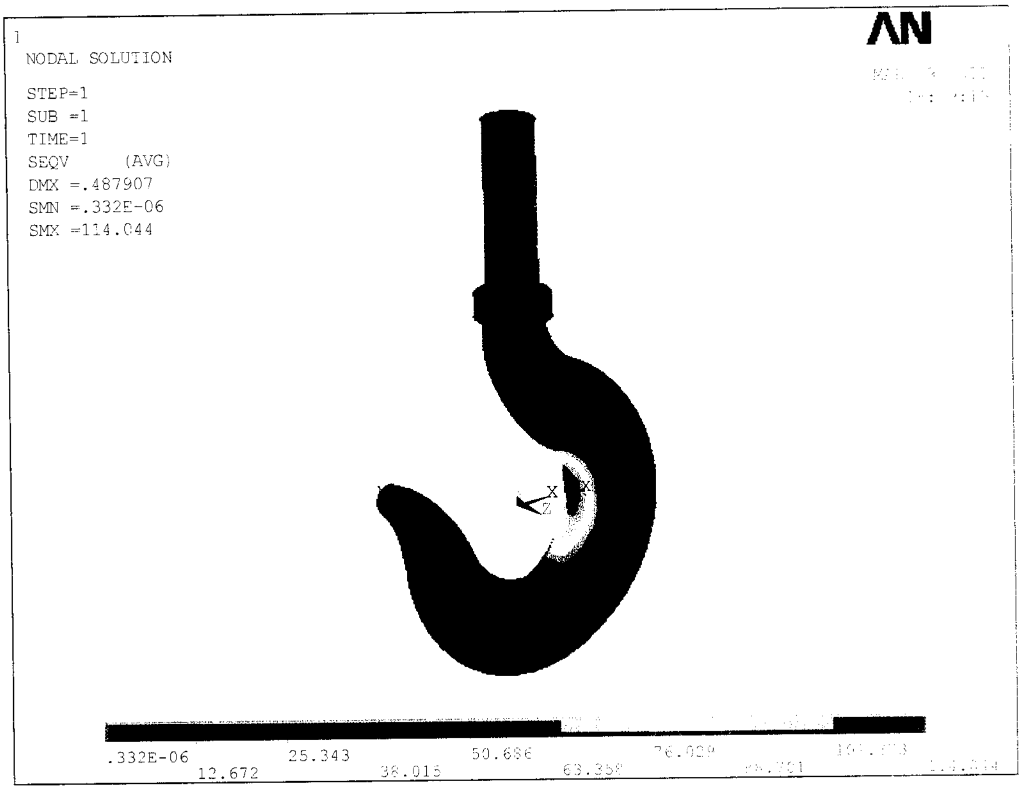
Hook Analysis – Displacement 1



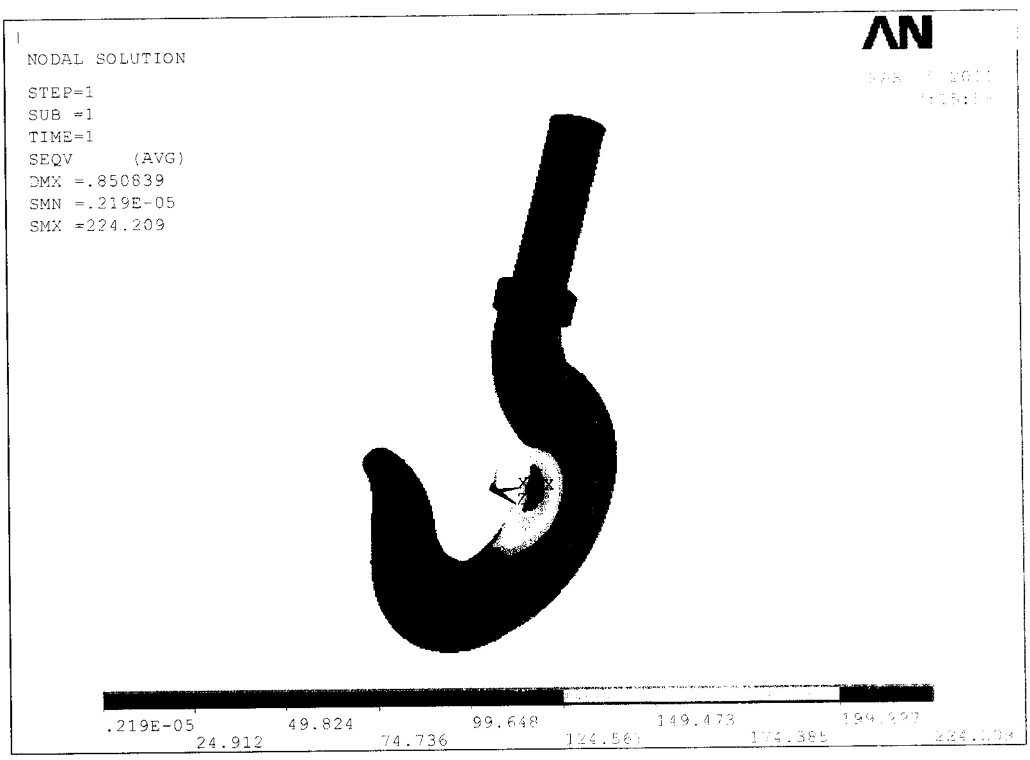
Hook Analysis – Displacement 2



Hook Analysis – Stress 1



Hook Analysis – Stress 2



6.3 RESULT

Based on the above hook analysis, a trapezoidal section of the hook was assumed with the sizes of 50 mm and 30 mm on both sides and a lateral distance of 50 mm was assumed for the initial section. This section proved to produce 224.2 N/sq.mm stress which is very closer to the yield point of structural steel. To overcome this, the depth of the section was increased to 60 mm which yielded a maximum stress of 114 N/sq.mm. This means the factor of safety increases to almost 2 and the maximum deformation yielded 0.5 mm.

Chapter 7

Conclusion

The main concern in this project is on three vital points. First is the frame skewing, second is the failure of the rope and the third is the hook design. In the present frame design it is noticed that though the frame is a safe design as far as the stress induced are concerned but fails in buckling and the frame gets skewed. The total load of door is 80,000N (8Tons), hence the total load on the rope is 8T, this 8T load comes on the piston through the three pulleys. This induces a multiple load on the frame. The critical buckling load is only 82,000N. Hence to avoid this we have introduced cross diagonals as shown in the figure. This has helped in increasing the critical bucking load to 125,000N hence this load is far above the applied load of 80,000N. This will help the frame from buckling and the skewing can be completely avoided.

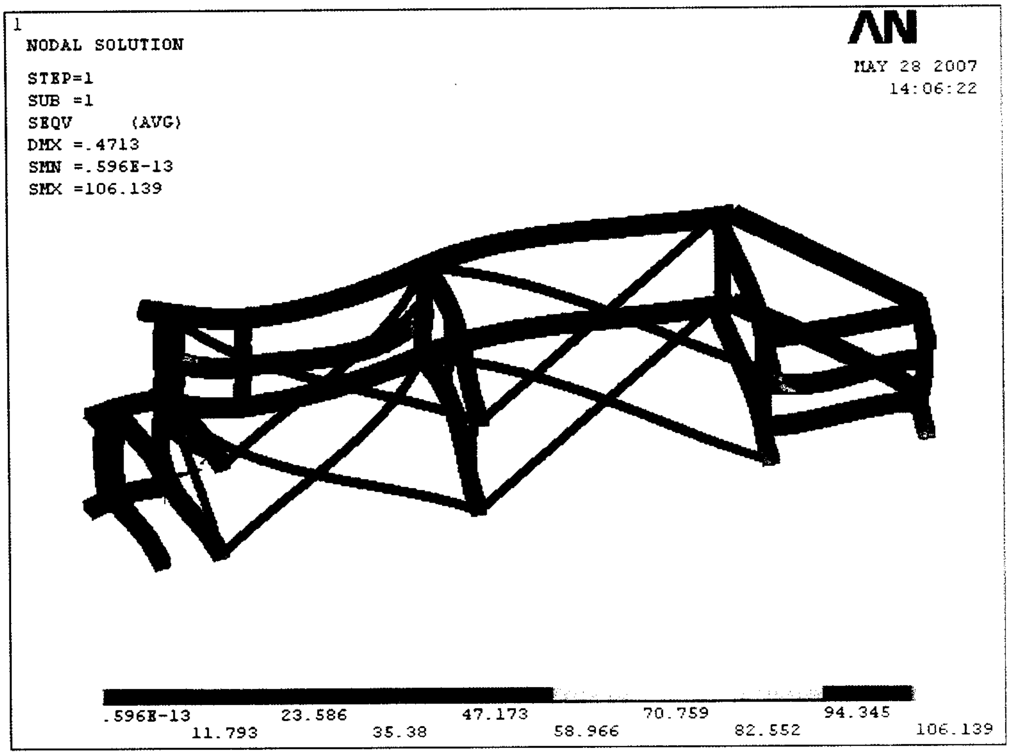
The main reason for the failure is the rope portion around the pulley. After stroke comes to pulley, the compression & tension fiber direction changes. Due to this, the design needs to be changed for ropes will be reverse cycled rather than repeated cycles. But the rope cannot be changed as it is a costly process. Instead, pulley realignment / repositioning will be considered as an alternate solution.

From these observations, it is clear that both the sides are subjected to reverse bending. But the rope was designed for repeated bending and not for reversed bending. To avoid this problem the bottom pulley is shifted by 290mm.Hence decreasing the angle of bend from 90 deg to 60 deg on either pulley will increase the life of the rope.

The previous design of the Hook had a functionality failure. Hence to avoid this, a new hook is being introduced whose holding direction will be in line with rope direction. This will avoid slipping of the hook. Based on the above, a trapezoidal section of the hook was assumed with the sizes of 50 mm and 30 mm on both sides and a lateral distance of 50 mm was assumed for the initial section.

This section proved to produce 224.2 N/sq.mm stress which is very closer to the yield point of structural steel. To overcome this, the depth of the section was increased to 60 mm which yielded a maximum stress of 114 N/sq.mm. This means the factor of safety increases to almost 2 and the maximum deformation yielded 0.5 mm.

Stress – Buckling



Chapter 5

Rope Analysis

5.1 INTRODUCTION - WIRE ROPE

It consists of several strands laid (or 'twisted') together like a helix. Each strand is likewise made of metal wires laid together like a helix. Initially wrought iron wires were used, but today steel is the main material used for wire ropes.

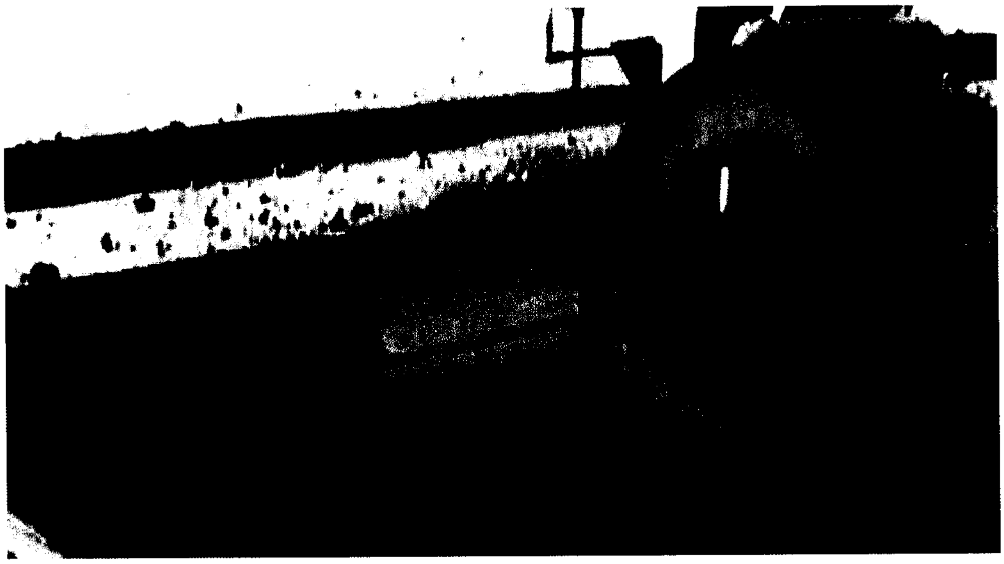


FIG 5.1 RHOL WIRE ROPE TERMINATED IN A LOOP WITH A THIMBLE AND FERRULE.

The end of a wire rope tends to fray readily, and cannot be easily connected to plant and equipment. A number of different mechanisms exist to secure the ends of wire ropes to make them more useful. The most common and useful type of end fitting for a wire rope is when the end is turned back to form a loop. The loose end is then fixed by any number of methods back to the wire rope.

5.2 THIMBLES

When the wire rope is terminated with a loop, there is a risk that the wire rope can bend too tightly, especially when the loop is connected to a device that spreads the load over a relatively small area. A thimble can be installed inside the loop to preserve the natural shape of the loop, and protect the cable from pinching and abrasion on the inside of the loop. The use of thimbles in loops is industry best practice. The thimble prevents the load from coming into direct contact with the wires.

5.3 WIRE ROPE CLAMPS

A wire rope clamp, also called a clip, is used to fix the loose end of the loop back to the wire rope. It usually consists of a u-shaped bolt, a forged saddle and two nuts. The two layers of wire rope are placed in the u-bolt. The saddle is then fitted over the ropes on to the bolt (the saddle includes two holes to fit to the saddle). The nuts secure the arrangement in place. Three or more clamps are usually used to terminate a wire rope.

5.4 SWAGED TERMINATIONS

Swaging is a method of wire rope termination that refers to the installation technique. A metal sleeve is fitted over the two wire rope ends, and a mechanical or hydraulic tool compresses and deforms the fitting, creating a permanent joint. A ferrule is a type of swaged joint.

5.5 SOCKETS

A socket termination is useful when the fitting needs to be replaced frequently. For example, if the end of a wire rope is in a high-wear region, the rope may be periodically trimmed, requiring the termination hardware to be removed and reapplied.

An example of this is on the ends of the drag ropes on a dragline. The end loop of the wire rope enters a tapered opening in the socket, wrapped around a separate component called the wedge. The arrangement is knocked in place, and load gradually eased onto the rope. As the load increases on the wire rope, the wedge becomes more secure, gripping the rope tighter.

5.6 EYE SPLICE

An eye splice may be used to terminate the loose end of a wire rope when forming a loop. The strands of the end of a wire rope are unwound a certain distance, and plaited back into the wire rope, forming the loop, or an eye, called an eye splice.

Displacement fields in h-elements are described by simple polynomials (first or second order), so it is easy to miss important displacement and stress gradients by placing too few elements in areas of interest. Low-order displacement fields call for many tiny elements to represent the expected displacement and stress patterns. Even though meshing geometry is most often done automatically, the user determines when the mesh is good enough to deliver needed results. Controlling error requires a tedious process of mesh refinement that is rarely done in practice. The h-method can be used for any type of analysis. The h-method usually requires a finer mesh.

Disadvantages

- Using the FEA h-method concurrently with a design process creates several difficulties. For example, users must prepare CAD geometry for analysis, and judge the correctness of mesh, accuracy, and quality of results.
- Geometry must be extensively defeatured, idealized and cleaned-up before it can be meshed.
- Even though meshing is most often done automatically, user's judgment is still necessary to decide if the mesh is acceptable.

b) P-Method

When the element polynomial order is varied while solving the FEM model in order to seek suitable accuracy of the computed FEM results it is termed the p-Method. The p-method obtains results such as displacements, stresses, or strains to a user-specified degree of accuracy.

To calculate these results, the p-method manipulates the polynomial level (p-level) of the finite element shape functions, which are used to approximate the real solution. This feature works by taking a finite element mesh, solving it at a given p-level, increasing the p-level selectively, and then solving the mesh again. The results are compared for convergence against a set of convergence criteria after each step of iteration. You can specify the convergence criteria to include displacement, rotation, stress or strain at a point (or points) in the model, and global strain energy.

The higher the p-level, the better the finite element approximation to the real solution. In order to capitalize on the p-method functionality, you don't have to work only within the confines of p-generated meshes. The p-method is most efficient when meshes are generated considering that p-elements will be used, but this is not a requirement. The p-method can improve the results for any mesh automatically.

In addition, the p-method adaptive refinement procedure offers error estimates that are more precise than those of the h-method, and can be calculated locally as well as globally (for example, stress at a point rather than strain energy). For example, if you need to obtain highly accurate solutions at a point, such as for fracture or fatigue assessments, the p-method offers an excellent means of obtaining these results to the required accuracy.

P-Element is generally used when Geometry is suitable for beam or shell element representation Geometry is suitable for solid representation, meshes with little preparation and meshing produces reasonably low number of elements. Structural and P-Method are selected as preferences.

3.9 PRE-PROCESSOR

a) Selection of Element Type

The elements quadrilateral 42, 182, 183, 82 are plane elements, which cannot be used for volume mesh. Hence these elements are ignored for this application. The quarter model of the wheel is transferred into ANSYS as a solid. Hence solid element is chosen as the element type. The types of solid elements available in H-method are solid 45, solid 185, 187, 92 and solid 46.

Solid 45: SOLID45 is used for the three-dimensional modelling of solid structures. Eight nodes having three degrees of freedom at each node define the element: translations in the nodal x, y, and z directions. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. A reduced integration option with hourglass control is available.

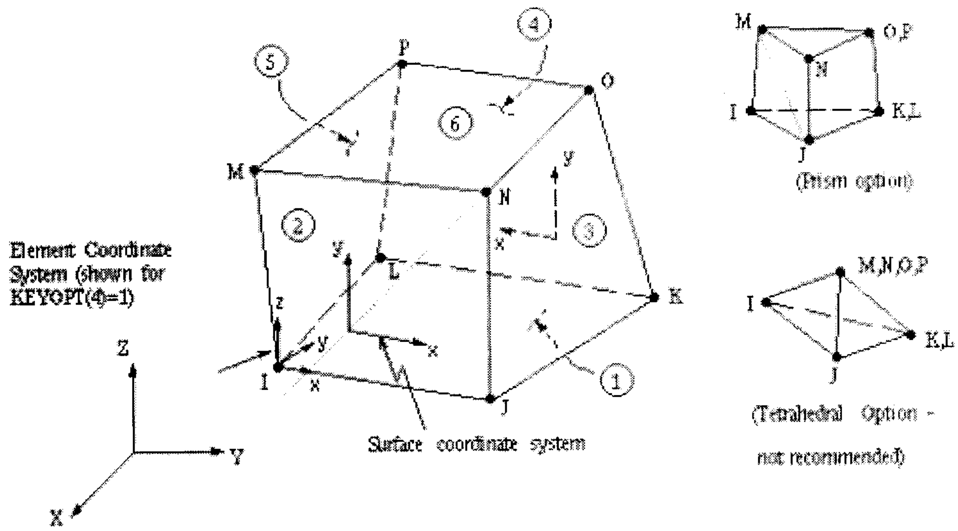


FIG 3.6 SOLID 45 ELEMENT

Solid 185 is a similar element like solid 45, except that it lacks swelling and creep behaviour. The 10 node elements were not suitable for this application (number of nodes exceeded the level while meshing).Since solid 45 have extra features than solid 185 and because it is an 8node element it is suitable for this application for a better mesh quality. Hence solid 45 are chosen as the element type.

Assumptions and Restrictions of Solid 45

Zero volume elements are not allowed. Elements may be numbered either as shown in Figure 3.6 or may have the planes IJKL and MNOP interchanged. Also, the element may not be twisted such that the element has two separate volumes. This occurs most frequently when the elements are not numbered properly. All elements must have eight nodes. Defining duplicate K & L and duplicate O & P node numbers may form a prism-shaped element. A tetrahedron shape is also available. The extra shapes are automatically deleted for tetrahedron elements.

b) Material Properties

Structural -> Linear -> Elastic -> Isotropic material is selected.

Material properties such as young's modulus and poison ratio are applied to the wheel. Steel is selected as the material.

c) Meshing the Solid Model

The procedure for generating a mesh of nodes and elements consists of three main steps:

a) Set the element attributes – The element is selected as SOLID 45.

b) Set the mesh controls (optional). ANSYS offers a large number of mesh controls, which you can choose from to suit your needs. The element size is selected as 8 for this application.

c) A free mesh is performed on the solid with element size as 8. The mesh could not be refined, because the node numbers are restricted to 32,000 in the ANSYS Student version.

d) Material properties like (Young's modulus $= 2.1 \times 10^5$ N/ m^2 and Thermal conductivity =0.3) are set

e) Mesh is generated.

d) Applying Loads and Constraints

Pre-processor-> Define Loads -> Apply -> Structural -> Displacement -> on Areas.

e) Constraints

The displacement at the bolthole is constrained. The areas in the bolthole are selected and the displacement of nodes in these areas are assigned zero. Thus constraining the holes from movement in all directions.

f) Loading Method

The total weight of a car is balanced with a vertical reaction force from the road through the tire. This load constantly compresses the wheel radially. While the car is running, the radial load becomes a cyclic load with the rotation of the wheel. Hence, the evaluation of wheel fatigue strength under radial load is an important performance characteristic for structural integrity.

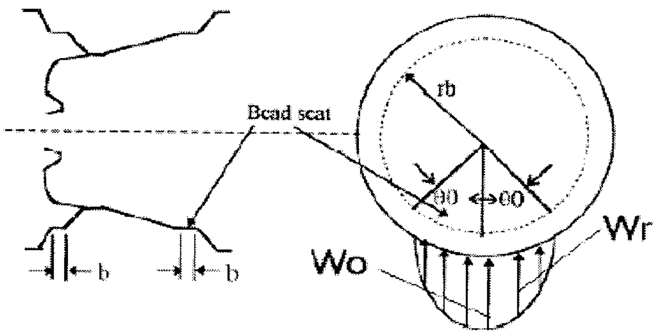


FIG 3.7 LOADING METHOD

Under a radial load, the strength of the rim usually determines the fatigue life of a wheel, so the stress evaluation is mainly focused on the rim.

In an actual wheel, since a radial load is applied to the wheel on the bead seats with the tire, the distributed pressure is loaded directly on the bead seats of the model in this analysis. The pressure is assumed to have a cosine function distribution mode within a central angle of 45 deg in a circumferential direction as shown in Figure 3.7.

Since the bead seats support the total weight, the total weight is equated to the sum of the loads distributed on the bead seats and is given by

W = (Wo + 2 (Wr1 + Wr2 + Wr3 +Wr4+....Wrn)) + (Wo + 2 (Wr1 + Wr2 + Wr3 +Wr4+...Wrn))..Equation 1

Where,

Wo = Maximum load acting at the center of the wheel bead seat area.

Wr = the distributed load acting at the bead seat given by

$$W_r = W_0 * \cos\left(\frac{\pi}{2} * \frac{\theta}{\theta_0}\right)$$

.... Equation 2

The loads Wo and Wr are calculated by using the Equations 1 and 2. These loads are applied to the selected node numbers. To select the nodes, the model is numbered using Plot controls -> Numbering -> Nodes -> Element Numbers. The node numbers on both the bead seat are tabulated. A graph is plot, to find the angular location of other nodes. Selection of Nodes is explained in Chapter 5 in detail.

The forces are applied by selecting Pre-processor -> Loads -> Define Loads – Apply -> structural -> Forces -> On Nodes.

3.10 SOLUTION

After applying loads and constraints, the model is solved using Solution -> Solve – Current LS. Solution is done successfully.

3.11 POST-PROCESSING

The results are reviewed in this stage. Post Processing –> Plot results –> contour plot -> Nodal solution is used to view the results of stress in x, y, z, xy, yz, xz and von-misses stress and strain.

Chapter 4

Door Lifting Mechanism

4.1 WORKING OF DLM

A door lifting mechanism works on a simple pulley system. The total weight of door to be lifted is 8 tons with a stroke of 3m. The setup consists of 128 ovens. There are totally 256 doors on either side of the oven. There are two door lifting mechanisms on either side. Hence each DLM has a requirement of lifting 128 doors. Not all doors are lifted together. Whichever door needs to be lifted is hooked to the main rope which is operated by the DLM. The DLM has a pulley & the main rope goes around that pulley as shown in the picture. A cylinder pushes this pulley so as to pull the rope.

4.1.1 Specifications of DLM

1.0: Operation	:	Door Lifting
2.0: Capacity	:	10 Tons
3.0: Total Operating Weight	:	8 Tons
4.0: Temperature of Environment	:	60-70 Deg.C
5.0: Operating	:	Out Door
6.0: Locking arrangement	:	Manual
7.0: Long Travel Drive	:	Hydraulic Piston-Cylinder
8.0: Stroke	:	1.5 m
9.0: Total Door Lift	:	2899 mm
10.0: Pulleys	:	Idler 4 numbers per Mechanism
11.0: Rope	:	24 x 7 Stranded Wire
10.0: Hook	:	256 nos, 12 tons capacity

4.2 REQUIRED TOOLS & TACKLES

1.0) Mobile Crane – 20 T

2.0) Derrick Posts

3.0) General Power Tools required for Machine Handling Equipment Erection

4.0) General Hand Tools like Hammer, Chisel, Files, Spanner Sets, and Torque Wrench Chain Pulley Block etc.

5.0) Welding Generator

6.0) Hand Trolley

4.3 REQUIRED MAN POWER & RESPONSIBILITIES

Site Manager: He should have the necessary experience in handling a similar Project/Site independently. He should prepare Bar Chart for the Completion of Erection and Installation. He has to coordinate and liaison the project with clients and other agencies involved in the project. He has to organize the necessary work force and the other materials required in executing the work as per schedule without any disturbances.

Site Engineer: He shall have also the necessary experience in handling a similar. Project/Site independently. He has to extract works from the workers and has to report to the Site manager.

Site Supervisor: 2 Nos.He has to extract works from the workers and has to report to the Site Engineer.

Qualified Welders	:	2 Nos.
Mill Wright Fitters	:	2 Nos.
Skillful Fitters	:	2 Nos.
Riggers	:	4 Nos.
Helpers	:	6 Nos.

4.4 SEQUENCE OF ERECTION

1.0) Bogie Assembly can be fixed with Carriage and after Alignment. Check the level using Shims

2.0) Carriages can be placed on a Rail at 10763mm apart

3.0) Place the Long Travel Chassis Frame (SS Fabricated Beam) along with the Bottom Frame and Side Frames on the Carriages and align assembly

4.0) Check the dimensions on all direction and the diagonal distances and if found OK, the same setup can be tack welded.

5.0) Remove the packing placed for the Guide Wheel locking and after alignment, The Tray along with the side frame and other frames can be placed on the Guide Wheel so that the rails at bottom of Tray rests on all the 12 Wheels uniformly. If necessary align the Assembly using necessary Shims.

6.0) Hydraulic Cylinder Mounting brackets shall be fixed with the frames on both Tray and Trolley

7.0) Fix the Skirt Boards and align them properly

8.0) Side Liners shall be placed referring the match marking and can be bolted with the Side Frames.

9.0) Side Platforms can be Tack Welded with the SS Chassis Frame and after Checking the levels, welding can be done.

10.0) Gratings can be fixed on the Platform structure as per the match marking

11.0) Hand Railing Posts and Piping can be welded with the platform frames on both sides

12.0) After Checking the Tray movement for a distance 1499mm by manual and stoppers location, the hydraulic cylinders for the Cross Travel can be fixed in Position

13.0) After checking the movement of both the gates by manual fix the Hydraulic Cylinders for both the Gates

14.0) Position the Electrical and Operator's Cabin and fix it after Alignment

15.0) Position the Hydraulic power Pack and fix it.

16.0) Fix the electrical items to the cabin inclusive of Air conditioner.

17.0) Lay the cables in the Cable trays and fix and connect the electrical Accessories

18.0) Check the functioning of Electrical Item.

19.0) Fix the Drive Motor on the drive base with RG units and Brakes. Connect the electrical supply and check for the functioning.

20.0) Fix the Drive base with Motor and RG unit with the Bogie Wheel Assembly and check the alignment.

21.0) Connect the Hydraulic Circuits and check the functioning for Individual system.

22.0) Fix the Rail lock frame and after alignment, fix the Hydraulic cylinders and connect with power pack. Check for the functioning.

23.0) Fix the Bottom Liners for Tray as per Match marking.

24.0) Final painting and painting touch up can be done.

4.5 CHECKING

1.0) Check all the bought out Items, Inspection Documents to ensure all are in line with the Ordered Specification.

2.0) Check all Items for External Abuse due to Transport before using the same.

3.0) Check for the alignment on all directions and the elevation

4.0) Check the tightness of all the fasteners and retight wherever necessary.

5.0) Check the Long Travel movement, speed and the functioning of the brakes etc.

6.0) Check the Rail lock functioning.

7.0) Check the Tray travel movement.

8.0) Check the Main gate Functioning.

9.0) Check the Small Gate functioning.

4.6 IS 808: INDIAN STANDARD DIMENSION FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL & ANGLE SECTIONS.

Scope:

This Indian Standard Covers the nominal dimensions, mass and sectional properties of hot rolled sloping flange beam and column sections; sloping and parallel flange channel sections and equal and unequal leg angle sections.

SECTION 1 – GENERAL

Definitions

Y-Y Axis - A line parallel to the axis of the web of the section (in the case of Beams and Channel) or parallel to the axis of the longer flange (in the case of unequal angles) or either flange (in he case of equal angles) and passing through the center of gravity of the profile of the section.

X-X Axis - A line passing through center of gravity of the profile of the section, and at right angles to the Y-Y.

U-U and V-V Axes - Line passing through the center of gravity of the profile of the section, representing the principal axes of angle sections.

4.7 SYMBOLS FOR DIMENSIONS

A, B	- respectively the longer and the shorter leg length of angle section
B	- flange width of beam, column or channel sections
D	- depth of beam, column or channel sections
R1	- radius at fillet or root
R2	- radius at toe
T	- thickness of flange of beam, column or channel section
t	- thickness of web of beam, column or channel section.

4.8 SYMBOLS FOR SECTIONAL PROPERTIES

a = Sectional area,

C (with subscripts x, y, u or v) = Distance of center of gravity,

Cx = A - ex

Cy = B - ey

ex =Distance of extreme fibre from X-X axis,

ey = Distance of extreme fibre from Y-Y axis,

Ix = moment of inertia about X-X axis

Iy = moment of inertia about Y-Y axis

Iu = moment of inertia (max.) about U-U axis

Iv = moment of inertia (min.) about V-V axis

M = mass of the section per meter length,

Zx = Ix/ex = Modulus of section about the X-X axis

Zy = Iy/ey = Modulus of section about the Y-Y axis

rx = $\sqrt{Ix/a}$ = Radius of gyration about X-X axis

ry = $\sqrt{Iy/a}$ = Radius of gyration about Y-Y axis

ru = $\sqrt{Iu/a}$ = Radius of gyration about U-U axis

rv = $\sqrt{Iv/a}$ = Radius of gyration about V-V axis

α = Angle between U-U and V-V axis of angle section;
Slope of flange in the case of beam, column or channel

4.9 CLASSIFICATION

Beams

Indian Standard Medium weight Beams (ISMB)

Channels

Indian Standard Medium weight Channels (ISMC)

Angles

Indian Standard Equal Leg Angles (ISA)
Indian Standard Unequal Leg Angles (ISA)

4.10 DIMENSIONS, MASS AND TOLERANCES

Nominal dimensions and mass of beam, column, column, channel and equal and unequal angles shell conform to the values given in section 2, section 3, section 4, section 5 and section 6 respectively of the standard. Dimensional and mass tolerances of the various sections shell conform to the appropriate values stipulated in IS 1852: 1985.The hoarding which we are going to design is in the V shape as shown in the picture. The structure will be holding two hoardings on either sides of V. Each banner will be of size 9m X 9m. . Our main objective is to design the hoarding structure, the base plates and the foundation bolt.

The main forces that normally act on the hoarding are due to the wind. The contribution of weight due to the banner is normally negligible. Ansys software will be used to do the analysis. The details of procedures used in ansys and the results obtained will be discussed in the latter section. This problem can also be solved using the basic strength of currently we will be discussing about the structure and various components in the structure. The various parts of the structure will be the Columns, Beams, Bracing Members, The Base plates, Foundation Bolts. Based on the loading condition we fix upon the following sizes.

Columns: ISMC 125

Beams: ISA 65 X 65 X 6

Bracing: ISA50 X 50 x 6

ISMC is a way of specifying the channel sections.

ISMB is a way of specifying the I Beam sections.

ISA is a way of specifying the Angle sections.

ISMC 125 means the Height of the beam is 125, ISA 65 x 65 x 6 means the width and height are 65 and the thickness of the web and flange is 6mm.

The section properties are as follows:

4.11 SECTIONAL PROPERTIES

Sectional properties the Beam Column Channel and equal and unequal leg angles are given in section 2 to section 6 for information.

ISMC 125

Mass, (M)		= 13.7 kg/m
Sectional area, (a)		= 17.5cm2
Dimensions,		
	D	= 125 mm
	B	= 66mm
	t	= 6.0mm
	T	= 8.1mm
	Flange slope, α	= 96 deg
	R1	= 9.5mm
	R2	= 2.4mm
Sectional properties,		
	Cy	= 1.92mm
	Ix	= 435cm4
	Iy	= 64.4cm4
	rx	= 4.98cm
	ry	= 1.92cm
	Zx	= 69.6cm3
	Zy	= 13.8cm3

ISA 65x65x6

Mass, (m)	= 5.4 kg/m
Sectional area, (a)	= 6.84 cm2
Dimensions,	
AxB	= 60x60mm
t	= 6.0mm
R1	= 6.5 mm
R2	= should be reasonably square
Sectional properties,	
Cx	= 1.69 mm
Cy	= 1.69 mm
Ix	= 22.6 cm4
Iy	= 22.6 cm4
Iu	= 36.0 cm4
Iv	= 9.1 cm4
rx	= 1.82 cm
ry	= 1.82 cm
ru	= 2.29 cm
rv	= 1.15 cm
Zx	= 5.2 cm3
Zy	= 5.2 cm3

ISA 50x50x6

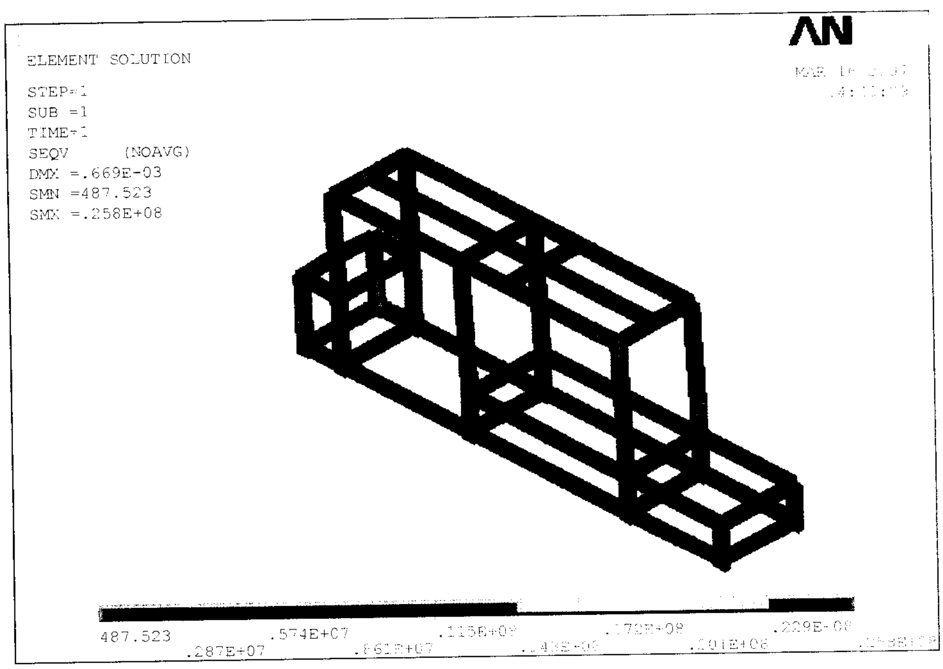
Mass,(m)	= 4.5kg/m
Sectional area, (a)	= 5.68 cm2
Dimensions,	
AxB	= 50x50 mm
t	= 6.0mm
R1	= 6.0 mm

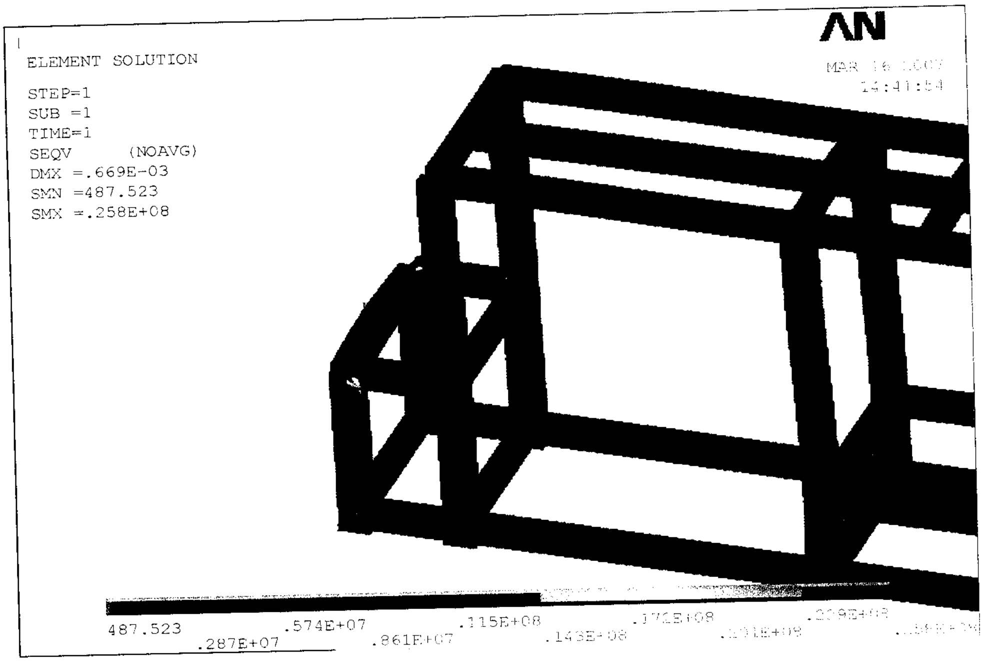
	R2	= should be reasonably square
Sectional properties,		
	Cx	= 1.45 mm
	Cy	= 1.45 mm
	Ix	= 12.9 cm4
	Iy	= 12.9 cm4
	Iu	= 20.6 cm4
	Iv	= 5.3 cm4
	rx	= 1.51 cm
	ry	= 1.51 cm
	ru	=1.90 cm
	rv	= 0.96 cm
	Zx	= 3.6 cm3
	Zy	= 3.6 cm3

4.12 FRAME STRUCTURE ANALYSIS

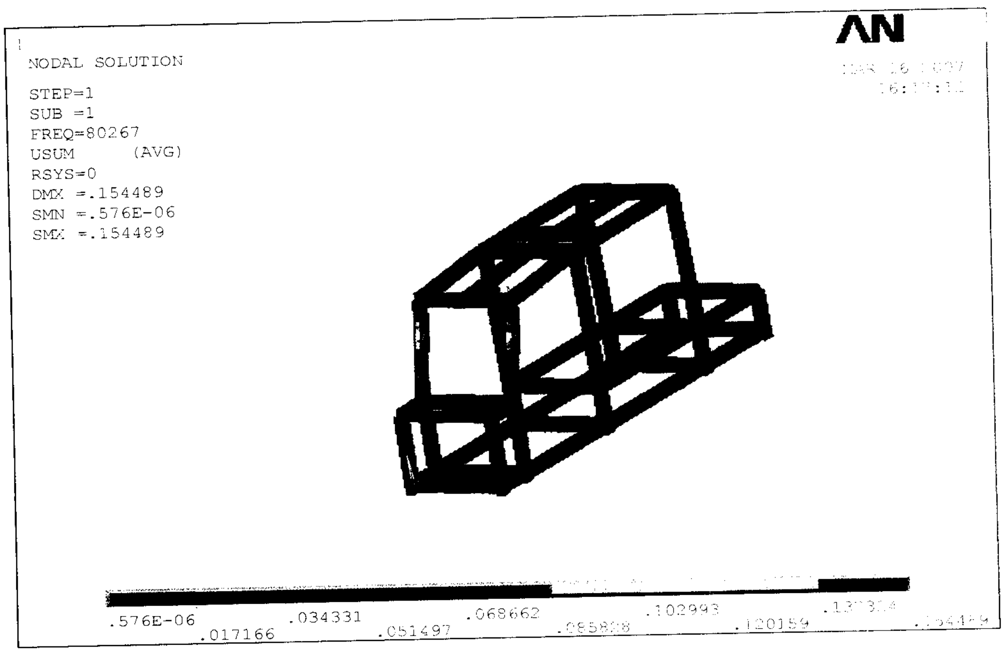
The problem we come across in the door lifting mechanism is the skewing or buckling. Though the frame structure is designed for 10 tons of total operating capacity, there occurs skewing in certain members of the frame. The skewing is due to the lifting of the massive load of 8 tons of total operating weight. Skewing of the structure is predictably due to the criteria. The critical backlog load of the structure is 8.2T which is close to the total load of 8T. The problem will be resolved by identifying the weak members in the frame structure and introduction of extra diagonal members of structure according to the behavior of the structure.

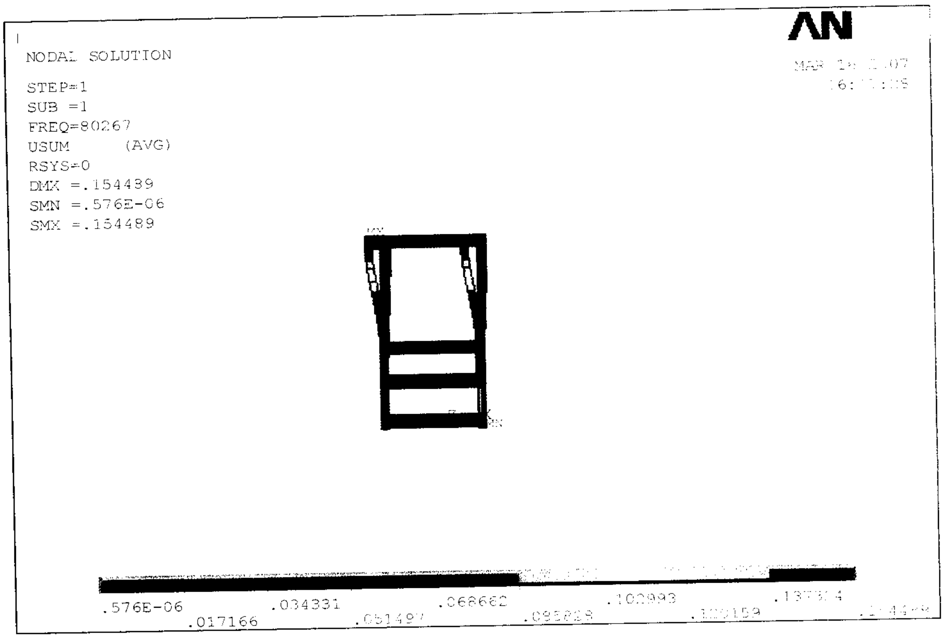
STATIC STRUCTURAL PROBLEM RESULTS



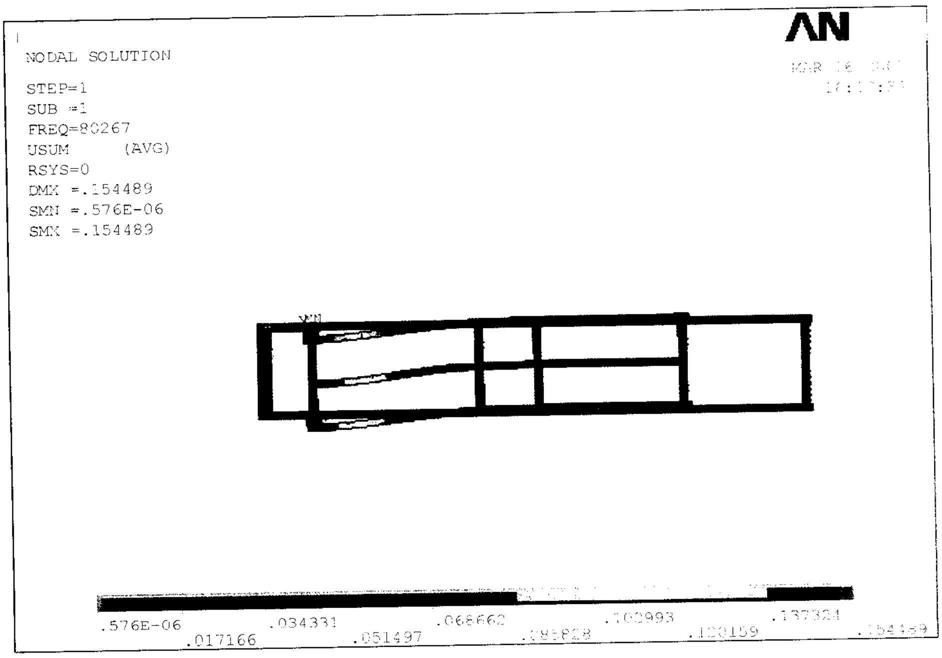


Skewing (Buckling) Problem





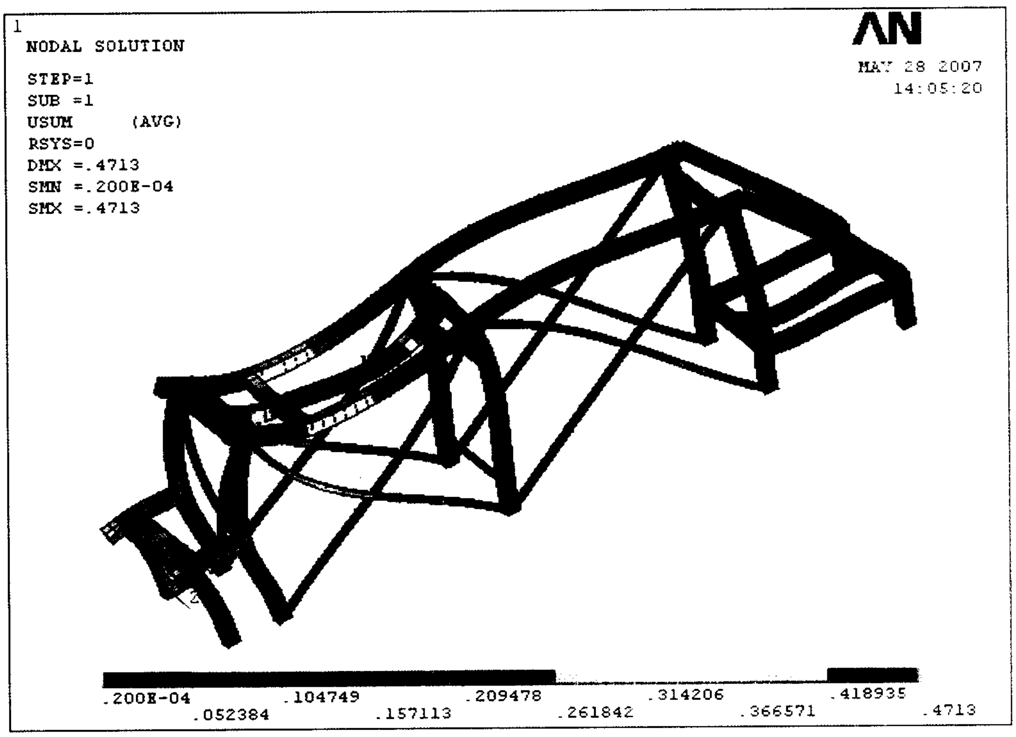
NODAL SOLUTION
STEP=1
SUB =1
FREQ=80267
USUM (AVG)
RSYS=0
DMX =.154489
SMN =.576E-06
SMX =.154489
.576E-06
.017166
.034331
.068662
.102993



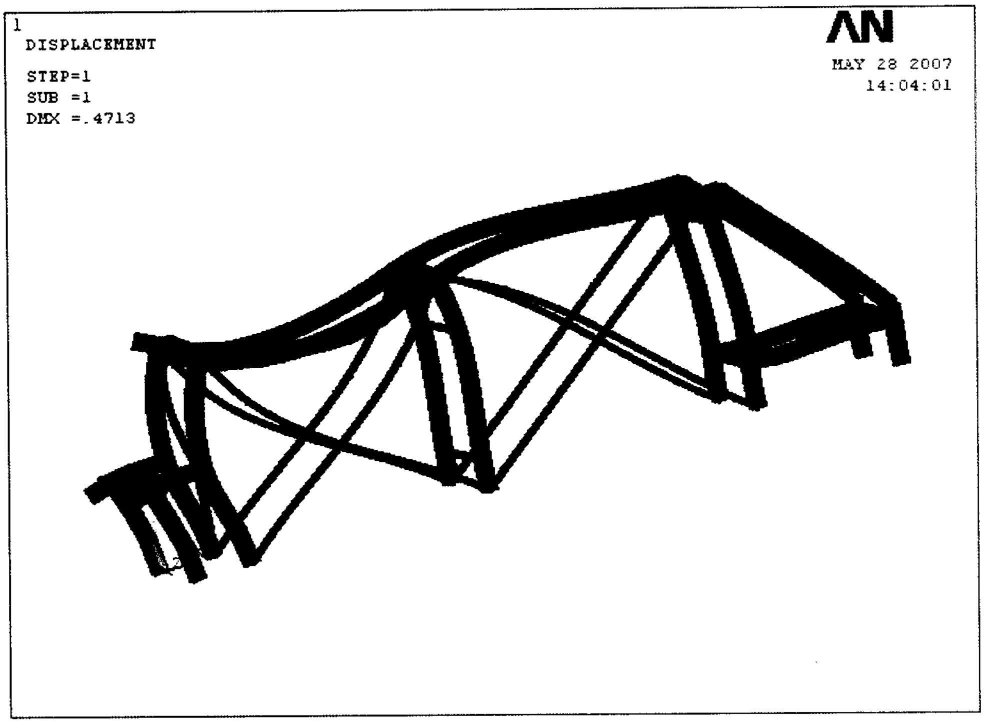
NODAL SOLUTION
STEP=1
SUB =1
FREQ=80267
USUM (AVG)
RSYS=0
DMX =.154489
SMN =.576E-06
SMX =.154489
.576E-06
.017166
.034331
.051497
.068662
.102993
.137324

From the structural analysis it is found that the stresses are within the limit. But the buckling analysis reveals that the critical load is 82,000N which is equal to the load applied. Hence the column tends to buckle due to buckling. To solve this problem we introduce a diagonal member to solve this problem.

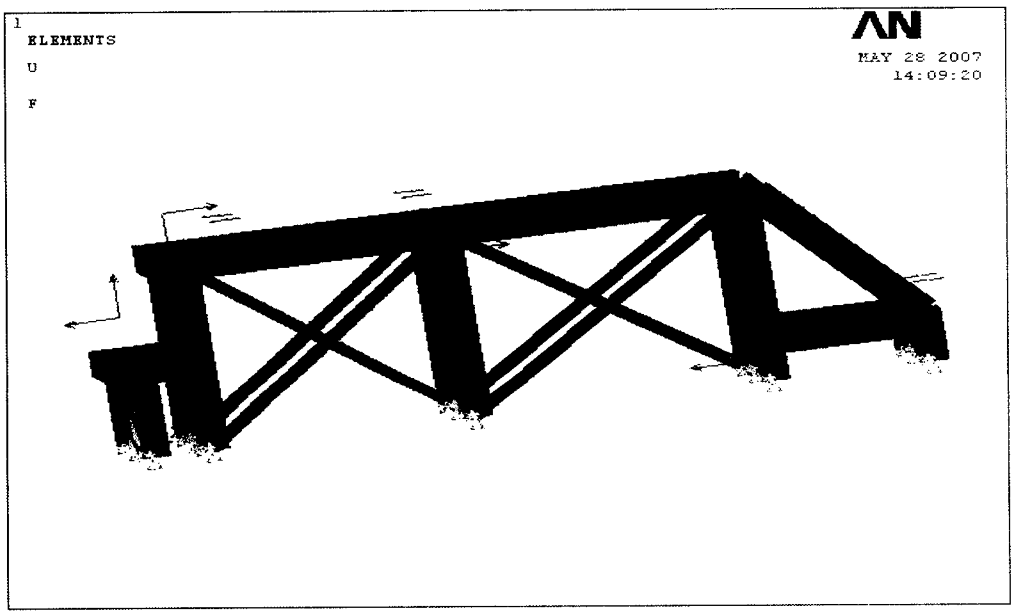
Deflection – Buckling



Deformation – Buckling



Loading – Buckling



References

1. Choo.Y.S, Ju.F.and Lee.K.H (1997) "Static Sling tensions in Heavy Lifts with Doubled Sling Arrangements" , *International Journal offshore and Polar Engineering,* Vol 7, No 4 (ISSN 1053-5381).

2. Toshihiro SAITO, Takemasa OKANO (2002) "Application of metal Pushing V – Belt Stress Simulation with CVT Pulley Stiffness", *2002 ABAQUS Users Conference.*

3. Benjamin Brooks (2006) "The Pulley Model-A Descriptive Model of Risky Decision-Making", *Safety Science Monitor,* Article 1, Vol 11.

4. Banfield,S.J.,Hearle,J.W.S.(1998)."Comparative Performance of Fibre Ropes", *Proceedings of International Conference on Offshore Mechanics and Artic Engineering,*ASME,NewYork, PaperOMAE98-358.

5. Stephen W.Attaway,(1996)"Rope System Analysis"

6. Strachan and Henshaw (1994),"Dynamic Simulation of 9 Dock RAH 45t Crane Rope Failure"

7. Mohammed Raoof,Ivana Kraincanic(1996),"Behaviour of Large Diameter Wire Ropes",*International Journal of Offshore and Polar Engineering,*Vol.6,No.3(ISSN 1053-5381).

8. Mohammed Raoof(1999),"Design of Spiral Strands Against Axial Fatigue",*International Journal of Offshore and Polar Engineering,*Vol.9,No.2(ISSN 1053-5381).

9. Klaassens .J.B,Honderd.G,ElAzzouzi.A,Cheok.C (1999),"3D Modelling Visualization for Studying Controls of the Jumbo Container Crane",*American Control Conference,* Vol 3, Issue , 1999 Page(s):1754 - 1758