STRESS ANALYSIS OF A FIXED WHEEL VERTICAL-LIFT GATE BY FINITE ELEMENT METHOD

A DISSERTATION

Submitted in partial fulfillment of the requirements for the award of the degree

of

MASTER OF TECHNOLOGY

in

WATER RESOURCES DEVELOPMENT (MECHANICAL)

By

MD. ABDUL WAHAB

DEPARTMENT OF WATER RESOURCES DEVELOPMENT & MANAGEMENT
INDIAN INSTITUTE OF TECHNOLOGY ROORKEE
ROORKEE - 247 667 (INDIA)

JUNE, 2005

CANDIDATE'S DECLARATION

I do hereby declare that the dissertation entitled "**STRESS ANALYSIS OF A FIXED WHEEL VERTICAL-LIFT GATE BY THE FINITE ELEMENT METHOD** (Using **ANSYS** Program)" is being submitted by me in partial fulfillment of requirement for the award of degree of "**MASTER OF TECHNOLOGY** in **WATER RESOURCES DEVELOPMENT (MECHANICAL)**" and submitted in the Department of Water Resources Development and Management, Indian Institute of Technology Roorkee, is an authentic record of my own work carried out during the period from July 2004 to June 2005 under the guidance of **Prof. Gopal Chauhan**, Department of Water Resources Development and Management, and **Dr. B K Mishra,** Associate Professor, Department of Mechanical and Industrial Engineering, Indian Institute of Technology Roorkee.

I have not submitted the matter embodied in this dissertation for the award of any other degree.

IIT Roorkee, India,

Dated: June, 2005

(MD. ABDUL WAHAB)

This is to certify that above statement made by the candidate is correct to the best of our knowledge.

(Dr. B K Mishra)
Associate Professor, MIED,
IIT Roorkee, UA, India.

(Gopal Chauhan)
Professor, WRDM,
IIT Roorkee, UA, India.

24/6/05

ACKNOWLEDGMENT

With great pleasure and proud privilege I express my deep sense of respect and gratitude to Prof. Gopal Chauhan, Department of Water Resources Development and Management, and Dr. B K Mishra, Associate Professor, Department of Mechanical and Industrial Engineering, Indian Institute of Technology Roorkee, for their valuable, inspiring and thorough guidance, constant assistance and encouragement in bring out this Dissertation. The valuable hours of discussions and suggestions that I have from them have undoubtedly helped me in supplementing my thoughts in right directions for attaining the desired objectives of completing this Dissertation in its present form. Working under their guidance will always remain a cherished experience in my memory and I will worship it throughout my life.

My heartful gratitude and indebtness goes to all the faculty members of WRDM who with their encouraging and kind words, constructive criticisms and suggestions or by simply doing their duty with sincerity and smile have contributed directly or indirectly in a significant way towards completion of this Dissertation.

My special and sincere thanks to batch mates whose support and encouragement have been constant sources of inspiration, guidance and strength.

I would also like to extend my thanks to all the staffs of WRDM Computer Centre, IIT Roorkee for their constant supports and cooperation in all respect for caring out FEM ANSYS Analyses and completion of this Dissertation.

Last but not the least I would like to express my greatest appreciation to my loving and helpful wife Jesmin Ara Begum (Lucky) and son Nazmus Sakib for their constant encouragement and inspiration to bring it to a success.

IIT Roorkee, India,

Dated: June, 2005

(MD. ABDUL WAHAB)

CONTENTS

CONTENTS

CONTENTS

CONTENTS

LIST OF TABLES

LIST OF TABLES IN APPENDIX - A

LIST OF FIGURES

LIST OF FIGURES IN APPENDIX - A

LIST OF GRAPHS

LIST OF GRAPHS IN APPENDIX - A

SYNOPSIS

The hydraulic gates are moving equipments installed in dams, barrages, hydropower projects, reservoir, canal etc. for regulating flow. They are most important components of any water resources projects. They need specialized expertise in regard to selection, planning and design, erection and maintenance. Any deviation in satisfying, the intended purposes can have disturbing results, and sometimes may cause danger to the entire project. Vertical lift gate and radial gate are frequently used in water resources projects. These gates are designed as per guidelines given in various handbooks of hydroelectric engineering, gates & valves and different Indian Standard Codes. These are based on generally two-dimensional design approaches. However, in actual practice, each structure behaves as a three-dimensional structure. Hence displacements and stresses are developed in all three directions. The accurate scenery of displacement (deflection) and stress in different components of the gate, such as, 'skin plate, horizontal girder, vertical end girder, vertical stiffener and their joints, wheel etc. for Vertical lift gate are relatively complex and can be examined through three dimensional analysis.

In this dissertation an attempt has been made to analyses the stresses and displacements at vital parts with useful form Finite Element Model through ANSYS software. A comparative study between conventional two-dimensional design and three-dimensional FEM results has been made to find out the effectiveness of two-dimensional design procedure. This study is limited to skin plate, horizontal girder, vertical end girder, vertical stiffener, and wheel of an existing "Fixed Wheel Vertical Lift Gate" fitted in Tailrace Channel of Dharasu Power House.

Two-dimensional and Three-dimensional Comparison Study shows that, gate can be designed with confidence, reliably and cost effectively with FEM as a three-dimensional structure because the three-dimensional results are more accurate than the two-dimensional design results, which indicates that two-dimensional design procedure is conservative.

LIST OF NOTATIONS

Sl No.	NOTATION	DETAILS
1.	UX	Displacement in X-direction
2.	UY	Displacement in Y-direction
3.	UZ	Displacement in Z-direction
4.	XROT	Rotation about X-direction
5.	YROT	Rotation about Y-direction
6.	ZROT	Rotation about Z-direction
7.	DOF	Degrees of Freedom
8.	$SX..or..\sigma_x$	Bending Stress in X-direction
9.	$SY..or..\sigma_y$	Bending Stress in Y-direction
10.	$SZ..or..\sigma_z$	Bending Stress in Z-direction
11.	$SXY..or..\tau_{xy}$	Shear Stress in X-Y plane
12.	$SYZ..or..\tau_{yz}$	Shear Stress in Y-Z plane
13.	$SXZ..or..\tau_{xz}$	Shear Stress in X-Z plane
14.	SF	Shear Force
15.	BM, or M	Bending Moment
16.	I_{xx}	Moment of Inertia about X-direction
17.	I_{yy}	Moment of Inertia about Y-direction
18.	I_{zz}	Moment of Inertia about Z-direction
19.	Z	Section Modulus
20.	mm	Millimeter
21.	cm	Centimeter
22.	kg	Kilogram
23.	kg-cm	Kilogram-Centimeter
24.	t	Tonne, Thickness
25.	E	Young's Modulus of Elasticity
26.	υ	Poison's Ratio
27.	ρ	Density
28.	P	Hydraulic Pressure
29.	f_t , f_b	Flexural or Bending Stress in Top or Bottom fibre
30.	Sc, fc	Contact Stress or Combined Stress

BASIC CONCEPT OF VERTICAL LIFT GATE

1.1 INTRODUCTION

Construction of hydraulic structures like barrage, dam, weir etc. across rivers for the purpose of flood control, irrigation, power generation, recreation etc. are very common. The gates are used in these hydraulic structures-to control and regulate the flow of water. They are also known as water regulating equipment.

Many type of gates are in successful operation. However, few of these may be suitable or economical for a specific situation. The tough job is to select and design the most appropriate type and size of gate, which will meet the hydraulic, operational, site condition and economic requirement. The conditions like water head which could be low, medium or high, the locations like spillway, barrage, intake, navigation channels etc., frequency and mode of operational requirement etc. are some of the basic considerations for the selection and successful installation of gates.

The failure of a gate may cause serious hazard and terrible situations, which may not only result in loss of life but may necessitate closing of project requiring high expenditure in rehabilitation and replacement of damaged gates. Therefore, the gates are the most important components on any water resources project. So, it needs specialization and expertise for selection, planning, designing, erection as well as maintenance.

1.2 VERTICAL LIFT GATE

The vertical lift gates are rectangular in shape and supported by guides in which gates move vertically in their own plane. The gates consist of a framework to which a skin plate is attached. The skin plate is normally placed on the upstream face in order to avoid accumulation of debris. The skin plate may also be placed on the downstream face in case of intake gates on face of dam. The main load of water pressure is transmitted through the framework to the piers. The hoisting mechanism of the gate must be strong enough to withstand the weight of the gate and overcome the friction developed due to water pressure and movement of the gate. Different types of vertical lift gate's frame structures are shown in Fig.1.1.

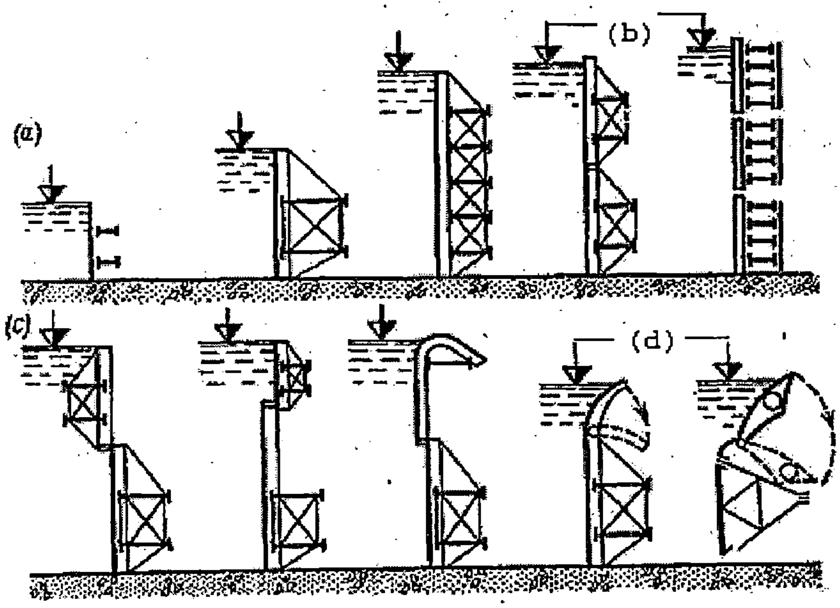


Figure: 1.1 Types of framework of vertical lift gate (Grishin, MM)
(a) unitary; (b) sectionalized; (c) double; (d) with flap.

To reduce the friction, various devices have been developed and fitted with the gates and some structural development and modification are done on the track (movement surfaces) which leads following sub-division of vertical lift gates.

i) Sliding type,
ii) Fixed wheel type,
iii) Stoney or Roller type, and
iv) Caterpillar or Coaster type.

In this dissertation study is focused to **fixed wheel type vertical lift gate**, which is commonly used now a days.

1.3 FIXED WHEEL VERTICAL LIFT GATE

The structural arrangement is similar in all vertical lift gates. A typical structural arrangement of a fixed wheel vertical lift gate is shown in Fig.1.2. The main components of this gate are:

1. Skin plate;
2. Horizontal stiffeners;
3. Vertical stiffeners;
4. Horizontal girders;

5. Cross ties (diaphragms);
6. Cross-bars;
7. Vertical end girders;
8. Wheels;
9. Guide rollers and
10. Rubber seals.

Horizontal girders are made in the form of continuous beams. For opening up to 5m I-beams are generally used but for wider openings, I-beams of composite sections or fabricated beams may be used. To reduce the width of the grooves, beams are curtailed at ends to extend of 0.40 – 0.65 of its height at the opening. Also spacing of the beam is progressively reduced from top to bottom as shown in fig.1.2 such that load on each beam is almost equal.

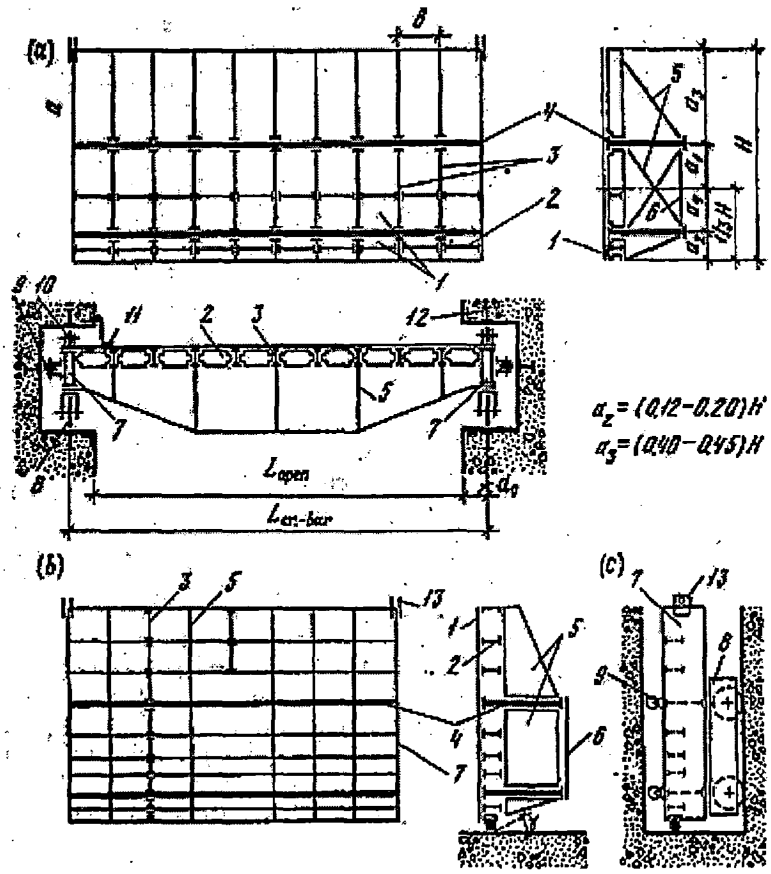


Figure: 1. 2 Fixed wheel vertical-lift gate (Grshin, MM)
(a) lateral arrangement; (b) longitudinal arrangement (cross ties are continuous diaphragms); (c) cross-section and side view.

Wheels are mounted on the vertical end girders of the gate leaf and move on the tracks, which are anchored into the piers, and thus carry the water loads, transmitted through the horizontal girders to the vertical end girders.

On small low head gates, double flanged wheels are used to guide the gate leaf while moving under loads. On the large size gates, separate guide rollers and wheels without flanges are used as shown in Fig.1.2. Different wheel arrangements are shown below in Fig.1.3.

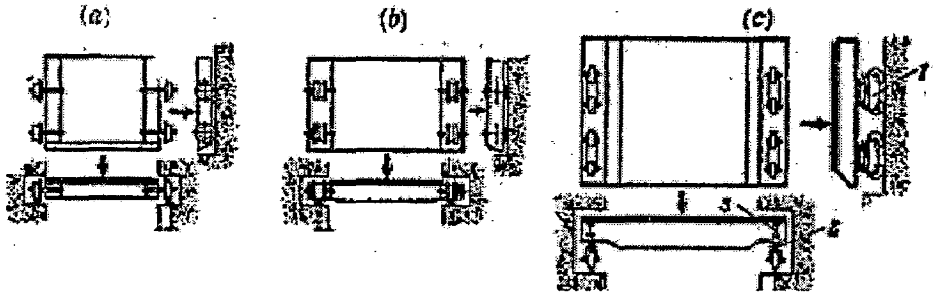


Figure: 1.3 Wheel support arrangement of VL gates (Grishin, MM)
(a) Cantilever wheels; (b) between vertical end girders; (c) wheels joined by carriages.

In the earlier designs, cast iron wheels were used, but with the technological advancement, surface hardened cast steel wheels and surface hardened tracks are being used, which may carry loads as much as 150 tonnes per wheel. By the use of self lubricated bronze bushing or anti-friction roller bearing in the wheels, the hoisting capacity is reduced effectively.

Different sealing arrangements are shown in fig.1.4.

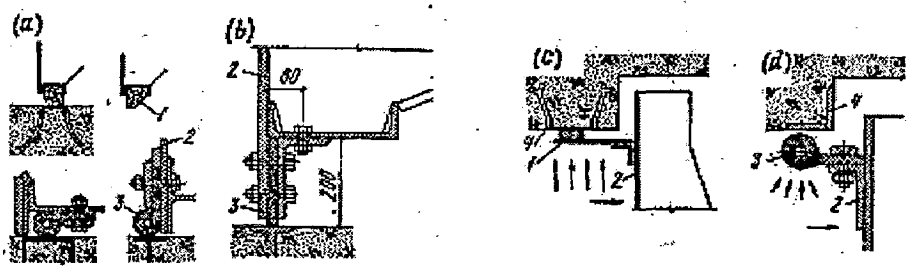


Figure: 1.4 Crest gate seal arrangement (Grishin, MM)
(a), (b) bottom seal; (c), (d) side seal;

The fixed wheel gates are frequently used in structures like medium and moderately high head barrages, spillways, sluices and outlets, tunnel intakes etc. for the advantages such as (a) Short length of piers, (b) Simplest in construction, (c) Easy of erection, (d) Provide good sealing, (e) Large span can provide good navigation and (f) Multi-tier gates reduce height of operation decks.

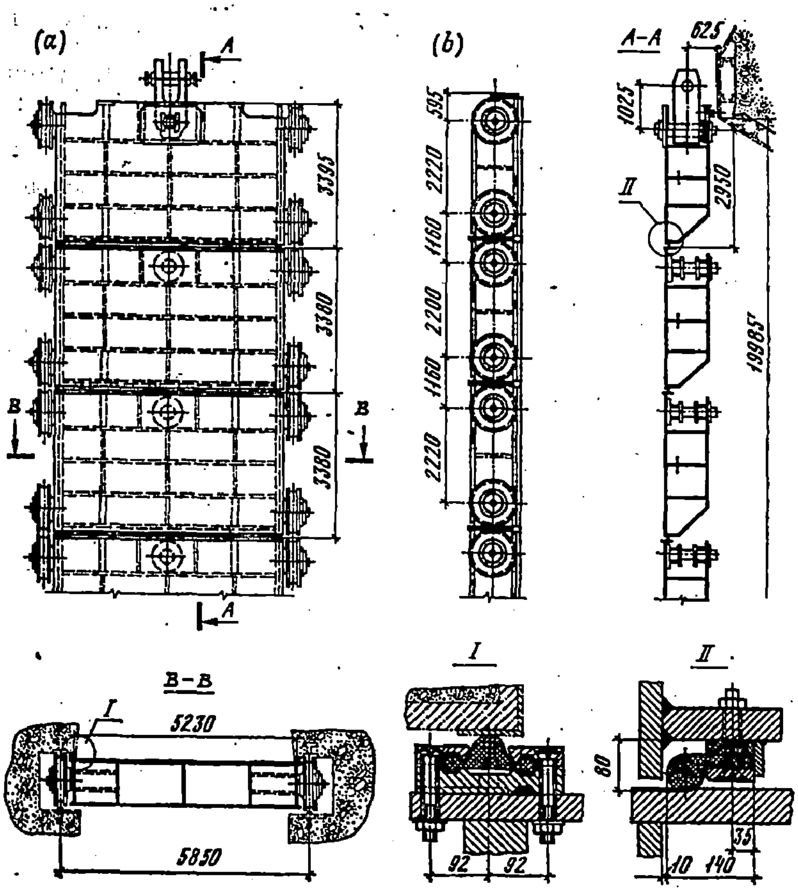


Figure: 1.5 Fixed wheel vertical lift gate with multiple sections (Grishin, MM)
(a) view from head side; (b) side view.

For minimizing the size and capacity of transporting, handling and hoisting equipment and greater ease in shipment and erection, the following developments have been made:

- The vertical lift gates can also be fitted with overflow sections, where limited overflow is required. When substantial overflow is required, a hook type gate is used.
- Larger vertical lift gates can be manufactured in sections with an articulated joint between sections. It is often possible to mount four wheels on each section to omit adjustment of wheels. For reducing high room of operation structure, the gate can also be divided into two or more leaves moving independently either in the same or separate grooves as shown in Fig.1.5. They are known as multi-tier vertical lift gates being used in USA, Europe and also in India. Small floods, floating debris or ice can pass over the upper leaf in its lowered position and rising of the heavier lower leaf is not required.
- In one leaf high head vertical lift gate, the large number of horizontal girders can be omitted by the use of box girder type vertical lift gate.

- For reducing hoisting loads, most of the vertical lift gates are counterbalanced. To prevent the counter weight from entering into water when the gate is lifted, the counter weight is reeled 2:1, so that it travels on half the distance. This results in an additional load on the superstructure of the order of 2.7 times the mass of the gate.

When the gates are fully closed and rest on sill the following forces act on it.

i. Hydrostatic pressure;
ii. Silt pressure;
iii. Self-weight etc.

When the gates are partially opened the following forces act on it

i. Hydrostatic pressure;
ii. Rope tension;
iii. Self-weight;
iv. Hydraulic down pull etc.

The loads are transferred from skin plate to vertical stiffener to horizontal girder to vertical end girders and finally to tracks and piers through wheels. Due to the loads deflections take place and therefore, stresses are developed in various parts of the gate. Each side of the gate carries half of the total load acts on it. The joints of skin plate with vertical stiffeners and horizontal girders are expected high stress zone. Similarly the joints between horizontal girders and vertical stiffeners as well as with vertical end girders are also expected to be high stress zone. Generally the gate is designed as per guidelines given in various handbooks of hydro-electric engineering, gates and valves and IS: 4622: 1992, Indian Standard Fixed Wheel Vertical-lift Gates Structural Design Recommendations (2nd. revision) etc. which are based on two-dimensional conventional approach. However these guidelines don't spell in clear terms about the stress conditions at the joints mentioned above. This can be visualized with the help of three-dimensional analysis, which may confirm that the design is safe and economical.

1.4 SCOPE OF STUDY

In this dissertation an attempt has been made to study Stresses and Displacements (deflection) in a **Fixed Wheel Vertical-lift Gate**'s different components like skin plate, horizontal girders, end girders, vertical stiffeners and wheel in two-dimensional as well as three-dimensional by the **Finite Element Method** when the gate rests on the sill. This analysis is carried out by the **ANSYS-5.4** software, which is based on FEM and to compare the results of three-dimensional analysis and two-dimensional analysis as well as conventionally designed for establishing the effectiveness and utility of these analyses.

The fixed wheel vertical lift gate adopted for this study is installed at the tailrace channel of Dharasu Power House, India. One gate is splitted into two equal parts as shown in Fig.1.6. Also the system of units adopted is kg for force instead of Newton because of prevailing procedures in Bangladesh.

The design of this gate is based on following IS codes:

i. IS – 4622 – 1992: Indian Standard Fixed Wheel Vertical-lift Gates Structural Design Recommendations.
ii. IS – 226 – 1975: Specification of structural steel standard quality.
iii. IS – 2062 – 1982: Specification for structural steel for fusion welding quality.
iv. IS – 823 – 1964: Code of practice for manual metal arc welding of mild steel.
v. IS-318- 1981: Specification for leaded tin bronze ingots and castings.
vi. IS-1030-1982: Carbon steel casting for general engineering purposes.
vii. IS-1570 (part V): 1985 Schedule for wrought steels for general engineering purposes.

1.5 ARRANGEMENT OF DISSERTATION

This dissertation consists of six chapters.

Chapter – 1: covers concept of vertical lift gate.

Chapter – 2: contains basic concept of finite element method.

Chapter – 3: contains design data, hydraulic loads, design considerations, design criteria for skin plate, vertical stiffeners, horizontal girders, wheel and summary of conventional results of the fixed wheel vertical lift gate. Details of design calculation for static condition is appended in APPENDIX – A.

Chapter – 4: contains basic FEM Model of vertical lift gate and wheel by ANSYS 5.4 program.

Chapter – 5: contains ANSYS analyses results and summary of comparison with conventional two-dimensional design results.

Chapter – 6: contains conclusion, suggestion and scope of further study.

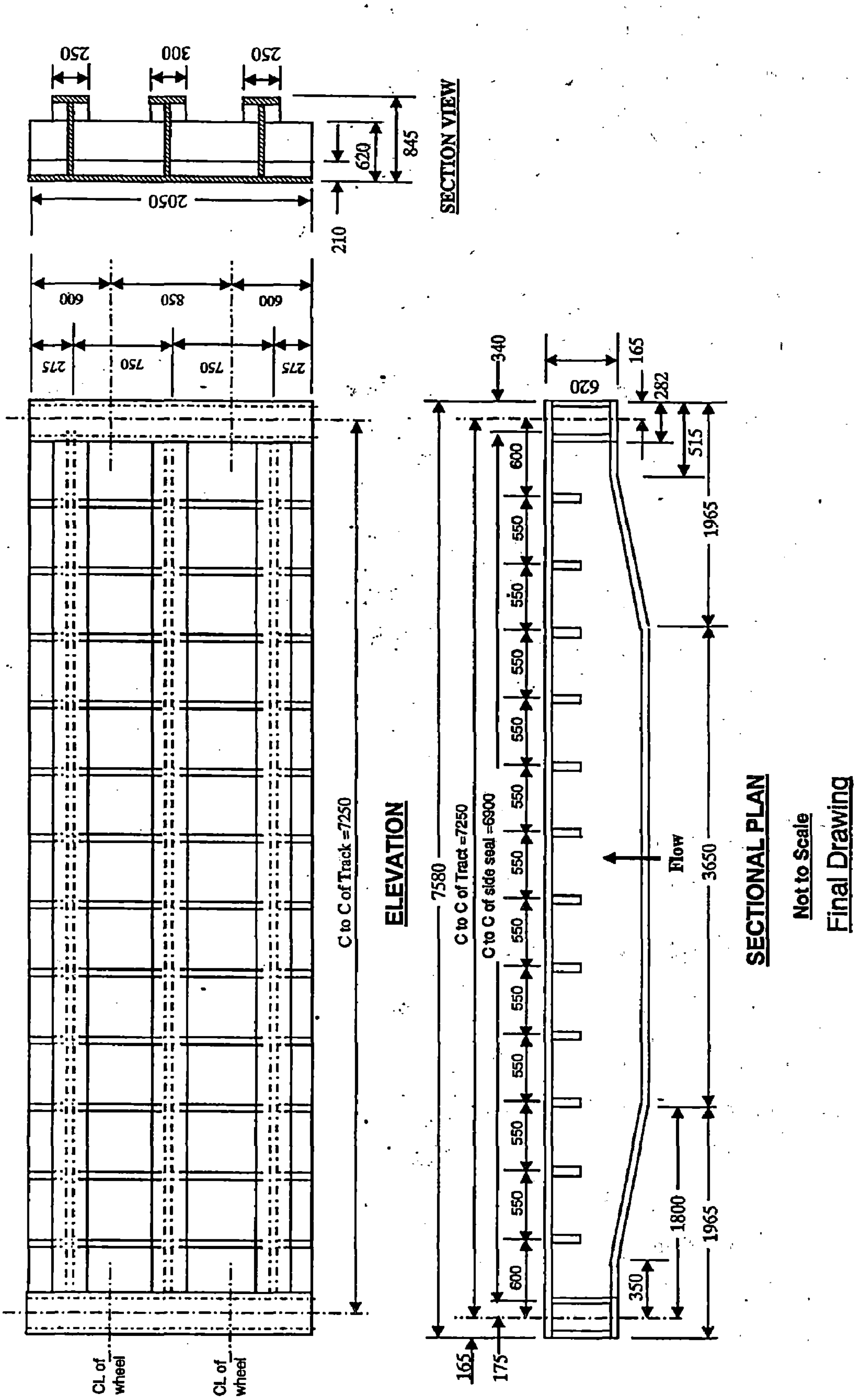


Figure: 1.6 FW Vertical-Lift Gate at Tailrace Channel of Dharasu Power House.

BASIC CONCEPT OF FINITE ELEMENT METHOD

2.1 INTRODUCTION

The finite element method has developed simultaneously with the increasing use of high-speed electronic digital computers.

In brief, the basis of the finite element method is the representation of a structure (say a gate) by an assemblage of subdivisions called finite elements. These elements are considered interconnected at joints, which are called nodes. Simple functions (usually, polynomials) are chosen to approximate the distribution of the actual displacements over each finite element. These functions are called displacement functions. The displacement functions are expressible in terms of the displacements at the nodal points. The final solution yields the approximate displacements at discrete locations i.e. nodal points in the body.

A variational principle of mechanics, such as the principle of minimum potential energy, is usually employed to obtain the set of equilibrium equations for each element. The equilibrium equations for the entire body are then obtained by combining the equations for individual elements in such a way that continuity of displacements is preserved at the interconnected nodes. These equations are modified for the given displacement boundary conditions and then solve to obtain the unknown displacements. By using these displacements, the strains and stresses are computed. This method provides approximate, but acceptable solutions.

2.2 SUMMARY OF THE FEM ANALYSIS PROCEDURE

The sequence of steps as stated below describes the actual solution process that is followed in setting up and solving any equilibrium structural problem in FEM. These steps are:

2.2.1 Discretization Of The Continuum

The continuum is the physical body or structure (say, gate) being analyzed. Discretization may be simply described as the process in which the given body is subdivided into an equivalent system of finite elements. Different type elements are used for different continuums. Line or bar shape (beam) elements are used for representing beam like structure in two-dimensional and three-dimensional; shell elements are used for plates in two-dimensional and three-dimensional; triangular, rectangular or quadrilateral shape solid elements are used in two-dimensional solid model and tetrahedron, rectangular prism or arbitrary hexahedron shape solid elements are used in three-dimensional solid model. One must decide, what type, size and arrangements of the finite elements will give an effective representation of a given continuum for a particular problem considered.

For choosing element size a general guideline is that, where stress or strain gradients are expected to be comparatively flat i.e. the variation is not fast, coarse mesh may be used for reducing the computation effort and where stress or strain gradients are expected to be relatively steeper fine mesh should be used to get more accurate results. Theoretically speaking, to get an exact solution the required number of nodal points is infinite. So, a compromise has to be made between computation effort and accuracy.

2.2.2. Selection Of The Interpolation Or Displacement Function

The assumed displacement functions represent only approximately the actual or exact distribution of the displacements. To ensure convergence to the correct results, the displacement function should be able to represent the true displacement distribution as closely as possible.

There are three interrelated factors, which influence the selection of a displacement function.

First, the type and degree of the displacement function must be chosen (Since, usually, a polynomial is chosen; only the degree of the polynomial is open to decision.).

As for example, for a 3 nodded triangular element the linear polynomial of degree one as stated below

$$\phi = a_1 + a_2 x + a_3 y \qquad \text{(i)}$$

is appropriate.

Where a_1, a_2, a_3 are constants which can be expressed in terms of ϕ at these nodes.

For a 4 nodded quadrilateral a polynomial of degree two as stated below

$$\phi = a_1 + a_2 x + a_3 y + a_4 xy \qquad \text{(ii)}$$

is appropriate.

An 8-nodded quadrilateral has 8 a_i in its polynomial expansion and can represent a parabolic function.

Second, the particular (nodal points) displacement magnitudes that describe the function must be selected.

Third, the function should satisfy **convergence criteria**, which ensure that the numerical results approach the correct solution. These criteria are:

Criterion 1: The displacement functions should be so chosen such that it must be continuous within the element and must be compatible with the adjacent elements.

The first part is automatically satisfied if displacement functions are polynomials. The second part implies that the adjacent elements must deform without causing openings, overlaps or discontinuities between them. This can be satisfied if displacements along the side of an element depend only upon displacements of the nodes occurring on that side. Since the displacements of nodes on the common boundary will be same, displacement for boundary line for both elements will be identical.

Criterion 2: The displacement functions must include the rigid body displacements of the element.

Basically this condition states that, there should exist such combinations of values of coefficients in displacement function that cause all points in the elements to experience the same displacement.

Criterion 3: The displacement functions must include the constant strain states of elements. This means that, there should exist such combinations of values of the coefficients in the displacement function that cause all points on the element to experience the same strain. The necessity of this requirement can be understood if we imagine that the continuum is divided into infinitesimally small elements. In such a case the strains in each element approach constant values all over the element. The terms a_3 and a_6 in the following equations

$$u(x) = a_1 + a_2 x + a_3 y$$

$$v(y) = a_4 + a_5 x + a_6 y \qquad (3.3)$$

provide for uniform strain in x and y directions.

The elements, which meet first criterion, are called compatible or conforming. The elements, which meet second and third criteria, are called complete. For plain strain and plain stress and three-dimensional elasticity the three conditions mentioned above are easily satisfied by linear polynomials.

2.2.3. Derivation Of The Element Stiffness Matrix

The stiffness matrix consists of the coefficients of the equilibrium equations derived from the material properties ([D]-elasticity matrix) and geometric properties ([B]-strain matrix) of an element (i.e. stiffness matrix $[K]^e = \int_v [B]^T[D][B]dv$) and obtained by the use of the principle of minimum potential energy.

The stiffness matrix relates the displacements at the nodal points to the applied forces at the nodal points. The distributed loads/forces applied to the structure are converted into equivalent concentrated forces at the nodes. The equilibrium relation between stiffness matrix $[K]^e$, nodal force vector $\{F\}^e$, and the nodal displacement vector $\{\delta\}^e$, is expressed as a set of simultaneous linear algebraic equations as

$$[K]^e\{\delta\}^e = \{F\}^e;$$

The stiffness matrix for an element depends upon:

i. The displacement function;
ii. The geometry of the element, and
iii. The local material properties (Young's modulus E andPoisson's ratio .).

The local material properties as stated above are one of the factors, which determine stiffness matrix. For an elastic isotropic body, Modulus of Elasticity (E) and Poisson's Ratio (v) define the local material properties. The stiffness matrix is essentially a symmetric matrix, which follows from the principle of stationary potential energy, that "In an elastic structure work done by internal forces is equal in magnitude to the change in strain energy". And also from Maxwell Betti reciprocal theorem which states that: "If two set of loads $\{F\}_1$ and $\{F\}_2$ act on a structure, work done by the first set in acting through displacements caused by the second set is equal to the work done by second set in acting through displacements caused by first set.

2.2.4. Assembly Of The Algebraic Equations For The Overall Discretized Continuum

This process includes the assembly of the overall or global stiffness matrix for the entire body from the individual element stiffness matrices, and the global force or load vector from the element nodal force vectors.

The most common assembly technique is known as the "**direct stiffness method**." In general, the basis for an assembly method is that the displacement at a node to be the same for all elements adjacent to that node. The overall equilibrium relation between the total stiffness matrix [K], the total load vector {F} and the nodal displacement vector for the entire body $\{\delta\}$ will again be expressed as a set of simultaneous equations.

$$[K]\{\delta\}=\{F\};$$

The global stiffness matrix [K] will be banded and also symmetric of size n x n;

where, n = total number of nodal degrees of freedom in the entire body.

The steps involved in generation of global stiffness matrix are:

i. All elements of global stiffness matrix [K] are assumed to be equal to zero;
ii. Individual element stiffness matrices [K] are determined successively;
iii. The element k_{ij} of element stiffness matrix are directed to the address of element K_{ij} of global stiffness matrix which means

$$K_{ij} = \Sigma\, k_{ij};$$

Similarly nodal load $\{F_i\}^e$ at a node 'i' of an element 'e' is directed to the address of $\{F_i\}$ of total load vector i.e.

$$\{F_i\} = \Sigma \{F_i\}^e$$

Then these equations can be solved after appropriate modification by imposing the geometric boundary conditions. A geometric boundary condition arises from the fact that displacements may be prescribed at the boundaries or edges of the body or structure.

Nodal Degrees of Freedom: The nodal displacements and rotations are necessary to specify deformation of the finite elements completely, are called degrees of freedom (DOF) of elements.

Nodal Forces and Loads: Generally when subdividing a structure we select nodal locations that coincide with the locations of the concentrated external forces. In case of distributed loading over the body such as water pressure on dam or the gravity forces the loads acting over an element are distributed to the nodes of that element by principle of minimum potential energy. If the body forces are due to gravity only then they are equally distributed among the nodes of each element.

Boundary Conditions: A problem in solid mechanics is not completely specified unless boundary conditions are prescribed. Boundary conditions arise from the fact that at certain points or near the edges, the displacements are prescribed. The physical significance of this is that a loaded body or a structure is free to experience unlimited rigid body motion unless some supports or kinematics constraints are imposed that will ensure the equilibrium of the loads. These constraints are called boundary conditions. There are two basic types of boundary conditions, geometric and natural. One of the principal advantages of Finite Element Method is, we need to specify only geometric boundary conditions, and the natural boundary conditions are implicitly satisfied in the solution procedure as long as we employ a suitable valid variational principle. In other numerical methods, solutions are to be obtained by trial and error method to satisfy boundary conditions whereas in Finite Element Method boundary conditions are inserted prior to solving algebraic equations and the solution is obtained directly without requiring any trial.

2.2.5. Solutions For The Unknown Displacements

The assembled algebraic equations are solved for the unknown displacements. In linear equilibrium problems, this is a relatively straightforward application of the matrix algebra techniques i.e. solved for {δ} wherein [K] and {F} are already determined. The equations can be solved either by iterative or elimination procedure. Once the nodal displacements are found, then element strains or stresses can be easily found from generalized Hook's law for a linear isotropic material. However, for non-linear problems, the desired solutions are obtained by a sequence of steps, each step involving the modification of the stiffness matrix and /or load vector.

2.2.6. Computation Of The Element Strains And Stresses From The Nodal Displacements

For certain cases, the magnitudes of the primary unknowns i.e. the nodal displacements will be all to meet the requirement for an engineering solution. However, strains and stresses can be derived and computed from the nodal displacements as follows:

Strain vector $\{\varepsilon\}=[B].\{\delta\}^e$;

Stress vector $\{\sigma\}=[D].\{\varepsilon\}$;

2.3 MATHEMATICAL MODEL OF FEM

2.3.1 General

Most Engineers and Scientists studying physical phenomena are involved with two major tasks:

i. Mathematical formulation of the physical process;
ii. Numerical analysis of the mathematical model;

Development of the mathematical model of a process is achieved through assumptions concerning how the process works. In a numerical simulation, we use a numerical method and a computer to evaluate the mathematical model. While the derivation of the governing equations for most problems is not unduly difficult, their solution by exact methods of analysis is a formidable task. In such cases, approximate methods of analysis provide alternative means of finding solution. Among this finite element method is most frequently used.

Finite element method has the following three basic features:

i. A geometrically complex domain of the problem is represented as a collection of geometrically simple sub domains called finite elements.
ii. Over each element the approximation functions are derived using the basic idea that any continuous function can be represented by a linear combination of algebraic polynomials.
iii. Algebraic relations among the undetermined coefficients (i.e. nodal values) are obtained by satisfying the governing equations over each element.

The approximation functions are derived using concepts from interpolation theory and are called interpolation functions. The degree of interpolation functions depends on the number of nodes in the element and the order of differential equation being solved.

2.3.2 Interpolation Function

The finite element approximation $U^e(x,y)$ of $u(x,y)$ over an element Ω^e must satisfy the following conditions in order for the approximate solution to be convergent to the true one.

i. U^e must be differentiable.
ii. The polynomials used to represent U^e must be complete (i.e. all terms beginning with a constant term up to the highest order used in the polynomial, should be included in U^e).
iii. All terms in the polynomial should be linearly independent.

The number of linearly independent terms in the representation of U^e dictates the shape and number of nodes in the element.

2.3.3. *Displacement Function*

For a typical finite element 'e' defined by nodes i,j,k etc. the displacements {f} within the element are expressed as:

$$\{f\} = [N]\{\delta\}^e \tag{2.1}$$

Where $[N] = [N_i \ N_j \ N_m \ldots]$

And $\{\delta\}^e = \{\delta_i \ \delta_I \ \delta_m \ldots\}$ (2.2)

The components of [N] are in general functions of position and $\{\delta\}^e$ represents a listing of nodal displacements for a particular element.

For the three dimensional element

$$\{f\} = \begin{Bmatrix} u \\ v \\ w \end{Bmatrix} \tag{2.3}$$

represents the displacements in x,y and z directions at a point within the element and

$$\{\delta_i\} = \begin{Bmatrix} u_i \\ v_i \\ w_i \end{Bmatrix} \tag{2.4}$$

are the corresponding displacements of node i.

$[N_i]$ is equal to [IN] where N_i is the shape function of node i and I is an identity matrix.

2.3.4. *Strains*

With displacements known at all points within the element, the strains at any point can be determined. Six strain components are relevant in three-dimensional analysis and the strain vector can be expressed as:

$$\{\in\} = \begin{Bmatrix} \in_x \\ \in_y \\ \in_z \\ \in_{xy} \\ \in_{yz} \\ \in_{zx} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial w}{\partial z} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \\ \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \end{Bmatrix} \qquad (2.5)$$

which can be further written as:

$$\{\in\} = [\mathbf{B}]\{\delta\}^e = [B_i B_j B_k \ldots\ldots]\{\delta\}^e \qquad (2.6)$$

in which $[B_j]$ is the strain displacement matrix.

$[B_j]$ is given by

$$[B_i] = \begin{bmatrix} \frac{\partial N_i}{\partial x} & 0 & 0 \\ 0 & \frac{\partial N_i}{\partial y} & 0 \\ 0 & 0 & \frac{\partial N_i}{\partial z} \\ \frac{\partial N_i}{\partial y} & \frac{\partial N_i}{\partial x} & 0 \\ 0 & \frac{\partial N_i}{\partial z} & \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} & 0 & \frac{\partial N_i}{\partial x} \end{bmatrix} \qquad (2.7)$$

with other sub matrices obtained in a similar manner simply by interchange of subscripts.

For isoparametric elements

$$x = \sum_{i=1}^{n} N_i x_i, y = \sum_{i=1}^{n} N_i y_i, z = \sum_{i=1}^{n} N_i z_i$$

$$u = \sum_{i=1}^{n} N_i u_i, v = \sum_{i=1}^{n} N_i v_i, w = \sum_{i=1}^{n} N_i w_i \tag{2.8}$$

the summation being over total number of nodes in an element. Because the displacement model is formulated in terms of the natural coordinates ξ, η and ς and it is necessary to related Eq. (2.7) to the derivatives with respect to these local coordinates.

The natural coordinatesξ, η and ς are functions of global coordinates x,y, z. Using the chain rule of partial differentiation we can write:

$$\frac{\partial N_i}{\partial \xi} = \frac{\partial N_i}{\partial x}.\frac{\partial x}{\partial \xi} + \frac{\partial N_i}{\partial y}.\frac{\partial y}{\partial \xi} + \frac{\partial N_i}{\partial z}.\frac{\partial z}{\partial \xi} \tag{2.9}$$

Performing the same differentiation with respect to the other two coordinates and writing in matrix form:

$$\begin{Bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \\ \frac{\partial N_i}{\partial \zeta} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & \frac{\partial z}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & \frac{\partial z}{\partial \eta} \\ \frac{\partial x}{\partial \zeta} & \frac{\partial y}{\partial \zeta} & \frac{\partial z}{\partial \zeta} \end{Bmatrix} \begin{Bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} \end{Bmatrix} = [J] \begin{Bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} \end{Bmatrix} \tag{2.10}$$

where [J] is given by:

$$[J] = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & \frac{\partial z}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & \frac{\partial z}{\partial \eta} \\ \frac{\partial x}{\partial \zeta} & \frac{\partial y}{\partial \zeta} & \frac{\partial z}{\partial \zeta} \end{bmatrix} \tag{2.11}$$

The matrix [J] is called the Jacobian matrix. The global derivatives can be found by inverting [J] as follows:

$$\begin{Bmatrix} \dfrac{\partial N_i}{\partial x} \\ \dfrac{\partial N_i}{\partial y} \\ \dfrac{\partial N_i}{\partial z} \end{Bmatrix} = [J]^{-1} \begin{Bmatrix} \dfrac{\partial N_i}{\partial \xi} \\ \dfrac{\partial N_i}{\partial \eta} \\ \dfrac{\partial N_i}{\partial \zeta} \end{Bmatrix} \tag{2.12}$$

Substituting Eq. (2.8) into Eq. (2.11) the Jacobian matrix is given by

$$[J] = \begin{bmatrix} \sum \dfrac{\partial N_i}{\partial \xi} x_i & \sum \dfrac{\partial N_i}{\partial \xi} y_i & \sum \dfrac{\partial N_i}{\partial \xi} z_i \\ \sum \dfrac{\partial N_i}{\partial \eta} x_i & \sum \dfrac{\partial N_i}{\partial \eta} y_i & \sum \dfrac{\partial N_i}{\partial \eta} z_i \\ \sum \dfrac{\partial N_i}{\partial \zeta} x_i & \sum \dfrac{\partial N_i}{\partial \zeta} y_i & \sum \dfrac{\partial N_i}{\partial \zeta} z_i \end{bmatrix} \tag{2.13a}$$

$$[J] = \begin{bmatrix} \dfrac{\partial N_1}{\partial \xi} & \dfrac{\partial N_2}{\partial \xi} \cdots\cdots \\ \dfrac{\partial N_1}{\partial \eta} & \dfrac{\partial N_2}{\partial \eta} \cdots\cdots \\ \dfrac{\partial N_1}{\partial \zeta} & \dfrac{\partial N_2}{\partial \zeta} \cdots\cdots \end{bmatrix} \begin{bmatrix} x_1 & y_1 & z_1 \\ x_2 & y_2 & z_2 \\ x_3 & y_3 & z_3 \\ . & . & . \\ . & . & . \\ . & . & . \end{bmatrix} \tag{2.13b}$$

2.3.5. Stresses

The stresses are related to the strains as:

$$\{\sigma\} = [D](\{\in\} - \{\in_0\}) + \{\sigma_0\} \tag{2.14}$$

Where,

[D] is an elasticity matrix containing the appropriate material properties.

$\{\in_0\}$ is the initial strain vector.

$\{\sigma\}$ is the stress vector given by

$\{\sigma\} = \{\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{zx}\}^T$ and

$\{\sigma_0\}$ is the initial stress vector.

For linear elastic, isotropic material the elasticity matrix is given by:

$$[D]=\frac{E}{(1+v)(1-2v)}\begin{bmatrix} 1-v & v & 0 & 0 & 0 & 0 \\ v & 1-v & v & 0 & 0 & 0 \\ v & v & 1-v & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2v}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2v}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2v}{2} \end{bmatrix} \qquad (2.15)$$

where, E is the Young's modulus of elasticity and v is the Poisson's ratio of the material of the element.

2.3.6. Stiffness Matrix

The stiffness matrix of the element is given by the following relation

$$\{F\}^e = [K]^e \{\delta\}^e \qquad (2.16)$$

where $\{F\}^e$ is the element nodal load vector, $\{\delta\}^e$ = nodal displacement vector and $[K]^e$ the element stiffness matrix given by:

$$[K]^e = \int\int [B]^T [D][B] dV \qquad (2.17)$$

where V refers to the volume of the element.

The equivalent nodal forces are obtained as

i) Forces due to pressure distribution $\{p_x, p_y, p_z\}^T$ per unit area given by:

$$\{F^e\}_p = \int_v [N]^T \{p\} dA \qquad (2.18)$$

For the complete structure relation of the form given below is obtained

$$\{K\}\{\delta\} = \{F\}$$

where $\{\delta\}$ is the vector of global displacements, $\{F\}$ the load vector and [K] the stiffness matrix.

The global stiffness matrix [K] is obtained by directly adding the individual stiffness coefficients in the global stiffness matrix. Similarly the global load vector for the system is also obtained by adding individual element loads at the appropriate locations in the global vector.

The mathematical statement of the assembly procedure is:

$$[K] = \Sigma_{0-1}^{E} [K]^{e}$$

$$\{F\} = \mathrm{E}_{0-1}^{E} \{F\}^{0} \qquad (2.19)$$

where E is the total number of elements.

To transform the variable and the region with respect to which the integration is made the relationship.

$$dA = dxdy = \det[J]d\xi d\eta \qquad (2.20)$$

$$dV = dxdydz = \det[J]d\xi d\eta d\zeta$$

is used.

Writing explicitly

$$\int_{v} dxdy = \int_{-1}^{+1}\int_{-1}^{+1} \det[J]d\xi \, d\eta \qquad (2.21)$$

and the characteristic element stiffness matrix can be expressed as

$$[K]^{e} = [B]^{T}[D][B]\int_{-1}^{+1}\int_{-1}^{+1} \det[J]d\xi \, d\eta \qquad (2.22)$$

A 2 x 2 x 2 integration has been used for the three dimensional analysis.

CONVENTIONAL DESIGN OF VL GATE

3.1 INTRODUCTION

The conventionally the fixed wheel vertical-lift gates are designed on the basis of guidelines given in various handbooks of hydro-electric engineering, gates & valves and IS: 4622: Indian Standard Fixed Wheel Vertical-lift Gates Structural Design Recommendations. Normally the skin plate is fitted on the upstream side of the frameworks and it can be fitted on the downstream side also when advantageous as the case of under study. The skin plate is supported by suitably spaced stiffeners either vertical or horizontal or both. If horizontal stiffeners are used, these are supported by suitably spaced vertical stiffeners, which are connected to horizontal girders transferring the load to the two vertical end girders. The wheels are fitted to the vertical end girders and ultimately loads are transferring to wheel tracks & piers.

The gate under study is consisting of three horizontal girders, two vertical end girders, twelve vertical stiffeners and four wheels two on each side.

3.2 DESIGN CRITERIA

3.2.1 Main Components Of The Gate

i. Skin plate;
ii. Vertical stiffener;
iii. Horizontal girder;
iv. Vertical end girder;
v. Wheel;
vi. Wheel axle;
vii. Track and
viii. Seal.

3.2.2 Design Of Skin Plate

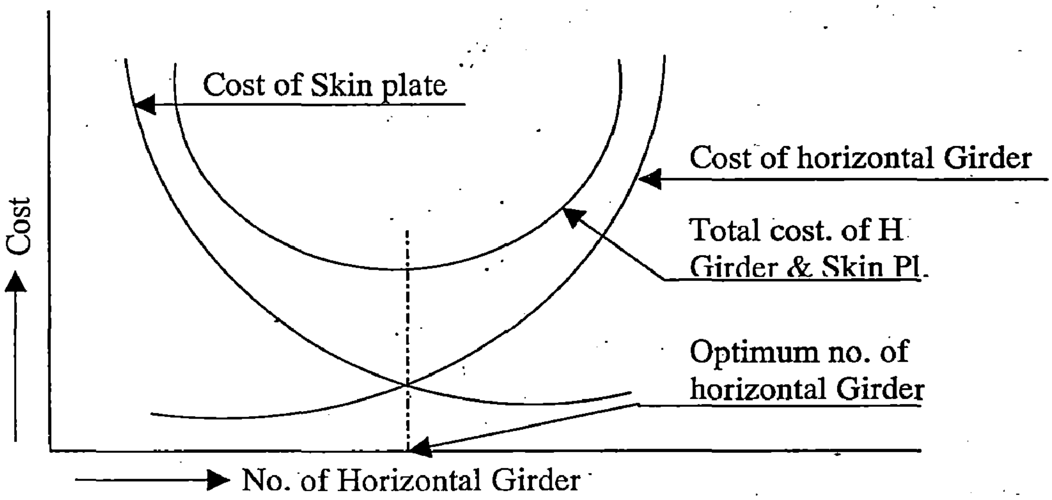
The skin plate and stiffeners are designed together in a composite manner. The skin plate is designed for either of the following two conditions unless more precise methods are available:

a) In bending across the stiffeners or horizontal girders as applicable, or

b) As panels in accordance with the procedure and support conditions given in Annex C. of IS 4622.

The maximum bending moment can be found only after knowing the number of horizontal girder and their location.

Number of horizontal girders is decided on economics. If the number of horizontal girders increases thickness of skin plate decreases as such cost of horizontal girder increases and skin plate decreases, and vice-versa as shown in graph.3.1.



Graph: 3.1

The number of girders depends on the total height of the gate but to be kept as minimum as possible to simplify fabrication and erection and to facilitate maintenance.

The spacing of horizontal girders is based on the criteria that, load on all girders is equal for economic reasons. If all girders carry equal load then cross section of each girder throughout the gate height will be same and easy of construction. The spacing of girders for equal loading can be found by:

i. Analytical method;

ii. Graphical method, or

iii. By trial and error.

Analytically for surface gate, on the basis of equal loading, spacing of the horizontal girders is given by the following formula as shown in Fig.3.1.

$$d_r = \frac{2}{3}\frac{H}{\sqrt{n}}\left[r^{3/2} - (r-1)^{3/2}\right] \qquad (3.1)$$

Where,

d_r = distance of C/L of horizontal girders from the top of the gate in m or cm;

H = total head of water in m or cm;

n = number of horizontal girders;

r = sequence no. of the horizontal girders (1, 2, ------n)

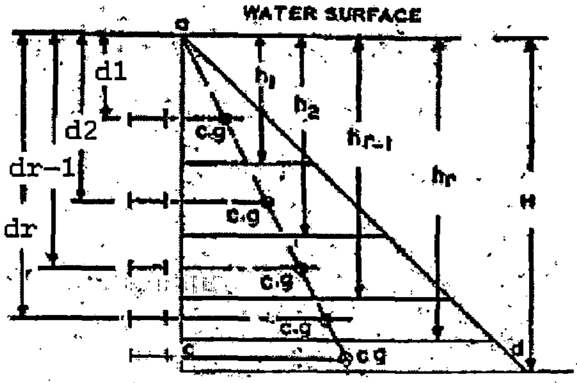


Figure: 3.1 Spacing of horizontal girders.

The analytical method can also be used for intake (submerged) gates with slite modification on the basis of equal loading. The final formulation is given as below.

$$d_r = \frac{2n}{3} \times \frac{\left\{\frac{r}{n}(H^2 - H_t^2) + H_t^2\right\}^{3/2} - \left\{\frac{r-1}{n}(H^2 - H_t^2) + H_t^2\right\}^{3/2}}{H^2 - H_t^2} \quad \text{-------- (3.2)}$$

Where,

d_r = distance of C/L of horizontal girders from the top of water level in m or cm;

H = depth of water at the bottom of the gate in m or cm;

H_t = depth of water at the top of the gate in m or cm;

n = number of horizontal girders;

r = sequence no. of the horizontal girders (1, 2, ------n)

Graphically, for barrage gates the problem of girder spacing can also be solved as follows given by Leliavsky. Let a b c in Fig.3.2 be the triangle representing the water pressure applied to the gate. Draw a semicircle with a b as diameter. Then divide a b into equal intervals a a_1, a_1 a_2 ... ...of the number of girders to be used. Through the points a_1, a_2,. ... ...draw a set of parallel lines at right angles to a b and mark the intersections of these lines with semicircle as b_1, b_2, ... Draw arcs with centre at point a through the points b_1, b_2, ... which cut a b at points c_1, c_2, ... yield the positions of the horizontal lines c_1 d_1, c_2 d_2, ... dividing the area of the basic triangle in equal parts. Find the centroid of the areas a c_1 d_1, d_1 c_1 c_2 d_2, ... and space the girders accordingly.

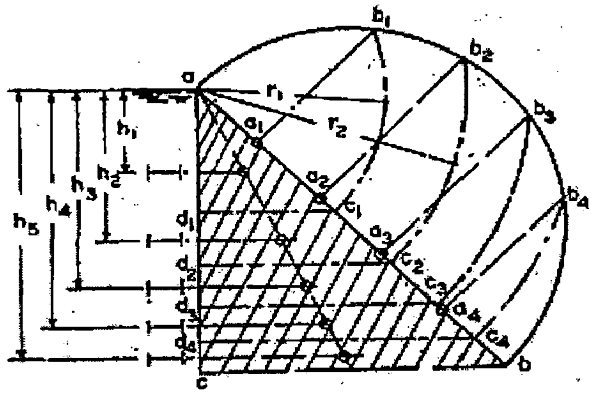


Figure: 3.2 Spacing of girders on a barrage gate (Graphically).

The graphical method is also applicable to two-tier and intake gates as shown in Fig. 3.3 where opening requirements are such that:

a) top girder of top tier is subjected to repeated impact.

b) same is true for:

   i. top girder of lower tier,

   ii. bottom girder of top tier.

In this case the top tier girders spacing is done as follows:

Divide the hypotenuse of water pressure triangle into $2n_{ti}+1$ equal parts.

Where, n_{ti} = number of intermediate girders in top tier = $n_t - 2$,

n_t = number of girders in top tier.

Similarly, for the bottom tier, number of division = $2n_{bi}+2$;

Where, n_{bi} = number of intermediate girders in bottom tier = $n_b - 2$,

n_b = number of girders in bottom tier.

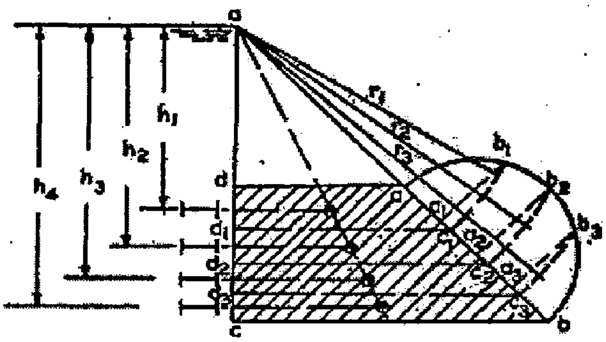


Figure: 3.3 Spacing of girders on an intake gate (Graphically).

A trial and error method can also be applied for spacing of girders. All the girders are of same section but the top and bottom girders each are supposed to carry half the load. While the top girder is flushed with the gate, the bottom girder is placed a little above the bottom in order that the bottom sealing is properly fixed.

Once horizontal girders are located to carry almost equal load, the skin plate can be designed on bending either across the horizontal girders or vertical stiffeners.

Now, with reference to IS 4622: 1992, page 12, fig. 5, Annex-C, Clause 5.2.3(a). We have the following equation for stresses in skin plate. (Note: symbols of stress & thickness have been changed).

$$S = \frac{K}{100} x \frac{Pxa^2}{t^2} \quad \text{-------------------------} \quad (3.3)$$

Where,

S = bending stress in flat plate in N/mm^2 or kg/cm^2;

K = non-dimensional factor depends on the values of a & b;

P = water pressure in N/mm^2 or kg/ cm^2 (relative to the plate centre);

a, b = bay width in mm or cm as shown in ref. and

t = plate thickness in mm or cm.

The values of 'K' are given with due consideration to biaxial bending in contrast to simple bending of plates. Once stresses are found, they are converted into equivalent stress as follows:

$$S_c = \sqrt{S_x^2 + S_y^2 - S_x \times S_y + 3S_s^2} \quad \text{-------------} \quad (3.4)$$

Where,

S_c = combined or equivalent stress in N/mm^2 or kg/cm^2;

S_x = stresses in 'X' direction in N/mm^2 or kg/cm^2;

S_y = stresses in 'Y' direction in N/mm^2 or kg/cm^2;

S_s = shear stresses in N/mm^2 or kg/cm^2;

In the above equation due consideration is to be given to proper sign of stresses i.e. compression or tension. Finally $S_c < S_{permissible}$. The values of permissible stresses are given in the IS in terms of S_{yp}. In case of skin plate design S_s is neglected. In considering the thickness of skin plate, due consideration is also given to corrosion allowance. Minimum corrosion allowance is 1.5 mm. The minimum thickness of the skin plate shall not be less than 8 mm, exclusive of corrosion allowance.

3.2.3 Design Of (Vertical) Stiffeners

The vertical stiffeners are designed as simply supported beam on horizontal girders as shown in Fig. 3.4.

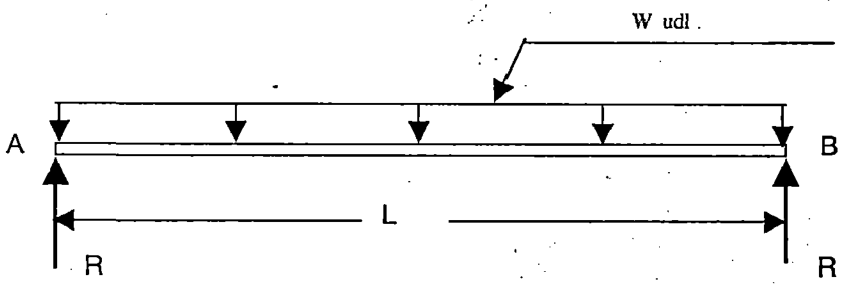


Figure : 3.4

Maximum Bending Moment, $M = W x L^2/8$ in kg-cm;

Where,

udl W = P.b in kg/cm;

P = water pressure in kg/cm^2;

B = spacing of vertical stiffeners in cm;

L = spacing of horizontal girders in cm.

Flexural or fibre stress is given by the following formula.

$$f = \frac{M.y}{I} = \frac{M}{Z} \qquad (3.5)$$

where,

y = distance of the fibre from the neutral axis in cm;

I = moment of inertia about neutral axis in cm^4;

Z = I/y, section modulus of the section in cm^3.

For taking into account the section, due consideration is to be given on coacting width of skin plate.

Coacting width can be found; as per ANNEX- 'D' page 15 of IS 4622: 1992.and also page 4, art. 5.2.4, coacting widths are given by:

i) 40 t + B;

Where, t = thickness of skin plate, and

B = width of stiffener flange in contact with the skin plate;

ii) 0.11 span;

iii) Centre to centre of stiffeners or girders;

The least of the above values shall be considered as coacting width.

3.2.4 *Design Of Horizontal Girder*

Number of horizontal girders, spacing or location is already mentioned in the section of design of skin plate. The horizontal girders are designed as simply supported beam on bending as shown in Fig. 3.5.

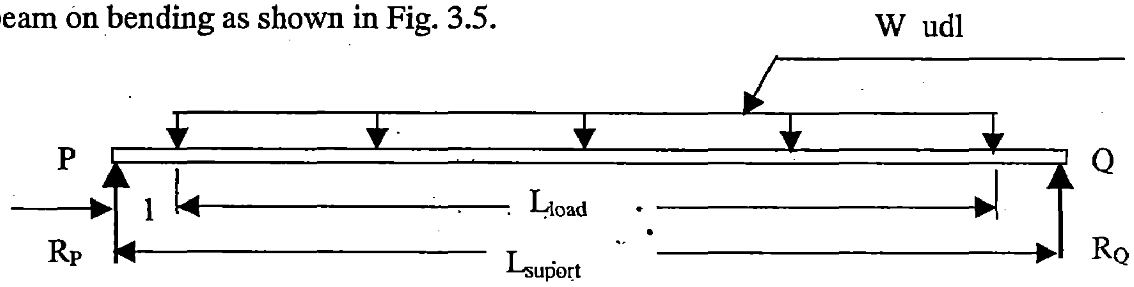


Figure : 3.5

Where,

P = water pressure in kg/cm^2;

B = Width of water pressure on beam in cm;

Then Water load per unit length (udl), W =PxB in kg/cm;

L_{load} = C/C of vertical seals, load span on beam in cm;

$L_{support}$= C/C of wheels, support span of the beam in cm;

Reaction force on each support in kg, $R_p = R_Q = W x L_{load}/2$;

l = Offset of load from point P in cm;

x = Distance of any point in X direction from point P in cm;

So, B.M at a distance x in kg-cm, $M = R_P.x - \{W.(x-l)^2/2\}$;

And max. BM in kg-cm, $M_{max}=R_p.L_{support}/2 - \{W(L_{support}/2 - l)^2 /2\}$;

Flexural or fibre stress is given by the following formula.

$$f = \frac{M.y}{I} = \frac{M}{Z} \quad \text{-----------------------------------} \quad (3.5)$$

where,

y = distance of the fibre from the neutral axis in cm;

I = moment of inertia about neutral axis in cm^4;

Z = I/y, section modulus of the section in cm^3.

For taking into account the section, due consideration is to be given on coacting width of skin plate as described in the above section.

Horizontal girders are made in the form of continuous beams. For opening up to 5 m I-beams are generally used and for wider openings I-beams of composite sections or fabricated beams may be used. To reduce the width of the grooves, beams heights at the ends are curtailed to extend of 0.40 – 0.65 of its height as shown in Fig: 1.2 because of lesser bending moment but are checked for end shear force.

Maximum deflection of the gate under normal conditions of loading is limited to 1/800 of the span. However, in case of gates with upstream top seals, the maximum deflection of the gate leaf at the top seal should not be more than 80% of the initial interference of the seal.

The actual deflection can be found from the following formula:

$$d_{actual} = \frac{5}{384} \times \frac{WL^4}{E.I} \quad \text{------------------------------} \quad (3.6)$$

Where, W = load per unit length in kg/cm;

L = supporting span of the girder;

E = modulus of elasticity of the materials in kg/cm^2 and

I = moment of inertia of the section in cm^4.

3.2.5 Design Of Vertical End Girder

Load comes on vertical end girders from the horizontal girders and supported on wheels as shown in Fig: 1.2 and 1.3. Design is carried out as continuous beam by using any of the methods used for analytical analysis of statistically indeterminate structures viz. theorem of 3 moments or, moment distribution methods in bending, such that maximum bending stress,

$$S_{bending} = \frac{M_{max}}{I} \times \overline{Y} < S_{permissible}$$

Then it is checked for shear as well as bearing stresses.

Also design is to be checked for:

i. Normal operating conditions (when all the wheel touches the track) and
ii. Emergency operating conditions (when one of the wheels is off the track).

For 2nd conditions, permissible stresses are increased by 33% (as per IS clause-6.2). However, before designing of vertical end girders, location of the wheels and tread width of the wheels and pin diameter has to be known.

The location of the wheels is decided by the following design criteria:

a) All wheels should carry equal loads and their physical similarity in wheel diameter etc. can be assumed;

b) Wheel pin should not interfere the horizontal girder.

In most cases wheels are conventionally located symmetrically on both sides of the horizontal girders. In some cases, the vertical end girders comprise of a flexible arrangement where the girder is discontinuous between adjoining wheel pairs such that the structure is statically determinate.

3.2.6 Design Of Wheel

The wheel capacity can be calculated on the basis of the stress in the tread. The product of wheel diameter and net tread width, which can be determined by the following formula, gives the required projected area:

Critical stress (kg/cm^2) on the projected area, S_c= 1.72 BHN – 154.93;

Allowable critical stress, S_{ac}= S_c/sf, (sf =safety factor, 2 – 3);

Projected area of the wheel = wheel load/allowable critical stress

i.e. $A_{pa} = D_w x l_w = P / S_{ac}$;

For an assumed suitable wheel diameter, net tread width can be found or vice versa. Then, wheel to be checked for contact stress. For line contact, contact stress between wheel and track shall be calculated in accordance with the following formula:

$$f_c = 0.418\sqrt{\frac{PE}{rl}} \quad \text{------------------------------} \quad (3.7)$$

where,

f_c = contact stress in N/mm^2 or kg/cm^2;

P = wheel load in N or kg;

E = modulus of elasticity of material in N/mm^2 or kg/cm^2;

r = radius of wheel in mm or cm and

l = tread width of wheel in mm or cm.

The permissible contact stress (as per IS: 4622, 1992, page-18) at the surface of the wheel is given by-

f_{pc}= 1.4 UTS;

3.2.7 Design Of Wheel Axle (Shaft)

Usually wheel axle is design as simply supported beam in bending as follow:

$$S_{bending} = \frac{M_{max}}{I} \times \overline{Y} < S_{permissible}$$

where,

$S_{bending}$ = bending stress in kg/cm²;

M_{max} = maximum bending moment in kg-cm;

for shaft, moment of inertia, $I = \pi D^4 / 64$ in cm⁴;

D = diameter of the shaft in cm;

$\overline{Y} = D/2$; and

$S_{permissible}$ = permissible bending stress in kg/cm²;

Permissible bending stress, S_{pb} = UTS/3; and

Permissible shear stress, $S_{ps} = 0.6\ S_{pb}$;

Then, it is checked for shear.

3.3 DESIGN CONSIDERATIONS OF THE VL GATE UNDER STUDY

Conventional design calculations are appended in Appendix –A. However, summary of the approach and results is given below for convenience.

3.3.1 Design Data (Given and Assumed)

The gate is divided into two equal parts to facilitate transportation and at the time of errection two pars is joined together by plate (in both sides) with rubber seal, bolts & nuts Each gate is provided with two wheels on either side and are positioned in such a way that they share the water load almost equally. Calculations have been done for bottom most gate unit and the same is used for the top part too.

i.	Type of gate:	Fixed wheel vertical lift gate;
ii.	Clear opening (height & span):	390 x 675 cm;
iii.	Actual size of gate (height & span):	410 x 758 cm;
iv.	Split size of gate (height & span):	205 x 758 cm;
v.	Design Head of water:	1640 cm;
vi.	Design water pressure, P:	1.64 kg/cm²;

vii.	No. of horizontal girder:	3;
viii.	C/C spacing of horizontal girders:	75 cm;
ix.	End spacing of horizontal girder:	27.5 cm;
x.	Load span, i.e. C/C of vertical seals:	690 cm;
xi.	Support span, i.e. C/C of wheels:	725 cm;
xii.	Final spacing of vertical stiffeners:	55 cm;
xiii.	No. of wheels:	4 (on each side: 2).

3.3.2 Skin Plate

Thickness, t = 2 cm;

Interior panel (ref. appendix – A, art. 5.6 & 13.1):

Max. bending stress in X direction, $S_x = S_{3x} = 529.14$ kg/cm^2 (with v stiffeners);

Max. bending stress in Y direction, $S_y = S_{4y} = 422.70$ kg/cm^2 (with h girders);

Av. shear stress in XY plane, $S_{(xy)av} = S_s = 30.75$ kg/cm^2 (with h girders);

Max. shear stress in XY plane, $S_{(xy)max} = 1.5S_s = 46.13$ kg/cm^2 (with h girders);

Bottom panel (ref. appendix – A, art. 5.7 & 13.6):

Max. bending stress in X direction, $S_x = S_{7y} = 145.73$ kg/cm^2 (with v stiffeners);

Max. bending stress in Y direction, $S_y = S_{6x} = 211.77$ kg/cm^2 (with h girders);

Av. shear stress in XY plane, $S_{(xy)av} = S_s = 22.55$ kg/cm^2 (with v stiffeners);

Max. shear stress in XY plane, $S_{(xy)max} = 1.5S_s = 33.83$ kg/cm^2 (with v stiffeners);

3.3.3 Vertical Stiffener

The section adopted for vertical stiffener is shown in Fig. 3.6.

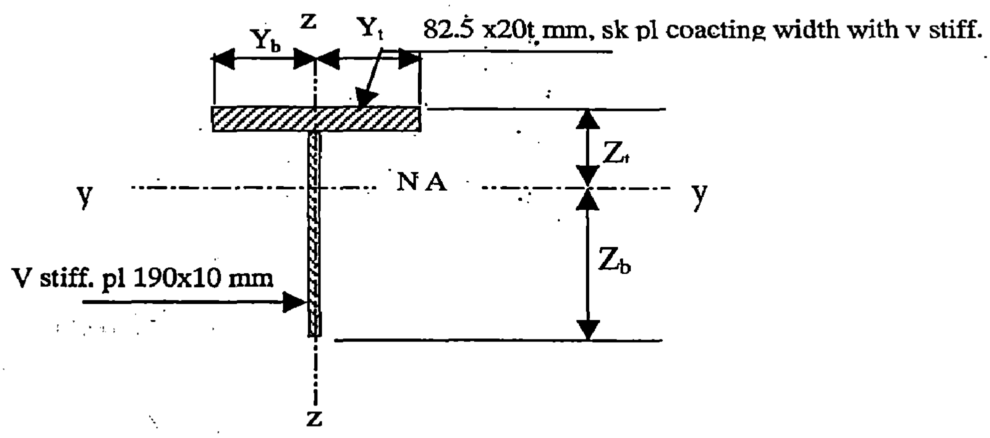


Figure : 3.6

Table: 3.1 Sectional properties:

A (cm^2)	Z_t (cm)	Z_b (cm)	$Y_{t/b}$ (cm)	I_{yy} (cm^4)	I_{zz} (cm^4)
35.50	6.62	14.38	4.125	1550.7	95.17

Maximum bending stresses:

at top, f_t = 270.74 kg/cm^2 (tension);

at bottom, f_b = 588.14 kg/cm^2 (compression).

Maximum shear stresses:

at NA, $f_{s(NA)}$ = 225.54 kg/cm^2;

at skin pl, $f_{s(sk\ pl)}$ = 202.26 kg/cm^2.

3.3.4 Central Horizontal Girder

The section adopted for central horizontal girder is shown in Fig. 3.7.

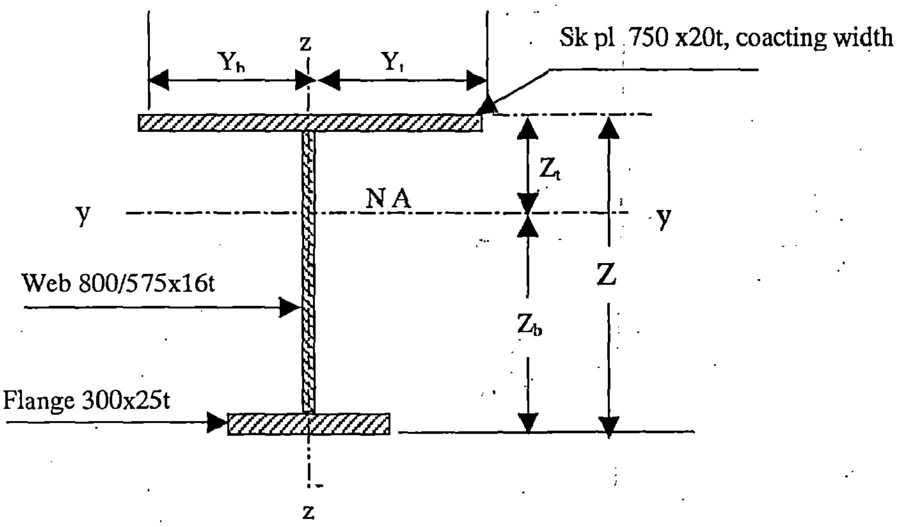


Figure : 3.7

Table: 3.2 Sectional properties:

Position	A (cm^2)	Z (cm)	Z_t (cm)	Z_b (cm)	$Y_{t/b}$ (cm)	I_{yy} (cm^4)	I_{zz} (cm^4)
Centre	353	84.5	33.34	51.16	37.5	421662	75964.8
End	317	62.0	23.77	38.23	37.5	210254	75957.1

Maximum bending stresses (at the centre of the span):

at top, f_t = 637.54 kg/cm^2 (tension);

at bottom, f_b = 978.20 kg/cm^2 (compression).

Maximum bending stresses (at the middle of curtain section):

at top, $f_t = 395.23$ kg/cm^2 (tension);

at bottom, $f_b = 635.64$ kg/cm^2 (compression).

Maximum deflection (at the centre of the span): $d_{actual} = 0.54$ cm.

3.3.5 Bottom Horizontal Girder

The section adopted for bottom horizontal girder is shown in Fig. 3.8.

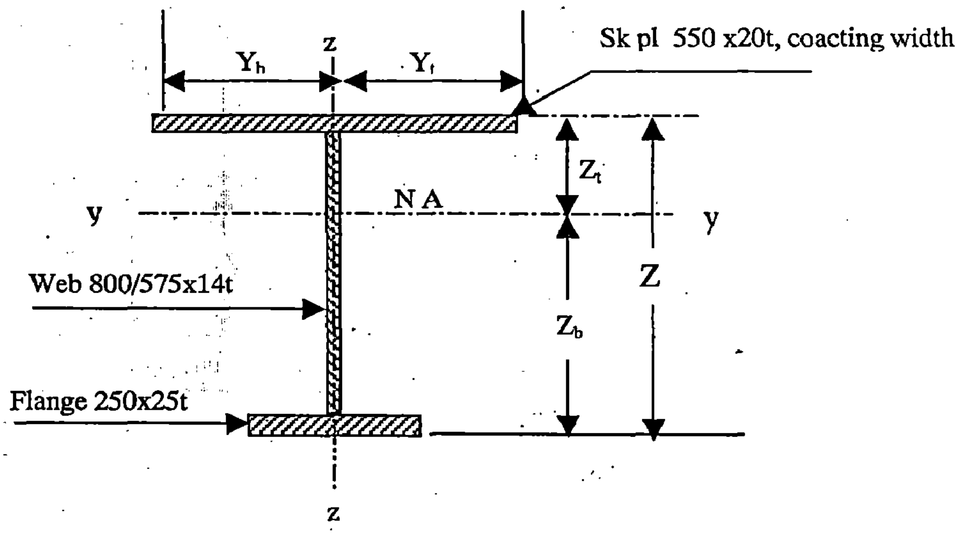


Figure : 3.8

Table: 3.3 Sectional properties:

Position	A (cm^2)	Z (cm)	Z_t (cm)	Z_b (cm)	$Y_{t/b}$ (cm)	I_{yy} (cm^4)	I_{zz} (cm^4)
Centre	284.5	84.5	35.21	49.29	27.5	337942	31002.67
End	253	62.0	25.23	36.77	27.5	168136	30997.52

Maximum bending stresses (at the centre of the span):

at top, $f_t = 728.03$ kg/cm^2 (tension);

at bottom, $f_b = 1019.18$ kg/cm^2 (compression).

Maximum bending stresses (at the middle of curtain section):

at top, $f_t = 454.57$ kg/cm^2 (tension);

at bottom, $f_b = 662.65$ kg/cm^2 (compression).

Maximum deflection (at the centre of the span): $d_{actual} = 0.54$ cm.

3.3.6 Vertical End Girder

The section adopted for vertical end girder is shown in Fig. 3.9.

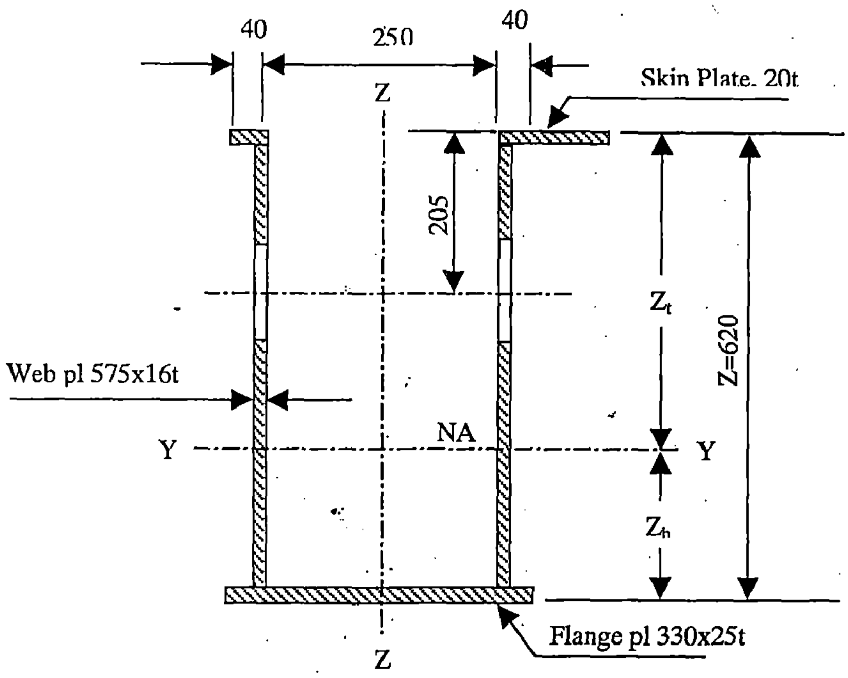


Figure: 3.9

Table: 3.4 Sectional properties:

Condition	A (cm^2)	Z (cm)	Z_t (cm)	Z_b (cm)	$Y_{t/b}$ (cm)	I_{yy} (cm^4)	I_{zz} (cm^4)
Without hole	282.5	62.0	37.83	24.17	16.5	125010	43459.2
With holes	245.7	62.0	40.42	21.58	16.5	111900	36957.52

Maximum bending stresses (at the location of wheel axle):

at top, $f_t = 431.75$ kg/cm^2 (compression);

at bottom, $f_b = 230.49$ kg/cm^2 (tension).

Maximum shear stress (at the NA): $f_{s(NA)} = 232.07$ kg/cm^2;

3.3.7 Wheel

Design water pressure, P = 1.64 kg/cm^2;

Load area, A = 690 x 205 = 141450 cm^2;

Total water load, F =AxP = 231978 kg;

Load on each wheel, $F_w = F/4 = 57994.5$ kg;

Wheel size adopted is shown in Fig: 3.10.

Diameter, D_w = 50 cm;

Tread width, L_w = 15 cm;

Net tread width, $L_{w(net)}$ =13 cm;

Width of the bearing, B = 12 cm;

Maximum:

Contact stress, f_c = 7532.93 kg/cm^2;

Shear stress, f_s = 2290 kg/cm^2;

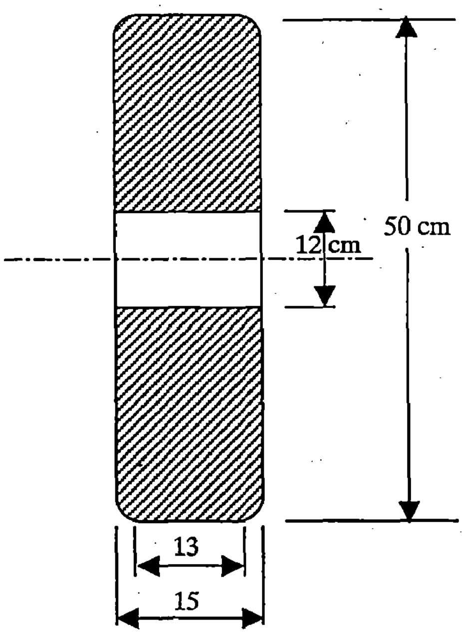


Figure: 3.10

3.4 SUMMARY OF CONVENTIONAL RESULTS

Table 3.5 Maximum value of Bending Stress:

S.N.	Part Name	Location/direction of max. stress.	Max. Ben. Stress (kg/cm^2)
1	Skin Plate	X-direction with V stiffener	529.14
		Y-direction with H girder	422.70
2	Vertical Stiffener	Top flânge (coacting with skin pl)	270.74
		Bottom of web	588.14
3	Central horizontal girder	Top flange (coacting with skin pl)	637.54
		Bottom flange	978.20
4	Top/Bottom horizontal girder	Top flange (coacting with skin pl)	728.03
		Bottom flange	1019.18
5	Vertical end girder	Top flange (coacting with skin pl)	431.75
		Bottom flange	230.49
6	Wheel	Contact stress	7532.93
		Shear stress	2290.00

Table 3.6 Maximum Value of Deflection of Horizontal Girders:

S.N.	Part Name	Max. Deflection (cm)
1	Centre horizontal girder	0.54
2	Top/Bottom Horizontal Girder	0.54

3.5 RESULT OF DESIGN ORGANISATION

Power House Design Unit – II, IDO, Roorkee, UA, India has designed the "Fixed Wheel Vertical Lift Gate" for Tailrace Channel of Dharasu Power House, Maneri Bhali Hydel Scheme Stage-II. The same calculations have been carried out in 'Excel Sheet', which is appended in 'Appendix-A'. It is found that maximum values of stresses in skin plate, vertical stiffener, centre horizontal girder, top/bottom horizontal girder, vertical end girder, wheel etc. are almost same. Deflections of the un-curtail horizontal girders are found exactly same. After curtailment, deflections should be slightly more than un-curtail girders and it's found accordingly.

FEM MODEL OF VERTICAL LIFT GATE BY ANSYS

4.1 INTRODUCTION

There are various finite element software and ANSYS is one of them. It is a powerful tool in FEM analysis. Any complicated structure can be analyzed suitably by ANSYS. It is reasonably flexible and has multi-physics capabilities i.e. options to analysis in various fields like mechanical, structural, thermal, fluid, electromagnetic etc. linear/nonlinear problems in various conditions like static, dynamic, transient, harmonic, etc.

The ultimate purpose of a finite element analysis is to realize the behavior of an actual engineering system. So, the prototype object must be modeled accurately in physical and mathematical sense for accurate results. A physical model consists of geometrical shapes like key points, lines, area and volumes and a mathematical model consists of nodes, elements, real constants and material properties. Therefore, selection of proper element type and feeding of real constant is important. Then, appropriate boundary conditions and loads are applied to represent the physical situation of the system. Convergence of the results is checked with coarse to fine meshes.

In the beginning of analysis, physical modeling technique should be planned out and the elements types should be decided.

A general guideline for deciding the element size is that, where stress/strain gradients are expected to be flat, coarse mesh and where stress/strain gradients are likely to be steeper fine mesh should be used to get more accurate results.

4.2 SELECTION OF ELEMENTS

Selection of proper element is most important in ANSYS to have desired results.

4.2.1 Element For Skin Plate

In three-dimensional analysis for skin plate three-dimensional SHELL element must be used. Diversified three-dimensional SHELL elements are available in ANSYS.

They are:

i. Elastic 4 node 63;
ii. Elastic 8 node 93;
iii. Hyper 4 node 181;
iv. Plastic 4 node 43;
v. 16-Layer 91;
vi. 100-Layer 99.

In brief capabilities and properties of these shell elements are mentioned below:

SHEEL63 'Elastic Shell' as shown in Fig.4.1 has both bending and membrane capabilities. Both in-plane and normal loads are permitted. The element has six degrees of freedom at each node: translations in the nodal x, y and z directions and rotations about the nodal x, y and z-axes. Stress stiffening and large deflection capabilities are included. A consistent tangent stiffness matrix option is available for use in large deflection analyses.

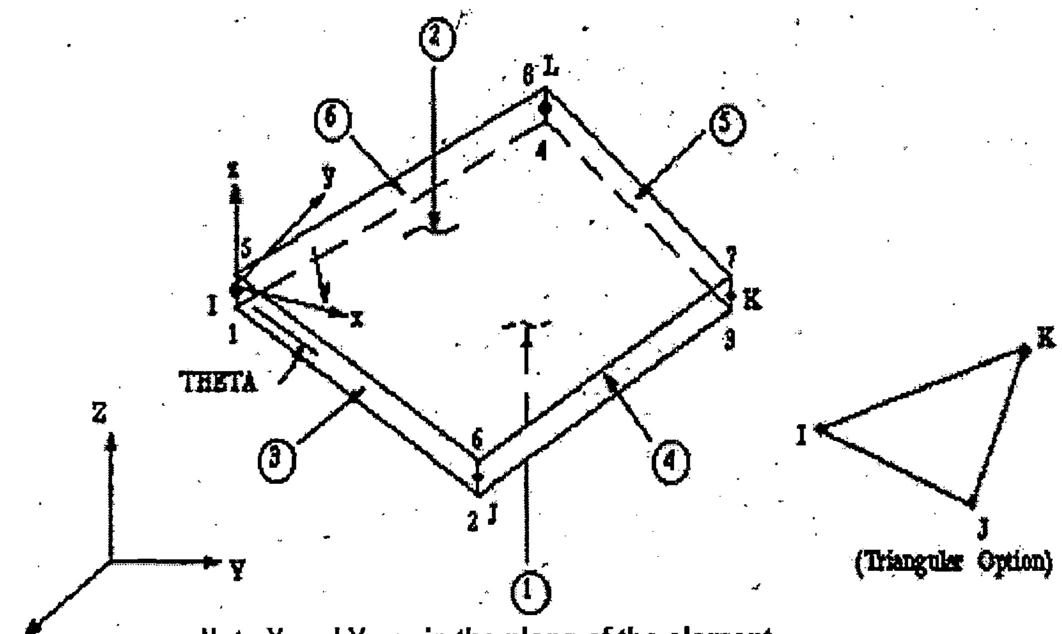


Fig: 4.1 SHELL63 Elastic Shell

SHELL93 '8-Node Structural Shell' as shown in Fig.4.2 is particularly well suited to model curve shells. The element has six degrees of freedom at each node: translations in the nodal x, y and z directions and rotations about the nodal x, y and z-axes. The deformation shapes are quadratic in both in-plane directions. The element has plasticity, stress stiffening, large deflection and strain capabilities.

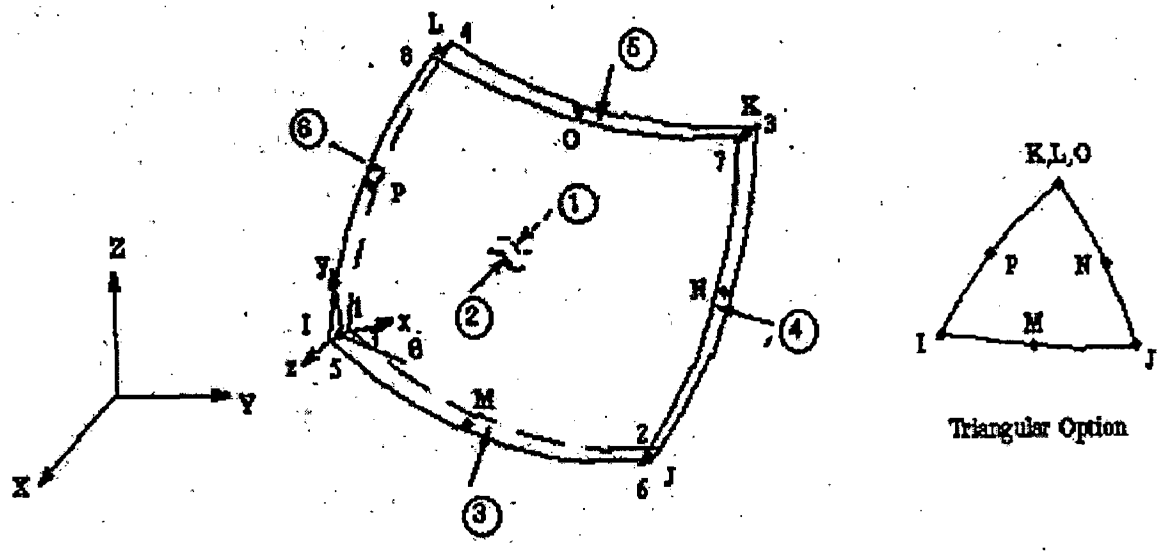


Fig: 4.2 SHELL93 8-Node Structural Shell

SHELL181 'Finite Strain Shell' as shown in Fig.4.3 is suitable for analyzing thin to moderately thick shell structures. It is a four-node element with six degrees of freedom at each node: translations in the x, y and z directions and rotations about the x, y and z-axes. The degenerate triangular option should only be used as filler elements in mesh generation. This shell is well suited for linear, large rotation, and/or large strain non-linear applications and accounts for follower effects of distributed pressure. Change in shell thickness is accounted for in nonlinear analyses. In the element domain, both full and reduced integration schemes are supported.

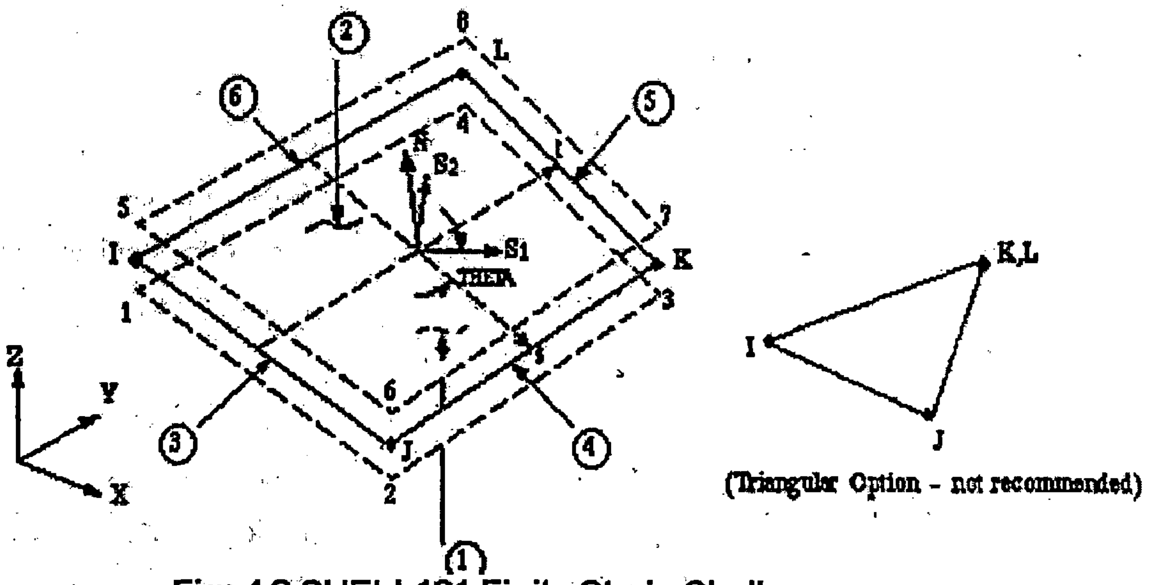


Fig: 4.3 SHELL181 Finite Strain Shell

SHELL43 'Plastic Large Strain Shell' as shown in Fig.4.4 is well suited to model linear, warped, moderately thick shell structures. The element has four-nodes with six degrees of freedom at each node: translations in the x, y and z directions and rotations about the x, y and z-axes. The deformation shapes are linear in both in-plane directions. For the out-of-plane motion, it uses a mixed interpolation of tonsorial components. The element has plasticity, creep, stress stiffening, large deflection, and large strain capabilities. For a thin shell capability or if plasticity or creep is not needed, shell63 may be used.

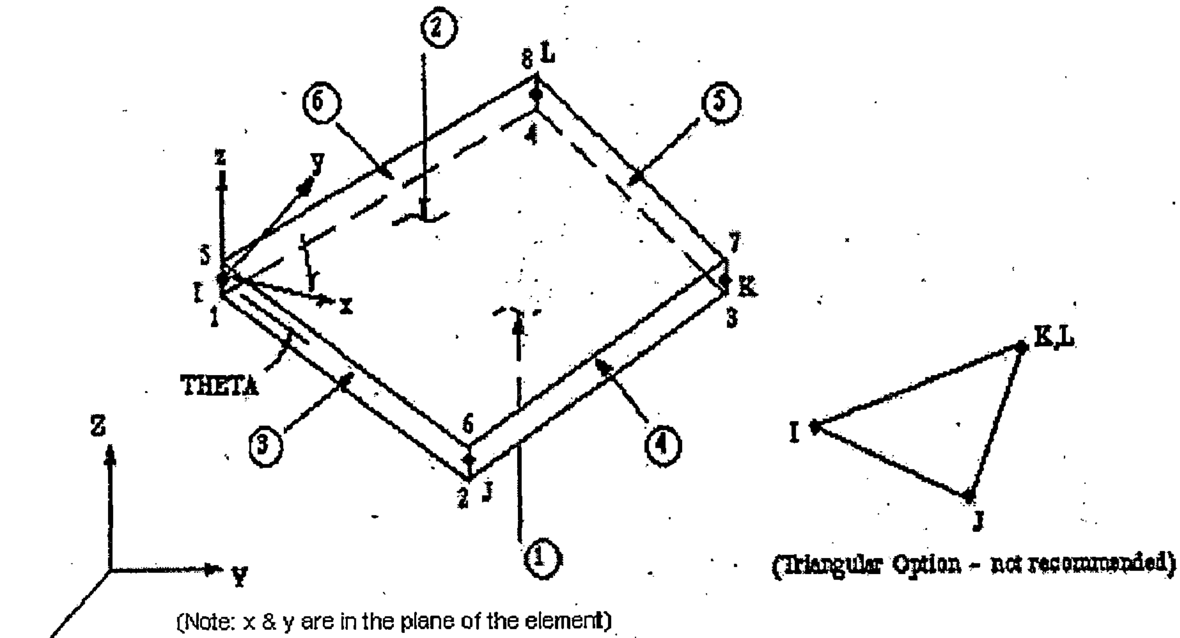


Fig: 4.4 SHELL43 Plastic Large Strain Shell.

SHELL91 '16-Layer Structural Shell' as shown in Fig.4.5 may be used for layered applications of a structural shell model or for modeling structural sandwich structures (fig. 4.5). Up to 16 different layers are permitted for applications with the sandwich option turned off. The element has four-nodes with six degrees of freedom at each node: translations in the x, y and z directions and rotations about the x, y and z-axes.

SHELL99 '100-Layer Structural Shell' may be used for layered applications of a structural shell model. This element is an extension of the 16-layer SHELL91 shell elements as shown in Fig.4.5 in that up to 100 different material layers are permitted. The element has four-nodes with six degrees of freedom at each node: translations in the x, y and z directions and rotations about the x, y and z-axes.

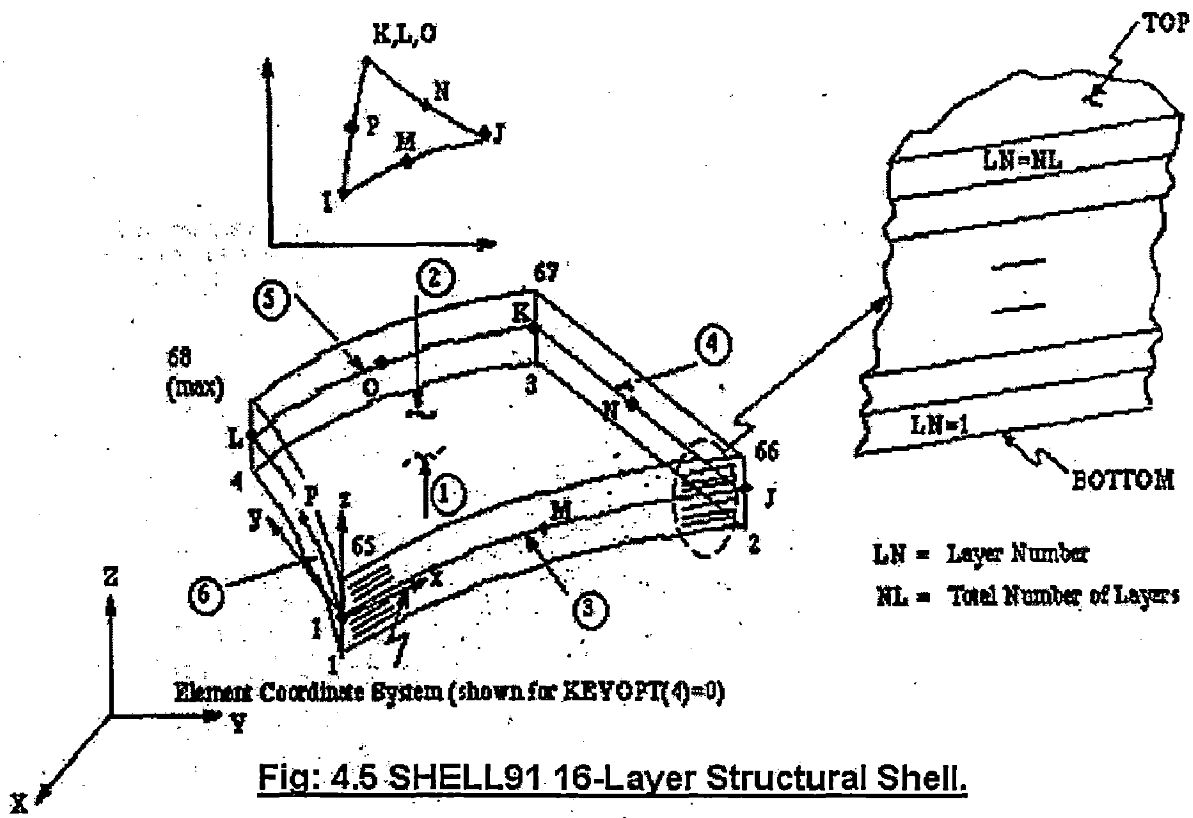


Fig: 4.5 SHELL91 16-Layer Structural Shell.

ASSESSMENT: From the above briefs it is concluded that for vertical lift gate's skin plate (flat shell), SHELL63 'Elastic Shell' element is a suitable element type.

4.2.2 Element For Girders/Beams

In three-dimensional analysis for girders/beams three-dimensional BEAM element must be used. Various three-dimensional beam elements are available in ANSYS. They are:

i. Elastic 4;

ii. Plastic 24;

iii. Tapered 44;

In brief capabilities and properties of these BEAM elements are mentioned below:

BEAM4 'three-dimensional Elastic Beam' as shown in Fig.4.6 is a uniaxial element with tension, compression, torsion and bending capabilities. The element has six degrees of freedom at each node: translations in the nodal x, y and z directions and rotations about the nodal x, y and z-axes. Stress stiffening and large deflection capabilities are included. A consistent tangent stiffness matrix option is available for use in large deflection analyses.

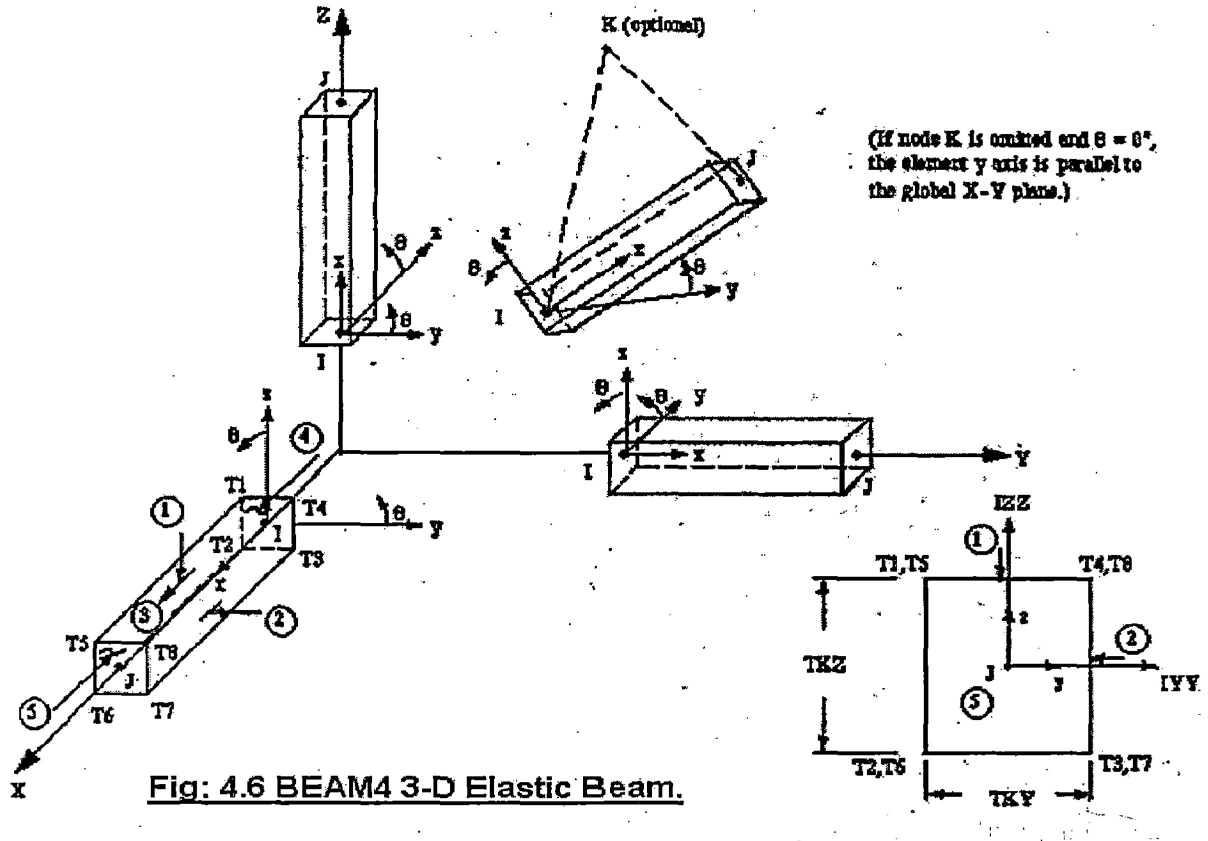


Fig: 4.6 BEAM4 3-D Elastic Beam.

BEAM24 'three-dimensional Thin-Walled Beam' as shown in Fig.4.7 is a uniaxial element of arbitrary cross-section (open or closed) with tension-compression, bending and St. Venant torsional capabilities. Any open cross section or single-celled closed section can be used. The element has six degrees of freedom at each node: translations in the nodal x, y and z directions and rotations about the nodal x, y and z-axes. The element has plastic, creep, and swelling capabilities in the axial direction as well as a user-defined cross section. If these capabilities are not needed, the elastic beams BEAM4 or BEAM44 may be used.

BEAM44 'three-dimensional Tapered Unsymmetric Beam' as shown in Fig.4.8 is a uniaxial element with tension, compression, torsion and bending capabilities. The element has six degrees of freedom at each node: translations in the nodal x, y and z directions and rotations about the nodal x, y and z-axes. This element allows a different unsymmetrical geometry at each end and permits the end nodes to be offset from the centroidal axis of the beam. If these features are not desired, the uniform symmetrical beam BEAM4 may be used. The effect of shear deformation is available as an option. Another option is available for printing the forces acting on the element in the element coordinate directions. Stress stiffening and large deflection capabilities are also included.

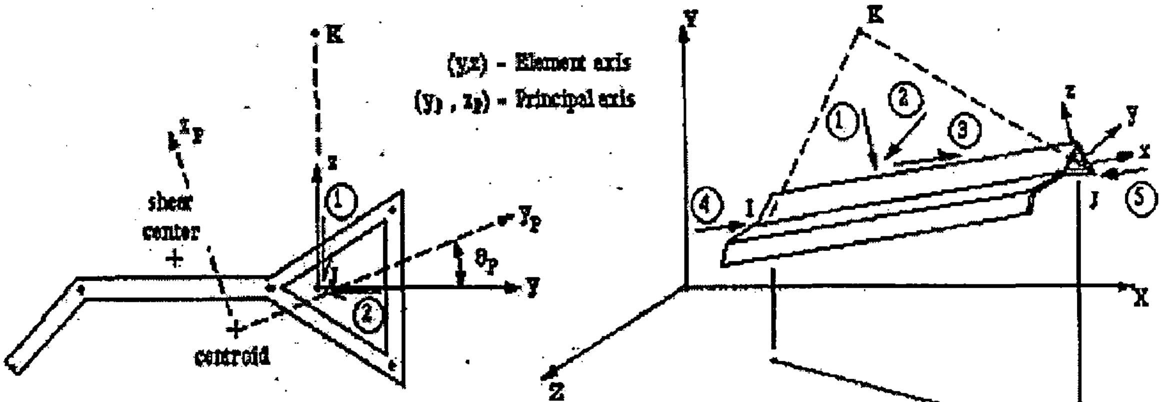


Fig: 4.7 BEAM24 3-D Thin-walled Beam.

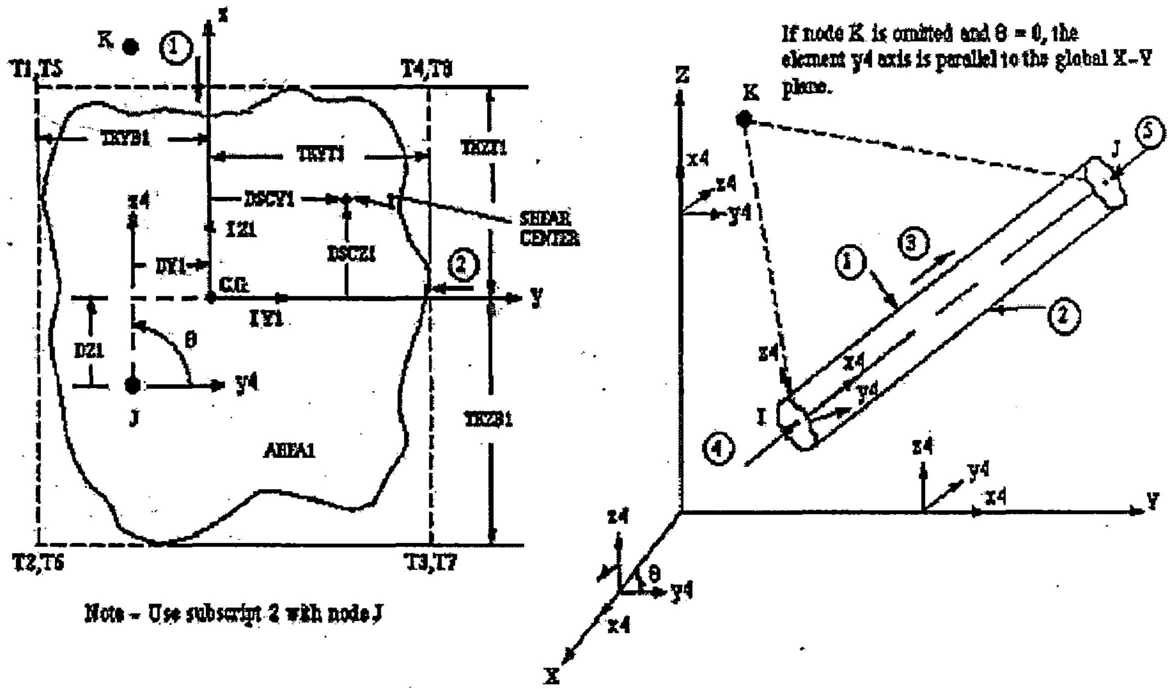


Fig: 4.8 BEAM44 3-D Tapered Unsymmetric Beam.

ASSESSMENT: From the above briefs it is obvious that Girder/Beam components like horizontal girders, vertical end girders, vertical stiffeners of the vertical lift gate under study posses unsymmetrical sections as shown in chapter-3, and therefore, BEAM44 'three-dimensional Tapered Unsymmetric Beam' element will be the most appropriate for modeling them.

NOTE: It is noted here that gate to be meshed with two different type of elements and their DOFs at the common nodes must be same. Both the selected SHELL63 and BEAM44 elements have same DOFs, which fulfill the required criteria.

4.2.3 Element For Wheel

In three-dimensional solid model analysis for wheel three-dimensional SOLID element must be used. A variety of three-dimensional SOLID elements are available in ANSYS. They are:

i. Brick 8 node 45;

ii. Brick 20 node 95;

iii. Tet 10 node 92;

Capabilities and properties of these three-dimensional SOLID elements are mentioned below:

SOLID45 'three-dimensional 8-Node Structural Solid' as shown in Fig.4.9 is used for the three-dimensional modeling of solid structures. The element is defined by eight-nodes having three degrees of freedom at each node: translations in the nodal x, y and z directions. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

SOLID95 'three-dimensional 20-Node Structural Solid' as shown in Fig.4.10 is a higher order version of the SOLID45. It can tolerate irregular shapes without as much loss of accuracy. This element has compatible displacement shapes and is well suited to model curved boundaries. The element is defined by 20-nodes having three degrees of freedom at each node: translations in the nodal x, y and z directions. The element may have any spatial orientation. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. Various printout options are also available.

SOLID92 'three-dimensional 10-Node Tetrahedral Structural Solid' as shown in Fig.4.11 is a quadratic displacement behavior and is well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element is defined by 10-nodes having three degrees of freedom at each node: translations in the nodal x, y and z directions. The element also has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. ANSYS has free mesh option with this element.

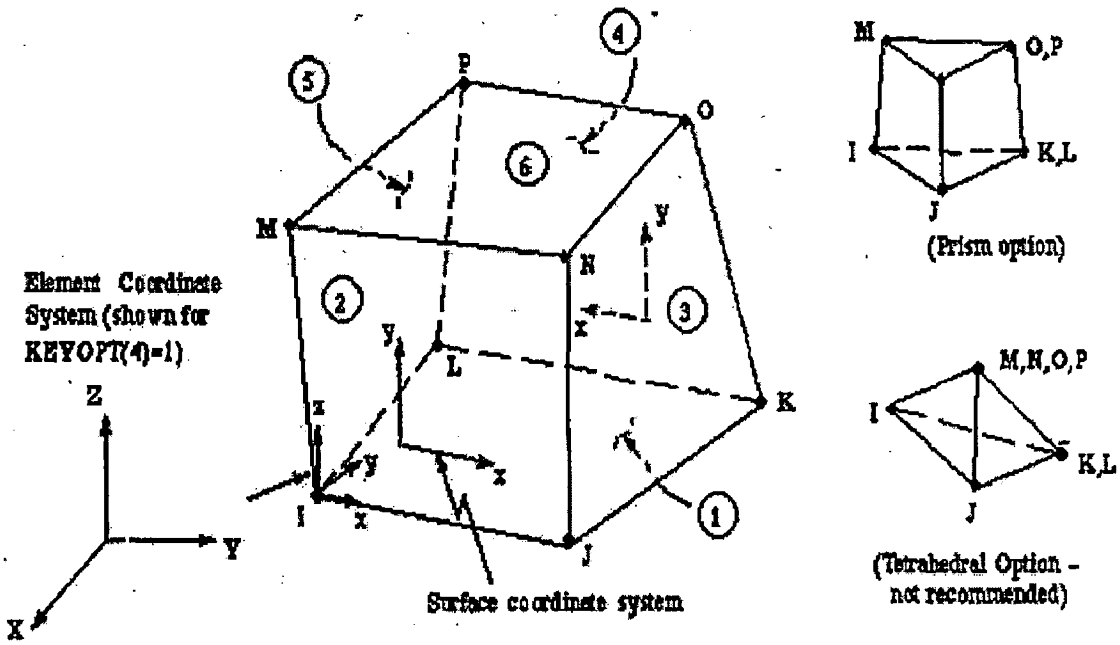


Fig: 4.9 SOLID45 3-D Structural Solid.

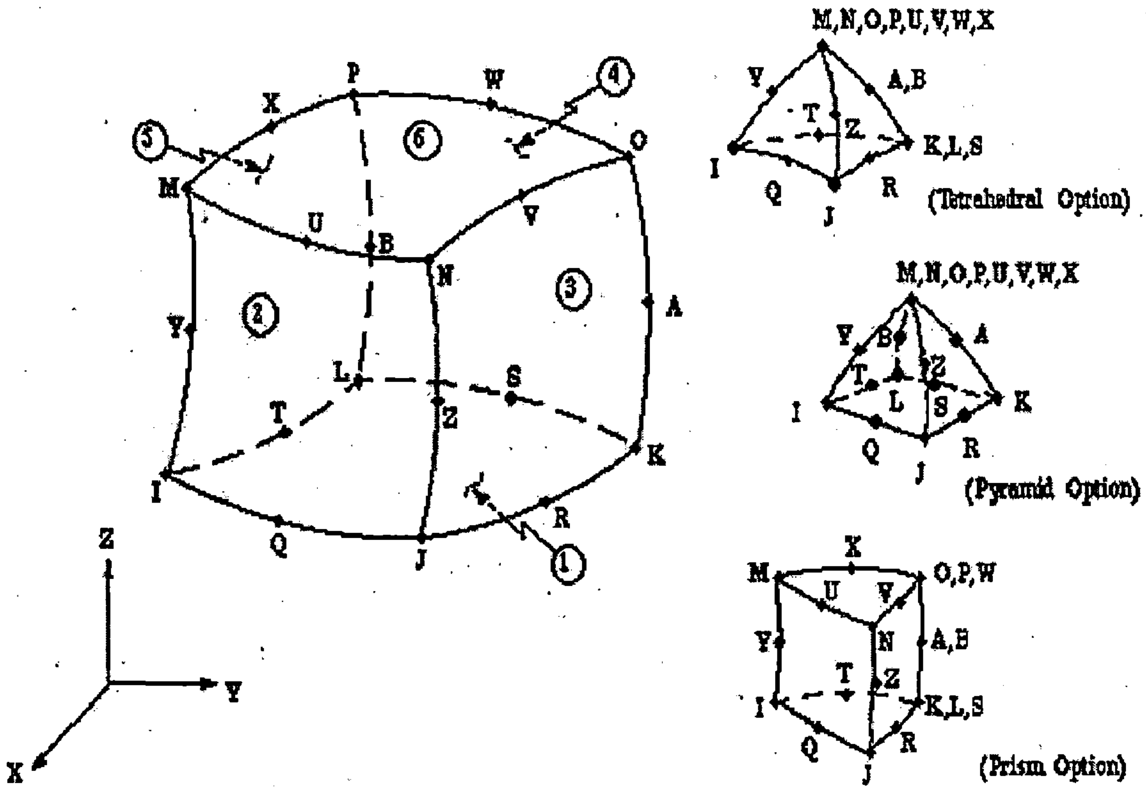


Fig: 4.10 SOLID95 3-D 20-Node Structural Solid.

Assessment: From the above briefs it is clear that Solid92 'three-dimensional 10-Node Tetrahedral Structural Solid' element will be most suitable for wheel model.

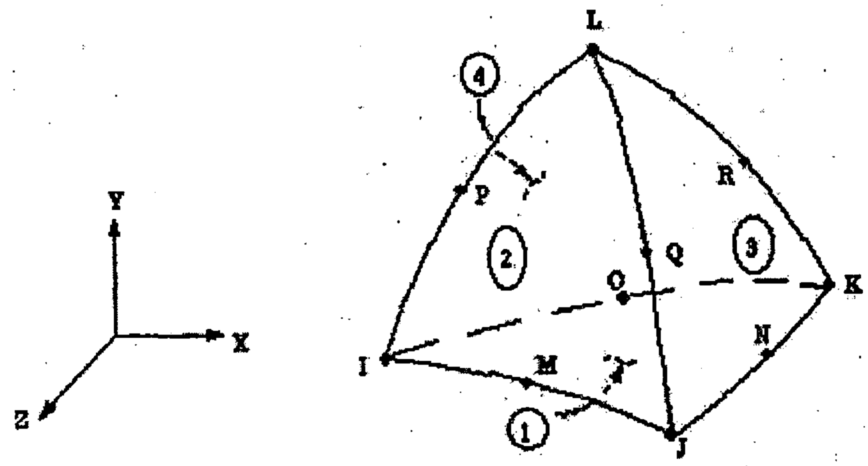


Fig: 4.11 SOLID92 3-D 10-Node Tetrahedral Structural Solid.

4.3 PARTICULARS OF THE SELECTED ELEMENTS

Particulars of the selected elements SHELL63, BEAM44 and SOLID92 are described below:

4.3.1 SHELL63 Three-Dimensional Elastic Shell

SHELL63 has been selected for meshing of skin plate of the fixed wheel vertical lift gate. The element is defined by four nodes I, J, K and L having six DOFs at each node as already mentioned above.

4.3.1.1 Input data for SHELL63

The geometry, node locations, and the coordinate system for this element are shown in Fig: 4.1. The element is defined by four nodes, four thicknesses, an elastic foundation stiffness, and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is used for orthotropic material input directions, applied pressure directions, and under some circumstances, stress output directions. For this element, SHELL63 the default orientation generally has x-axis aligned with element i-j side, the z axis is normal to the shell surface and the y axis perpendicular to the x and z axes.

Real Constant: TK(I), TK(J), TK(K), EFS, THETA ... ...;

The thickness is assumed to vary smoothly over the area of the element, with the thickness input at the four nodes. If the element has a constant thickness, only TK(I) need to be input. If the thickness is not constant, all four thicknesses must be input. The elastic foundation stiffness (EFS) is defined, as the pressure required producing a unit normal deflection of the foundation. The EFS capability is bypassed if EFS is less than, or equal to, zero.

Material Properties: EX, EY, ... NUXY, DENS, GXY ..;
For isotropic material, only EX, NUXY, DENS are required to input.

Surface Loads: pressure: face 1 (I-J-K-L) (bottom, in +Z direction), face 2 (I-J-K-L) (top, in – Z direction), face 3 (J-I), face 4 (K-J), face 5 (L-K) and face 6 (I-L).

4.3.1.2 Output data for SHELL63

The solution output associated with the element is in two forms: 1) nodal displacements included in the overall nodal solution, and 2) additional element output as shown in Fig: 4.12; stress output SX (Top), SX(Bottom), SY(Top), SY(Bottom) and moments output.

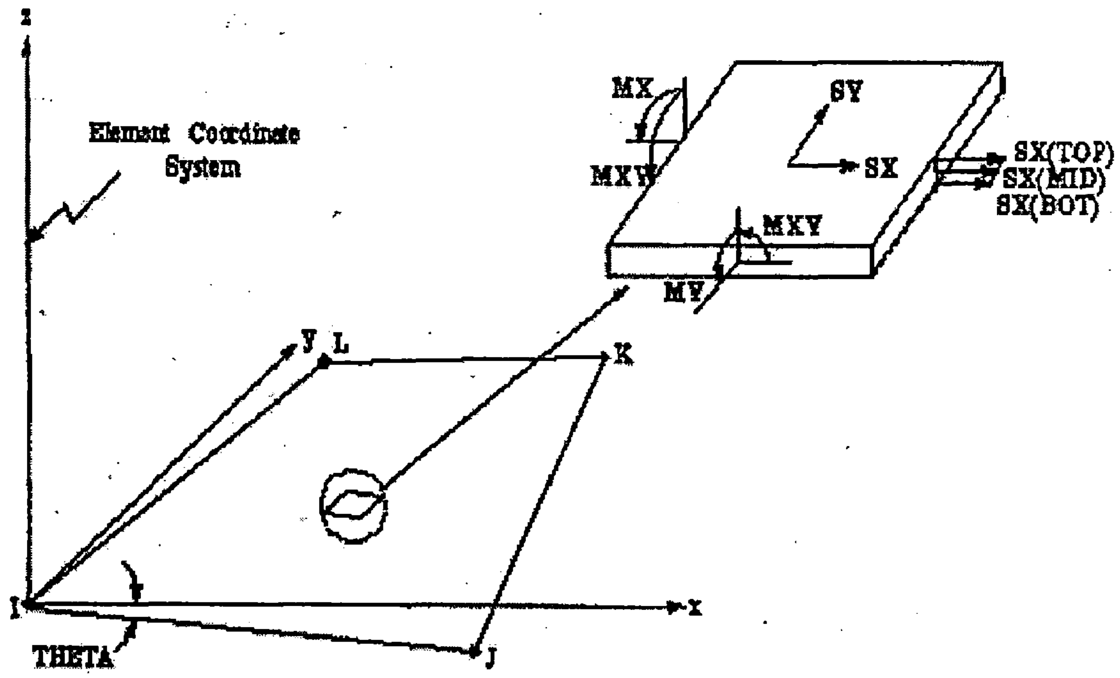


Fig: 4.12 SHELL63 Stress Output.

4.3.1.3 Assumptions and restrictions for SHELL63

Zero area elements are not allowed. This occurs most often whenever the elements are not numbered properly. Zero thickness elements or elements tapering down to a zero thickness at any corner are not allowed. An assemblage flat shell element can produce a good approximation to a curved shell surface provided that each flat element does not extend over more than a 15^0 arc. Shear deflection in not included in this thin-shell element. The four nodes defining the element should lie in an exact flat plane; however, a small out-of-plane tolerance is permitted so that the element may have a slightly warped shape. If the warped is too severe, a fatal massage results and a triangular element should be used.

4.3.1.4 Shape function for three-dimensional 4-node quadrilateral shells

These shape functions are for three-dimensional 4-node quadrilateral shell elements with rotational degrees of freedom (RDOFs) but without shear deflection and without extra shape functions, such as SHELL63 with KEYOPT(3)=1 when used as a quadrilateral:

$$u = \frac{1}{4}\left(u_1(1-s)(1-t) + u_j(1+s)(1-t) + u_K(1+s)(1+t) + u_L(1-s)(1+t)\right) \quad \dots\dots\dots\dots\dots \quad (4.1)$$

$$\mathrm{v} = \frac{1}{4}(\mathrm{v}_1(1-\mathrm{s})....(\text{analogous to u}) \quad \dots\dots\dots\dots\dots\dots\dots\dots \quad (4.2)$$

w = not explicitly defined. Four overlaid triangles (IJK, JKL, KLI, and LIJ) are defined as DKT elements (Batoz (56), Razzaque (57)).------ 12.5.3-3

The shape function is as per Fig: 4.13.

4.3.1.5 Stiffness matrix for three-dimensional 4-node quadrilateral shells

The stiffness matrix for membrane stress having quadrilateral geometry, the shape function equations (4.1) and (4.2) are used with points of integration 2x2. For bending four triangles that are overlaid are used. Their sub triangles are referred to equation 12.5.3-3 (ANSYS Theory Ref.) and points of integration 3 (for each triangle).

4.3.1.6 Numerical integration for 3-D 4-node quadrilateral shells

The numerical integration that ANSYS uses is given below.

For 4-Noded Shell Elements (2x2 or 3x3)

The numerical integration of shell element is given by

$$\int_{-1}^{1}\int_{-1}^{1} f(x,y)\,dxdy = \sum_{j=1}^{m}\sum_{i=1}^{l} H_j H_i f(x_i, y_j) \quad \dots\dots\dots\dots\dots\dots\dots\dots \quad (4.3)$$

where,

f(xy) = function to be integrated;

Hi, Hj = weighing factors and

1, m = number of integration (Gaussian Points).

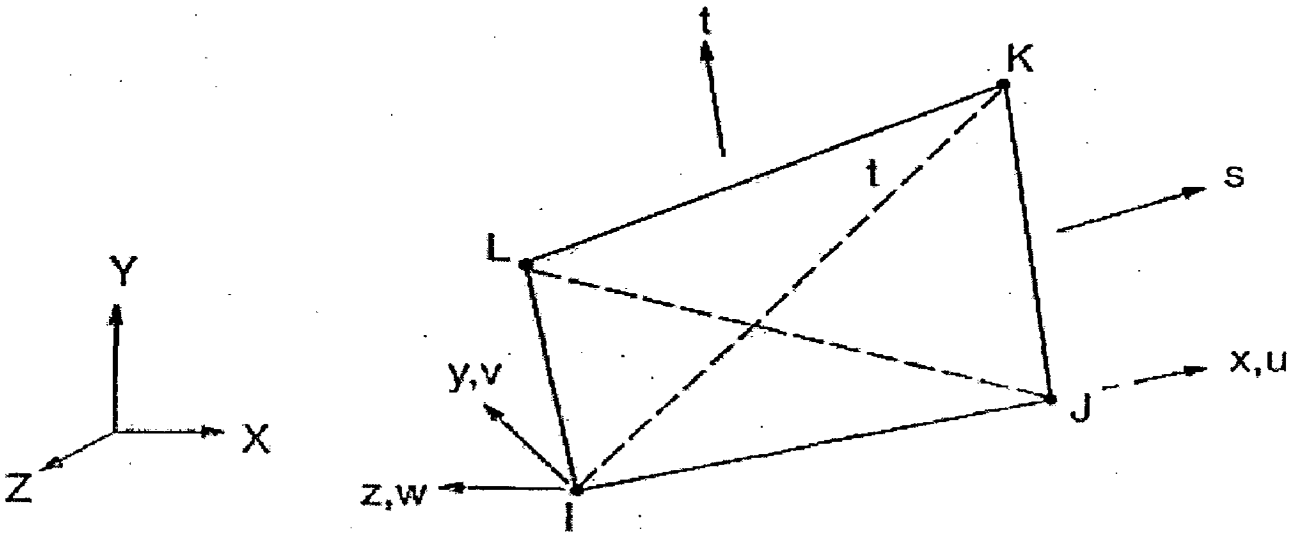


Fig: 4.13 SHELL63 Shape Function.

Table: 4.1 Gauss integration constants:

No. of Int. Points	Integration Point Locations (xi)	Weighting Factor (Hi)
1	0.00000 00000 00000	2.00000 00000 00000
2	± 0.57735 02691 89626	1.00000 00000 00000
3	± 0.77459 66692 41483 0.00000 00000 00000	0.55555 55555 55556 0.88888 88888 88889

4.3.2 BEAM44 Three-Dimensional Tapered Unsymmetric Beam

BEAM44 has been selected for meshing of horizontal girders, vertical end girders and vertical stiffeners of the fixed wheel vertical lift gate. It is a uniaxial element with tension, compression, torsion and bending capabilities. The element is defined by two or three nodes I, J and K(optional) and it allows a different unsymmetrical geometry at each node. The element has six degrees of freedom at each node: other properties and capabilities are already mentioned at the time of element selection.

4.3.2.1 Input data for BEAM44

The geometry, node locations, and coordinate system for this element are shown in fig: 4.8. The element is located by a reference coordinate system (X,Y,Z) and offsets. The reference system is defined by nodes I, J and K(optional) or an orientation angle. The principal axes of the beam are in the element coordinate system (x.y,z) with x along the cross section centroid (C.G.). The element x-axis is oriented from node I (end 1) toward node J (end 2). For the two-node option, the default ($\theta = 0^0$) orientation of the element y-axis is automatically calculated to be parallel to the global X-Y plane.

Real Constants: at node I: AREA1, IZ1, IY1, TKZB1, TKYB1,.... TKZT1, TKYT1; and at node-J: AREA2, IZ2, IY2, TKZB2, TKYB2, ..., TKZT2, TKYT2 .. (basic set).
The element real constants describe the beam in terms of the cross-sectional area, the area moments of inertia, the extreme fibre distances from the centroid, the centroid offset, and the shear constants. If any values at end J are blank, default to the corresponding end I.

Material Properties: EX, ... DENS, GXY, NUXY, ...

Surface Loads: Pressures on face 1 (I-J) (-Z normal direction), face 2 (I-J) (-Y normal direction), face 3 (I-J) (+X tangential direction), face 4 (I) (+X axial direction), face 5 (J) (-X axial direction) and negative value for opposite loading.

4.3.2.2 Output data for BEAM44

The solution output associated with the element is in two forms: 1) nodal displacements included in the overall nodal solutions, and 2) additional element output as shown in table 4.2 (element output definitions). Several items are illustrated in Fig: 4.14. At each gross-section, the computed output consists of the direct (axial) stress and four bending components. Then these five values are combined to evaluate maximum and minimum stresses, assuming a rectangular cross-section.

UX, UY and UZ are deflections in X, Y and Z directions respectively. In the present analysis deflection in Z direction is important and a matter of concern.

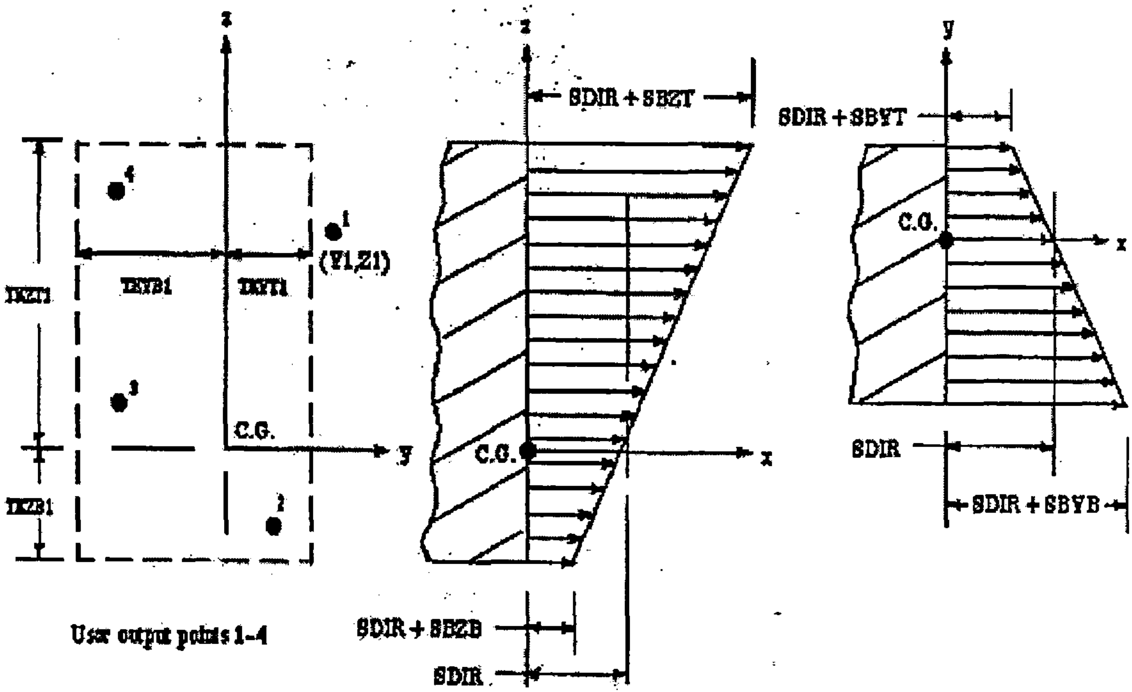


Fig: 4.14 BEAM44 Stress Output.

Table: 4.2 Beam44 Element Output Definitions (partial);

Name	Definition	O	R
SDIR	Axial direct stress	1	1
SBYT	Bending stress on the element +Y side of the beam	1	1
SBYB	Bending stress on the element -Y side of the beam	1	1
SBZT	Bending stress on the element +Z side of the beam	1	1
SBZB	Bending stress on the element -Z side of the beam	1	1
SMAX	Maximum stress (direct stress + bending stress)	1	1
SMIN	Minimum stress (direct stress - bending stress)	1	1
MFOR(X,Y,Z)	Member forces in the element coordinate system x, y, z directions.	1	1
MMOM(X,Y,Z)	Member moments in the element coordinate system x, y, z directions.	1	1

For element definition table results, we have to define element table with different key options item and sequence nos.

Table: 4.3 Beam44 (KYEOPT(9)=0) Item and sequence numbers for the ETABLE and ESOL Commands:

KYEOPT(9)=0				
Name	Item	E	I	J
SDIR	LS		1	6
SBYT	LS		2	7
SBYB	LS		3	8
SBZT	LS		4	9
SBZB	LS		5	10
SMAX	NMISC		1	3
SMIN	NMISC		2	4
MFORX	SMISC		1	7
MFORY	SMISC		2	8
MFORZ	SMISC		3	9
MMOMX	SMISC		4	10
MMOMY	SMISC		5	11
MMOMZ	SMISC		6	12
SXY	SMISC		13	16

4.3.2.3 Assumptions and restrictions for beam elements

The beam must not have a zero length or zero area. The beam can have any cross-sectional shape for which the moments of inertia can be computed. The element thicknesses are used in locating the extreme fibers for the stress calculations. Tapers within an element, if any, should be gradual. If AREA2/AREA1 or I2/I1 is not between 0.1 and 10.0, a warning message is printed. The element should not taper to a point (zero thickness). The shear stresses are calculated independently of the shear deflection.

(i) Cross-sectional area is averaged based on the areas by:

$$A_{AV} = \left(A_1 + \sqrt{A_1 A_2} + A_2\right)/3 \text{ ;}$$

(ii) Moments of inertia are averaged based on the end moments of inertia by:

$$I_{AV} = \left(I_1 + \sqrt[4]{I_1^3 I_2} + \sqrt{I_1 I_2} + \sqrt[4]{I_1 I_2^3} + I_2\right)/5$$

4.3.2.4 Shape functions for beam elements

The shape functions as shown in Fig.4.15 for three-dimensional line elements with RDOFs, such as BEAM44 are:

$$u = \frac{1}{2}\left(u_I(1-s) + u_J(1+s)\right) \quad \text{(4.4)}$$

$$v = \frac{1}{2}\left(v_I\left(1-\frac{s}{2}(3-s^2)\right) + v_J\left(1+\frac{s}{2}(3-s^2)\right)\right)$$
$$+\frac{L}{8}\left(\theta_{z,I}(1-s^2)(1-s) - \theta_{z,J}(1-s^2)(1+s)\right) \quad \text{(4.5)}$$

$$w = \frac{1}{2}\left(w_I\left(1-\frac{s}{2}(3-s^2)\right) + w_J\left(1+\frac{s}{2}(3-s^2)\right)\right)$$
$$-\frac{L}{8}\left(\theta_{y,I}(1-s^2)(1-s) - \theta_{y,J}(1-s^2)(1+s)\right) \quad \text{(4.6)}$$

$$\theta_x = \frac{1}{2}\left(\theta_{x,I}(1-s) + \theta_{x,J}(1+s)\right) \quad \text{(4.7)}$$

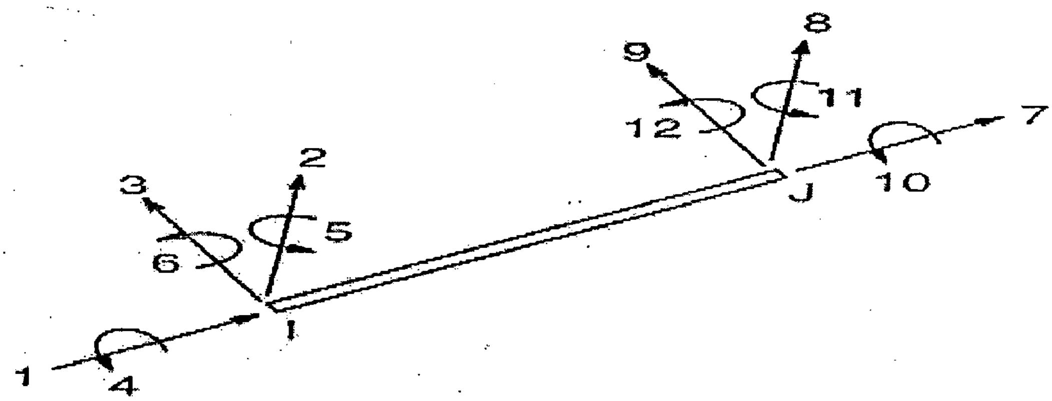


Fig: 4.15 BEAM44 Shape Function.

The stiffness matrix is derived from the shape function equations (4.4), (4.5), (4.6) and (4.7). The stress stiffness matrix is derived using equations (4.5) and (4.6), and Load vector (pressures) using equations (4.4), (4.5) and (4.6).

4.3.2.5 Stiffness matrix for beam elements

The order of degrees of freedom (DOFs) is shown below in matrix form. The stiffness matrix in element coordinates is (Przemieniecki (28)).

$$
[K_l] = \begin{bmatrix}
AE/L & & & & & & & & & & & \\
0 & a_z & & & & & & & & & & \\
0 & 0 & a_y & & & & & & & & & \\
0 & 0 & 0 & GJ/L & & & & & \text{Symmetric} & & & \\
0 & 0 & d_y & 0 & e_y & & & & & & & \\
0 & c_z & 0 & 0 & 0 & e_z & & & & & & \\
-AE/L & 0 & 0 & 0 & 0 & 0 & AE/L & & & & & \\
0 & b_z & 0 & 0 & 0 & d_z & 0 & a_z & & & & \\
0 & 0 & b_y & 0 & c_y & 0 & 0 & 0 & a_y & & & \\
0 & 0 & 0 & -GJ/L & 0 & 0 & 0 & 0 & 0 & GJ/L & & \\
0 & 0 & d_y & 0 & f_y & 0 & 0 & 0 & c_z & 0 & c_y & \\
0 & c_z & 0 & 0 & 0 & f_z & 0 & d_y & 0 & 0 & 0 & c_z
\end{bmatrix}
$$

where,

A = cross-section area (input as AREA on R command)

E = Young's modulus (input as Ex on MP command)

L = element length

G = Shear modulus (input as GXY on MP command)

J = torsional moment of inertia = $\begin{Bmatrix} J_x & \text{if} & I_x & -0 \\ I_x & \text{if} & I_x & \neq 0 \end{Bmatrix}$

I_x = Input as IXX on RMORE command

J_x = polar moment of inertia = $I_y + I_z$

$z = a(I_z, \phi_y)$

$a_y = a(I_y, \phi_z)$

$b_z = b(I_z, \phi_y)$

$f_z = f(I_z, \phi_y)$

$f_y = b(I_y, \phi_z)$

and where; $a(I, \phi) = \dfrac{12EI}{L^3(1+\phi)}$;

$$b(I, \phi) = \frac{-12EI}{L^3(1+\phi)}$$

$$c(I, \phi) = \frac{6EI}{L^2(1+\phi)}$$

$$d(I,\phi)=\frac{-6EI}{L^2(1+\phi)}$$

$$e(I,\phi)=\frac{(4+\phi)EI}{L(1+\phi)}$$

$$f(I,\phi)=\frac{(2-\phi)EI}{L(1+\phi)}$$

and where: $$\phi_y=\frac{12EI_z}{GA_z^sL^2}$$

$$\phi_z=\frac{12EI_y}{GA_y^sL^2}$$

I_i = moment of inertia normal to direction i (input as Iii on r command)

A_i^s = shear area normal to direction $i = A/F_i^s$

F_i^s = shear coefficient (input as SHEARi on RMORE command).

4.3.2.6 Stress calculation for BEAM44

The axial stresses are computed analogously to BEAM4 as below:

Centroidal stress is given by:

$$\sigma_i^{dir} = \frac{F_{x,i}}{A};$$

where, σ_i^{dir} = centroidal stress (output quantity SDIR)

$F_{x,i}$ = axial force (output quantity FX)

The bending stresses are given by:

$$\sigma_{z,i}^{bnd} = \frac{M_{y,i}}{2I_y} t_z$$

$$\sigma_{y,i}^{bnd} = \frac{M_{z,i}}{2I_z} t_y$$

where, $\sigma_{z,i}^{bnd}$ = bending stress in element x direction on the element +z side of the beam at end I (output quantity SBZ);

$\sigma_{y,i}^{bnd}$ = bending stress in element y direction on the element -y side of the beam at end I (output quantity SBY);

$M_{y,i}$ = moment about the element y axis at end i

$M_{z,i}$ = moment about the element z axis at end i

t_z = thickness of beam in element z direction (input as TKZ on R command)

t_y = thickness of beam in element y direction (input as TKY on R command)

The maximum stress at cross-section **i** is computed by:

$$\sigma_i^{\max} = \text{maximum\_of}\begin{cases} \sigma_i^{dir} + \sigma_{zt,i}^{bnd} + \sigma_{yt,i}^{bnd} \\ \sigma_i^{dir} + \sigma_{zt,i}^{bnd} + \sigma_{yb,i}^{bnd} \\ \sigma_i^{dir} + \sigma_{zb,i}^{bnd} + \sigma_{yb,i}^{bnd} \\ \sigma_i^{dir} + \sigma_{zb,i}^{bnd} + \sigma_{yt,i}^{bnd} \end{cases}$$

where,

$$\sigma^{dir} = output\_quantity\_SDIR$$

$$\sigma_{yt}^{bnd} = output\_quantity\_SBYT$$

$$\sigma_{yb}^{bnd} = output\_quantity\_SBYB$$

$$\sigma_{zt}^{bnd} = output\_quantity\_SBZT$$

$$\sigma_{zb}^{bnd} = output\_quantity\_SBZB$$

The minimum stress is analogously defined.

The assumption has been made that the cross-section is a rectangle, so that maximum and minimum stresses occur at the extreme fibers.

4.3.3 SOLID92 Three-Dimensional 10-Node Tetrahedral Structural Solid

SOLID92 has been selected for meshing of **wheel** of the fixed wheel vertical lift gate. It has a quadratic displacement behavior and is well suited to mesh irregular shape. The element is defined by ten nodes having three DOFs at each node: translations in the nodal x, y and z directions. The element also has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. (These are already mentioned at the time of element selection.)

4.3.3.1 Input data for SOLID92

The geometry, node locations, and the coordinate system for this element are shown in Fig: 4.11. Beside the nodes, the element input data includes the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is used for orthotropic material input directions, applied pressure directions, and under some circumstances, stress output directions.

Real Constant: None.

Material Properties: EX, EY, ... NUXY, DENS, GXY .. etc.

For isotropic material, only EX, NUXY, DENS are required to input.

Surface Loads: pressure;

4.3.3.2 Output data for SOLID92

The solution output associated with the element is in two forms: 1) nodal displacements included in the overall nodal solution, and 2) additional element output. Several items are illustrated in Fig: 4.16. The element stress directions are parallel to the element coordinate system and surface stress outputs are in the surface coordinate system.

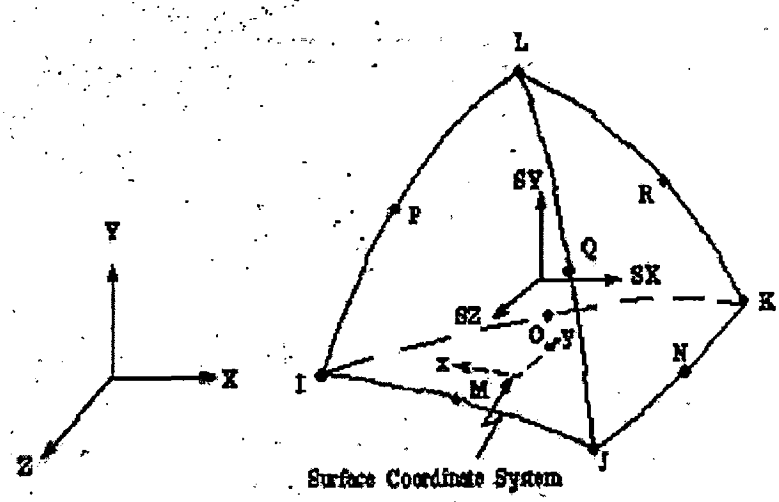


Fig: 4.16 SOLID92 Stress Output.

4.3.3.3 Assumptions and restrictions for SOLID92

The element must not have a zero volume. The element may be numbered either as shown in fig: 4.11 or may have node L below the IJK plane. An edge with a removed mid-side node implies that the displacement varies linearly, rather than parabollically, along the edge.

4.3.3.4 Shape function for SOLID92

These shape functions are for three-dimensional 10-node tetrahedral elements such as SOLID92:

$$u = u_I(2L_1-1)L_1 + u_J(2L_2-1)L_2 + u_K(2L_3-1)L_3 + u_L(2L_4-1)L_4$$
$$+4(u_M L_1 L_2 + u_N L_2 L_3 + u_O L_1 L_3 + u_P L_1 L_4 + u_Q L_2 L_4 + u_R L_3 L_4) \text{----} \quad (4.8)$$

$$v = v_I(2L_1 - 1)L_1 ......(analogoust\_to\_u) \quad \text{---------------------} \quad (4.9)$$

$$w = w_I(2L_1 - 1)L_1 ......(analogoust\_to\_u) \quad \text{---------------------} \quad (4.10)$$

4.3.3.5 Stiffness, stress and pressure load matrices

The Stiffness Matrix, Stress Stiffness Matrix, Mass Matrix are derived using shape function equations (4.8), (4.9) and (4.10) with 4 points of integration, and Pressure Load Vector is also derived using the same equations with 3 points of integration.

4.4 ANSYS PROCEDURE OUTLINE

The ANSYS program has finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, non-linear, transient dynamic analysis in Multiphysics, Mechanical, Structural, Fluid, Thermal, Electro-magnetic etc.

A typical ANSYS analysis has three distinct steps:

i. Building the model;
ii. Applying loads and obtaining the solution, and
iii. Reviewing the results.

i) Building A Model:

Building a finite element model requires more time than any other part of the analysis. This is carried out by the use of the ANSYS **PREP7** preprocessor. It includes:

- Specifying a **Job name** and **Analysis Title;**
- Setting **preference**;
- Defining **element type**(s);
- Defining element **real constants;**
- Defining **material properties** and
- Creating the **model geometry**:
  - describing the geometric shape of the model, and
  - meshing the geometry with nodes and elements.

ii) Applying Loads And Obtaining The Solution:

In this step, the following tasks are carried out by the use of the ANSYS **SOLUTION** processor-

- Defining the **Analysis Type** and **Analysis Options**;
- Applying **Boundary Conditions** i.e. displacement constraints;
- Applying **Loads**;
- Specifying **Load Step Options** (if needed) and
- Initiating the **Solution.**

iii) Reviewing The Results:

In this step, by the use of the ANSYS General Postprocessor we get:

- **Deformed shapes;**
- **Contour display of results;**
- Tabular listings to review and interpret the results of the analysis.

4.4.1 Setting Preference

In ANSYS, different preferences are available for GUI filtering like Structural, Thermal, ANSYS Fluid etc. Preference setting in the present works is **Structural** with Structural discipline options: **h – Method**.

4.4.2 Defining Element Type

In ANSYS, versatile elements are available in two-dimensional and three-dimensional analyses. In the present works, following element types are used:

For gate: –

Skin plate: SHELL63 three-dimensional Elastic Shell;

Girders and stiffeners: BEAM44 three-dimensional Elastic Tapered Unsymmetric Beam;

Wheel: - SOLID92 three-dimensional 10-Node Tetrahedral Structural Solid.

4.4.3 Defining Real Constants

In ANSYS, depending on the element type, real constant sets are prescribed. These consist of different geometrical properties such as cross sectional area, moment of inertia, added mass per unit length, initial strain etc. thickness of element at different nodes etc.

Real Constant Sets used in the present work:

a) Real constant set for **skin plate's SHELL63 element:**

Real constant set No.=1;

Shell thickness at node I, J, K and L, TK(I,J,K,L) = 2 cm;

Dist from mid surface to top and bottom, CTOP and CBOT =1 cm;

b) Real constant set for **central horizontal girder's BEAM44 element:**

The positions of the real constant sets are shown in Fig: 4.17.

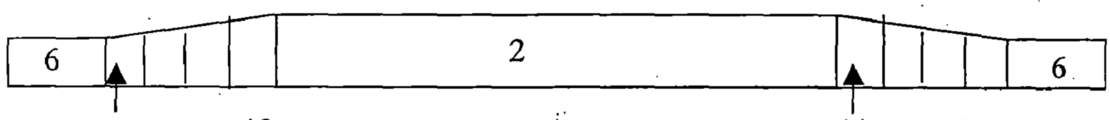


Fig: 4.17 CH Girder RC Set

Table: 4.4 real constant sets (section properties) for **central horizontal girder**:

Set No.	Node - I							Node - J						
	A_1 (cm)	I_{ZZ} (cm⁴)	I_{YY} (cm⁴)	TK_{ZB} (cm)	TK_{YB} (cm)	TK_{ZT} (cm)	TK_{YT} (cm)	A_2 (cm)	I_{ZZ} (cm⁴)	I_{YY} (cm⁴)	TK_{ZB} (cm)	TK_{YB} (cm)	TK_{ZT} (cm)	TK_{YT} (cm)
2	353	7596 4.81	4216 62.0	51.1 6	37.5	33.3 4	37.5	-	-	-	-	-	-	-
6	317	7595 7.13	2102 54.0	38.2 3	37.5	23.7 7	37.5	-	-	-	-	-	-	-
7	317	7595 7.13	2102 54.0	38.2 3	37.5	23.7 7	37.5	326	7595 9.05	2554 09.0	41.5	37.5	26.1 3	37.5
8	326	7595 9.05	2554 09.0	41.5	37.5	26.1 3	37.5	335	7596 0.97	3055 94.0	44.7 4	37.5	28.5 1	37.5
9	335	7596 0.97	3055 94.0	44.7 4	37.5	28.5 1	37.5	344	7596 2.89	3609 61.0	47.9 6	37.5	30.9 2	37.5
10	344	7596 2.89	3609 61.0	47.9 6	37.5	30.9 2	37.5	353	7596 4.81	4216 62.0	51.1 6	37.5	33.3 4	37.5
11	353	7596 4.81	4216 62.0	51.1 6	37.5	33.3 4	37.5	344	7596 2.89	3609 61.0	47.9 6	37.5	30.9 2	37.5
12	344	7596 2.89	3609 61.0	47.9 6	37.5	30.9 2	37.5	335	7596 0.97	3055 94.0	44.7 4	37.5	28.5 1	37.5
13	335	7596 0.97	3055 94.0	44.7 4	37.5	28.5 1	37.5	326	7595 9.05	2554 09.0	41.5	37.5	26.1 3	37.5
14	326	7595 9.05	2554 09.0	41.5	37.5	26.1 3	37.5	317	7595 7.13	2102 54.0	38.2 3	37.5	23.7 7	37.5

c) Real constant set for **top/bottom horizontal girder's BEAM44 element:**

The positions of the real constant sets are shown in Fig: 4.18.

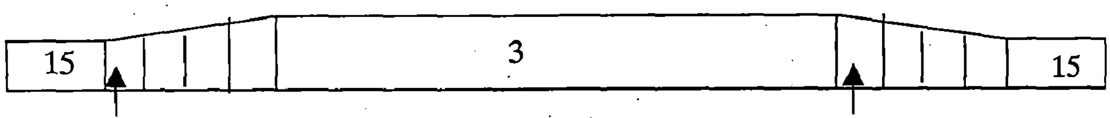


Fig: 4.18 T/B H Girder RC Set

Table: 4.5 real constant sets (section properties) for top/bottom horizontal girders:

Set No.	Node - I							Node - J						
	A_1 (cm)	I_{ZZ} (cm^4)	I_{YY} (cm^4)	TK_{ZB} (cm)	TK_{YB} (cm)	TK_{ZT} (cm)	TK_{YT} (cm)	A_2 (cm)	I_{ZZ} (cm^4)	I_{YY} (cm^4)	TK_{ZB} (cm)	TK_{YB} (cm)	TK_{ZT} (cm)	TK_{YT} (cm)
3	284.5	31002.67	337942.0	49.29	27.5	35.21	27.5	-	- -	-	-	-	-	-
15	253	30997.52	168136.1	36.77	27.5	25.23	27.5	-	-	-	-	-	-	-
18	253	30997.52	168136.1	36.77	27.5	25.23	27.5	260.88	30998.81	204355.3	39.93	27.5	27.69	27.5
19	260.88	30998.81	204355.3	39.93	27.5	27.69	27.5	268.75	31000.1	244643.1	43.07	27.5	30.18	27.5
20	268.75	31000.1	244643.1	43.07	27.5	30.18	27.5	276.63	31001.38	289128.9	46.19	27.5	32.69	27.5
21	276.63	31001.38	289128.9	46.19	27.5	32.69	27.5	284.5	31002.67	337942.0	49.29	27.5	35.21	27.5
22	284.5	31002.67	337942.0	49.29	27.5	35.21	27.5	276.63	31001.38	289128.9	46.19	27.5	32.69	27.5
23	276.63	31001.38	289128.9	46.19	27.5	32.69	27.5	268.75	31000.1	244643.1	43.07	27.5	30.18	27.5
24	268.75	31000.1	244643.1	43.07	27.5	30.18	27.5	260.88	30998.81	204355.3	39.93	27.5	27.69	27.5
25	260.88	30998.81	204355.3	39.93	27.5	27.69	27.5	253	30997.52	168136.1	36.77	27.5	25.23	27.5

d) Real constant set for **vertical end girder's BEAM44 element:**

The positions of the real constant sets are shown in Fig: 4.19.

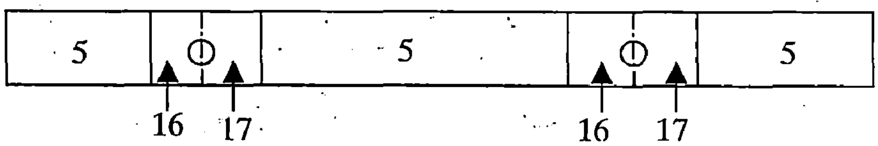


Fig: 4.19 Vertical End Girder RC Set.

Table: 4.6 real constant sets (section properties) for vertical end girder:

Set No.	Node - I							Node - J						
	A_1 (cm)	I_{ZZ} (cm^4)	I_{YY} (cm^4)	TK_{ZB} (cm)	TK_{YB} (cm)	TK_{ZT} (cm)	TK_{YT} (cm)	A_2 (cm)	I_{ZZ} (cm^4)	I_{YY} (cm^4)	TK_{ZB} (cm)	TK_{YB} (cm)	TK_{ZT} (cm)	TK_{YT} (cm)
5	282.5	43459.2	125010.0	24.17	16.5	37.83	16.5	-	-	-	-	-	-	-
16	282.5	43459.2	125010.0	24.17	16.5	37.83	16.5	245.7	36957.52	111900.0	21.58	16.5	40.42	16.5
17	245.7	36957.52	111900.0	21.58	16.5	40.42	16.5	282.5	43459.2	125010.0	24.17	16.5	37.83	16.5

e) Real constant set for **vertical stiffener BEAM44 element:**

Table: 4.7 real constant set (section properties) for vertical stiffener:

Set No.	Node - I							Node - J						
	A_1 (cm)	I_{ZZ} (cm^4)	I_{YY} (cm^4)	TK_{ZB} (cm)	TK_{YB} (cm)	TK_{ZT} (cm)	TK_{YT} (cm)	A_2 (cm)	I_{ZZ} (cm^4)	I_{YY} (cm^4)	TK_{ZB} (cm)	TK_{YB} (cm)	TK_{ZT} (cm)	TK_{YT} (cm)
4	35.5	95.17	1550.7	14.38	4.12	6.62	4.12	-	-	-	-	-	-	-

f) Real constant set for **Wheel SOLID92 element:**

Not required:

4.4.4 Defining Material Properties

In ANSYS, available material properties options are:

Constant:

Isotropic;

Orthotropic;

Temp dependent:

Linear;

Nonlinear;

Defined material properties under study in **Isotropic:**

i. For **gate** made of **Structural Steel:**

Young's modulus, EX = 2.01 x 10^6 kg/cm^2;

Poisson's ratio, NUXY = 0.27;

Density of material, DENS = 0.00785 kg/cm^3.

ii. For **Wheel** made of **Cast Steel IS 1030-1998, Grade C:**

Young's modulus, EX = 2.10 x 10^6 kg/cm^2;

Poisson's ratio, NUXY = 0.27;

Density of material, DENS = 0.00785 kg/cm^3.

4.4.5 Creating The Solid Model Geometry

The ultimate purpose of a finite element analysis is to re-create mathematically the behavior of an actual engineering system. In other words, the analysis must be an accurate mathematical model of a physical prototype. In the broadest sense, this model comprises of all the nodes, elements, material properties, real constants, boundary conditions, and other features that are used to represent the physical system. In ANSYS terminology, Model generation mean the process of defining the geometric configuration of the model's nodes and elements. The ANSYS program has the following approaches for model creation:

a) Creating a solid model within ANSYS;
b) Using direct generation and
c) Importing a model created in a CAD system;

a) Creating a solid model within ANSYS

In this approach, solid model can be created either by Bottom up or Top down techniques.

Bottom up means creation of solid model entities from lowest order to higher order i.e. 1st, create keypoints, then use those keypoints to define lines, then areas by lines, and then volumes by areas.

Top down means generation of model using geometric primitives, which are fully defined lines, areas, and volumes. If a primitive is created, the program automatically creates all the 'lower' entities associated with it.

b) Using direct generation

Direct generation is the approach in which we can define the nodes and elements of a model directly. In some cases, this approach is suitable but usually it is inconvenient since it commonly require about ten times as many data entities to define a model as compared to solid modeling.

c) Creation of solid model geometry in the present study

For the VL Gate and Wheel both the models are created by bottom up approach.

4.4.6 Meshing The Solid Model Geometry

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

i. Setting the element attributes (i.e. element type, material property, real constant (if applicable), element coordinate);
ii. Setting mesh controls (optional). ANSYS offers a large number of mesh control options (default, smart size, manual size) that we can choose from to suit our needs.
iii. Generating the mesh by free or mapped mesh.

Meshing solid model *in the present Study*:

For the **VL Gate**: mesh sizes are controlled by setting number of divisions on the lines manually, and then areas and appropriate lines are free meshed.

For the **Wheel**: setting suitable SmartSize option mesh size is controlled, and then the volume is free meshed.

4.4.7 Applying Loads On The Model

The main goal of a finite element analysis is to examine how a structure responds to certain loading conditions. Specifying the proper loading condition is, therefore, a key step in the analysis. We can apply loads on the model in a variety of ways in the ANSYS program. We can apply loads either on the solid model (on keypoints, lines, areas) or on the finite element model (on nodes and elements).

The word loads as used in ANSYS includes boundary conditions (DOF constraints) as well as other externally and internally applied loads. They are divided into six categories:

- DOF constraints (like UX,.... RX, ...);
- Forces (such as concentrated loads);
- Surface loads (like pressure loads);
- Body loads (such as gravity loads);
- Inertia loads (like inertia due to motion) and
- Coupled field loads.

Applying loads on the model *in the present study*:

a) Boundary conditions for the **VL gate** model:

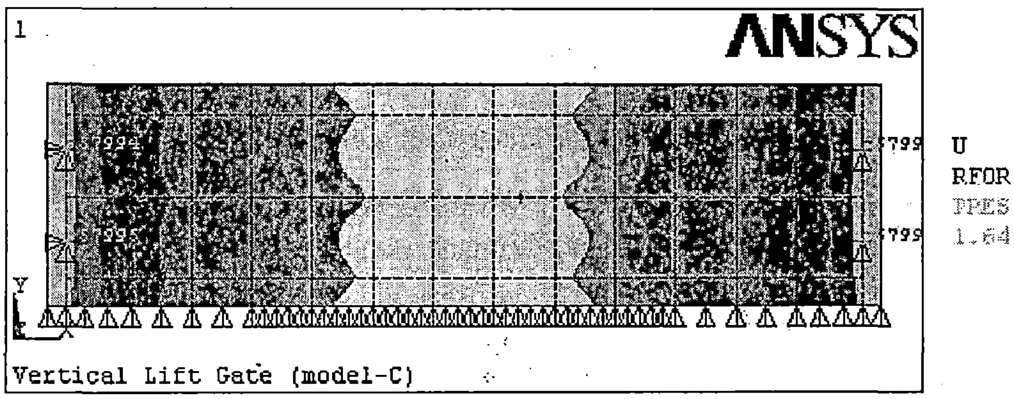


Fig: 4.20 VL Gate Applied BC, Pressure and Reaction Forces.

For two nodes located at the wheel fixation points of the left hand side on the vertical end girder where UX, UY and UZ =0; and two nodes on the other side at the same location UY and UZ = 0; and all the nodes at the bottom of the gate UY = 0; which are shown in the Fig: 4.20.

b) Boundary conditions for the **wheel model:**

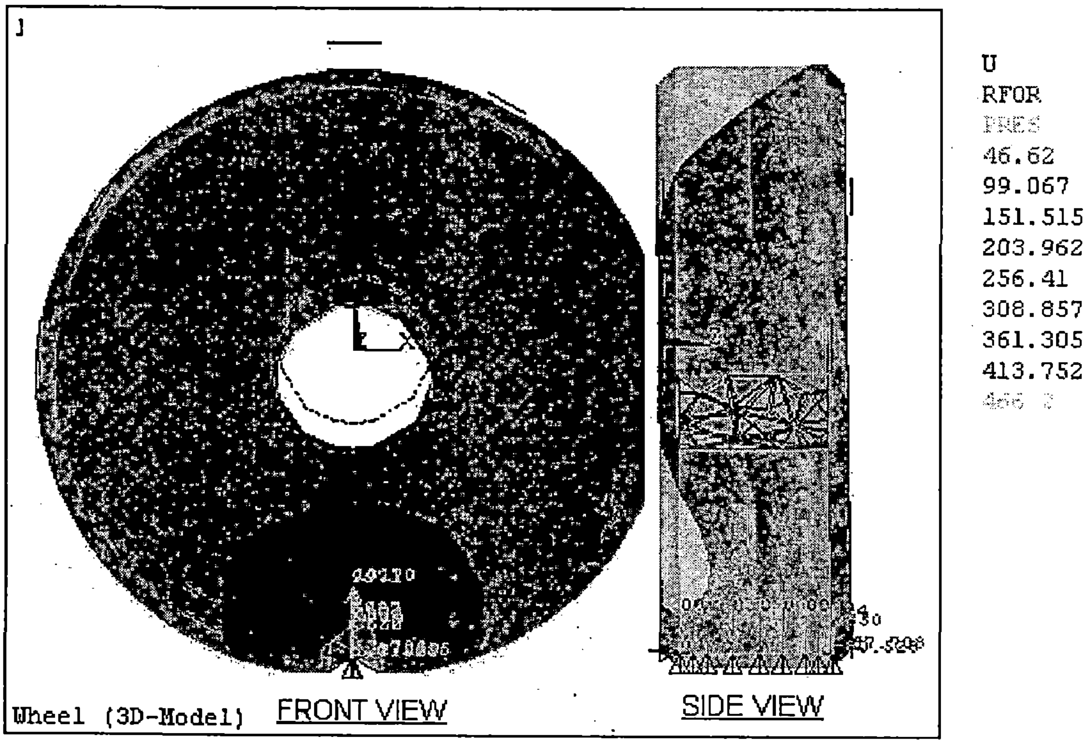


Fig: 4.21 Wheel Applied BC, Pressure and Reaction Forces.

On the bottom contact line, for the most left node UX, UY and UZ =0; for the most right node UX, UY =0; and for other nodes on the same line UY = 0; and for two nodes on chamfering zone UX = 0, which are shown in the Fig: 4.21.

c) Pressure loads on the **VL gate model**:

A constant pressure 1.64 kg/cm^2 is applied on the skin plate's SHELL63 elements (face-2) between the two vertical side seals. The total water loads on the gate comes to (690x205x1.64) 231978 kg. Since variation of pressure is negligibly small and therefore, constant pressure has been used in conventional design. The applied pressure value and reaction forces are seen in the Fig: 4.20.

d) Pressure Loads on the **wheel model:**

Load on each wheel is 57994.5 kg. This load is coming from shaft-bearing on the wheel. It is assumed that this load will act on the bottom half area under the bearing on the wheel hole as a distributed load varying ten times from horizontal middle (P_1=46.62 kg/cm^2) to bottom (P_2=466.17 kg/cm^2) with gradient <-69.93> ('-' sign relates with the coordinate system). The applied pressure values and reaction forces are seen in Fig: 4.21.

4.4.8 Comparison Of Reaction Forces And Applying Loads

To check the validity of the FEM model a comparison of the resultant of reaction forces and applied loads are made.

a) Reaction forces on the **VL gate** model:

ANSYS Listing of reaction forces are given below:

```
PRINT F     REACTION SOLUTIONS PER NODE

  ***** POST1 TOTAL REACTION SOLUTION LISTING *****

  LOAD STEP=      1  SUBSTEP=      1
   TIME=     1.0000      LOAD CASE=    0

  THE FOLLOWING X,Y,Z SOLUTIONS ARE IN GLOBAL COORDINATES

    NODE       FX           FY           FZ
       1                  .00000
       6                  .00000
      17   .00000         .00000       -57995.
      36   .00000         .00000       -57994.
      53                  .00000
       .                    .
             .                   .
    1072                  .00000
    1076                  .00000       -57994.
    1088                  .00000       -57995.
    1108                  .00000

 TOTAL VALUES
 VALUE     .00000       .00000       -.23198E+06
```

The total value of the reaction forces is same as applied total water loads.

b) Reaction forces on the **wheel** model:

ANSYS listing of reaction forces are given below:

```
PRINT F     REACTION SOLUTIONS PER NODE

  ***** POST1 TOTAL REACTION SOLUTION LISTING *****

  THE FOLLOWING X,Y,Z SOLUTIONS ARE IN GLOBAL COORDINATES

    NODE       FX           FY           FZ
      29   -41.426       3601.4
      30    33.725
      31    36.529       3714.8       -.23337E-02
      32   -28.527
     400                 5385.2
```

```
401                 3245.2
402                 5268.5
403                 4193.1
404                 7752.8
405                 4592.4
406                 6942.1
407                 3545.8
408                 5617.9

TOTAL VALUES
VALUE    .30177      53859.      -.23337E-02
```

The total value of the reaction forces is 7.13% less than the applied total loads that is within acceptable limit <= 10%. The comparison indicates that the loads applied on the model have been correctly incorporated.

4.4.9 Checking Of Convergence Of The ANSYS Results

Checking of Convergence of the ANSYS results with coarse to fine meshes is required for confirmation of the ANSYS convergence criteria. For this the gate and wheel models are meshed with three different sizes elements varying from coarse to fine and is distinguished as Model-A, B and C as stated below:

a) Convergence of the **VL gate** model results:

Model-A: Consists of number SHELL63 elements = 102 and BEAM44 elements = 129; Total number of elements = 231; in BHG, max. stress, SBZB = -984.79;

Model-B: Consists of number SHELL63 elements = 372 and BEAM44 elements = 255; Total number of elements = 627; in BHG, max. stress, SBZB = -988.41;

Model-C: Consists of number SHELL63 elements =1066 and BEAM44 elements = 427; Total number of elements =1493; in BHG, max. stress, SBZB= -988.62;

Design stress, f_b = 1019.18 kg/cm^2 (compression).

b) Convergence of the **wheel** model results:

Model-A: Consists of total number of SOLID92 elements = 2530 (by SmartSize 8); And max. stress, SEQV = 8431;

Model-B: Consists of total number of SOLID92 elements = 11464 (by SmartSize 6); And max. stress, SEQV = 6390;

Model-C: Consists of total number of SOLID92 elements = 14663 (by SmartSize 4); And max. stress, SEQV = 6580;

Design contact stress, f_c = 7532.93 kg/cm^2;

From the above it is observed that the ANSYS results are convergent.

VALIDATION OF FEM RESULT

5.1 INTRODUCTION

Essentially two geometrical models are built within ANSYS program; one for Fixed Wheel Vertical Lift Gate and another for Wheel. Then both the models are meshed with three different sizes element for convergence test as already mentioned in chapter-4. Here, for both the objects only the fine mesh 'Model-C's results are represented for analysis and validation since it would give more accurate results, and those would be close to the design results.

The validation of the results depends on professional experience and engineering judgment. When conventional design results are available then we can compare and validate the FEM RESULTS taking these results as a guideline.

At first FEM RESULTS for Vertical Lift Gate's are represented and then the same for Wheel are discussed.

5.2 VALIDATION OF THE RESULTS OF THE VL GATE

First of all, hydraulic loads are applied on the ANSYS model of VL gate. Deflections and stresses in skin plate, horizontal girders, vertical end girder, vertical stiffeners are obtained from FEM analysis are compared with the results of two-dimensional conventional approach.

5.2.1 Loads On The VL Gate

A constant hydrostatic pressure 1.64 kg/cm^2 is applied between the vertical side seals in the +Z direction (i.e. back side of the gate like design case) as seen in Fig: 5.1.

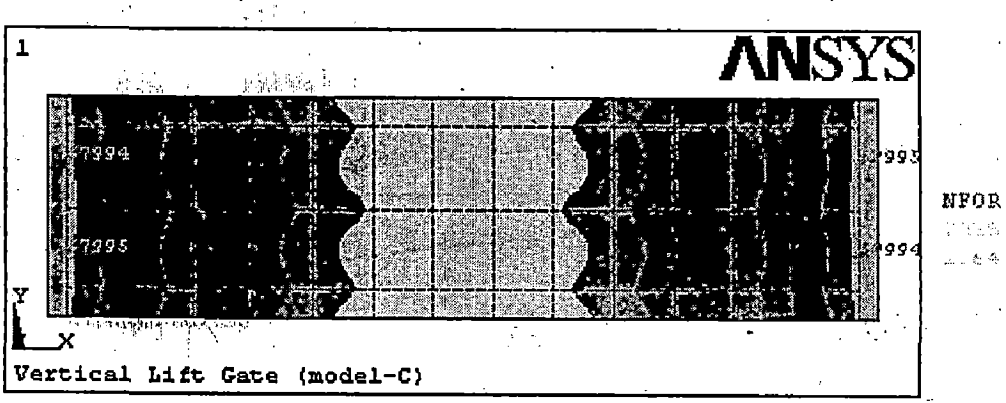


Fig: 5.1 VL Gate Applied Pressure & Reaction Forces.

The reaction forces are found in the –Z direction at the four-wheel fixing points. The reaction forces are obtained, as -57995.0 and -57994.0 are one side, and -57994.0 and -57995.0 on the other side as shown in Fig: 5.1. The total hydrostatic load on the gate is (690x205x1.64 =) 231978 kg (Appendix-A), which is same as ANSYS total reaction force. So, hydrostatic loads acting on the gate is properly accounted in the FEM model.

5.2.2 Skin Plate FEM Result

In conventional approach the skin plate was designed as panels in accordance with procedure and support conditions given in Annex-C of IS: 4622. Maximum stresses and deflection are mentioned in table: 5.1 for ready reference.

Table: 5.1 Maximum stresses in skin plate (ref. Appendix-A).

Location & direction of maximum stress/deflection	Max. Value	Unit
X-direction with V stiffener (bending stress)	529.14	kg/cm^2
Y-direction with H girder (bending stress)	422.70	kg/cm^2
XY-plane (shear stress)	46.13	kg/cm^2
Deflection at the centre of the horizontal girder	0.54	cm
Limited deflection (L/800)	0.90625	Cm

FEM RESULTS (deflection and stresses) are presented in contour plots (nodal & element solutions) as well as graphically in different sections as here after:

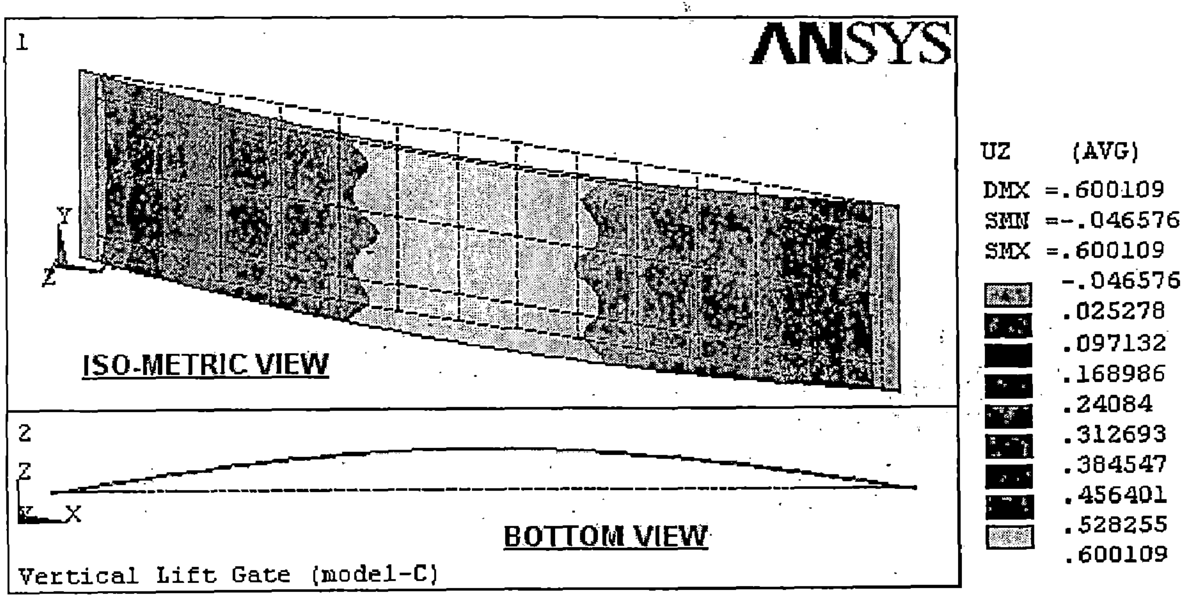


Fig: 5.2 VL Gate Contour Plots Deflections.

It is seen from the Fig: 5.2 and contour legend that deflections at the centre span of the gate vary from 0.528255 to 0.600109 cm.

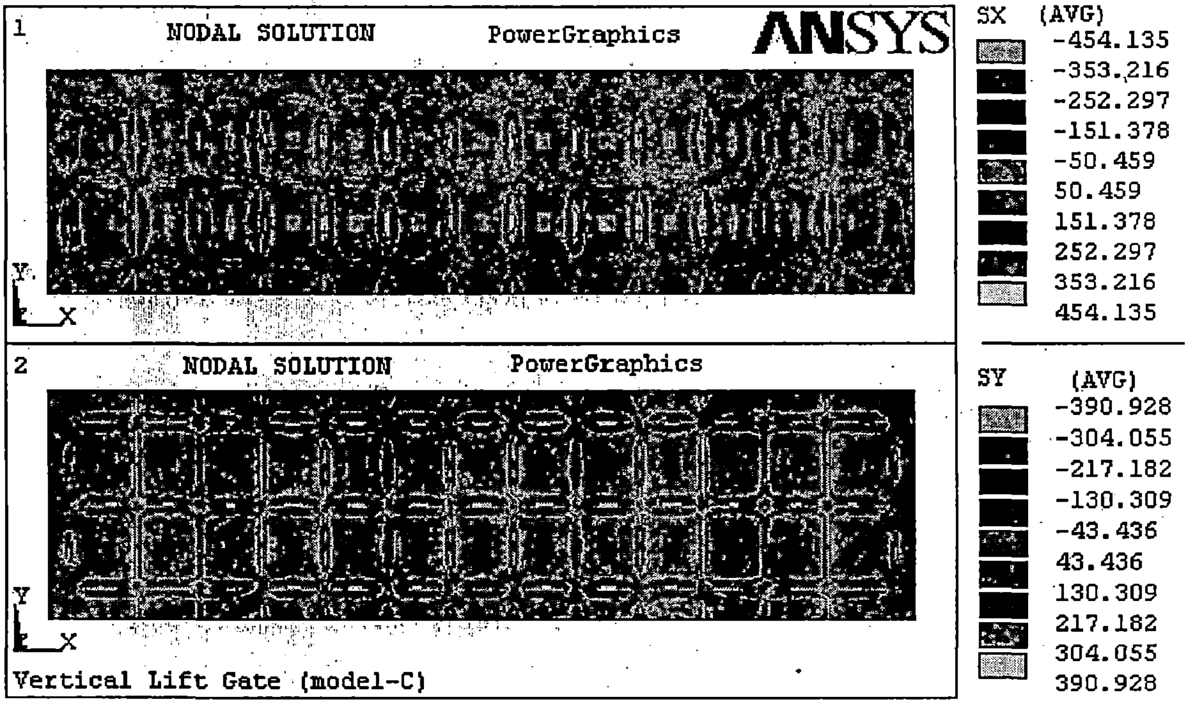


Fig: 5.3 Skin Plate Contour Plots Stresses SX, SY (Nodal Solution).

It is visible from the Fig: 5.3 & contour legends that, maximum stresses in the X-direction (window=1) are varying from +/-353.216 to +/-454.135 kg/cm^2 those are occurring with vertical stiffeners and in Y-direction (window=2) are varying from +/-304.055 to +/-390.928 kg/cm^2 with horizontal girders.

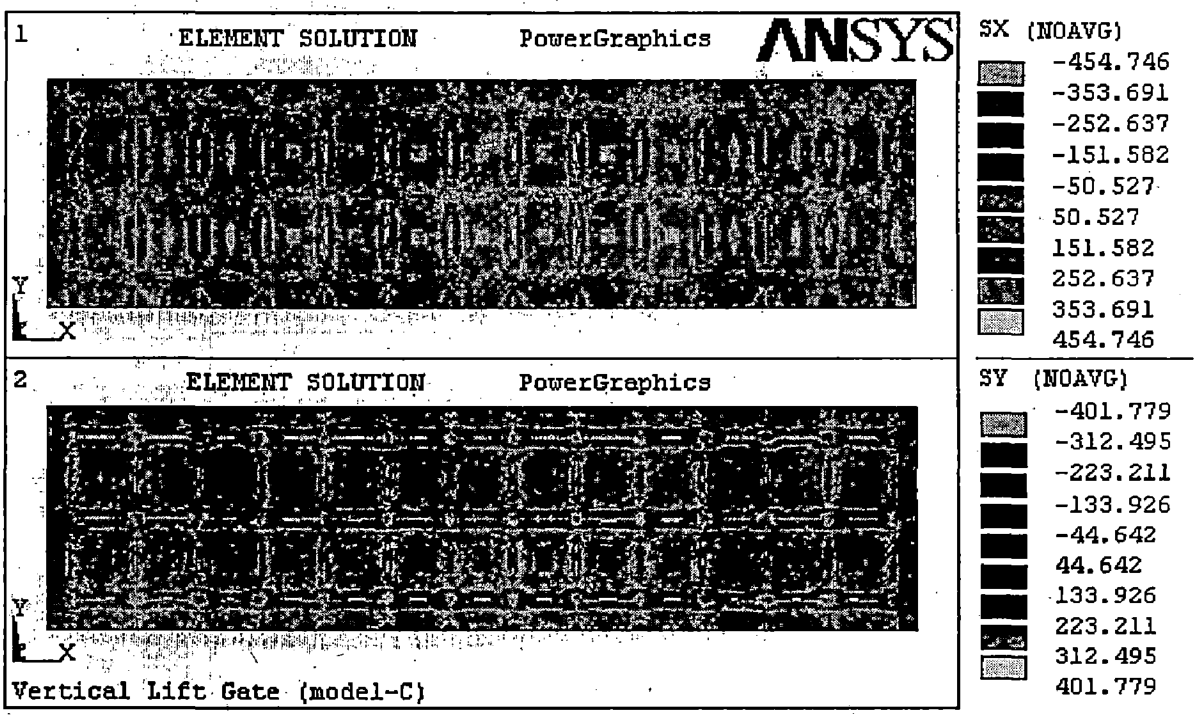


Fig: 5.4 Skin Plate Contour Plots Stresses SX, SY (Element Solution).

It is apparent from the Fig: 5.4 and contour legends that, maximum stresses in the X-direction (window=1) are varying from +/-353.491 to +/-454.746 kg/cm^2 those are happening with vertical stiffeners and in Y-direction (window=2) are varying from +/-312.495 to +/-401.779 kg/cm^2 with horizontal girders.

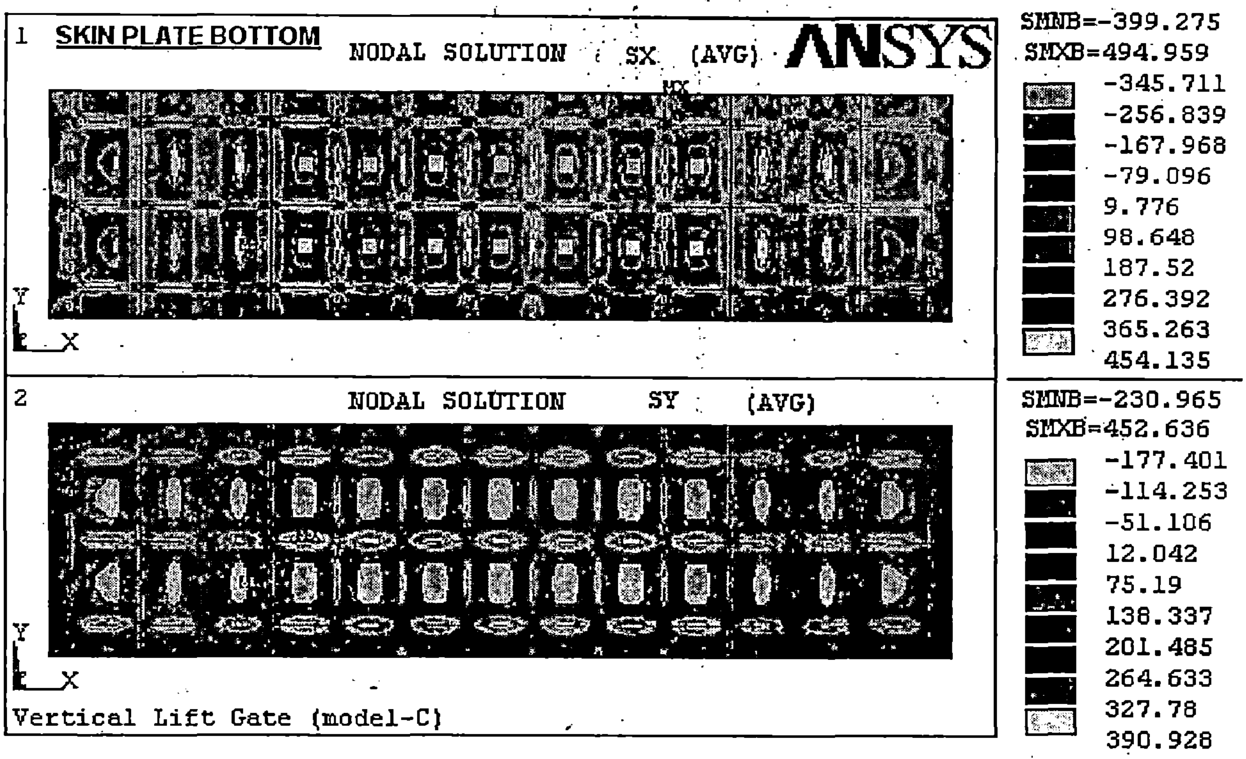


Fig: 5.5 Skin Plate Contour Plots Stresses SX, SY with Error Estimation (NS).

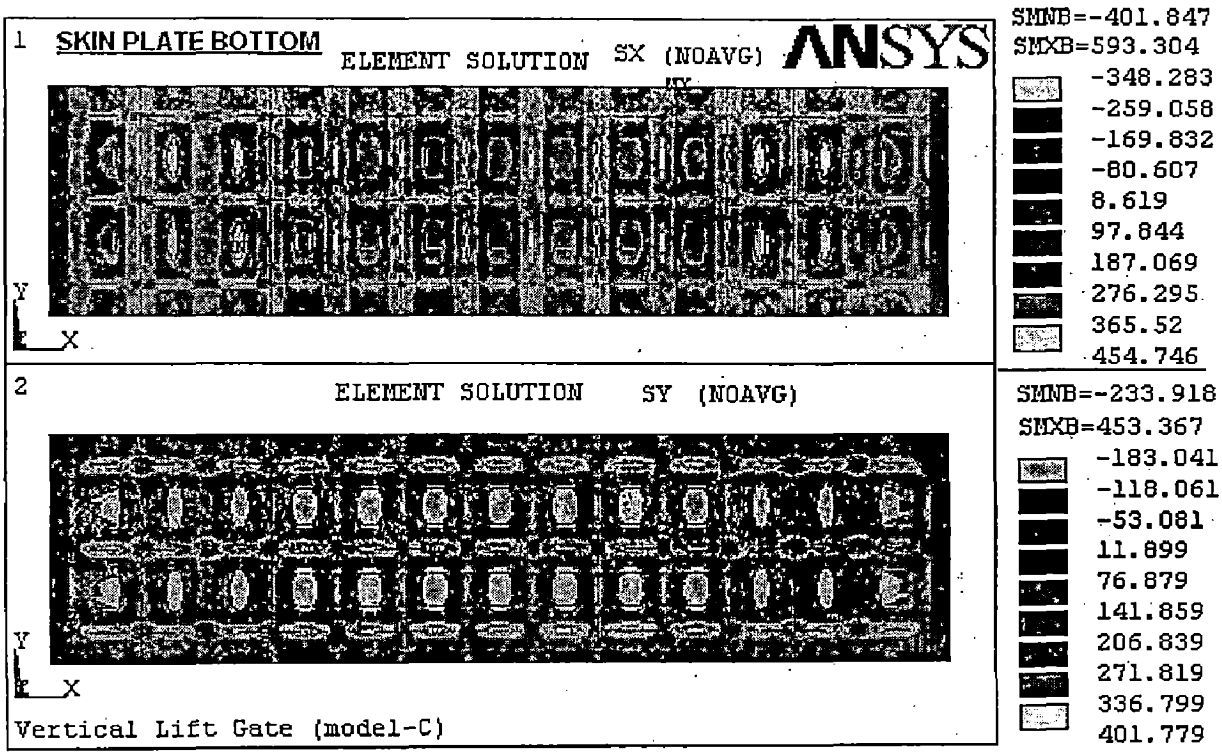


Fig: 5.6 Skin Plate Contour Plots Stresses SX, SY with Error Estimation (ES).

It is noticeable from the Fig: 5.5, 5.6 and contour legends that maximum stresses on the **bottom side** of skin plate in the X direction (wind=1): at centre of the panels are varying from –256.839 to –345.711 kg/cm^2 and with error estimation (SMNB=) –399.275 kg/cm^2 (nodal solution); –259.058 to –348.283 kg/cm^2 and SMNB= – 401.847 kg/cm^2 (element soln.)('-' Comp.), whereas with the vertical stiffeners are varying from 365.263 to 454.135 kg/cm^2 and SMXB= 494.959 kg/cm^2 (nodal solution); 365.52 to 454.746 kg/cm^2 and SMXB= 593.304 kg/cm^2 (element soln.)(tension), and in the Y direction (wind=2): at centre of the panels are varying from –114.253 to – 177.401 kg/cm^2 and SMNB= –230.965 kg/cm^2 (nodal soln.); –118.061 to –189.041 kg/cm^2 and SMNB= –233.918 kg/cm^2 (element soln.), whereas with the horizontal girders are varying from 336.799 to 401.779 kg/cm^2 and SMXB= 453.367 kg/cm^2.

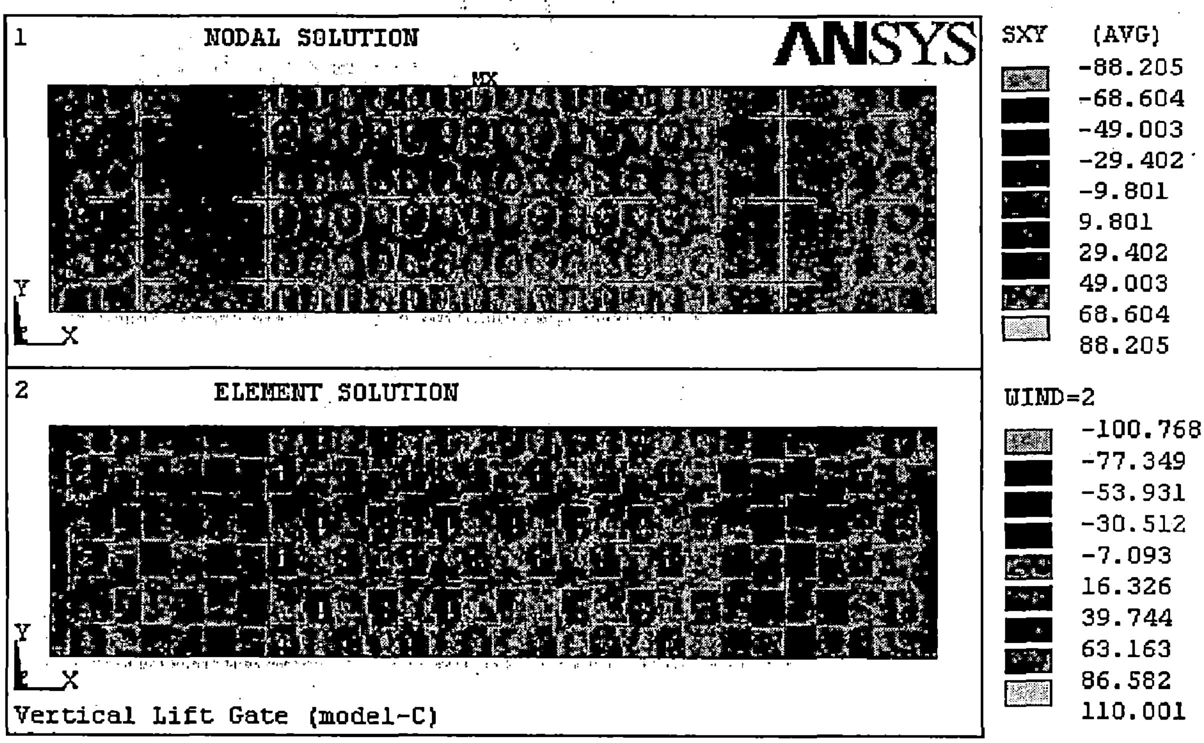


Fig: 5.7 Skin Plate Contour Plots Stresses Shear Stresses SXY.

It come into view from the Fig: 5.7 and contour legends that maximum shear stresses in the skin plate are happening (wind=1, nodal soln): 88.205 kg/cm^2 and (wind=2, element soln): 100.001 kg/cm^2; and these are taking place neither on the horizontal girders nor on the vertical stiffeners rather inside the panels. Shear stresses on the horizontal girders and vertical stiffeners are negligibly small.

It is noticeable from Fig: 5.8 and contour legends that maximum Von Mises SEQV stresses are also taking place with the vertical stiffeners and horizontal girders ranging from 362.81 to 408.144 kg/cm^2, SMXB= 542.776 kg/cm^2 in element solution (wind=1); and at same locations ranging from 356.91 to 401.474 kg/cm^2, SMXB= 442.297 kg/cm^2 in nodal solution (wind=2).

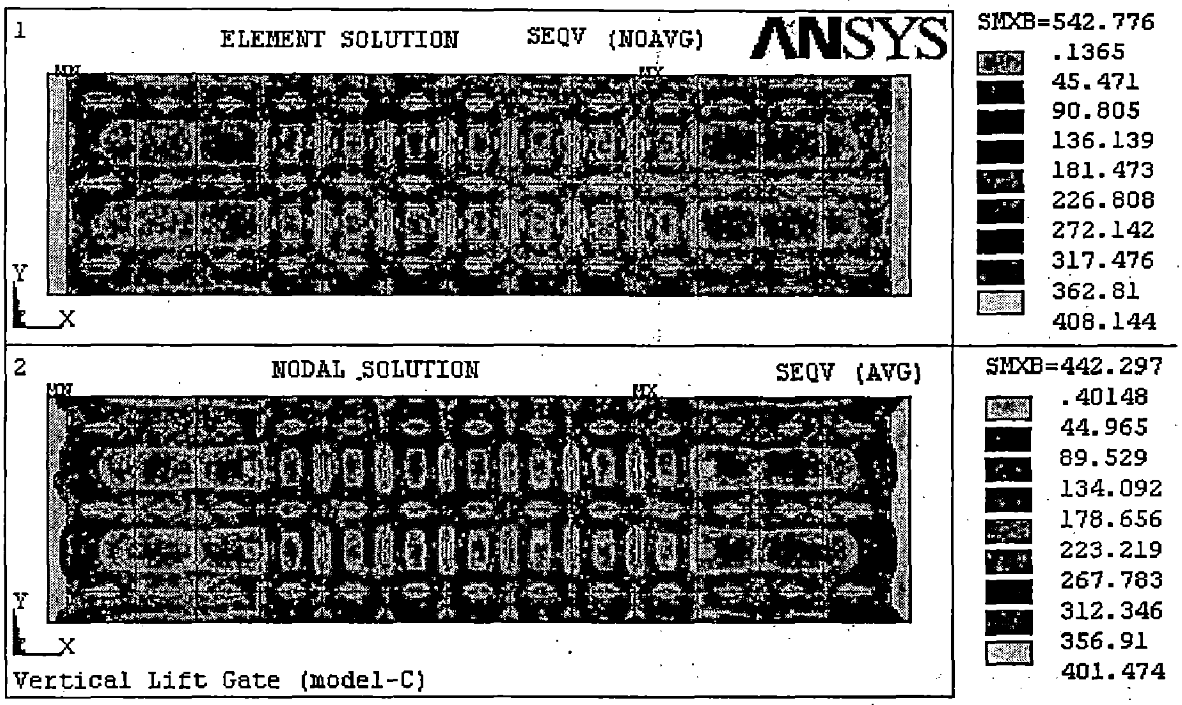


Fig: 5.8 Skin Plate Contour Plots Von Mises SEQV Stresses.

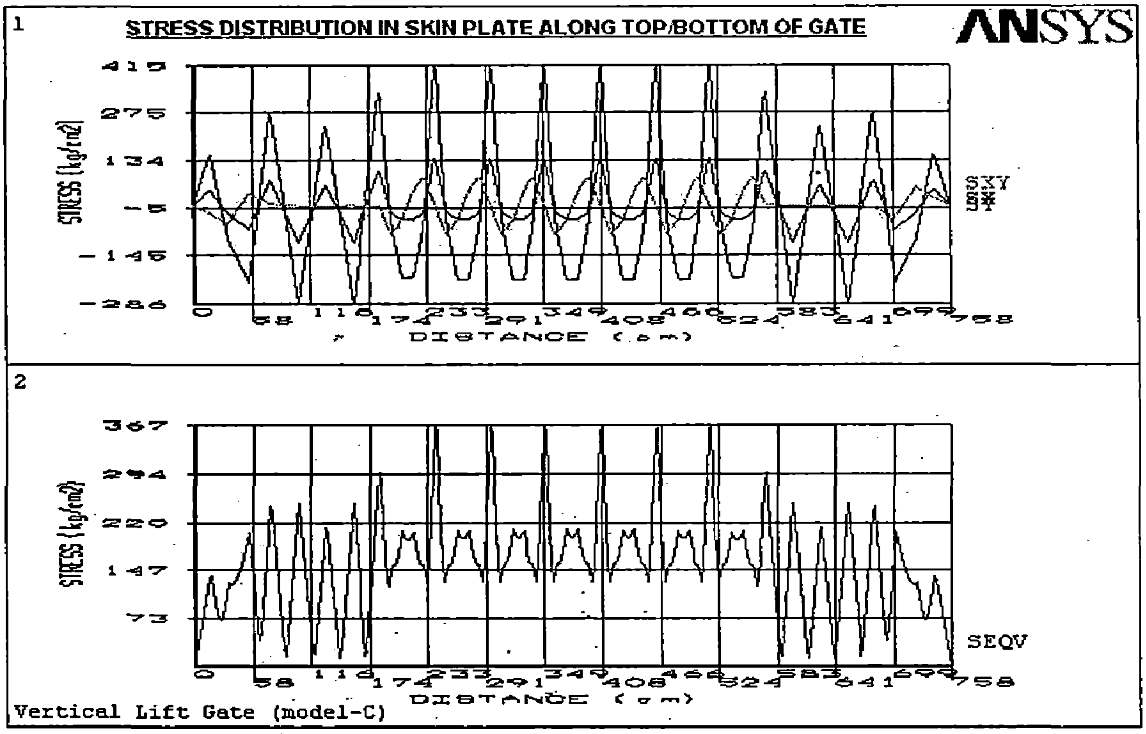
From the FEM RESULTS it appears that element solutions are closer to the design results and it is also observed during convergence study that, when mesh sizes are fine, element and nodal solution results come closer.

ANSYS provides tools to make sections in the model and analyses displacement, various stresses at different directions as well as VON MISES SEQV stress. So, FEM RESULTS are represented graphically along different sections of the skin plate. At the time of analyses it is observed that FEM RESULTS are same along top/bottom of the gate, along top/bottom horizontal girders and along sections between top and central as well as central & bottom horizontal girders due to same spacing and constant pressure. The gate and therefore, results are also symmetric on both sides of the mid-span. As such, ANSYS graphical results are represented along: top/bottom of the gate, top/bottom and central horizontal girders, between: top/bottom of the gate & top/bottom horizontal girders, central and top/bottom horizontal girders, and along vertical end girder, vertical stiffeners as well as between vertical stiffeners up to stiffener no. - 6.

Note: All graphical results are presented from bottom (pressure) side of the skin plate and so +ve results are with H girders/ V stiffeners and -ve results are in between.

Section Along Horizontal Direction (X – Direction): Total path length 758.00

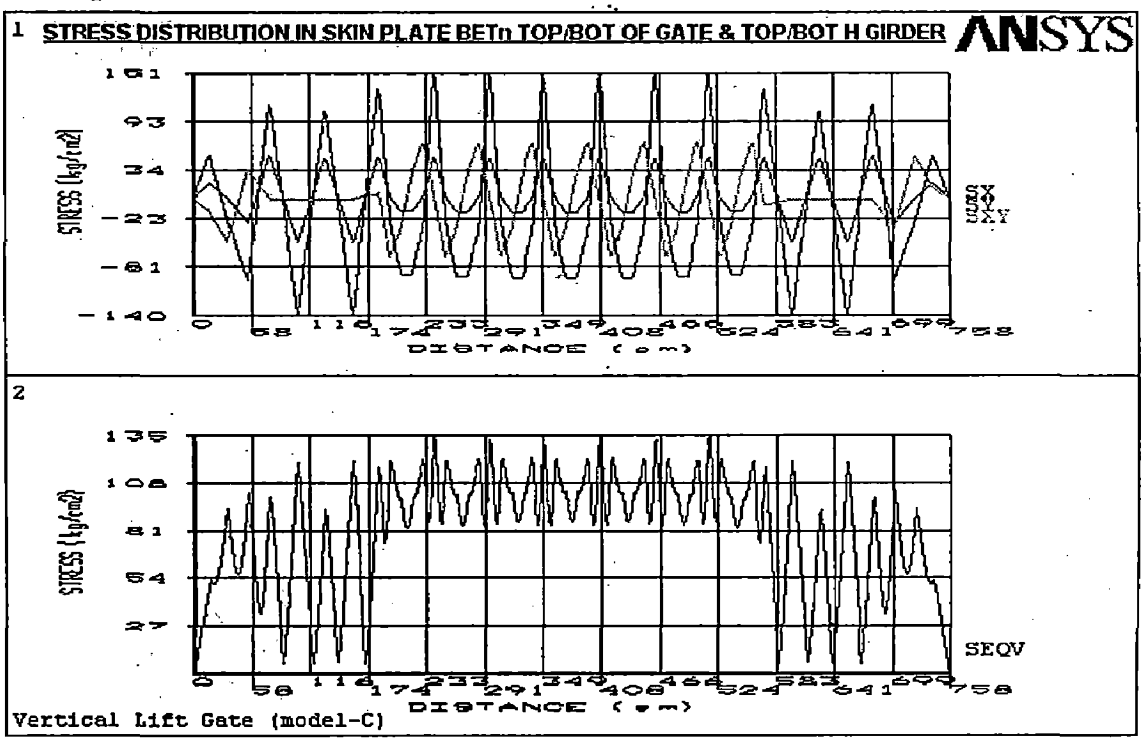
Graph: 5.1



SUMMARY OF THE GRAPH: 5.1 RESULTS (PATH VARIABLES: ABOG)

LABEL	MAX	MIN
SX	415.82	-286.30
SY	139.80	-104.39
SXY	84.275	-84.282
SEQV	367.91	.41454

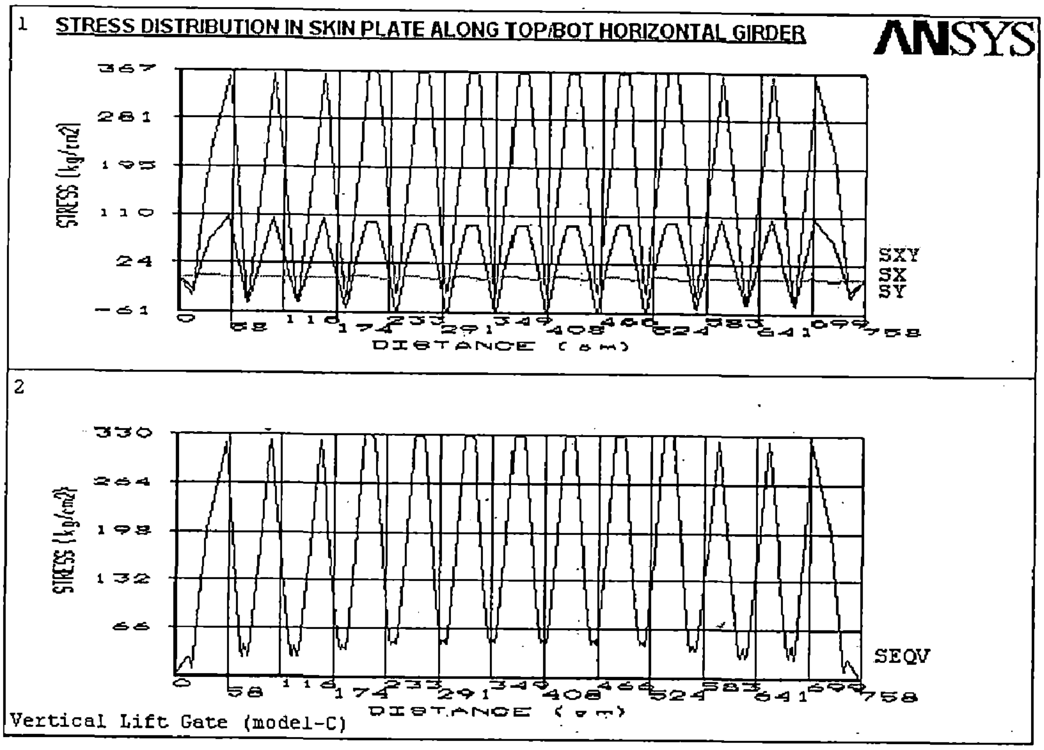
Graph: 5.2



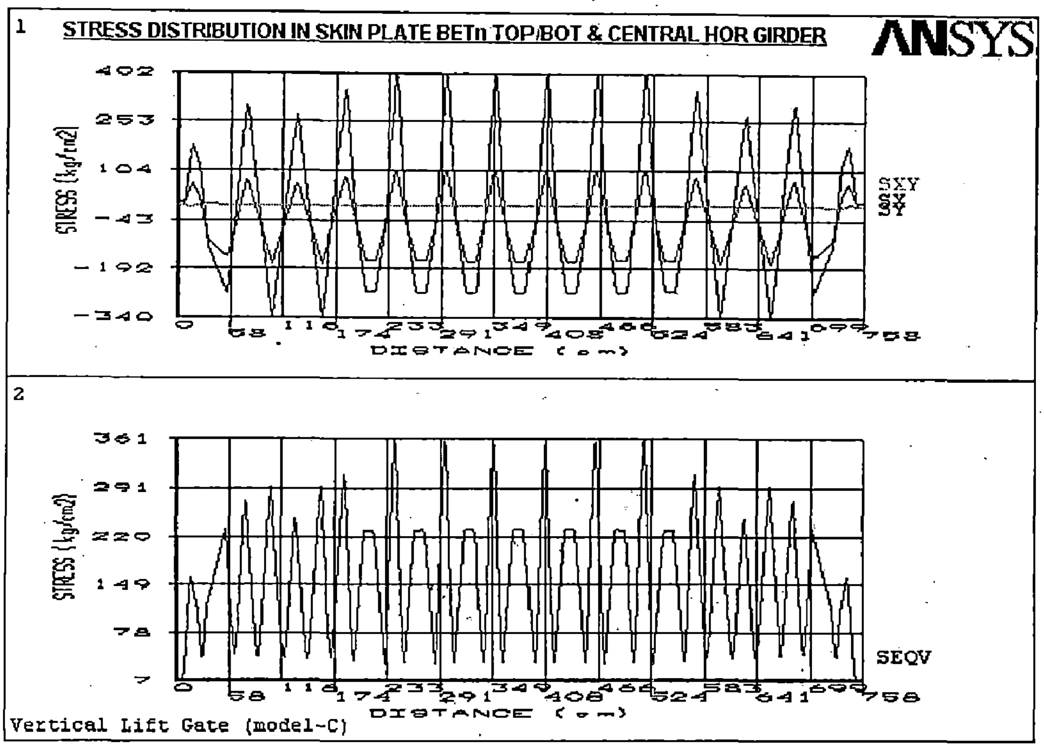
SUMMARY OF THE GRAPH: 5.2 RESULTS (PATH VARIABLES: COBP)

LABEL	MAX	MIN
SX	151.59	-140.22
SY	51.795	-52.082
SXY	68.302	-68.309
SEQV	135.37	.57158

Graph: 5.3



Graph: 5.4



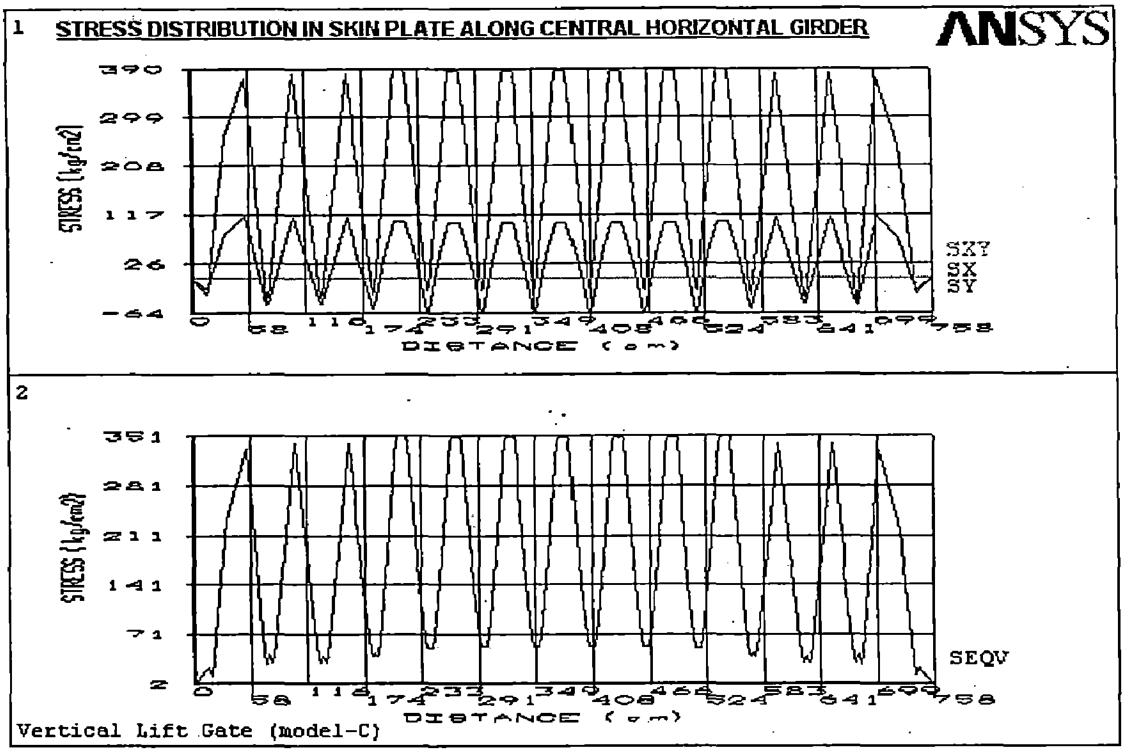
SUMMARY OF THE GRAPH: 5.3 RESULTS (PATH VARIABLES: ABHG)

LABEL	MAX	MIN
SX	105.22	-61.666
SY	367.67	-45.523
SXY	3.3993	-3.4049
SEQV	330.96	.87277

SUMMARY OF THE GRAPH: 5.4 RESULTS (PATH VARIABLES: BBCHG)

LABEL	MAX	MIN
SX	402.09	-340.92
SY	106.94	-175.08
SXY	5.4150	-5.5633
SEQV	361.87	7.6886

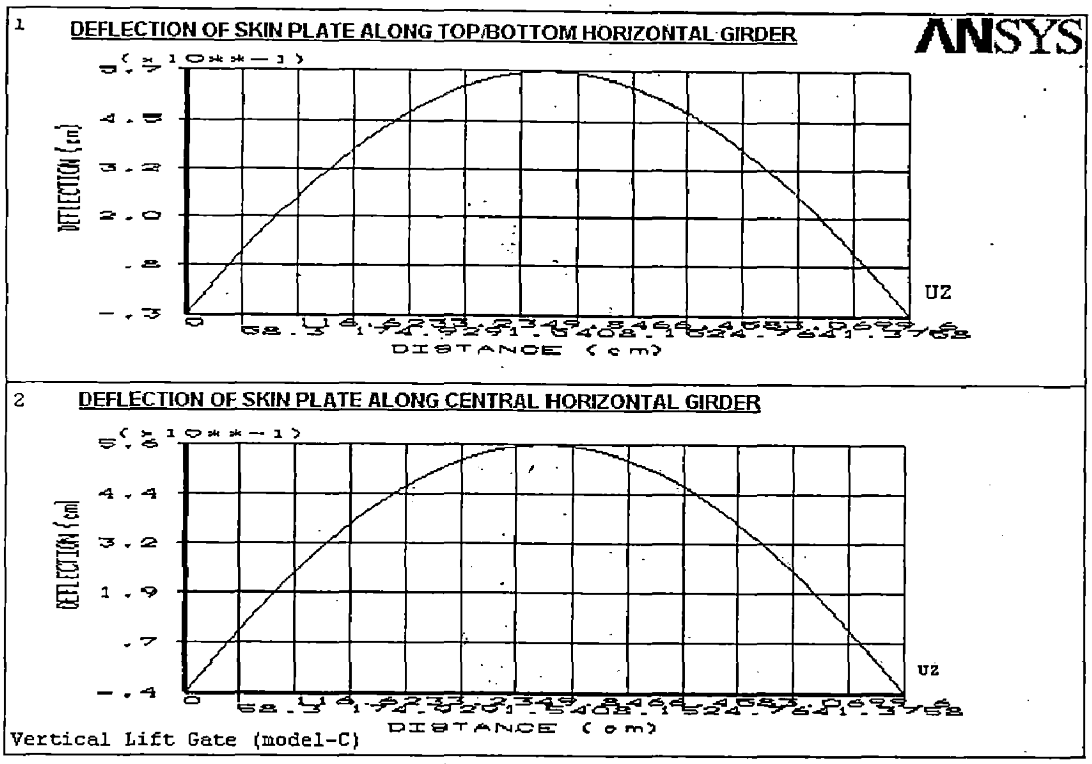
Graph: 5.5



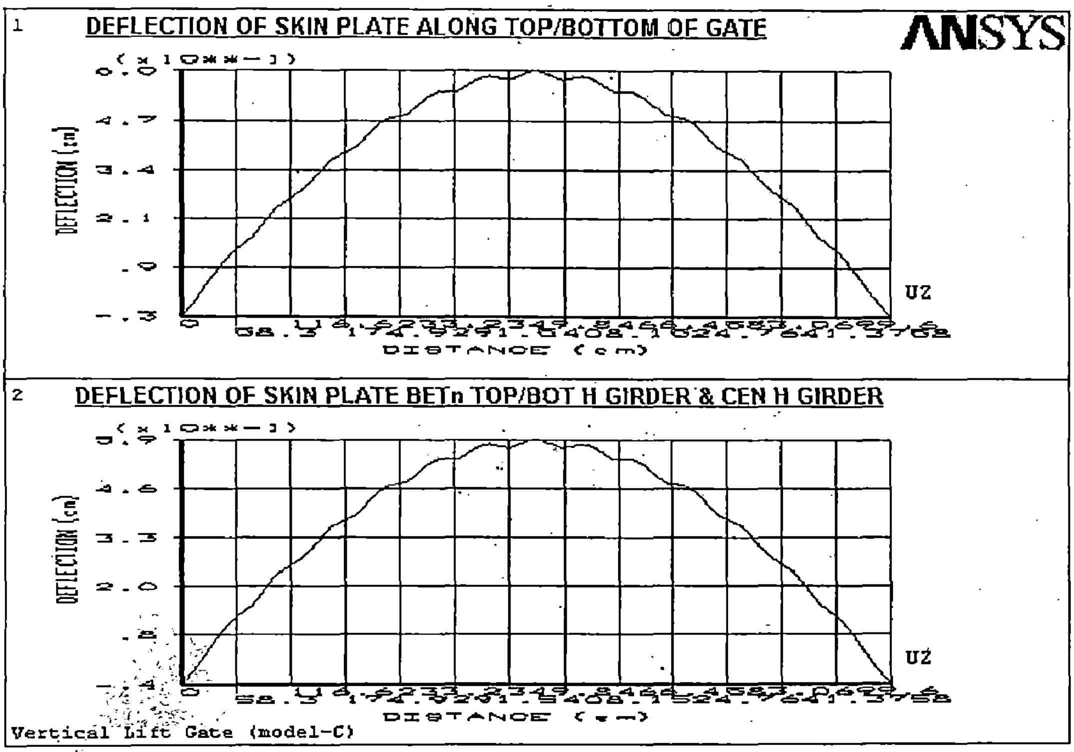
SUMMARY OF THE GRAPH: 5.5 RESULTS (PATH VARIABLES: ACHG)

LABEL	MAX	MIN
SX	112.83	-64.214
SY	390.72	-42.611
SXY	1.4964	-.36948E-01
SEQV	351.22	2.0701
UZ	.56870	-.46576E-01

Graph: 5.6



Graph: 5.7

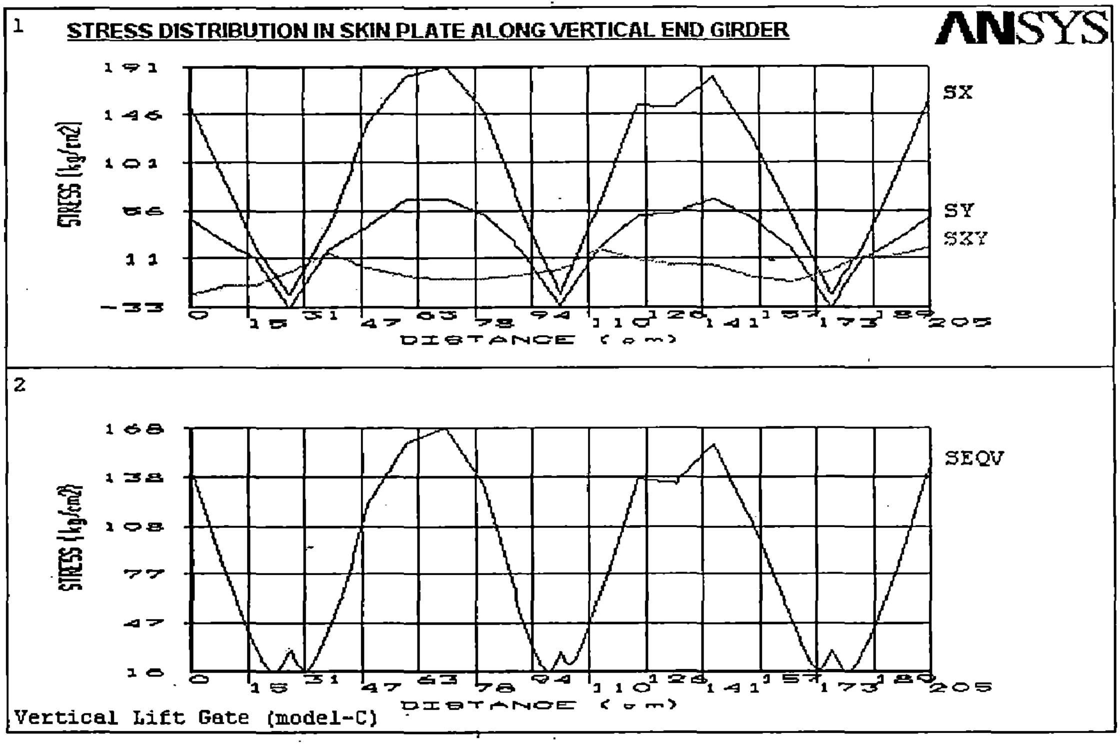


SUMMARY OF THE GRAPH: 5.6 & 5.7 RESULTS (PATH VARIABLES)

	LABEL	MAX	MIN
ALONG TOP/BOT OF GATE:	UZ	.60011	-.34482E-01
ALONG TOP/BOT H GIRDER:	UZ	.57601	-.39567E-01
BETn T/B HG & CEN HG:	UZ	.59208	-.45191E-01
ALONG CEN H GIRDER:	UZ	.56870	-.46576E-01

Section Along Vertical Direction (Y – Direction): Total path length 205.00

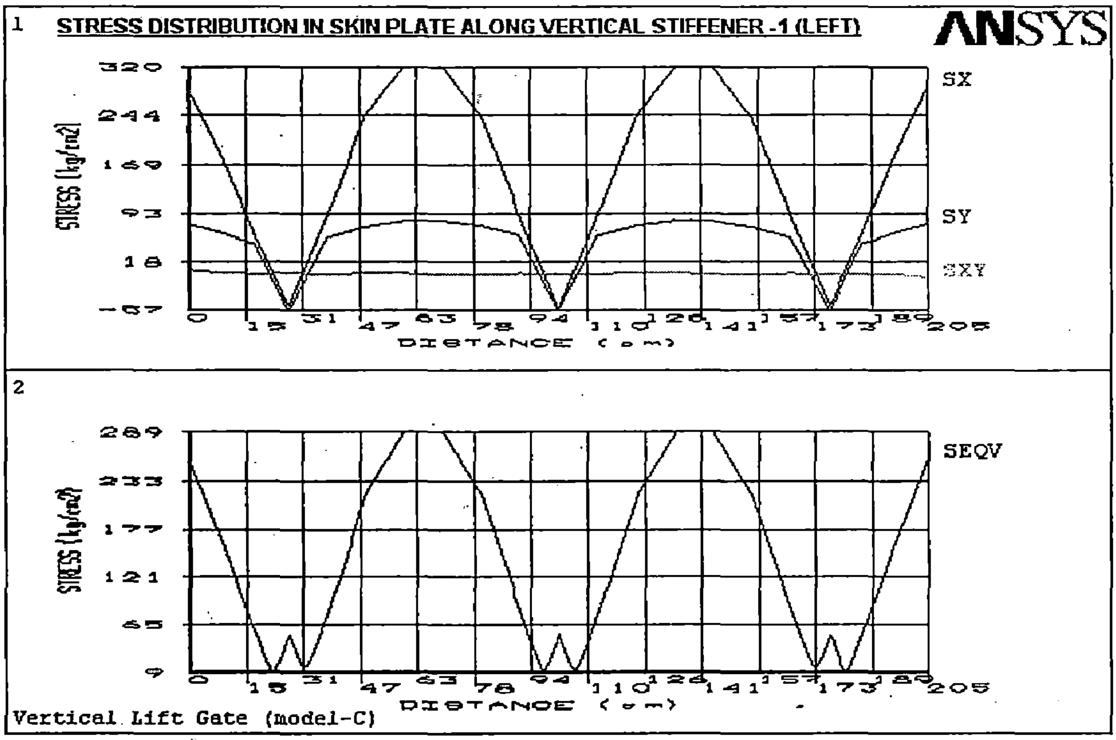
Graph: 5.8



SUMMARY OF THE GRAPH: 5.8 RESULTS (PATH VARIABLES: AVEG)

LABEL	MAX	MIN
SX	191.42	-21.282
SY	67.029	-33.554
SXY	22.132	-22.131
SEQV	168.86	16.740

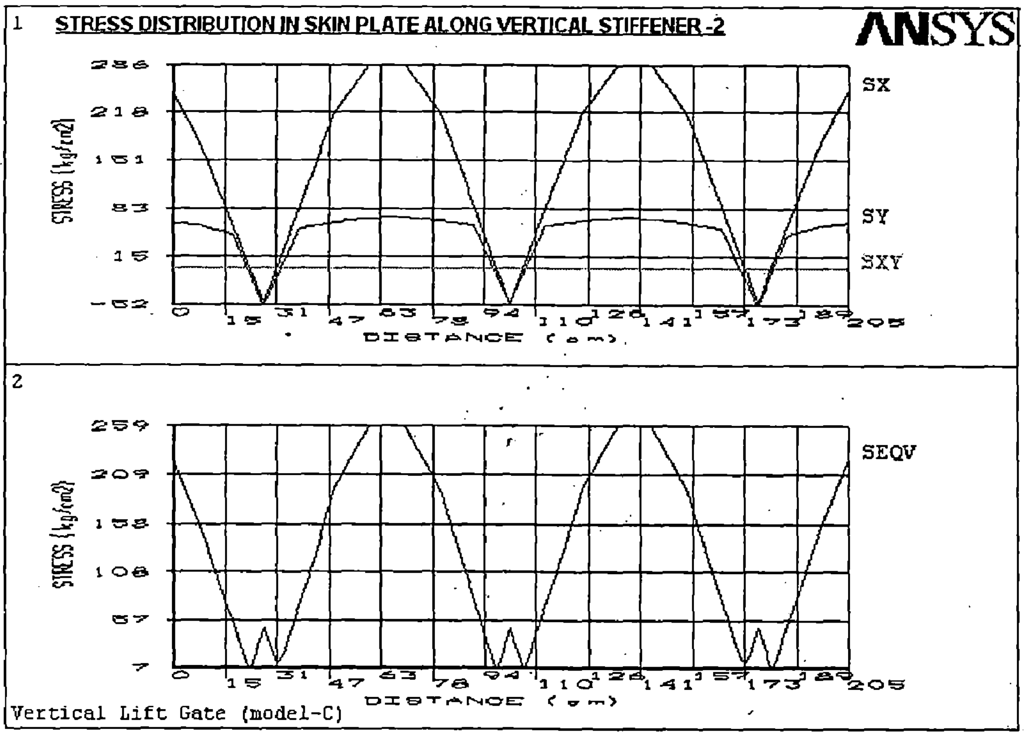
Graph: 5.9



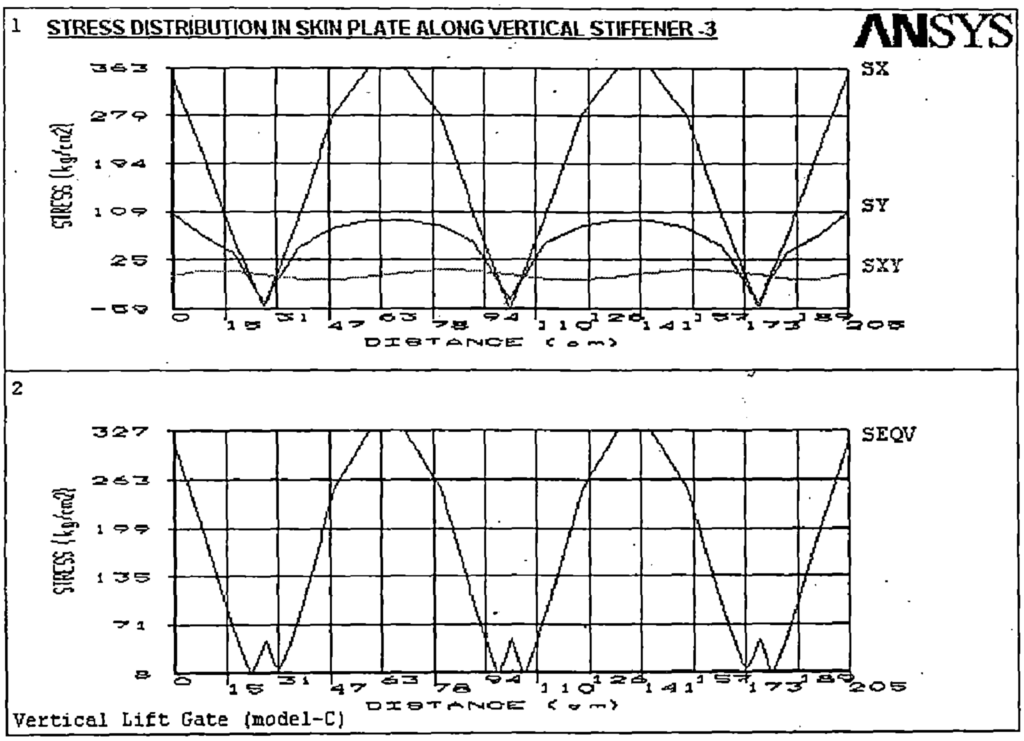
SUMMARY OF THE GRAPH: 5.9 RESULTS (PATH VARIABLES: AVS1)

LABEL	MAX	MIN
SX	320.30	-53.783
SY	82.556	-57.305
SXY	5.9442	-5.9443
SEQV	289.19	9.7408

Graph: 5.10



Graph: 5.11



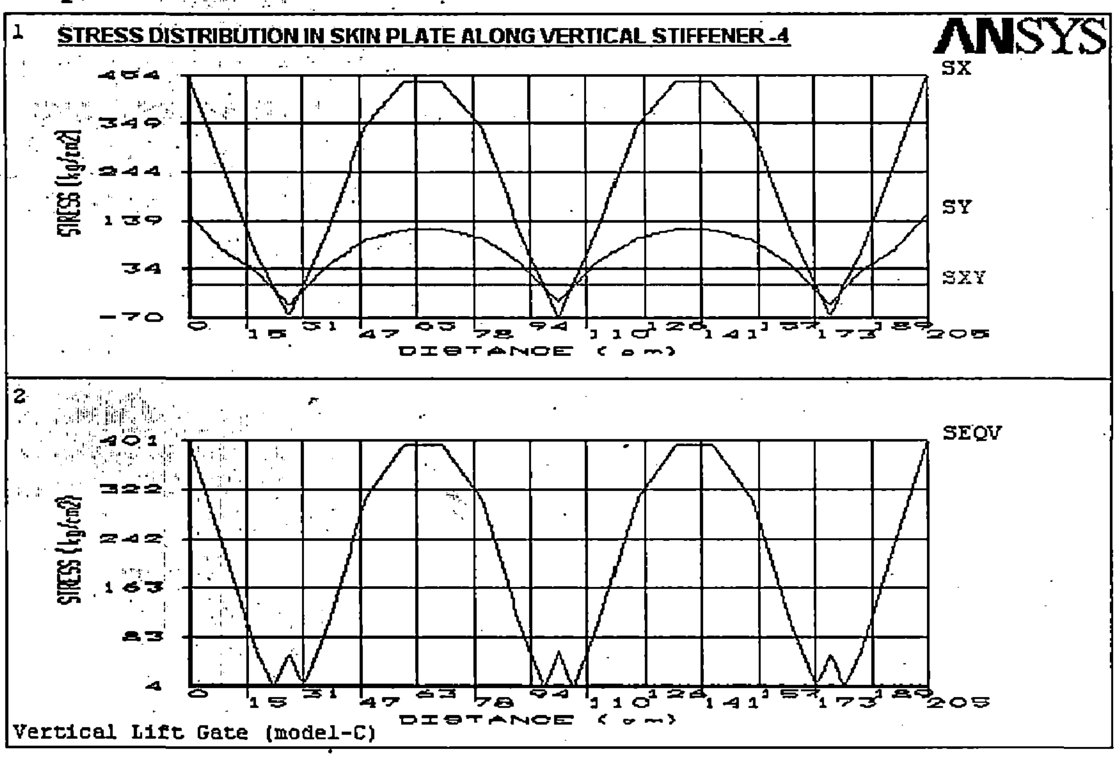
SUMMARY OF GRAPH: 5.10 RESULTS (PATH VARIABLES: AVS2)

LABEL	MAX	MIN
SX	286.66	-51.301
SY	70.723	-52.356
SXY	.24592	-.24595
SEQV	259.79	7.1059

SUMMARY OF GRAPH: 5.11 RESULTS (PATH VARIABLES: AVS3)

LABEL	MAX	MIN
UZ	.40580	.39252
SX	363.81	-59.695
SY	108.53	-46.662
SXY	8.6007	-8.6007
SEQV	327.81	8.0216

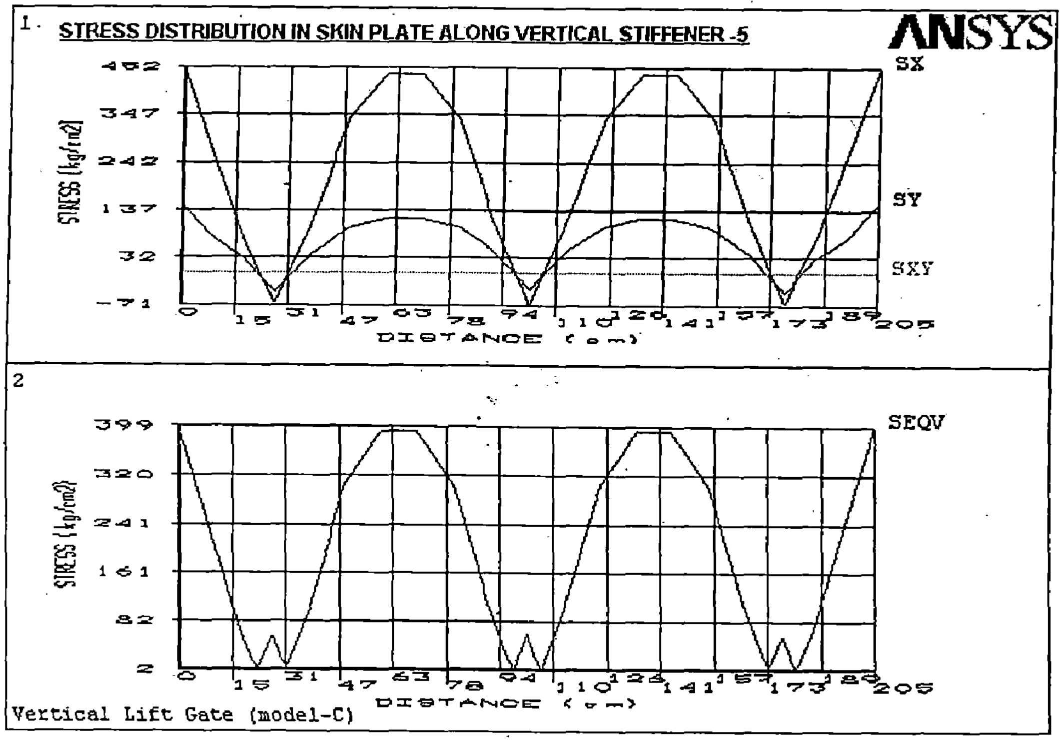
Graph: 5.12



SUMMARY OF GRAPH: 5.12 RESULTS (PATH VARIABLES: AVS4)

LABEL	MAX	MIN
UZ	.48985	.47671
SX	454.13	-70.041
SY	152.88	-41.305
SXY	.49554E-01	-.49564E-01
SEQV	401.47	4.2628

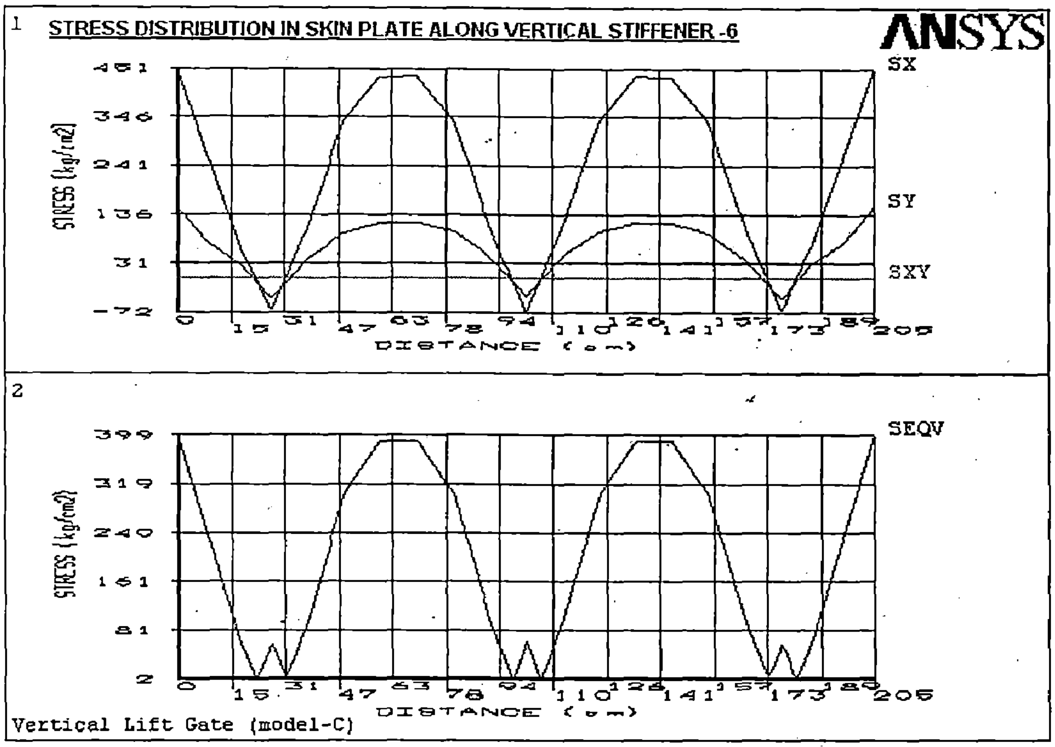
Graph: 5.13



SUMMARY OF GRAPH: 5.13 RESULTS (PATH VARIABLES: AVS-5)

LABEL	MAX	MIN
SX	452.36	-71.969
SY	152.38	-41.922
SXY	.47797E-01	-.47798E-01
SEQV	399.89	2.8622

Graph: 5.14



From the Table: 5.2 it is obvious that, although FEM RESULTS are varying from panel to panel but well below the permissible values and the significant values **(bold)** are quiet close to design results with little variations due to some assumptions made in design methods as well as some ANSYS iteration errors, e.g. SMXB= 542.776 kg/cm^2, SMXB= 442.297 kg/cm^2 (Fig: 5.8), which may be considered within acceptable range.

5.2.4 Horizontal Girders FEM Result

The vertical lift gate under study consists of three horizontal girders. The top and bottom girders are geometrically same and central girder is heavy, and spacing is equal for all girders (ref. appendix-A, page-16). Therefore, for constant applied pressure, load behavior of the top and bottom girders are same, and subsequently FEM RESULTS are presented for the central and top/bottom girders. For ready reference, design results are also mentioned hereunder.

Central Horizontal Girder: Maximum values are: Shear force = 42435.0 kg, Bending moment = 8062650.0 kg-cm, Deflection = 0.54 cm, Bending stress, on top flange = 637.54 kg/cm^2, bottom flange = -978.20 kg/cm^2 (ref: Appendix –A, page –16).

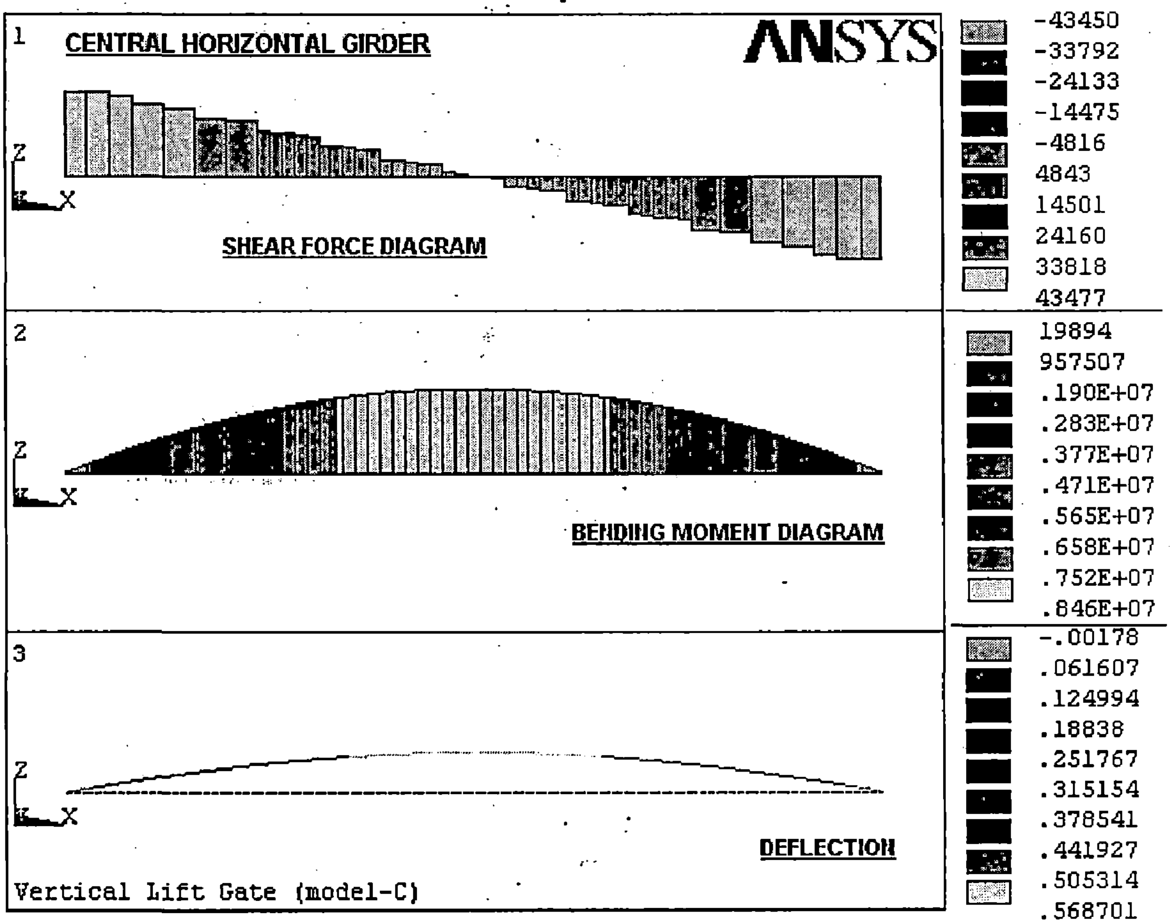


Fig: 5.9 Central H Girder Contour Plots Shear Force, Bending Moment & Deflection.

It is evident from the Fig: 5.9 and contour legends that maximum values of shear force 43477 kg, bending moment 8460000 kg-cm and deflection 0.568701 cm.

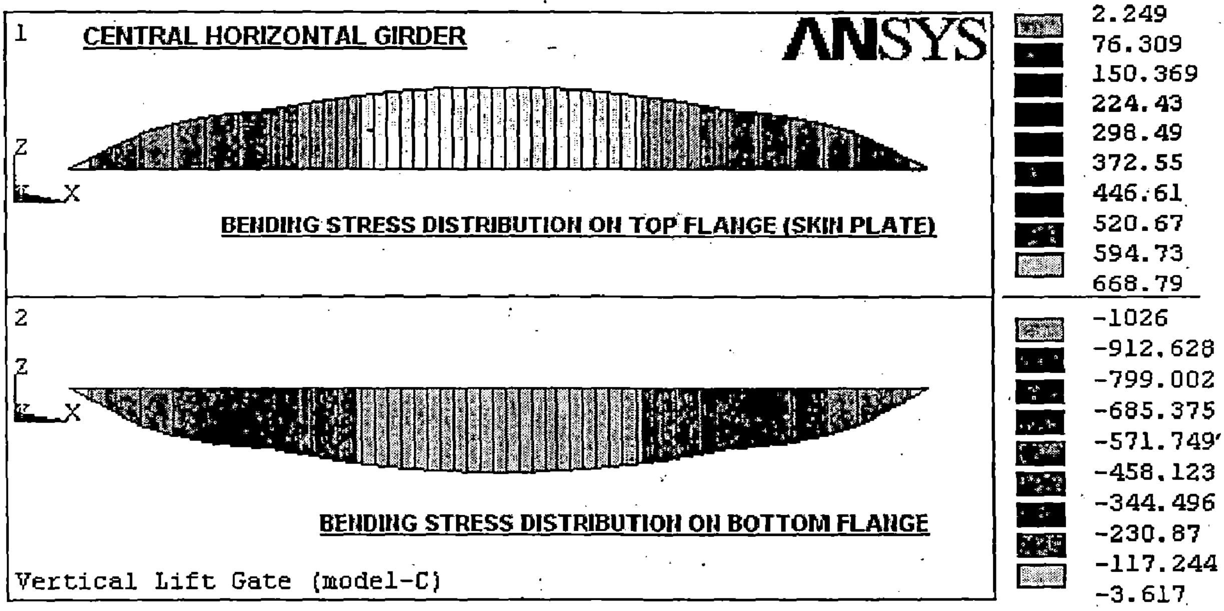


Fig: 5.10 Central H Girder Contour Plots Bending Stresses on Top & Bottom Flange.

It is clear from the Fig: 5.10 and contour legends that maximum values of bending stresses at the centre of span on top flange (skin plate): 668.79 kg/cm^2 (tension) and bottom flange: -1026 kg/cm^2 (compression). For details Element Table results given in ET.5.1.

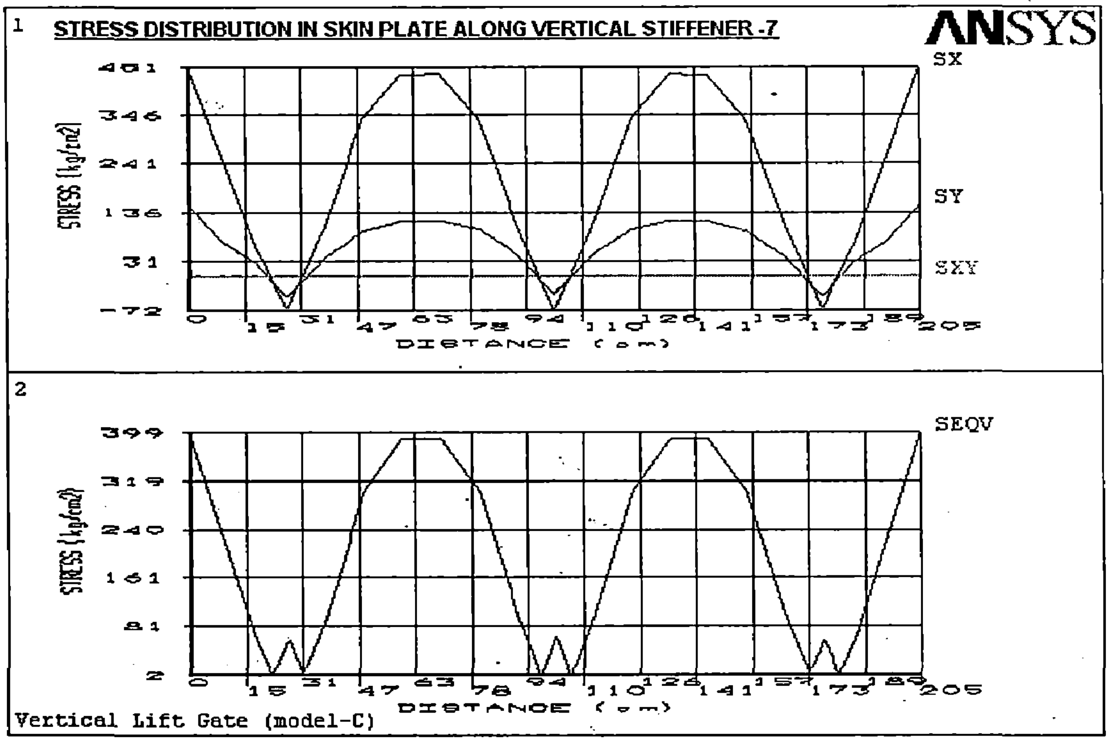
ET: 5.1 PRINT ELEMENT TABLE ITEMS PER ELEMENT **(CHG)**
***** POST1 ELEMENT TABLE LISTING *****

STAT ELEM	CURRENT SBZT	CURRENT SBZB	CURRENT SFZ	CURRENT BMY	CURRENT BMX
1067	2.2491	-3.6173	43477.	19894.	91.127
1068	88.350	-142.10	42670.	.78149E+06	-35.223
1069	190.77	-306.82	41385.	.16874E+07	-16.523
1070	290.13	-466.63	36254.	.25663E+07	-9.6532
1071	364.55	-578.99	34696.	.35633E+07	-9.7178
1072	421.47	-661.40	29226.	.45176E+07	-7.6791
1073	455.83	-707.04	27670.	.53214E+07	-7.6542
1074	481.06	-738.18	22634.	.60841E+07	-5.8558
1075	500.78	-768.45	22203.	.63336E+07	-5.8471
1076	520.10	-798.10	21361.	.65779E+07	-5.8385
1077	538.67	-826.59	20520.	.68128E+07	-5.8299
1078	556.48	-853.92	20095.	.70380E+07	-5.8216
1079	573.96	-880.74	15514.	.72590E+07	-4.2701
1080	587.49	-901.50	15089.	.74302E+07	-4.2627
1081	600.62	-921.65	14250.	.75963E+07	-4.2553
1082	613.01	-940.65	13410.	.77529E+07	-4.2480
1083	624.63	-958.49	12985.	.78999E+07	-4.2408
1084	635.93	-975.82	8391.1	.80428E+07	-2.9022
1085	643.26	-987.08	7965.9	.81355E+07	-2.8957
1086	650.20	-997.73	7126.6	.82233E+07	-2.8893
1087	656.39	-1007.2	6287.2	.83015E+07	-2.8829

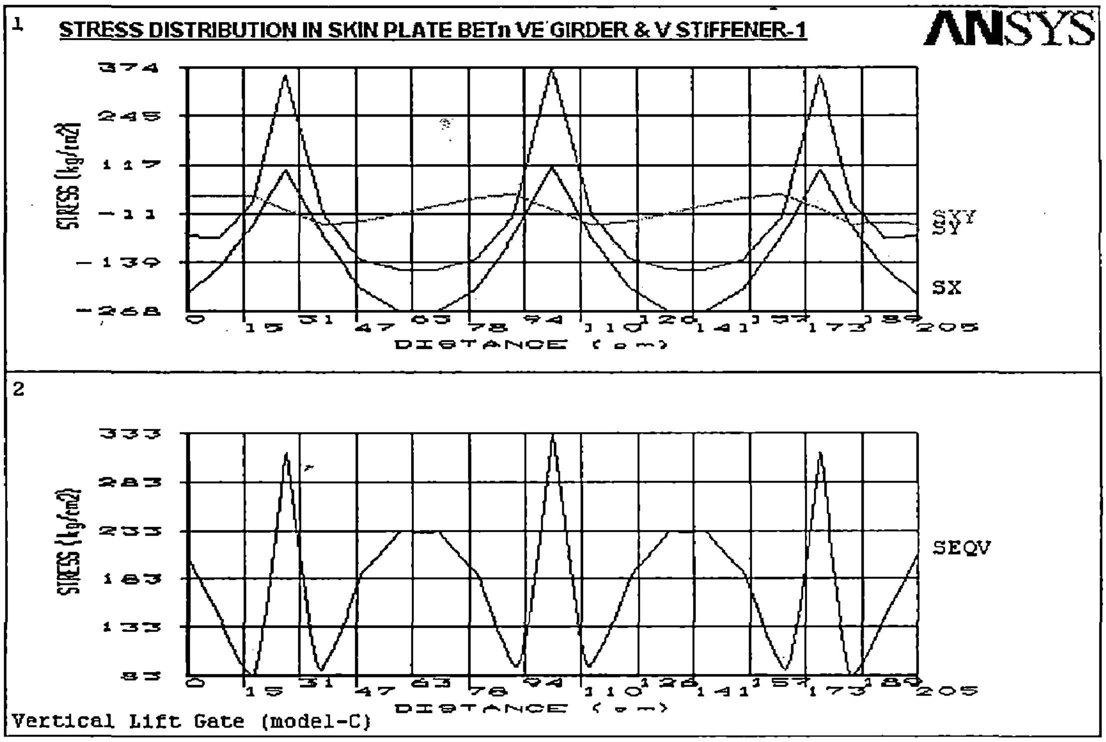
SUMMARY OF GRAPH: 5.14 RESULTS (PATH VARIABLES: AVS-6)

LABEL	MAX	MIN
SX	451.69	-72.961
SY	152.22	-42.239
SXY	.16514E-01	-.16514E-01
SEQV	399.29	2.3922

Graph: 5.15



Graph: 5.16



FEM RESULT

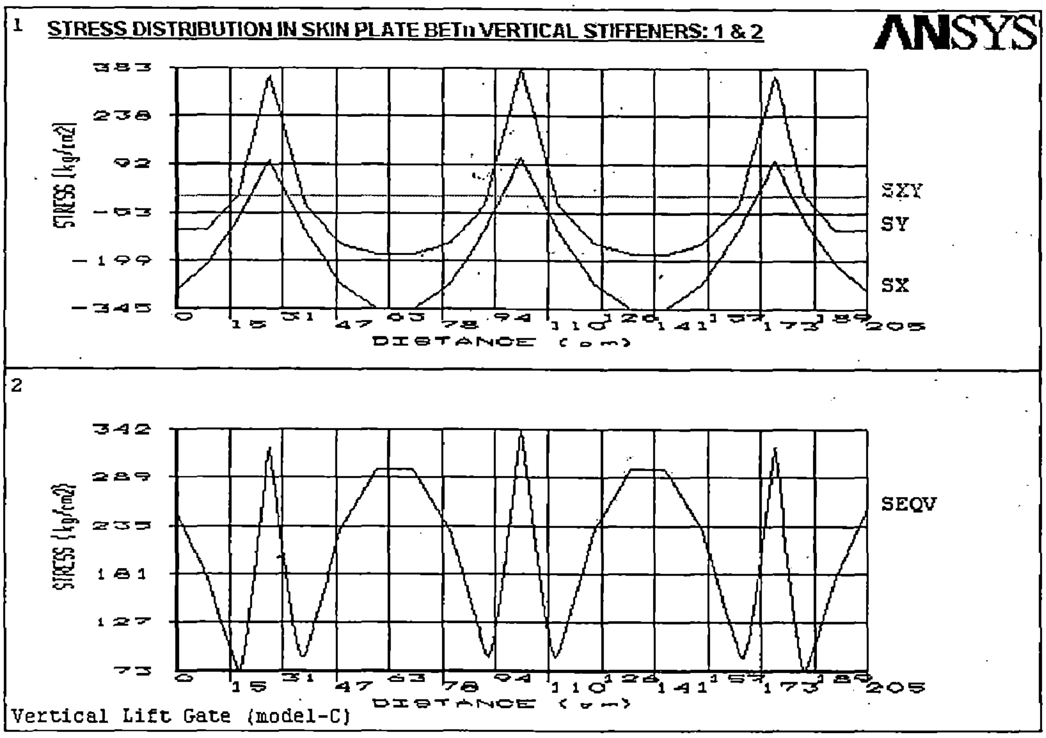
SUMMARY OF GRAPH: 5.15 RESULTS (PATH VARIABLES: AVS7)

LABEL	MAX	MIN
UZ	.57813	.56508
SX	451.69	-72.961
SY	152.22	-42.235
SXY	.14798E-01	-.14797E-01
SEQV	399.29	2.3937

SUMMARY OF GRAPH: 5.16 RESULTS (PATH VARIABLES: BVEG&VS1)

LABEL	MAX	MIN
UZ	.13197	.10208
SX	113.30	-268.20
SY	374.39	-158.70
SXY	40.594	-40.359
SEQV	333.75	83.228

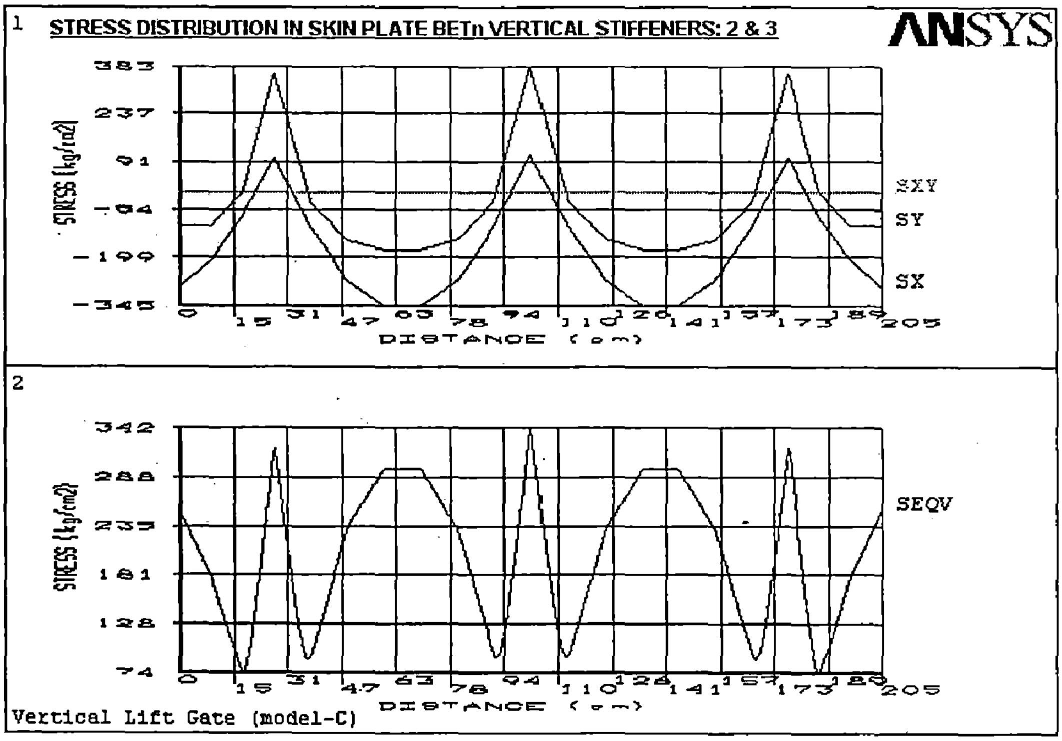
Graph: 5.17



SUMMARY OF GRAPH: 5.17 RESULTS (PATH VARIABLES: BVS1_2)

LABEL	MAX	MIN
UZ	.25301	.22385
SX	112.76	-345.01
SY	383.82	-177.25
SXY	.48642	-.48571
SEQV	342.89	73.614

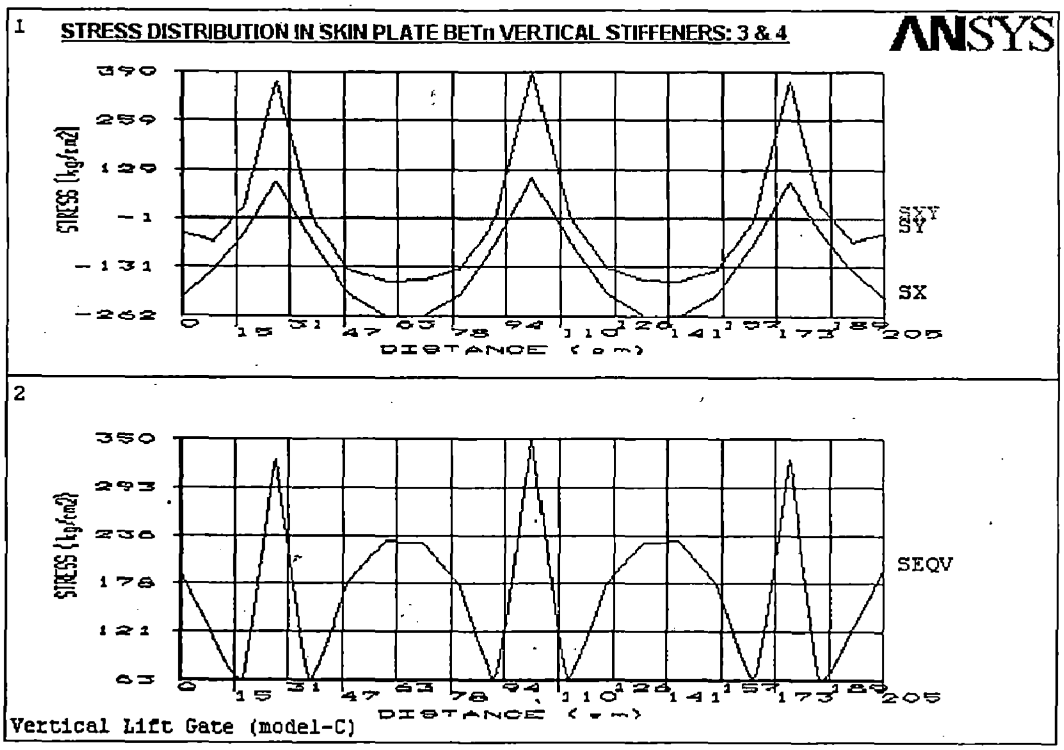
Graph: 5.18



SUMMARY OF GRAPH: 5.18 RESULTS (PATH VARIABLES: BVS2_3)

LABEL	MAX	MIN
UZ	.37079	.34184
SX	111.77	-345.70
SY	383.10	-177.37
SXY	.49809	-.49814
SEQV	342.42	74.398

Graph: 5.19

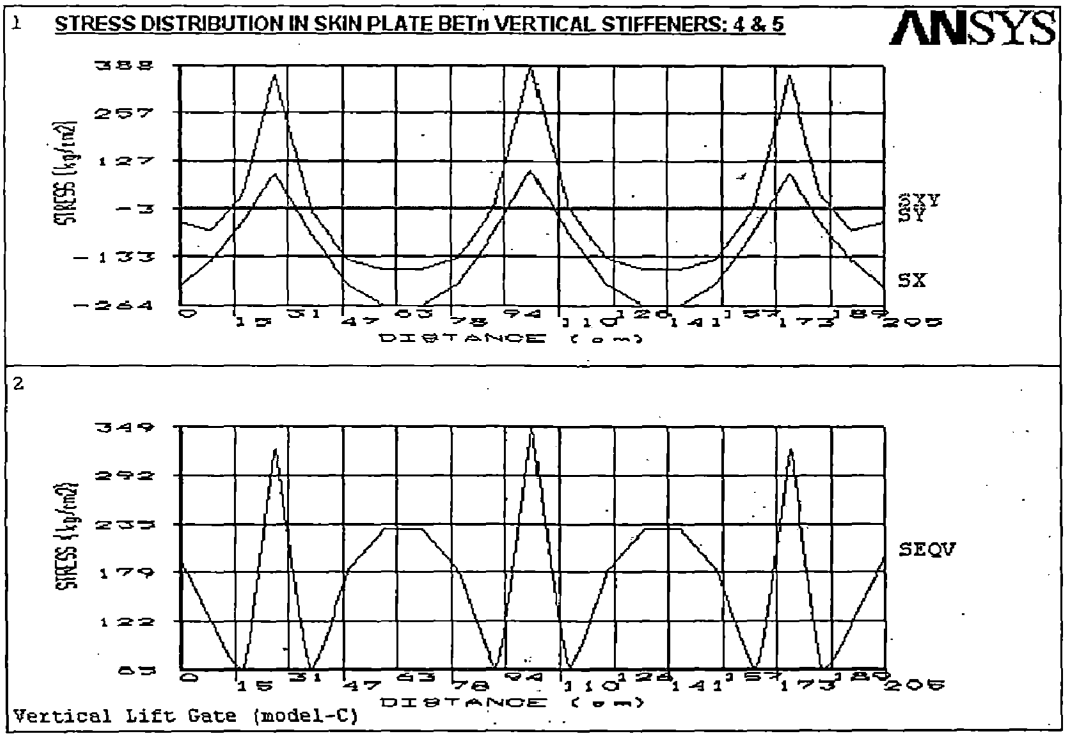


FEM RESULT

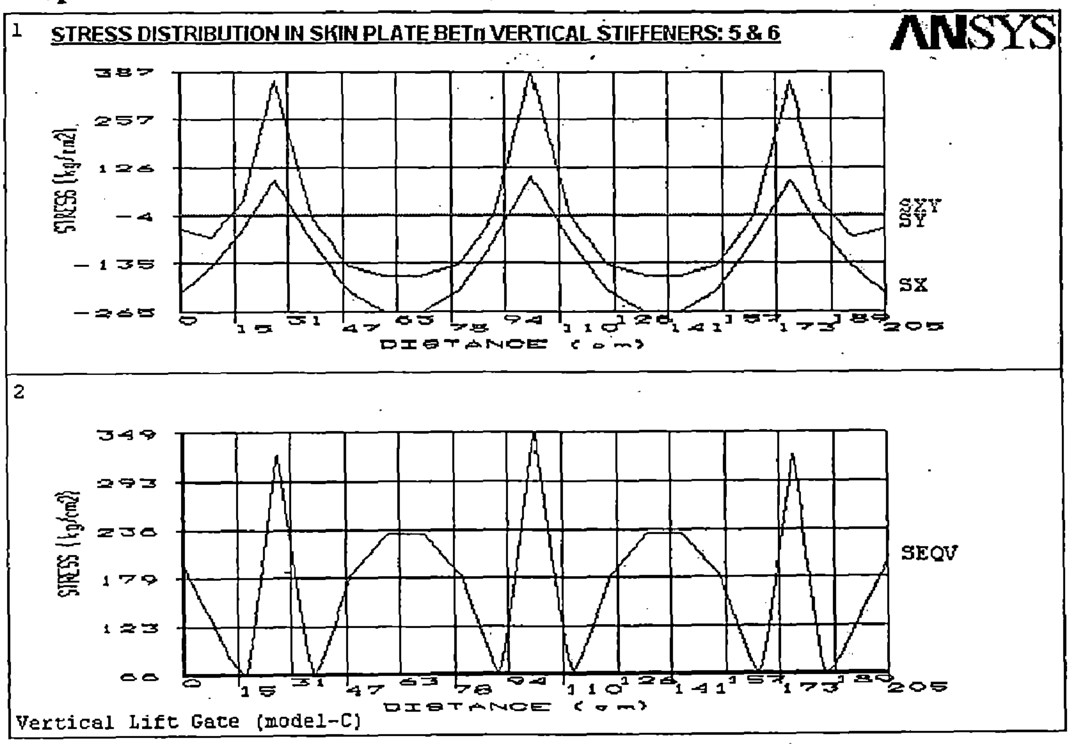
SUMMARY OF GRAPH: 5.19 RESULTS (PATH VARIABLES: BVS3_4)

LABEL	MAX	MIN
SX	105.36	-262.44
SY	390.30	-168.08
SXY	.38556	-.38559
SEQV	350.90	63.891

Graph: 5.20



Graph: 5.21



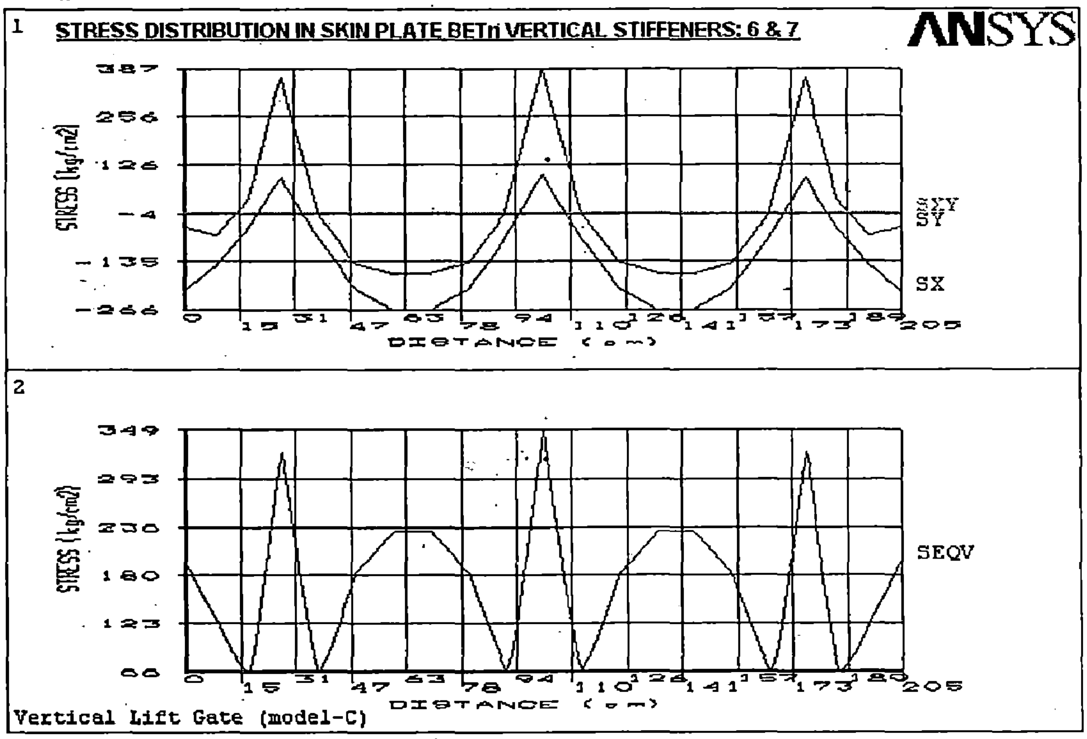
SUMMARY OF GRAPH: 5.20 RESULTS (PATH VARIABLES: BVS4_5)

LABEL	MAX	MIN
UZ	.54067	.50922
SX	102.46	-264.38
SY	388.21	-168.01
SXY	.89810E-01	-.89825E-01
SEQV	349.62	65.515

SUMMARY OF GRAPH: 5.21 RESULTS (PATH VARIABLES: BVS5_6)

LABEL	MAX	MIN
UZ	.58507	.55366
SX	100.98	-265.90
SY	387.89	-168.48
SXY	.42155E-01	-.42160E-01
SEQV	349.70	66.627

Graph: 5.22



SUMMARY OF GRAPH: 5.21 RESULTS (PATH VARIABLES: BVS6_7)

LABEL	MAX	MIN
UZ	.60011	.56870
SX	100.49	-266.41
SY	387.77	-168.64
SXY	.49478E-02	-.49426E-02
SEQV	349.72	66.999

5.2.3 Validation Of Skin Plate FEM Result

A comparison between conventionally design results and FEM results of skin plate for interior and top/bottom panels at centers, with vertical stiffeners and horizontal girders is presented in table: 5.2.

Table: 5.2 Comparison of DESIGN and FEM results in skin plate.

Location & direction of results		**Design results** (kg/cm²)	**FEM**		**Variation** (%)
			Results (kg/cm²)	**Ref. graph /figure**	
a) **Interior panels** (ref. Appendix –A, page-8, art. 5.6) **(FEM results for 14 panels)**					
Centre of the panel	SX	251.77	264.38 - 345.70	Gr: 5.4, 5.16-5.21	+5.01 to +37.31
	SY	160.11	168.01 - 177.37	,,	+4.93 to +10.78
	SXY	33.33	40.36 (max)	,,	+21.09
Mid-point with vertical stiffeners	SX	**529.14**	320.30 – **454.13**	Gr: 5.4, 5.9-5.15	-39.47 to -14.18
	SY	**158.74**	82.56 – **152.88**	,,	-47.99 to -3.69
	SXY	0	5.94 (max)	,,	---
Mid-point with central horizontal girder	SX	**126.81**	100.49 – **113.30**	Gr: 5.5, 5.16-5.21	-20.76 to -10.65
	SY	**422.70**	374.39 – **390.72**	,,	-11.43 to -7.57
	SXY	0	40.59 (max)	,,	---
b) **Top/Bottom panel** (ref. Appendix –A, page-9, art. 5.7) **(FEM results for 7 panels)**					
Centre of the panel	SX	104.80	89.21 – 140.78	Gr: 5.2, 5.16-5.21	-14.88 to +34.33
	SY	48.06	13.62 – 52.08	,,	-71.66 to +8.36
	SXY	33.33	35.41 (max)	,,	+6.24
Mid-point with vertical stiffeners	SX	145.73	110.99 – 164.68	Gr: 5.2, 5.9-5.15	-23.84 to +13.00
	SY	43.72	51.87 – 54.50	,,	+18.64 to +24.66
	SXY	0	7.05 (max)	,,	---
Mid-point with top/bottom horizontal girder	SX	63.53	82.08 – 102.91	Gr: 5.3, 5.16-5.21	+29.20 to +61.99
	SY	211.77	334.40 – 358.44	,,	+57.91 to +69.26
	SXY	0	1.96 (max)	,,	---
Deflection at the centre span of the gate, UZ (cm)		0.54	0.576 – 0.600	Gr: 5.6-5.7	+6.67 to +11.11

FEM RESULTS are analyzed for fourteen interior and seven top/bottom panels.

STAT ELEM	CURRENT SBZT	CURRENT SBZB	CURRENT SFZ	CURRENT BMY	CURRENT BMX
1088	661.82	-1015.6	5862.0	.83702E+07	-2.8766
1089	666.92	-1023.4	1262.1	.84347E+07	-1.6819
1090	668.05	-1025.1	836.91	.84491E+07	-1.6760
1091	**668.79**	**-1026.3**	**-2.4851**	**.84584E+07**	**-1.6701**
1092	668.78	-1026.2	-841.88	.84582E+07	-1.6642
1093	668.01	-1025.1	-1267.1	.84485E+07	-1.6584
1094	666.91	-1023.4	-5866.5	.84346E+07	-.52199
1095	661.84	-1015.6	-6291.7	.83705E+07	-.51627
1096	656.38	-1007.2	-7131.1	.83014E+07	-.51049
1097	650.17	-997.68	-7970.5	.82229E+07	-.50465
1098	643.20	-986.98	-8395.6	.81347E+07	-.49879
1099	635.90	-975.78	-12988.	.80424E+07	.65884
1100	624.64	-958.50	-13413.	.79000E+07	.66479
1101	612.98	-940.61	-14253.	.77526E+07	.67084
1102	600.57	-921.58	-15092.	.75957E+07	.67697
1103	587.41	-901.38	-15517.	.74292E+07	.68315
1104	573.92	-880.67	-20095.	.72585E+07	1.9139
1105	556.48	-853.91	-20520.	.70379E+07	1.9202
1106	538.64	-826.54	-21361.	.68124E+07	1.9267
1107	520.05	-798.01	-22203.	.65773E+07	1.9333
1108	500.70	-768.33	-22634.	.63326E+07	1.9399
1109	480.87	-737.90	-27665.	.60818E+07	3.2484
1110	455.79	-706.98	-29221.	.53209E+07	3.2654
1111	421.43	-661.34	-34689.	.45172E+07	4.5513
1112	364.54	-578.97	-36247.	.35632E+07	4.5275
1113	290.21	-466.75	-41383.	.25670E+07	2.5818
1114	190.88	-307.00	-42665.	.16884E+07	3.7511
1115	88.267	-141.96	-43450.	.78075E+06	9.2778

MINIMUM VALUES					
ELEM	1067	1091	1115	1067	1068
VALUE	2.2491	-1026.3	-43450.	19894.	-35.223

MAXIMUM VALUES					
ELEM	1091	1067	1067	1091	1067
VALUE	668.79	-3.6173	43477.	.84584E+07	91.127

Top/Bottom Horizontal Girder: Maximum values of the design results are: Shear force = 36777.0 kg, Bending moment = 6987630.0 kg-cm, Deflection = 0.53 cm, Bending stress, on top flange = 728.03 kg/cm^2, bottom flange = -1019.18 kg/cm^2 (ref: Appendix –A, page –23).

It is distinct from the FEM RESULTS in Fig: 5.11, 5.12 and contour legends that, maximum values of shear force 35222 kg, bending moment 6780000 kg-cm, deflection 0.576006 cm, bending stresses at the centre of span on top flange (skin plate): 706.213 kg/cm^2 (tension) and bottom flange: -988.618 kg/cm^2 (compression). For details Element Table results are also listed in ET.5.2.

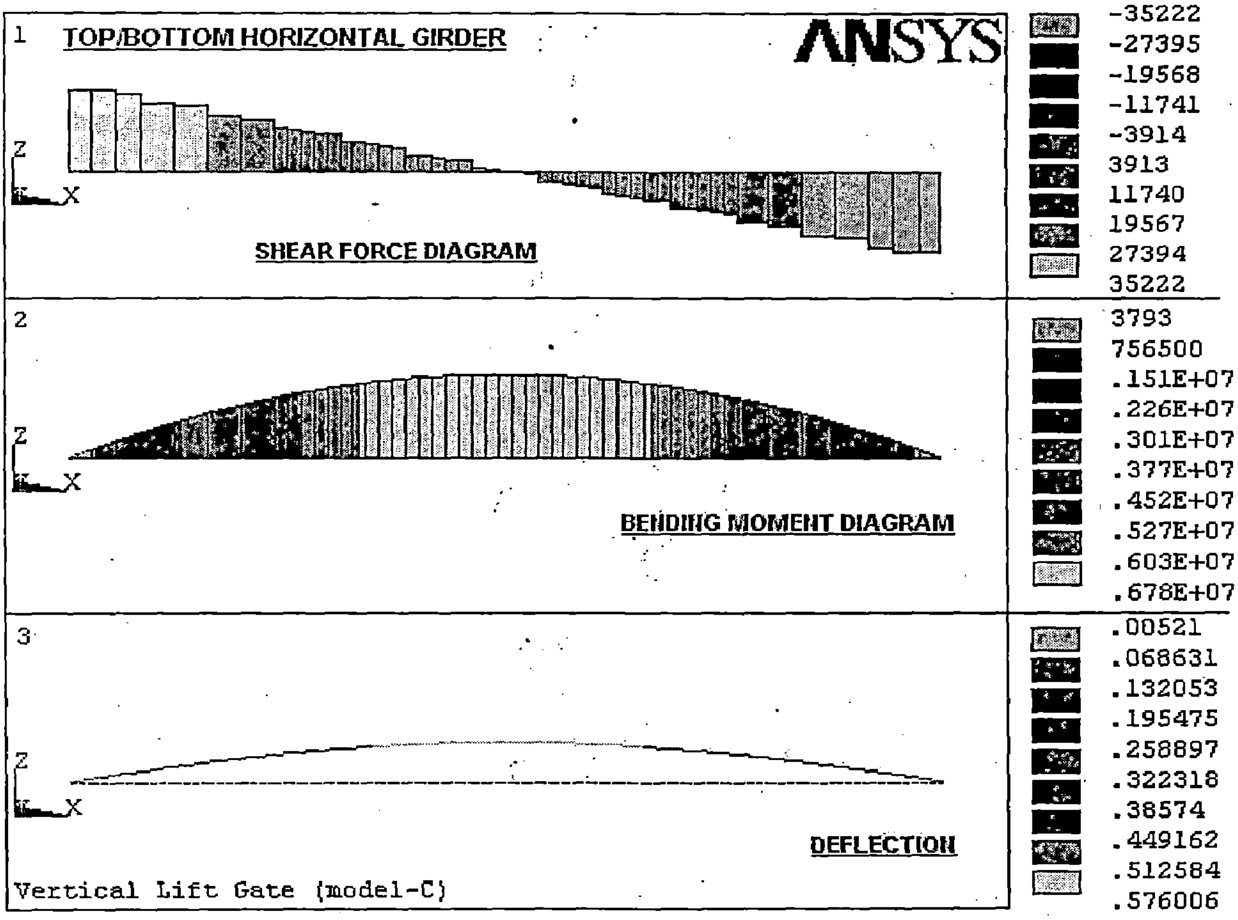


Fig: 5.11 Top/Bottom H Girder Contour Plots Shear Force, Bending Moment & Deflection.

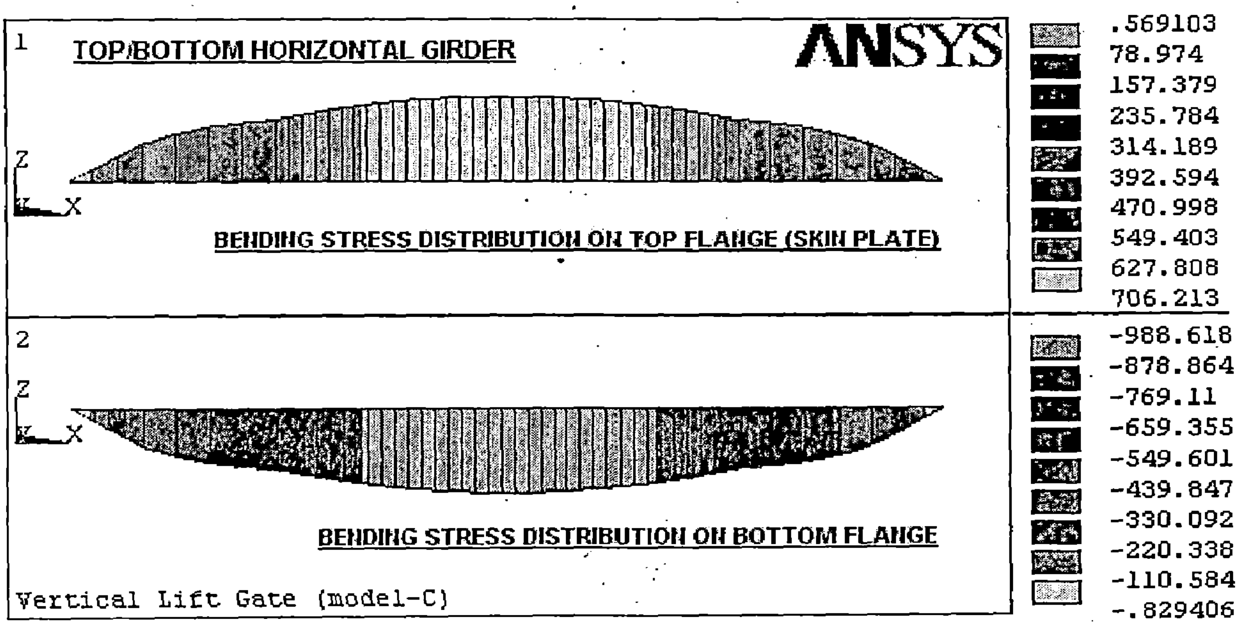


Fig: 5.12 Top/Bottom H Girder Contour Plots Bending Stresses on Top & Bottom Flange.

ET: 5.2 PRINT ELEMENT TABLE ITEMS PER ELEMENT **(THG/BHG)**

***** POST1 ELEMENT TABLE LISTING *****

STAT ELEM	CURRENT SBZT	CURRENT SBZB	CURRENT SFZ	CURRENT BMY	CURRENT BMX
1116	.59055	-.86067	35222.	3935.5	46437.
1117	93.222	-135.86	34505.	.62125E+06	46060.
1118	203.14	-296.05	33273.	.13537E+07	45607.
1119	309.14	-450.54	29235.	.20602E+07	35206.
1120	388.09	-559.64	27727.	.28642E+07	34823.
1121	447.40	-638.49	23513.	.36267E+07	26572.
1122	483.16	-682.69	22008.	.42733E+07	26215.
1123	508.45	-711.77	18262.	.48800E+07	19317.
1124	529.42	-741.13	17855.	.50814E+07	19172.
1125	549.90	-769.80	17055.	.52779E+07	18929.
1126	569.43	-797.14	16257.	.54654E+07	18688.
1127	588.02	-823.16	15855.	.56437E+07	18547.
1128	606.19	-848.59	12563.	.58181E+07	12467.
1129	620.63	-868.81	12161.	.59568E+07	12331.
1130	634.58	-888.34	11364.	.60907E+07	12097.
1131	647.59	-906.56	10566.	.62155E+07	11862.
1132	659.66	-923.44	10165.	.63313E+07	11725.
1133	671.31	-939.75	6879.8	.64431E+07	6025.6
1134	679.24	-950.86	6478.2	.65193E+07	5890.3
1135	686.68	-961.27	5680.9	.65906E+07	5657.1
1136	693.17	-970.36	4883.7	.66530E+07	5423.5
1137	698.72	-978.13	4482.2	.67063E+07	5287.7
1138	703.86	-985.32	1200.0	.67556E+07	-245.75
1139	705.28	-987.32	798.47	.67692E+07	-381.00
1140	**706.21**	**-988.62**	**1.2463**	**.67781E+07**	**-614.07**
1141	706.20	-988.60	-795.98	.67780E+07	-847.10
1142	705.24	-987.26	-1197.5	.67688E+07	-982.29
1143	703.87	-985.33	-4479.9	.67556E+07	-6528.9
1144	698.78	-978.21	-4881.4	.67068E+07	-6664.9
1145	693.20	-970.40	-5678.7	.66533E+07	-6898.6
1146	686.68	-961.27	-6476.0	.65907E+07	-7132.0
1147	679.21	-950.81	-6877.6	.65190E+07	-7267.3
1148	671.33	-939.78	-10163.	.64433E+07	-13009.
1149	659.72	-923.54	-10565.	.63320E+07	-13146.
1150	647.63	-906.61	-11362.	.62159E+07	-13382.
1151	634.59	-888.36	-12160.	.60908E+07	-13616.
1152	620.61	-868.79	-12561.	.59566E+07	-13752.
1153	606.22	-848.64	-15855.	.58184E+07	-19913.
1154	588.09	-823.26	-16257.	.56445E+07	-20054.
1155	569.48	-797.20	-17055.	.54658E+07	-20296.
1156	549.92	-769.82	-17855.	.52780E+07	-20540.
1157	529.41	-741.11	-18262.	.50812E+07	-20685.
1158	508.32	-711.59	-22011.	.48788E+07	-27716.
1159	483.18	-682.72	-23515.	.42735E+07	-28074.
1160	447.41	-638.51	-27731.	.36268E+07	-36387.
1161	388.09	-559.64	-29239.	.28642E+07	-36770.
1162	309.22	-450.66	-33274.	.20607E+07	-47031.
1163	203.23	-296.19	-34506.	.13544E+07	-47481.
1164	93.063	-135.63	-35222.	.62018E+06	-47856.

MINIMUM VALUES					
ELEM	1116	1140	1164	1116	1164
VALUE	.59055	-988.62	-35222.	3935.5	-47856.

MAXIMUM VALUES					
ELEM	1140	1116	1116	1140	1116
VALUE	706.21	-.86067	35222.	.67781E+07	46437.

5.2.5 Validation Of Horizontal Girders FEM Result

A comparison between conventionally design results and FEM results of horizontal girders is presented in table: 5.3.

Table: 5.3 Comparison of DESIGN and FEM results of horizontal girders.

Name and Location of maximum value	Design results	FEM		Variation (%)
		Results	Ref. fig./ET	
a) Central horizontal girder:				
Shear force in kg (Begin/end)	+/-42435.0	+/-43477.0	Fig: 5.9, ET: 5.1	+2.45
Bending moment in kg-cm (Centre of span)	8062650.0	8458400.0	,,	+4.91
Bending stresses in kg/cm^2 (centre of span)				
Top flange (skin plate)	637.54	668.79	Fig: 5.10, ET: 5.1	+8.04
Bottom flange	-978.20	-1026.30	,,	+4.92
Deflection (centre of span)	0.54	0.568701	Fig: 5.9	+5.31
b) Top/Bottom horizontal girder:				
Shear force in kg (Begin/end)	+/-36777.0	+/-35222	Fig: 5.11, ET: 5.2	-4.23
Bending moment in kg-cm (Centre of span)	6987630.0	6778100.0	,,	-3.00
Bending stresses in kg/cm^2 (centre of span)				
Top flange (skin plate)	728.03	706.21	Fig: 5.12, ET: 5.2	-3.00
Bottom flange	-1019.18	-988.62	,,	-3.00
Deflection (centre of span)	0.54	0.576006	Fig: 5.11	+6.67

From the Table: 5.3 it appears that FEM shear force as well bending moment on central horizontal girder is more than design values whereas on the top/bottom horizontal girders are less; this indicates, equal sharing of load for equal spacing by the horizontal girders as assumed in design consideration is deviating. So, FEM results are reflecting accordingly. However, FEM RESULTS are quiet close to design results with little variations, which are within acceptable limit.

5.2.6 Vertical End Girder FEM Result

The vertical lift gate under study consists of two vertical end girders at the left and right most end of the gate. In all respect, such as, geometry, loads, both the VE girders are same (ref. appendix-A, page-30). Therefore, load behavior of both the girders is same, and in that case FEM RESULTS are presented only for the left/right girder. For ready reference, maximum values of the design results are also mentioned here. Shear force (T/B HG): +/-36777.0 kg, Bending moment (wheel support point): -1195253 kg-cm, Bending stresses, top flange (skin plate): -431.75 kg/cm^2 (compression) & bottom flange: 230.49 kg/cm^2 (tension).

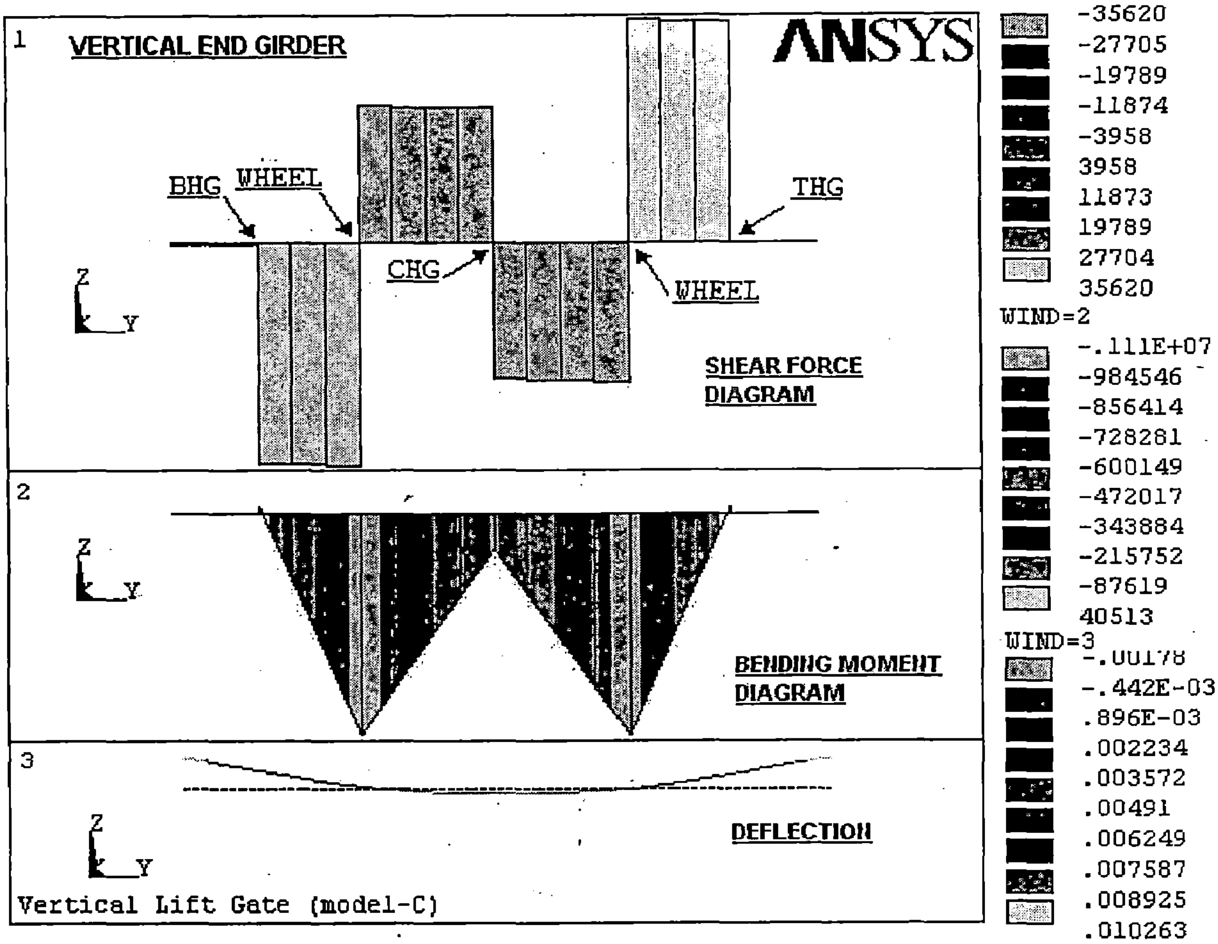


Fig: 5.13 VE Girder Contour Plots Shear Force, Bending Moment & Deflection.

It is distinct from the FEM RESULTS in Fig: 5.13 and respective contour legends that, maximum values of shear force at the points of bottom/top horizontal girder are +/-35620 kg, bending moment at the wheel support points are –0.111E+07 kg-cm, deflection at the both ends are 0.010263 cm and centre –0.00178 cm.

It is noticeable from Fig: 5.14 and contour legends that, bending stresses at the wheel support points on top flange (skin plate): -401.917 kg/cm^2 (compression) and bottom flange: 214.581 kg/cm^2 (tension). For details Element Table results are also listed in ET: 5.3.

FEM RESULT

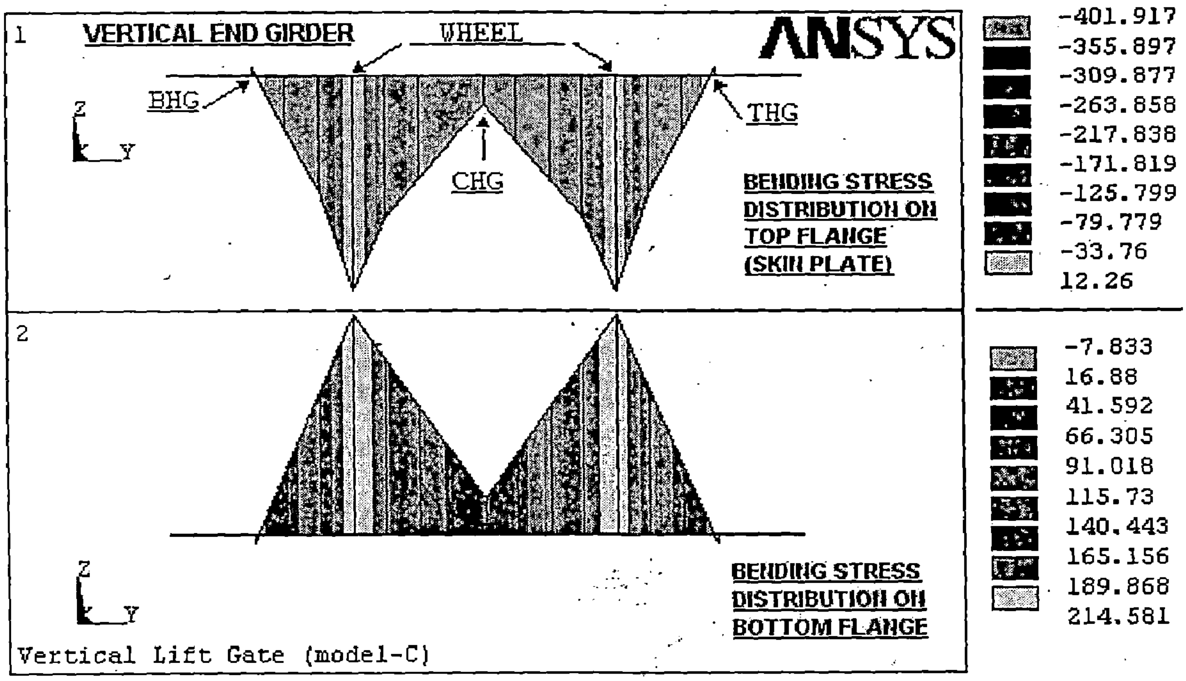


Fig: 5.14 VE Girder Contour Plots Bending Stresses on Top & Bottom Flange.

ET: 5.3 PRINT ELEMENT TABLE ITEMS PER ELEMENT (VEG)

***** POST1 ELEMENT TABLE LISTING *****

STAT ELEM	CURRENT SBZT	CURRENT SBZB	CURRENT SFZ	CURRENT BMY	CURRENT BMX
1214	.21929E-01	-.14010E-01	-155.11	72.463	1118.7
1215	-.38768	.24769	-251.93	-1281.1	2452.7
1216	-1.1046	.70574	-249.66	-3650.2	3001.1
1217	12.260	-7.8328	-35411.	40512.	-780.42
1218	-103.80	66.316	-35438.	-.34300E+06	57.414
1219	-219.92	140.51	-35620.	-.72672E+06	2053.2
1220	**-401.91**	**214.58**	**22131.**	**-.11127E+07**	**4615.2** (WHEEL)
1221	-265.59	169.69	21888.	-.87764E+06	7244.1
1222	-195.25	124.75	21714.	-.64521E+06	9203.5
1223	-125.47	80.162	21686.	-.41461E+06	9968.3
1224	-55.741	35.614	-21694.	-.18420E+06	-9967.1
1225	-125.40	80.119	-21728.	-.41438E+06	-9215.6
1226	-195.21	124.72	-21889.	-.64507E+06	-7274.2
1227	-265.55	169.66	-22133.	-.87752E+06	-4648.8
1228	**-401.89**	**214.57**	**35620.**	**-.11126E+07**	**-2085.3** (WHEEL)
1229	-219.97	140.54	35437.	-.72690E+06	-94.629
1230	-103.83	66.338	35411.	-.34311E+06	748.29
1231	-1.7973	1.1483	249.72	-5939.4	-3000.4
1232	-1.0865	.69421	251.94	-3590.5	-2452.5
1233	-.40834	.26089	155.11	-1349.4	-1118.7

MINIMUM VALUES					
ELEM	1220	1217	1219	1220	1224
VALUE	-401.91	-7.8328	-35620.	-.11127E+07	-9967.1
MAXIMUM VALUES					
ELEM	1217	1220	1228	1217	1223
VALUE	12.260	214.58	35620.	40512.	9968.3

5.2.7 Validation Of Vertical End Girder FEM Result

A comparison between conventionally design results and FEM results of vertical end girder left/right is presented in Table: 5.4.

Table: 5.4 Comparison of DESIGN and FEM RESULTS of vertical end girder.

Name and Location of maximum value	Design results	FEM		Variation (%)
		Results	Ref. fig./ET	
Shear force in kg (T/B H Girders)	+/-36777.0	+/-35620.0	Fig: 5.13, ET: 5.3	-3.15
Bending moment in kg-cm (Wheel support points)	-1195253.0	-1112700.0	,,	-6.91
Bending stresses in kg/cm^2 (wheel support points)				
Top flange (skin plate)	-431.75	-401.91	Fig: 5.14, ET: 5.3	-6.91
Bottom flange	230.49	214.58	,,	-6.90

From the Table: 5.4 it appears that FEM shear force as well bending moment at the respective locations on vertical end girder is slightly less than design results and therefore, FEM stresses are reflecting accordingly. However, FEM RESULTS are quiet close to design results with a few variations due to FEM iteration errors that are within acceptable boundary.

5.2.8 Vertical Stiffeners FEM Result

The vertical lift gate under study consists of twelve vertical stiffeners. All the stiffeners are equally spaced and geometrically same (ref. appendix-A, page-5, 10). The gate and therefore, results are symmetric on both sides of the mid-span. As such, FEM RESULTS are presented for six set vertical stiffeners from the left side.

For ready reference, design results are re-mentioned here. Maximum values are: Shear force = 3382.50 kg, Bending moment = 63421.90 kg-cm, Bending stress, on top flange (skin plate) = 270.74 kg/cm^2, bottom web = 588.14 kg/cm^2 (ref: Appendix –A, page –14, art. 6.8).

Vertical Stiffener-1: It is obvious from the FEM RESULTS as presented in Fig. 5.15 and contour legends that, maximum values of shear force at the location of central horizontal girder are +/-2376 kg and bending moment at the same location is – 44546 kg-cm.

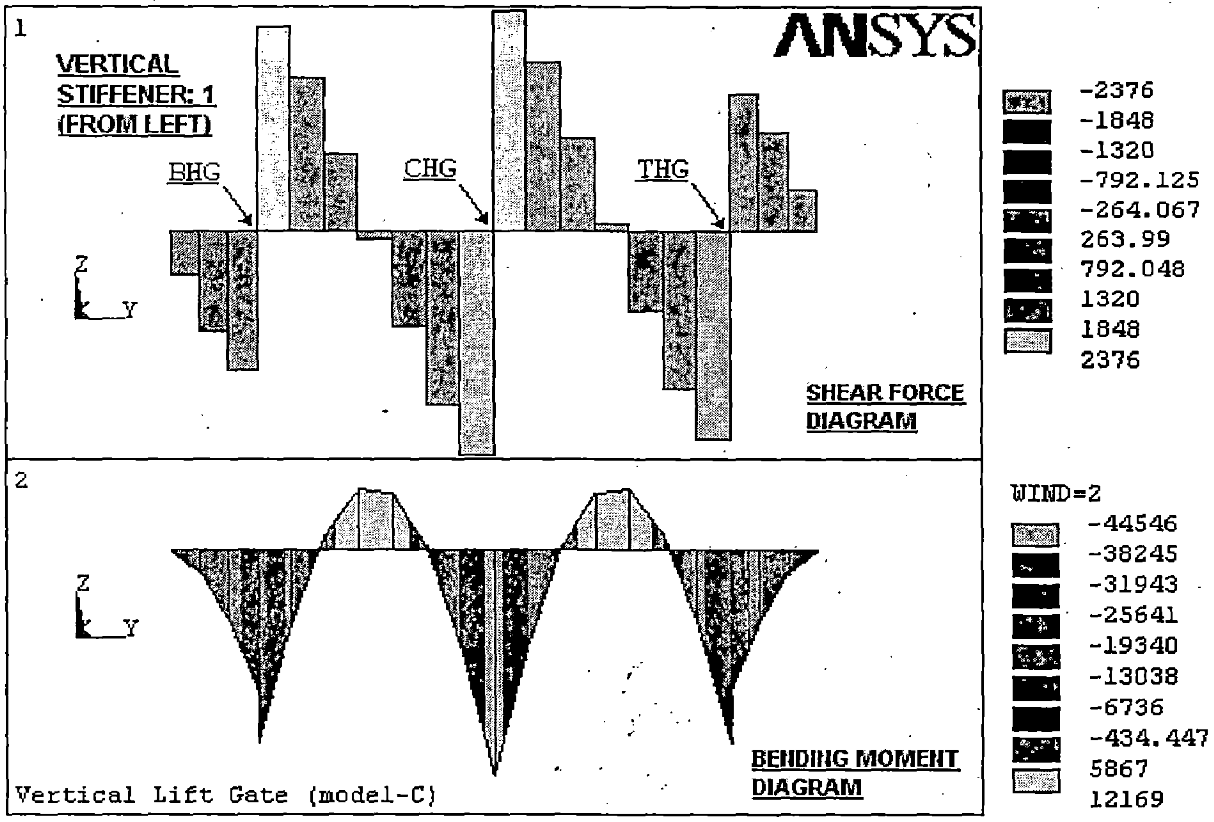


Fig: 5.15 V stiffener-1 Contour Plots Shear Force, Bending Moment.

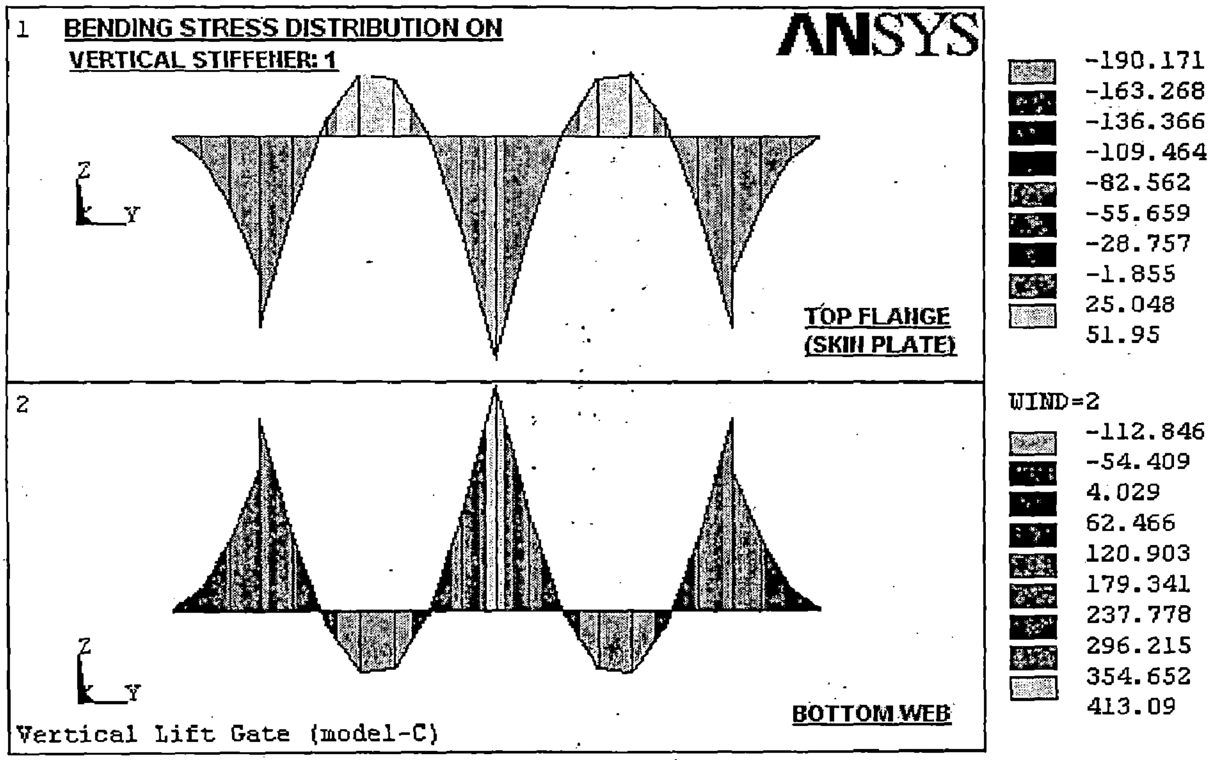


Fig: 5.16 V stiffener-1 Contour Plots Bending Stresses on Top flange & Bottom web.

It is visible from Fig: 5.16 and contour legends that, maximum bending stresses at top flange (skin plate) -190.17 kg/cm^2 (comp) and bottom web 413.09 kg/cm^2 (tension).

For details Element Table results are also listed in ET. 5.4.

ET: 5.4 PRINT ELEMENT TABLE ITEMS PER ELEMENT (VS-1)
(As per model position of horizontal girders are shown here only)

STAT ELEM	CURRENT SBZT	CURRENT SBZB	CURRENT SFZ	CURRENT BMY	CURRENT BMX	
1254	-2.0611	4.4772	-466.22	-482.81	-216.24	
1255	-18.870	40.989	-1077.2	-4420.2	-369.11	
1256	-60.506	131.43	-1491.0	-14173.	-323.42	(BHG)
1290	-162.77	353.58	2204.5	-38129.	466.82	
1291	-62.274	135.27	1667.5	-14587.	490.67	
1292	12.782	-27.765	856.32	2994.1	317.91	
1293	51.433	-111.72	-85.769	12048.	49.502	
1294	48.052	-104.38	-1027.9	11256.	-221.58	
1295	2.2799	-4.9523	-1839.2	534.04	-401.01	
1296	-81.479	176.99	-2376.3	-19086.	-382.55	(CHG)
1374	-190.15	413.04	2376.2	-44541.	378.00	
1375	-81.825	177.74	1838.7	-19167.	396.32	
1376	1.0370	-2.2525	1027.6	242.91	218.63	
1377	47.496	-103.17	85.803	11126.	-48.856	
1378	51.937	-112.82	-856.18	12166.	-314.82	
1379	13.992	-30.393	-1667.3	3277.5	-486.35	
1380	-61.938	134.54	-2204.3	-14509.	-462.00	(THG)
1458	-118.85	258.17	1491.0	-27841.	323.42	
1459	-61.025	132.56	1077.2	-14295.	369.11	
1460	-20.306	44.108	466.22	-4756.5	216.24	

MINIMUM VALUES

ELEM	1374	1378	1296	1374	1379
VALUE	-190.15	-112.82	-2376.3	-44541.	-486.35

MAXIMUM VALUES

ELEM	1378	1374	1374	1378	1291
VALUE	51.937	413.04	2376.2	12166.	490.67

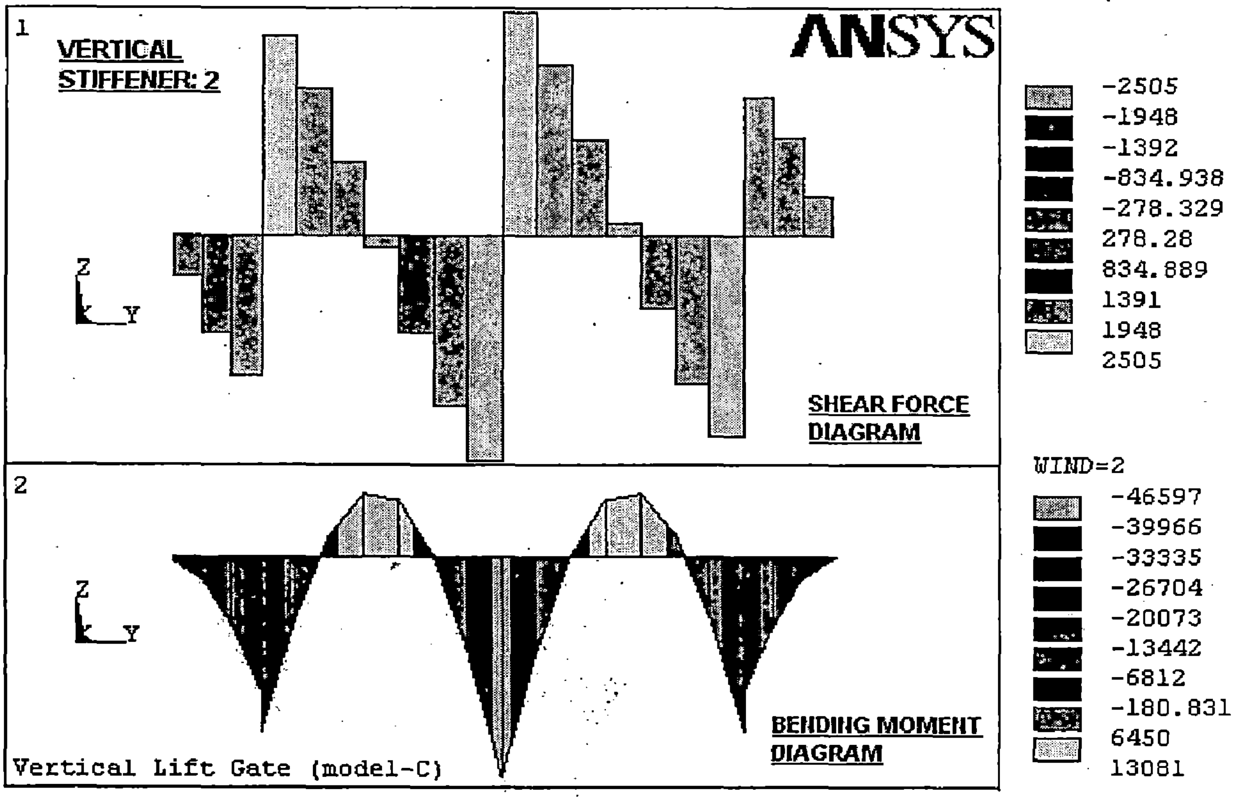


Fig: 5.17 V stiffener-2 Contour Plots Shear Force, Bending Moment.

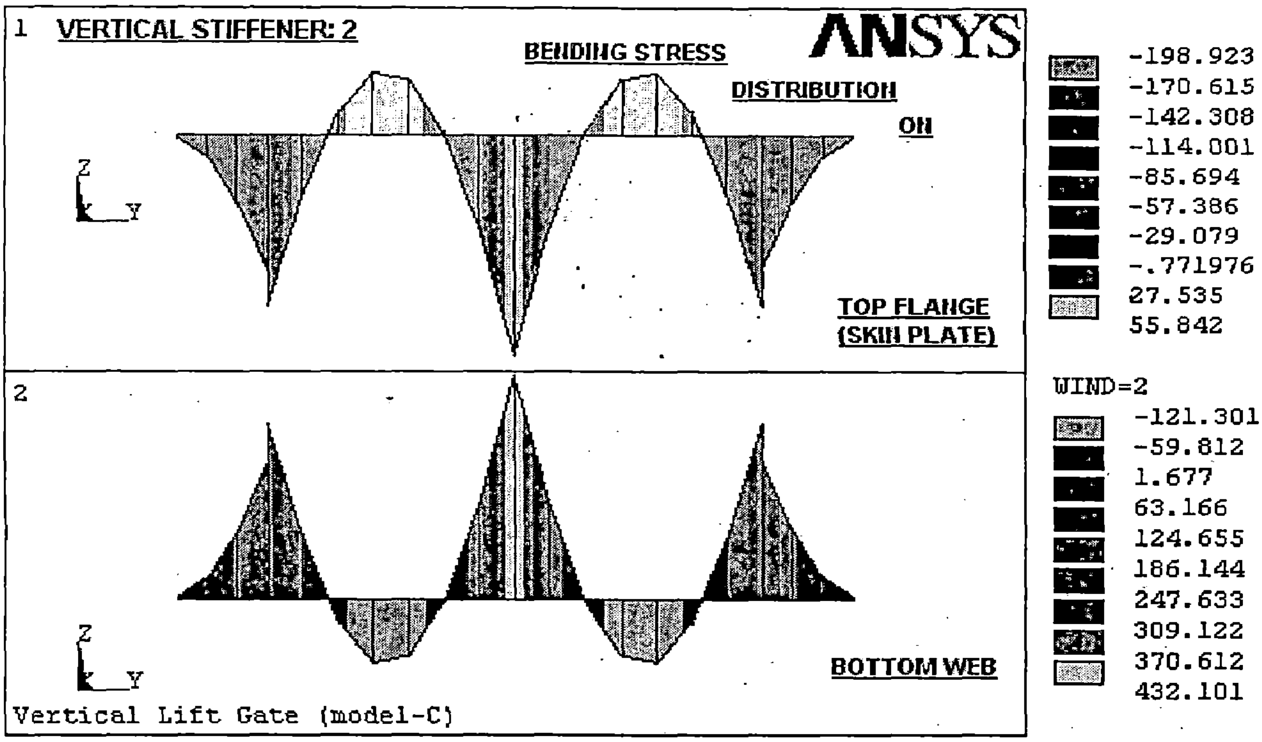


Fig: 5.18 V stiffener-2 Contour Plots Bending Stresses on Top flange & Bottom web.

ET: 5.5 PRINT ELEMENT TABLE ITEMS PER ELEMENT (VS-2)

STAT ELEM	CURRENT SBZT	CURRENT SBZB	CURRENT SFZ	CURRENT BMY	CURRENT BMX
1257	-3.2384	7.0344	-445.99	-758.58	-3.8870
1258	-19.053	41.388	-1089.2	-4463.1	-3.4973
1259	-60.701	131.86	-1558.5	-14219.	5.8311
1297	-156.55	340.06	2238.8	-36672.	-20.385
1298	-55.140	119.78	1646.1	-12916.	-2.9101
1299	18.548	-40.290	815.36	4344.8	10.643
1300	55.196	-119.90	-132.76	12929.	22.152
1301	49.827	-108.23	-1080.9	11672.	31.975
1302	2.0467	-4.4459	-1911.9	479.43	41.304
1303	-84.355	183.24	-2504.8	-19760.	55.420
1381	**-198.91**	**432.08**	**2504.7**	**-46595.**	**-55.337**
1382	-85.395	185.50	1911.8	-20003.	-41.228
1383	.38933	-.84571	1080.9	91.199	-31.914
1384	49.126	-106.71	132.71	11507.	-22.113
1385	55.842	-121.30	-815.41	13081.	-10.627
1386	20.149	-43.769	-1646.2	4719.9	2.9115
1387	-54.158	117.64	-2238.8	-12686.	20.379
1461	-121.69	264.33	1558.5	-28505.	-5.8271
1462	-61.678	133.98	1089.2	-14448.	3.5005
1463	-20.691	44.945	445.99	-4846.8	3.8885

MINIMUM VALUES					
ELEM	1381	1385	1303	1381	1381
VALUE	-198.91	-121.30	-2504.8	-46595.	-55.337

MAXIMUM VALUES					
ELEM	1385	1381	1381	1385	1303
VALUE	55.842	432.08	2504.7	13081.	55.420

Note: Since shear force and bending moment diagrams are almost similar with tiny variation of magnitude, so only stress contours and element table listings are presented.

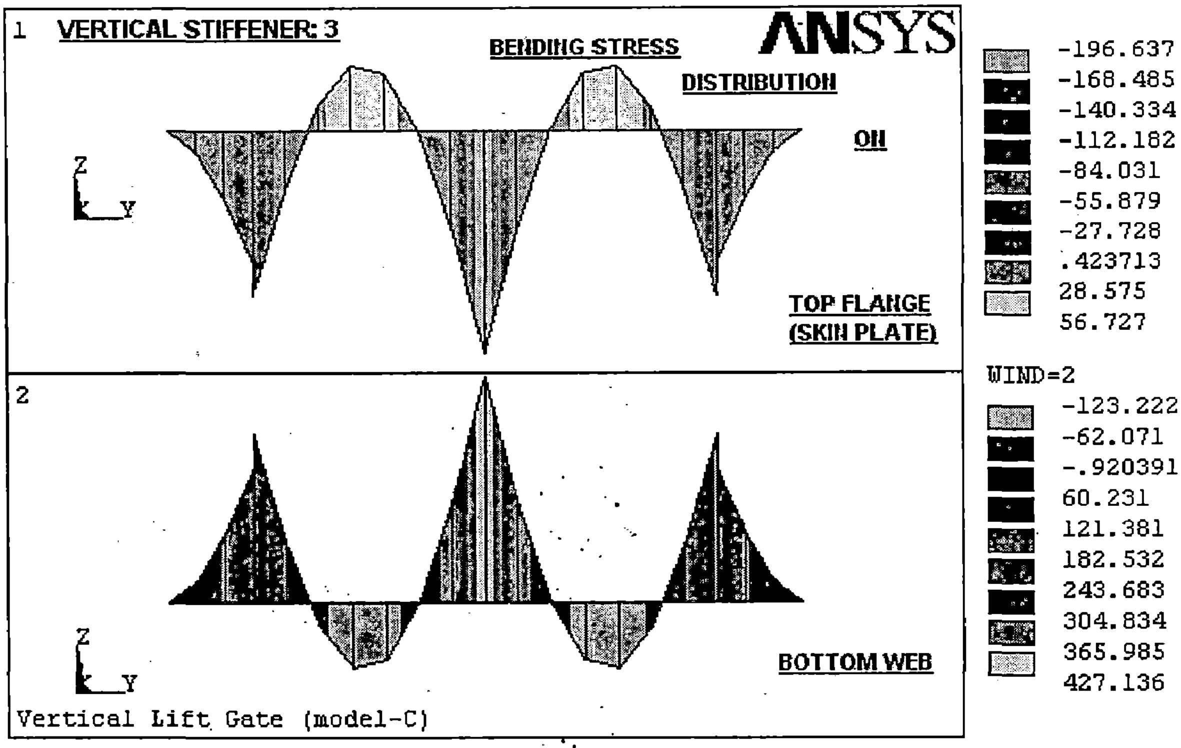


Fig: 5.19 V stiffener-3 Contour Plots Bending Stresses on Top flange & Bottom web.

ET: 5.6 PRINT ELEMENT TABLE ITEMS PER ELEMENT (VS-3)

STAT ELEM	CURRENT SBZT	CURRENT SBZB	CURRENT SFZ	CURRENT BMY	CURRENT BMX
1260	-.93127	2.0229	-481.00	-218.14	464.39
1261	-19.039	41.358	-1080.8	-4459.9	873.57
1262	-61.318	133.20	-1436.3	-14363.	805.47
1304	-146.68	318.62	2072.7	-34359.	-1172.5
1305	-51.520	111.91	1592.0	-12068.	-1172.1
1306	20.759	-45.094	786.35	4862.8	-648.35
1307	56.462	-122.65	-156.33	13226.	6.2583
1308	49.612	-107.77	-1099.0	11621.	659.83
1309	-.83740E-01	.18190	-1904.7	-19.616	1180.9
1310	-87.523	190.12	-2385.5	-20502.	1178.8
1388	**-196.63**	**427.12**	**2385.5**	**-46059.**	**-1178.8**
1389	-87.200	189.42	1904.7	-20426.	-1180.9
1390	-.65351	1.4195	1099.0	-153.08	-659.82
1391	49.314	-107.12	156.28	11551.	-6.2445
1392	56.726	-123.22	-786.39	13288.	648.36
1393	21.294	-46.254	-1592.0	4987.9	1172.1
1394	-51.879	112.69	-2072.7	-12152.	1172.5
1464	-117.52	255.28	1436.3	-27529.	-805.47
1465	-61.336	133.23	1080.8	-14368.	-873.57
1466	-19.754	42.910	481.00	-4627.3	-464.39
MINIMUM VALUES					
ELEM	1388	1392	1310	1388	1389
VALUE	-196.63	-123.22	-2385.5	-46059.	-1180.9
MAXIMUM VALUES					
ELEM	1392	1388	1388	1392	1309
VALUE	56.726	427.12	2385.5	13288.	1180.9

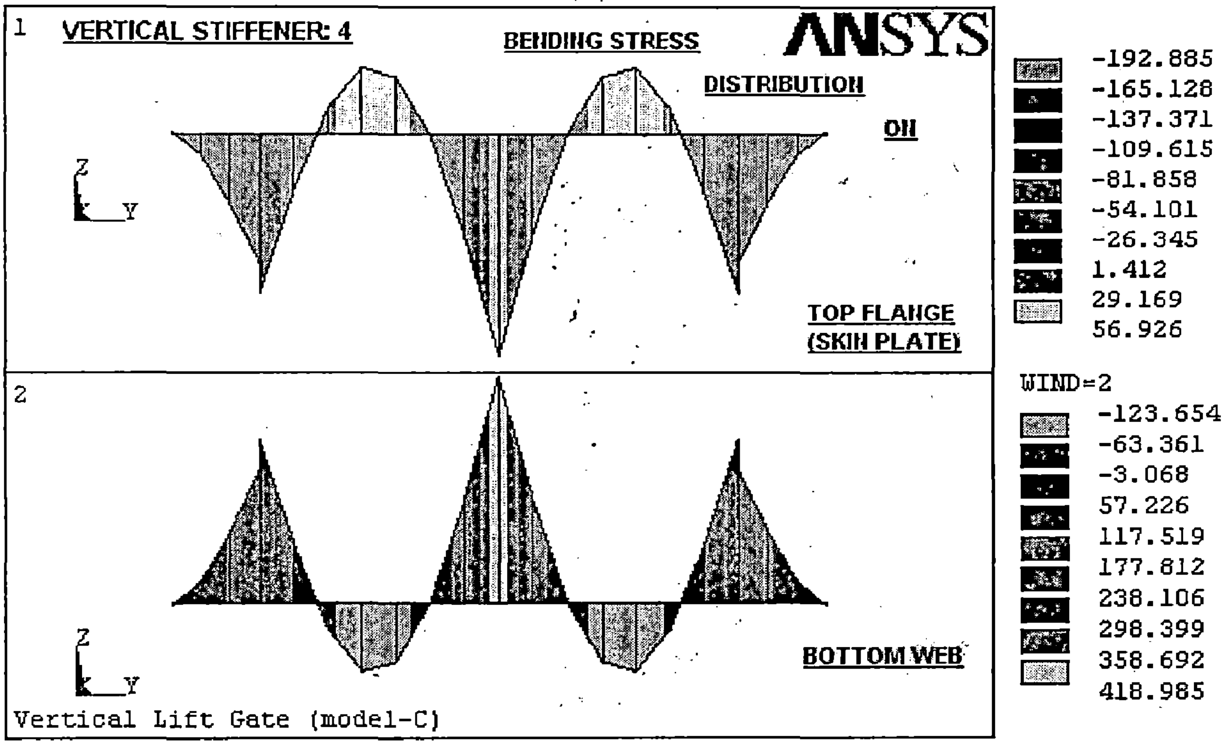


Fig: 5.20 V stiffener-4 Contour Plots Bending Stresses on Top flange & Bottom web.

ET: 5.7 PRINT ELEMENT TABLE ITEMS PER ELEMENT (VS-4)

STAT	CURRENT	CURRENT	CURRENT	CURRENT	CURRENT
ELEM	SBZT	SBZB	SFZ	BMY	BMX
1263	1.3790	-2.9954	-515.27	323.02	-3.0701
1264	-19.019	41.313	-1071.9	-4455.1	-7.8377
1265	-61.915	134.49	-1314.4	-14503.	-10.113
1311	-139.03	302.01	1917.4	-32568.	7.8715
1312	-49.632	107.81	1547.4	-11626.	8.6667
1313	21.694	-47.123	767.62	5081.6	5.8087
1314	56.926	-123.65	-168.20	13335.	-.41259
1315	49.117	-106.69	-1104.0	11505.	-7.2502
1316	-1.9223	4.1757	-1883.9	-450.29	-11.780
1317	-89.788	195.04	-2254.0	-21032.	-12.903
1395	**-192.88**	**418.97**	**2254.0**	**-45181.**	**12.909**
1396	-88.088	191.35	1883.9	-20634.	11.785
1397	-1.3790	2.9954	1104.0	-323.01	7.2542
1398	49.233	-106.94	168.16	11533.	.41591
1399	56.803	-123.39	-767.66	13306.	-5.8057
1400	21.145	-45.931	-1547.5	4953.1	-8.6637
1401	-51.338	111.52	-1917.4	-12026.	-7.8682
1467	-113.35	246.22	1314.4	-26552.	10.115
1468	-60.964	132.43	1071.9	-14281.	7.8387
1469	-18.785	40.805	515.27	-4400.2	3.0705

MINIMUM VALUES					
ELEM	1395	1314	1317	1395	1317
VALUE	-192.88	-123.65	-2254.0	-45181.	-12.903

MAXIMUM VALUES					
ELEM	1314	1395	1395	1314	1395
VALUE	56.926	418.97	2254.0	13335.	12.909

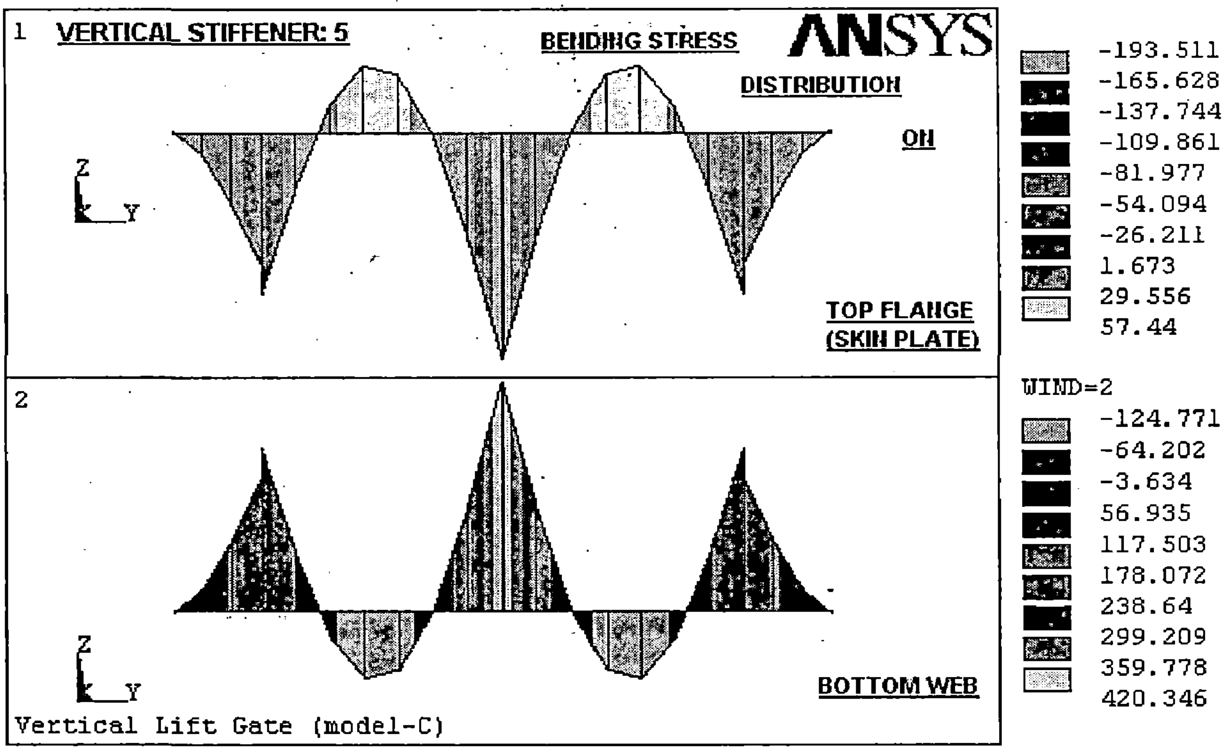


Fig: 5.21V stiffener-5 Contour Plots Bending Stresses on Top flange & Bottom web.

ET: 5.8 PRINT ELEMENT TABLE ITEMS PER ELEMENT (VS-5)

STAT ELEM	CURRENT SBZT	CURRENT SBZB	CURRENT SFZ	CURRENT BMY	CURRENT BMX
1266	1.3617	-2.9579	-515.89	318.97	-.20008
1267	-19.057	41.396	-1072.0	-4464.1	-1.6017
1268	-61.959	134.59	-1314.5	-14513.	-2.7183
1318	-137.46	298.60	1909.7	-32200.	-5.7616
1319	-48.414	105.17	1539.6	-11341.	-4.7237
1320	22.552	-48.988	760.10	5282.7	-3.4596
1321	57.440	-124.77	-175.08	13455.	-2.3443
1322	49.317	-107.13	-1110.3	11552.	-1.5447
1323	-2.0049	4.3550	-1889.8	-469.63	-1.1364
1324	-90.138	195.80	-2260.0	-21114.	-1.0793
1402	**-193.51**	**420.33**	**2260.0**	**-45328.**	**1.0811**
1403	-88.441	192.11	1889.8	-20717.	1.1375
1404	-1.4640	3.1800	1110.2	-342.93	1.5451
1405	49.433	-107.38	175.04	11579.	2.3441
1406	57.318	-124.51	-760.14	13426.	3.4591
1407	22.005	-47.800	-1539.7	5154.6	4.7232
1408	-50.117	108.87	-1909.7	-11740.	5.7613
1470	-113.40	246.32	1314.5	-26563.	2.7189
1471	-61.009	132.52	1072.0	-14291.	1.6022
1472	-18.827	40.895	515.89	-4410.0	.20030

MINIMUM VALUES					
ELEM	1402	1321	1324	1402	1318
VALUE	-193.51	-124.77	-2260.0	-45328.	-5.7616

MAXIMUM VALUES					
ELEM	1321	1402	1402	1321	1408
VALUE	57.440	420.33	2260.0	13455.	5.7613

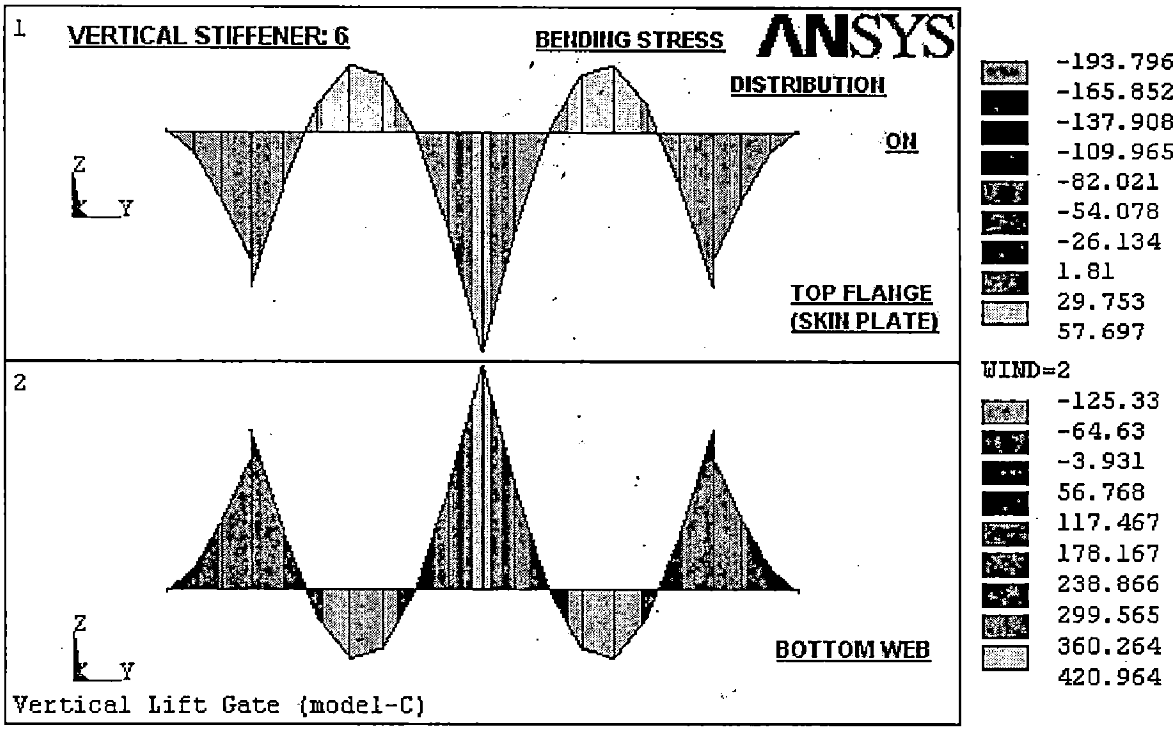


Fig: 5.22 V stiffener-6 Contour Plots Bending Stresses on Top flange & Bottom web.

ET: 5.9 PRINT ELEMENT TABLE ITEMS PER ELEMENT (VS-6)

STAT ELEM	CURRENT SBZT	CURRENT SBZB	CURRENT SFZ	CURRENT BMY	CURRENT BMX
1269	1.3540	-2.9412	-516.37	317.17	.37716E-02
1270	-19.082	41.451	-1072.3	-4470.0	-.37250
1271	-61.995	134.66	-1314.6	-14522.	-.69373
1325	-136.80	297.16	1906.7	-32045.	-1.7099
1326	-47.887	104.02	1536.7	-11217.	-1.4243
1327	22.944	-49.839	757.15	5374.5	-1.0820
1328	57.697	-125.33	-178.03	13515.	-.78690
1329	49.439	-107.39	-1113.2	11581.	-.57940
1330	-2.0182	4.3840	-1892.8	-472.76	-.47449
1331	-90.287	196.12	-2263.0	-21149.	-.46066
1409	**-193.79**	**420.95**	**2263.0**	**-45394.**	**.46212**
1410	-88.591	192.44	1892.8	-20752.	.47551
1411	-1.4777	3.2099	1113.2	-346.14	.57993
1412	49.554	-107.64	178.00	11608.	.78701
1413	57.575	-125.07	-757.19	13487.	1.0818
1414	22.397	-48.651	-1536.7	5246.4	1.4240
1415	-49.591	107.72	-1906.8	-11616.	1.7096
1473	-113.44	246.42	1314.6	-26573.	.69373
1474	-61.045	132.60	1072.3	-14300.	.37250
1475	-18.853	40.952	516.37	-4416.2	-.37736E-02

MINIMUM VALUES					
ELEM	1409	1328	1331	1409	1325
VALUE	-193.79	-125.33	-2263.0	-45394.	-1.7099

MAXIMUM VALUES					
ELEM	1328	1409	1409	1328	1415
VALUE	57.697	420.95	2263.0	13515.	1.7096

5.2.9 Validation Of Vertical Stiffeners FEM Result

It is obvious from the Fig: 5.15 - 5.22 and ET: 5.4 – 5.9 that, maximum values are occurring with central horizontal girder which also varying from: shear force: 2376.2 - 2504.7 kg, bending moment: -44541 - -46595 kg-cm, bending stress, top flange (skin plate): -190.15 - -198.91 kg/cm^2 and bottom web: 413.04 - 432.08 kg/cm^2.

A comparison between conventionally designed results and maximum FEM results of vertical stiffeners is presented in Table: 5.5.

Table: 5.5 Comparison of DESIGN and FEM RESULTS of vertical stiffeners.

Name and Location of maximum value	Design results	FEM		Variation (%)
		Results	Ref. fig./ET	
Shear force in kg (Central H Girders)	+/-3382.50	+/-2504.7	Fig: 5.17, ET: 5.5	-25.95
Bending moment in kg-cm (Central H Girders)	-63421.90	-46595	„	-26.53
Bending stresses in kg/cm^2 (Central H Girders)				
Top flange (skin plate)	-270.74	-198.91	Fig: 5.18 ET: 5.5	-26.53
Bottom web	588.14	432.08	„	-26.53

From the Table: 5.5 it appears that, FEM maximum shear force as well bending moment with the central horizontal girder are 25.95% & 26.53% less respectively than design results and therefore, FEM stresses are reflecting accordingly. This indicates that, assumption made in design calculation is deviating to some extent to conservation side. However, FEM results are parallel to design results with 26.53% variations due to some FEM iteration errors and as such, may be considered within acceptable boundary.

5.3 VALIDATION OF THE RESULTS OF VL GATE'S WHEEL

The wheel model is built up from SOLID92, three-dimensional 10-node tetrahedral structural solid element. From article 5.3.1 'Loads on the VL Gate' it is clear that, the reaction force at the four-wheel fixing points: 57995.0 kg which is exactly same as conventional design load. So, the same amount of varying distributed load is applied on the wheel axle bore as described in the previous chapter-4.

5.3.1 Wheel FEM Result

In conventional approach the wheel was designed as flat wheel with line contact in accordance with procedure given in Annex-F of IS: 4622. For ready reference maximum value stresses are mentioned here. Contact stress, f_c =7532.93 kg/cm^2 (permissible 9800 kg/cm^2), Shear stress, f_s = 2290.01 kg/cm^2 (permissible 4900 kg/cm^2).

In SOLID92, FEM results are available in X, Y & Z directions, XY, YZ & XZ planes as well as Von Mises Seqv in Nodal Solution & Element Solution, and therefore, results are presented accordingly. Since loads are applied in Y-direction so, FEM stresses are the most important in Y-direction as well as in XY plane.

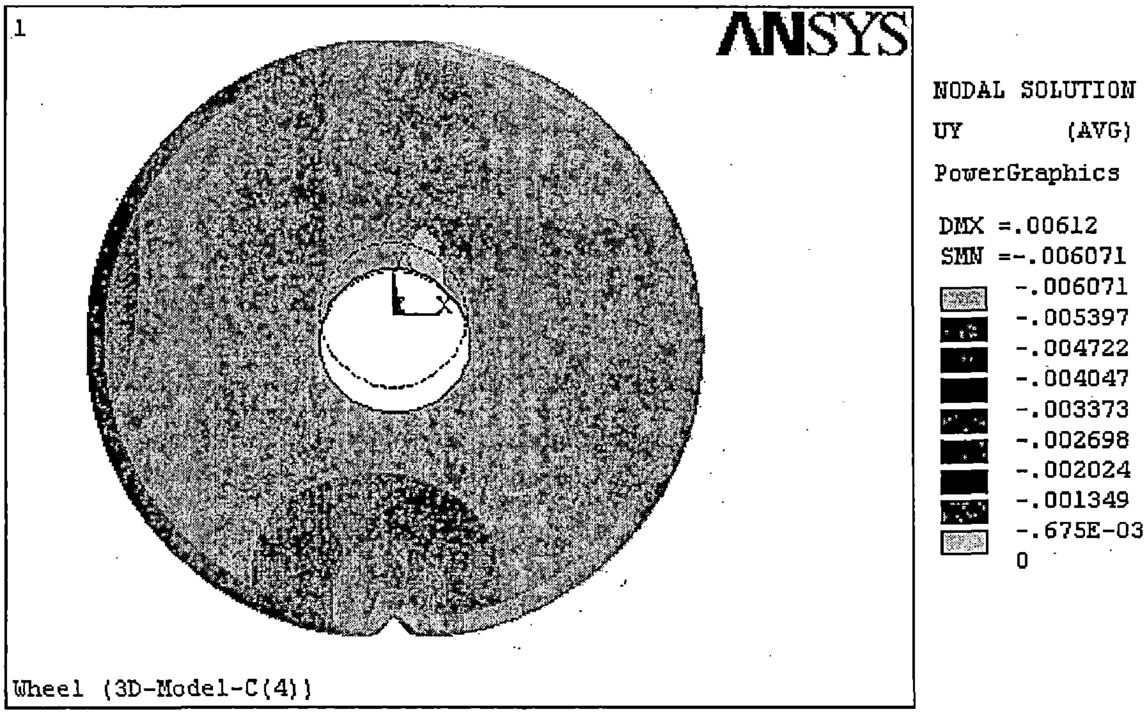


Fig: 5.23 Wheel Contour Plots Deformation UY.

From the Fig: 5.23 and contour legend it is noticeable that, maximum deformation: DMX = 0.00612 cm.

From the Fig: 5.24, 5.25, 5.26 and contour legends it is observed that, maximum stresses on the line of contact of wheel in X-direction, SX: -2207 kg/cm^2 (Nodal Solution) & -4182 kg/cm^2 (Element Solution), Y-direction, SY: -4728 kg/cm^2 (Nodal Solution) & -9070 kg/cm^2 (Element Solution), Z-direction, SZ: -1450 kg/cm^2 (Nodal Solution), -2603 kg/cm^2 (Element Solution) respectively.

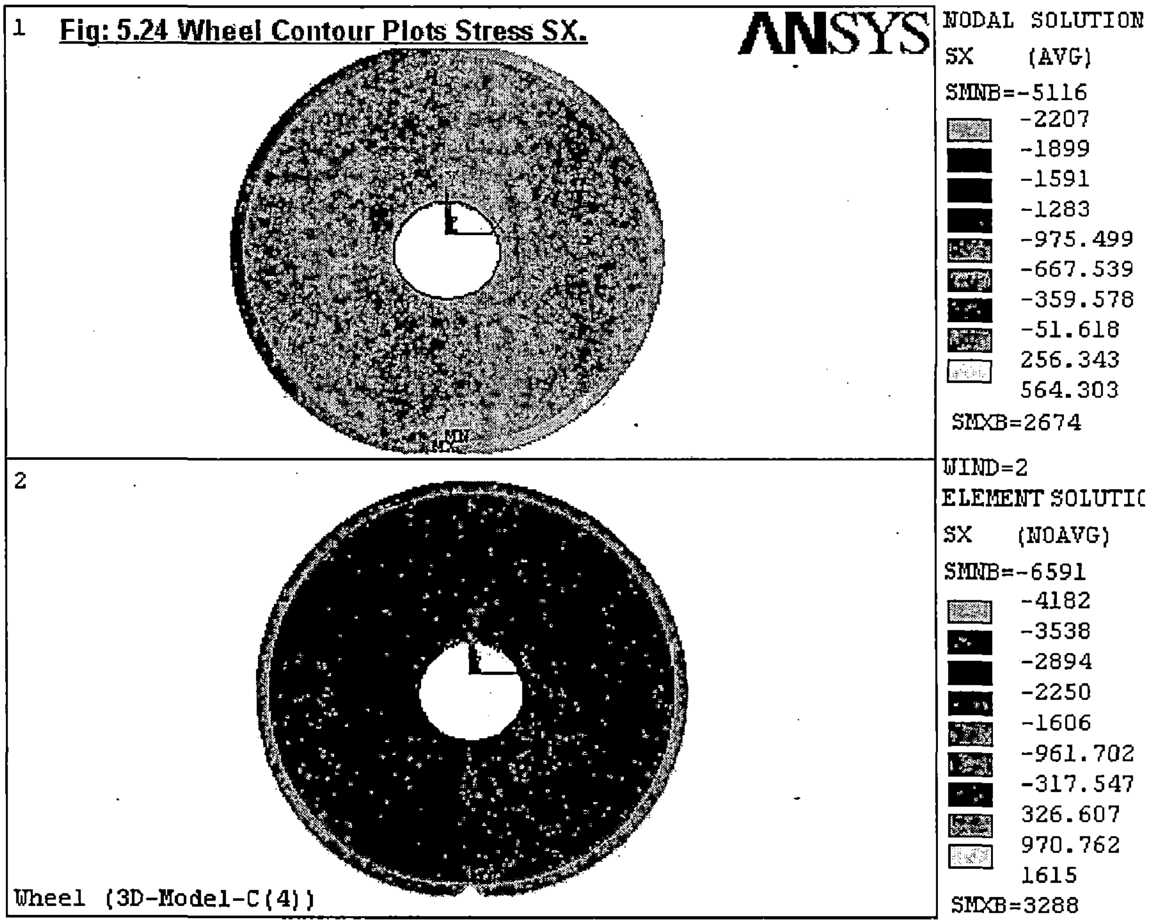


Fig: 5.24 Wheel Contour Plots Stress SX.

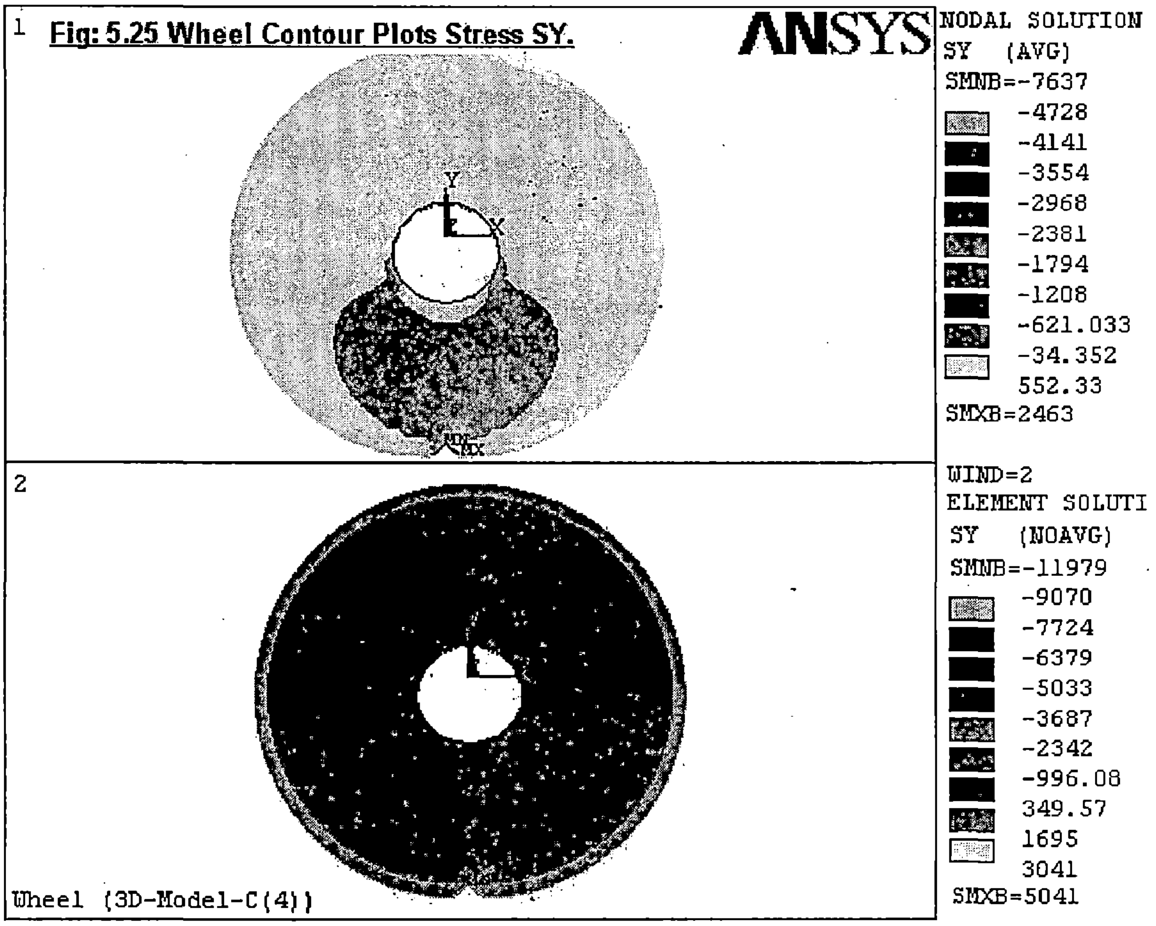


Fig: 5.25 Wheel Contour Plots Stress SY.

FEM RESULT

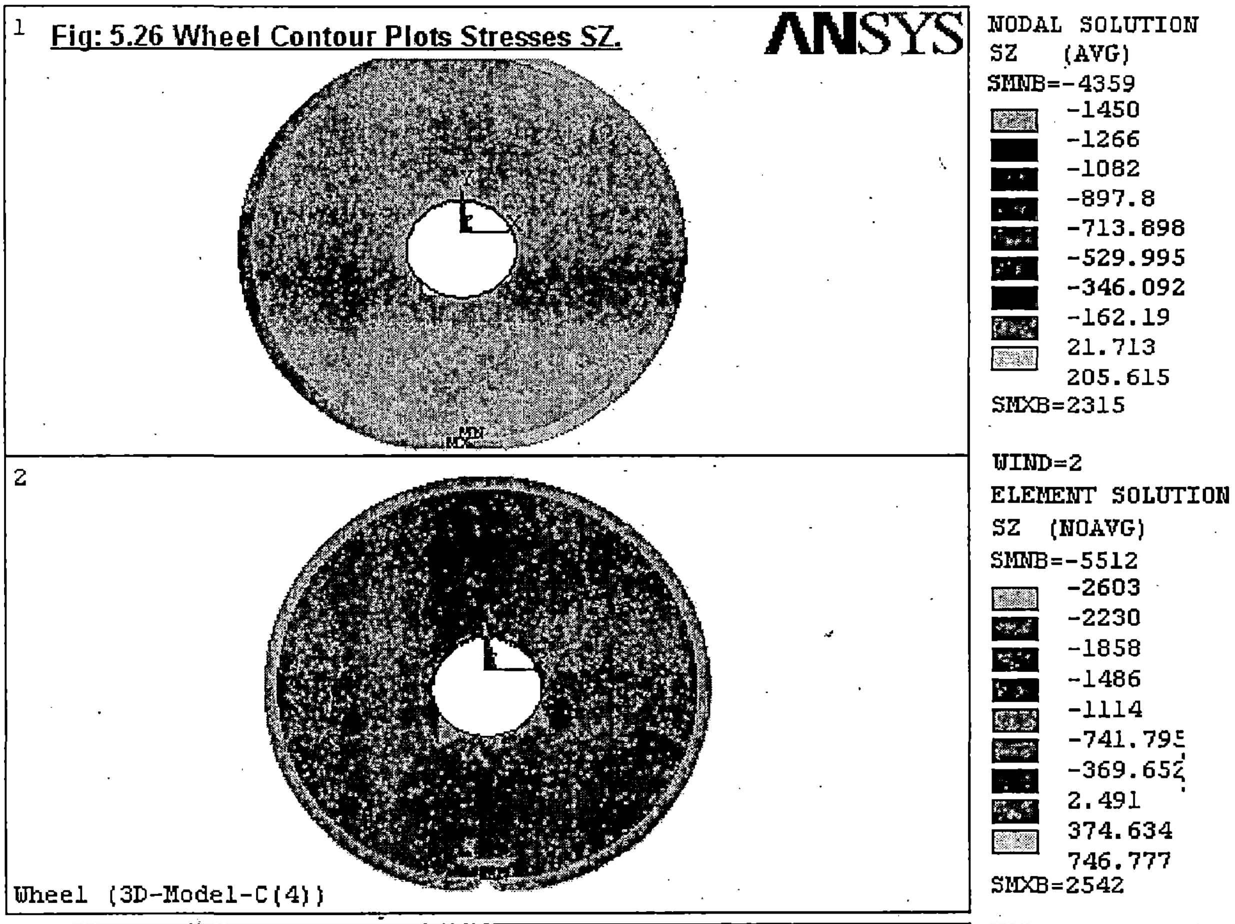


Fig: 5.26 Wheel Contour Plots Stresses SZ.

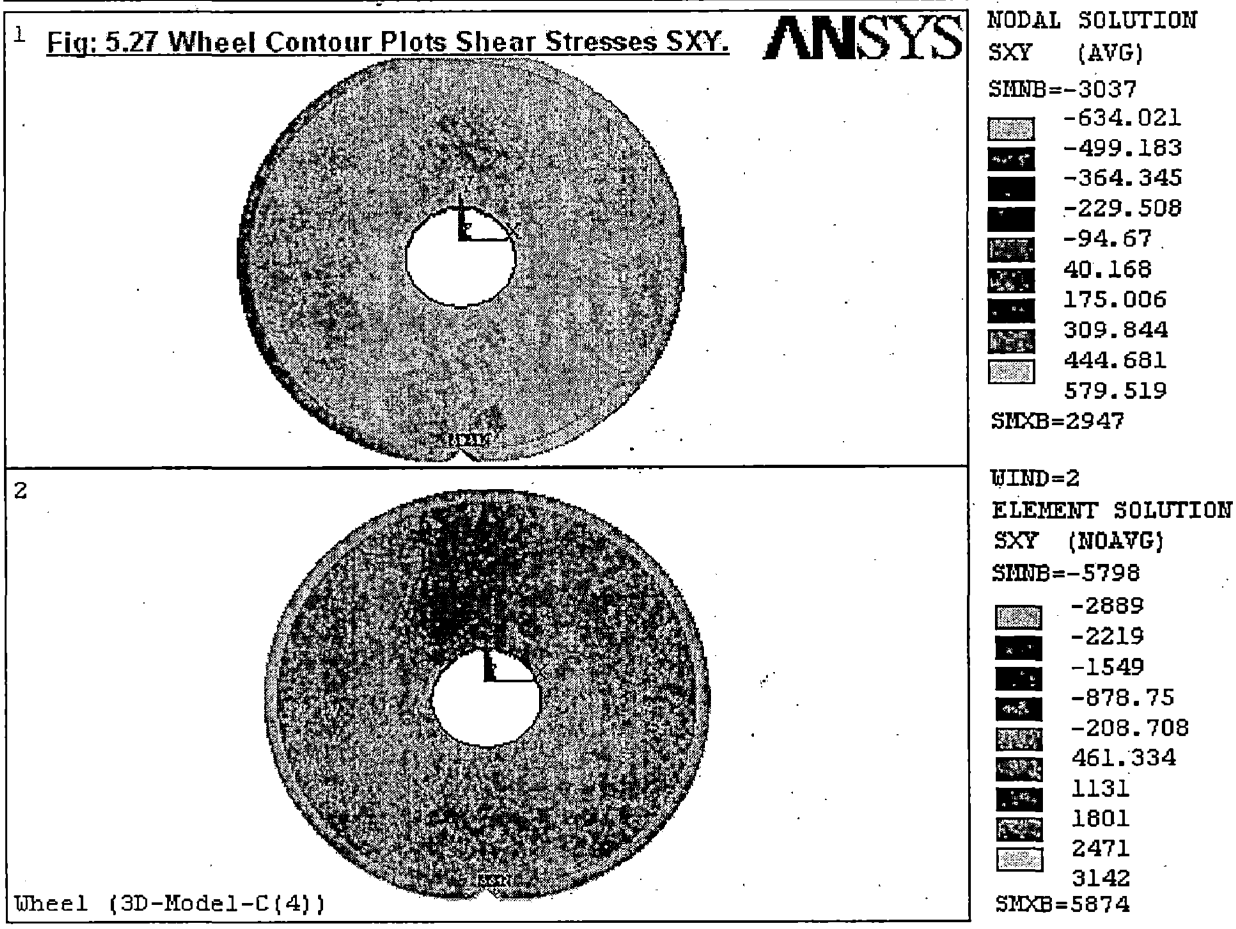


Fig: 5.27 Wheel Contour Plots Shear Stresses SXY.

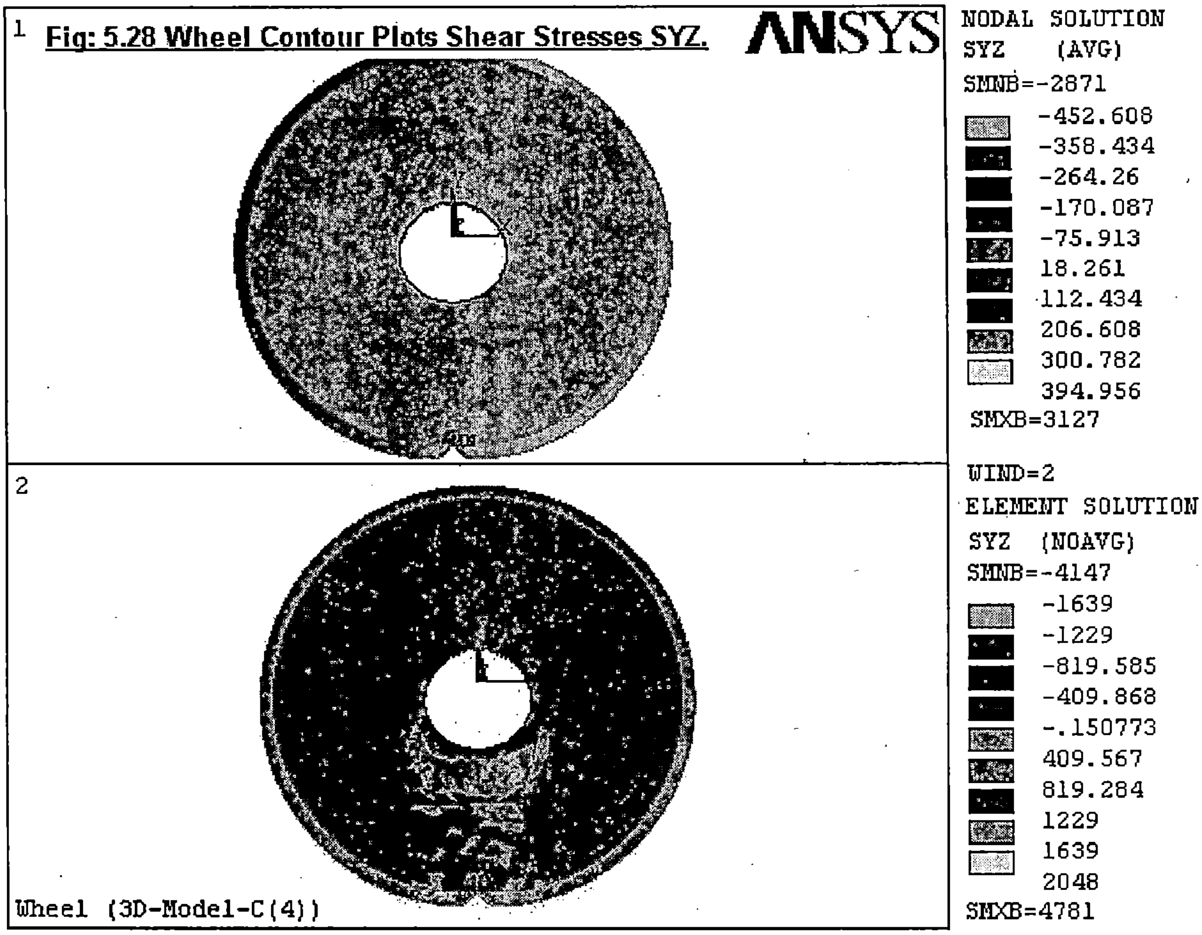


Fig: 5.28 Wheel Contour Plots Shear Stresses SYZ.

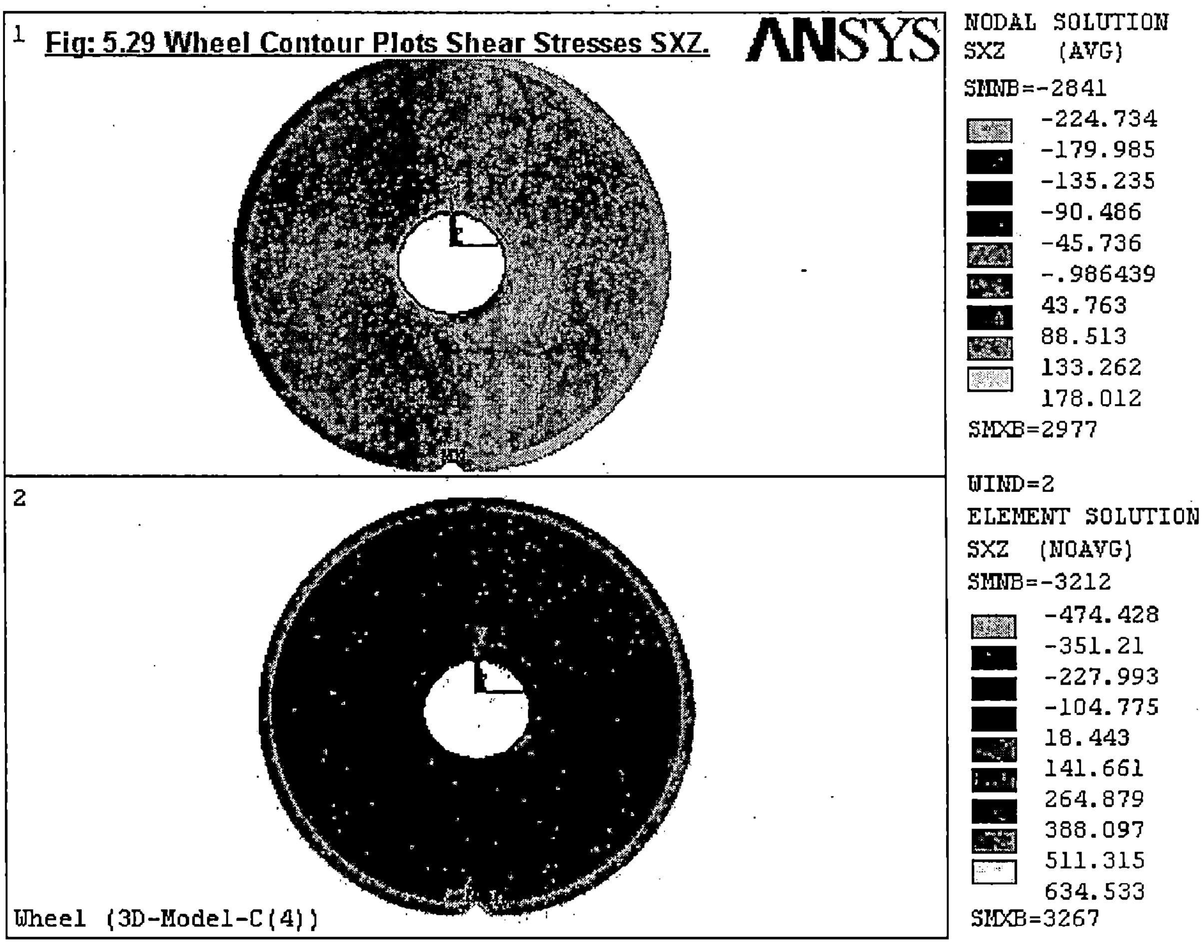


Fig: 5.29 Wheel Contour Plots Shear Stresses SXZ.

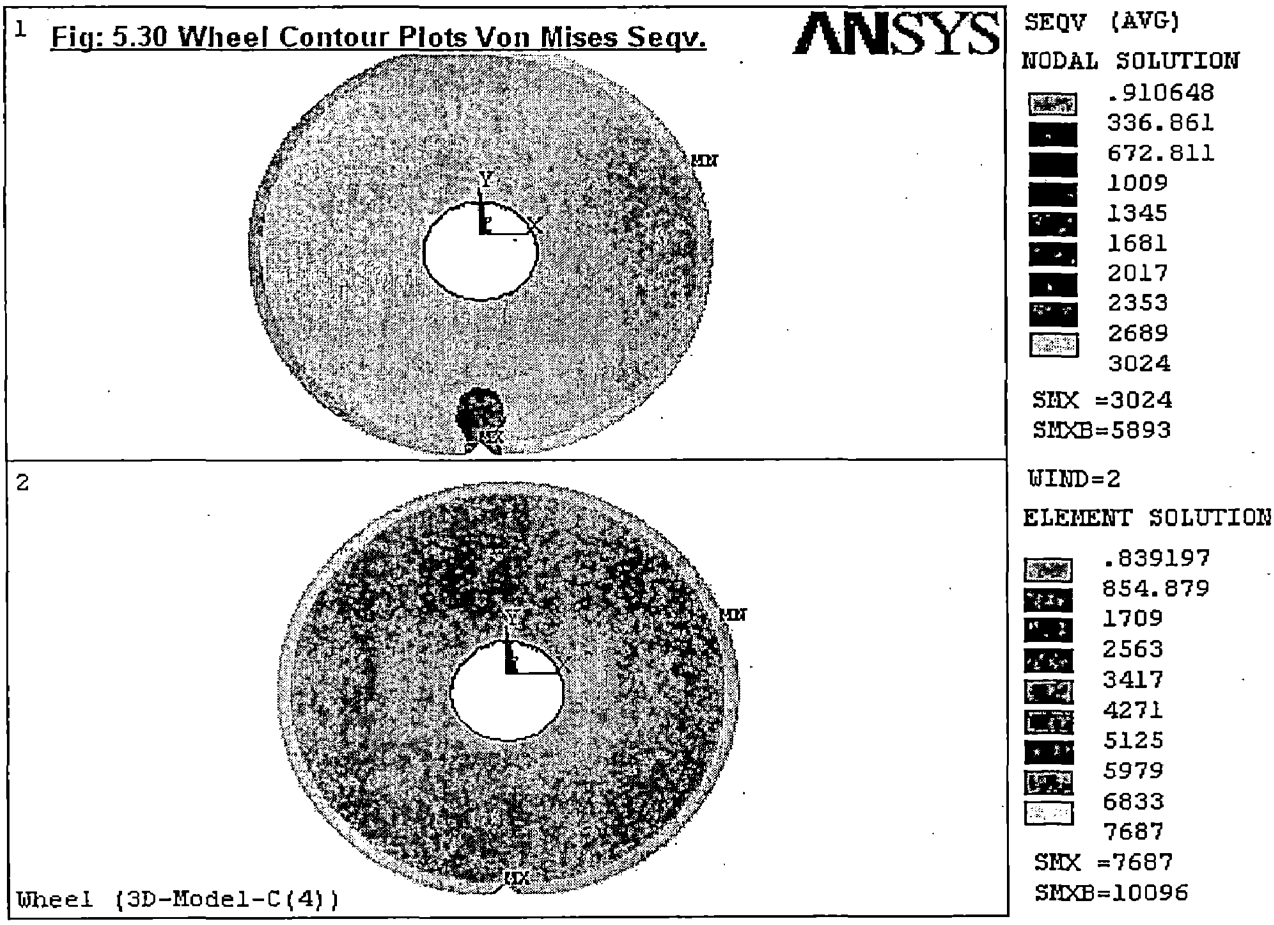


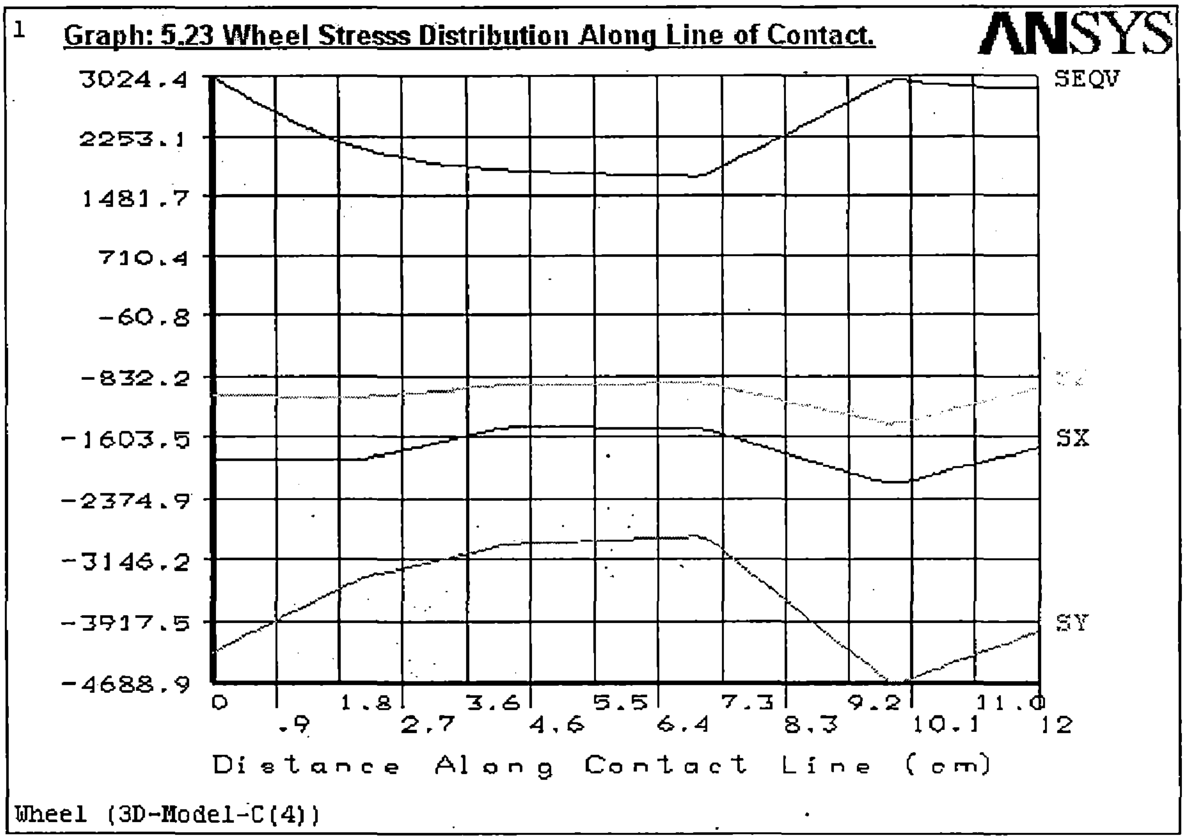
Fig: 5.30 Wheel Contour Plots Von Mises Seqv.

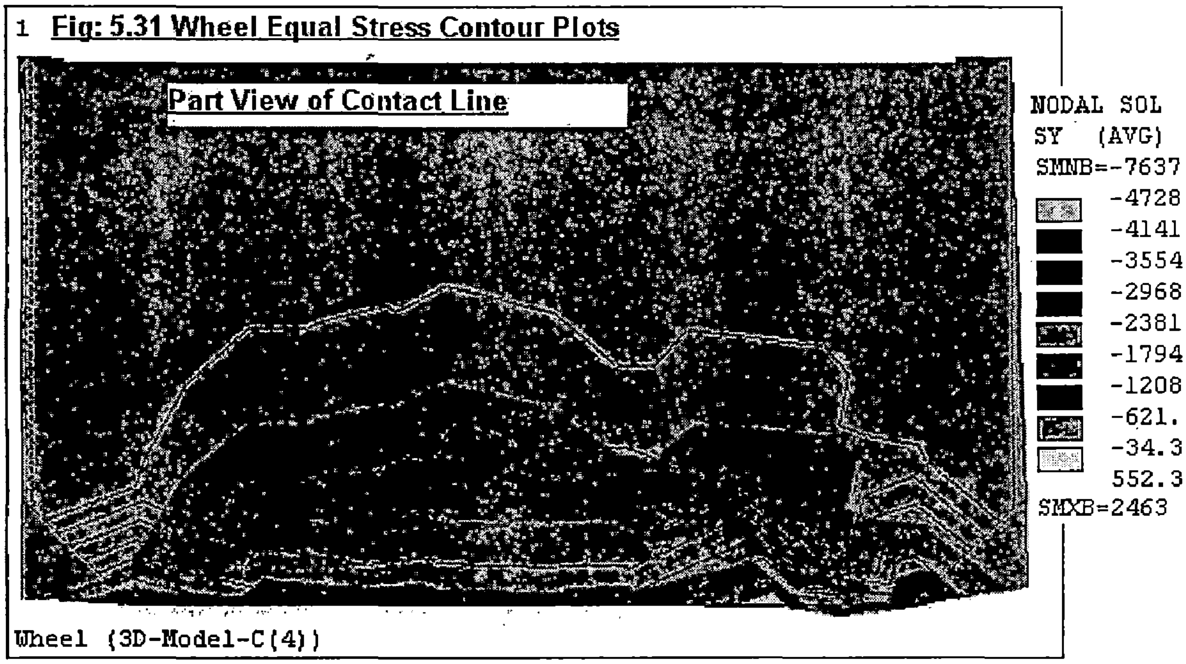
From the Fig: 5.27, 5.28, 5.29 and contour legends it is in plain sight that, maximum shear stresses on the line of contact of the wheel in **XY-plane**, SXY: -634.021 kg/cm^2 (Nodal Solution) and **3142** kg/cm^2 (Element Solution), YZ-plane, SYZ: -452.608 kg/cm^2 (Nodal Solution) and 2048 kg/cm^2 (Element Solution), XZ-plane, SXZ: -224.734 kg/cm^2 (Nodal Solution) and 634.533 kg/cm^2 (Element Solution) respectively. From the Fig: 5.30 and contour legends it is observed that, maximum Von Mises Equivalent stresses on the line of contact of the wheel: Seqv = 3024 kg/cm^2 (Nodal Solution) and Seqv = **7687** kg/cm^2 (Element Solution).

Now ANSYS graphical (path) results along the line of contact of the wheel and **equal stress contour plots** are presented for clear understanding of the distribution of stresses in the wheel.

SUMMARY OF THE PATH VARIABLES (Along Contact Line for gr: 5.23)

LABEL	MAX	MIN
SX	-1468.8	-2187.2
SY	**-2862.4**	**-4688.9**
SZ	-926.69	-1434.6
SXY	214.75	-293.12
SYZ	394.96	-452.61
SXZ	67.623	-34.171
SEQV	3024.5	1733.4

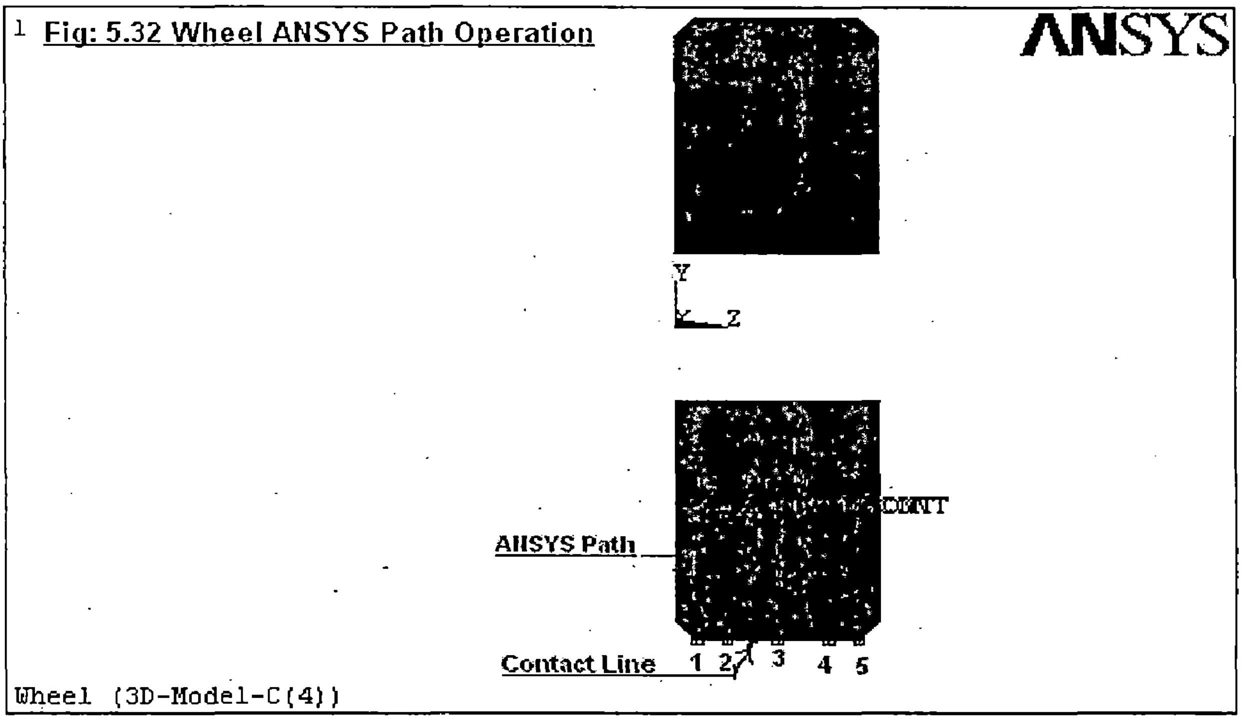


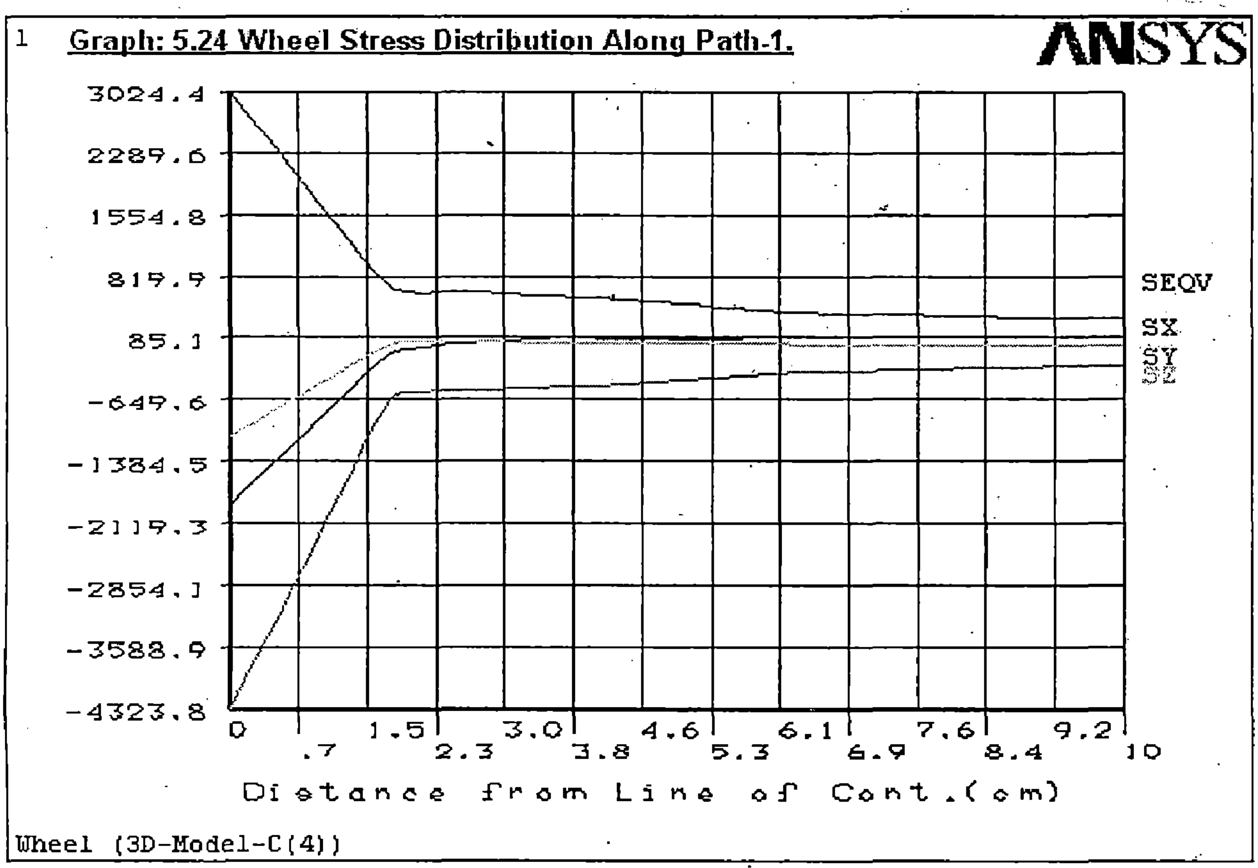


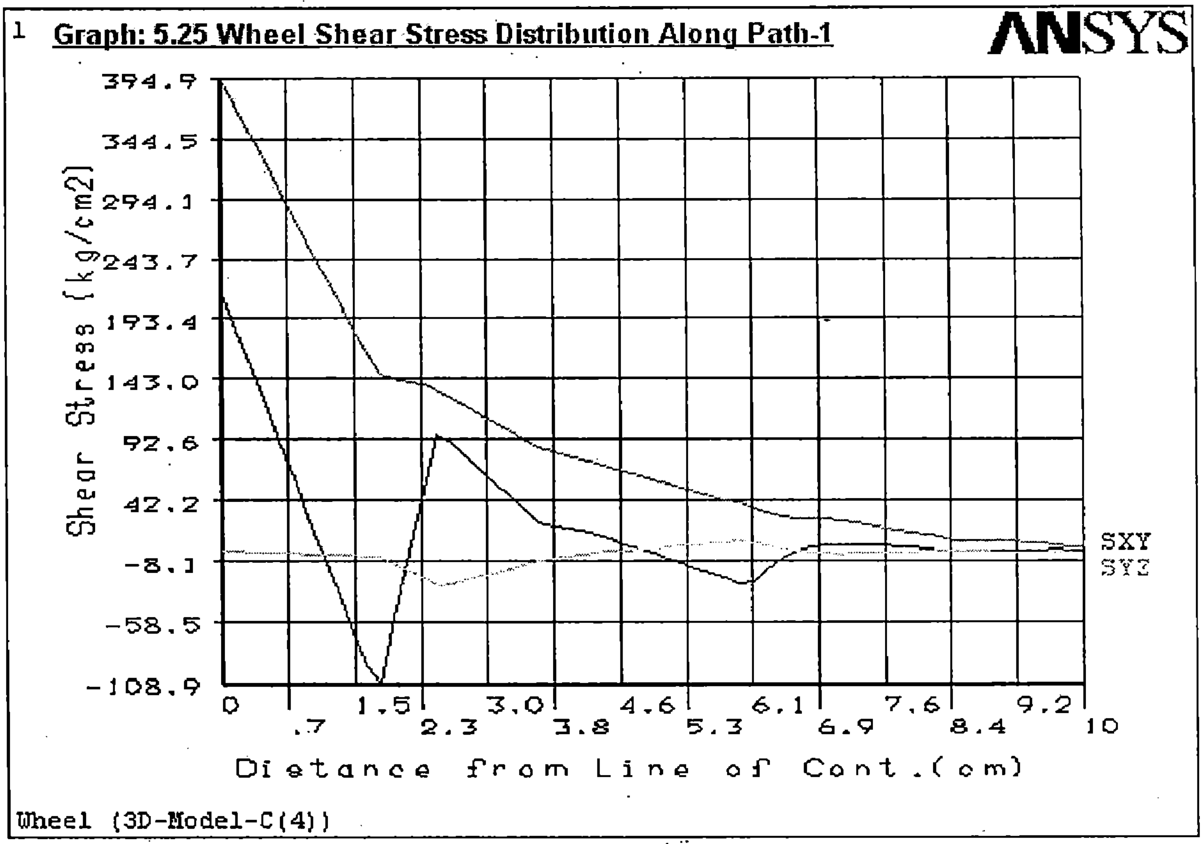
From the Graph: 5.23 and Fig: 5.31 it is observed that maximum stresses are occurring at the two ends of the contact line.

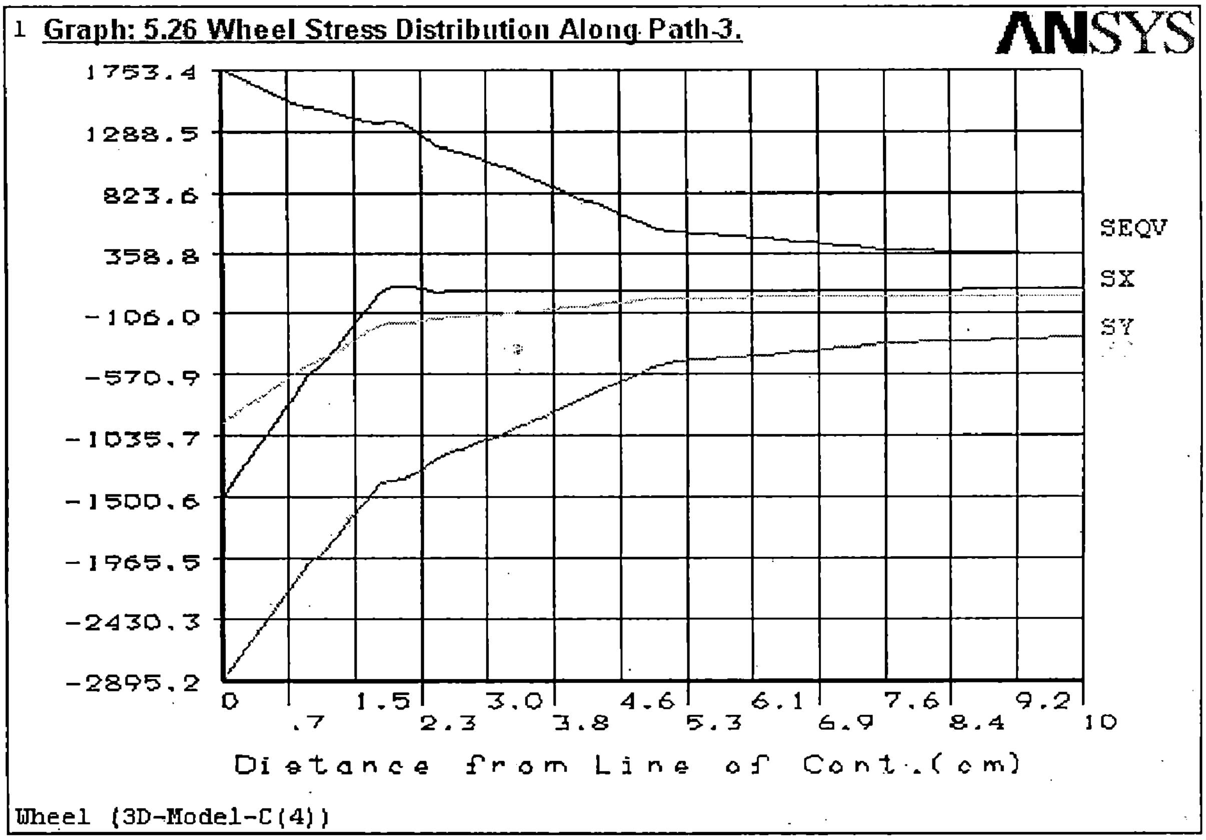
Finding the Depth of influence of maximum Stresses:

For this in ANSYS model a section is made in YZ-plane along the line of contact of the wheel. Then ANSYS Paths are defined & stresses are mapped on it from the contact line to vertically upward to 10 cm at two ends & mid-point of the contact line and in between as shown in Fig: 5.31 marked as 1, 2, 3, 4 & 5.

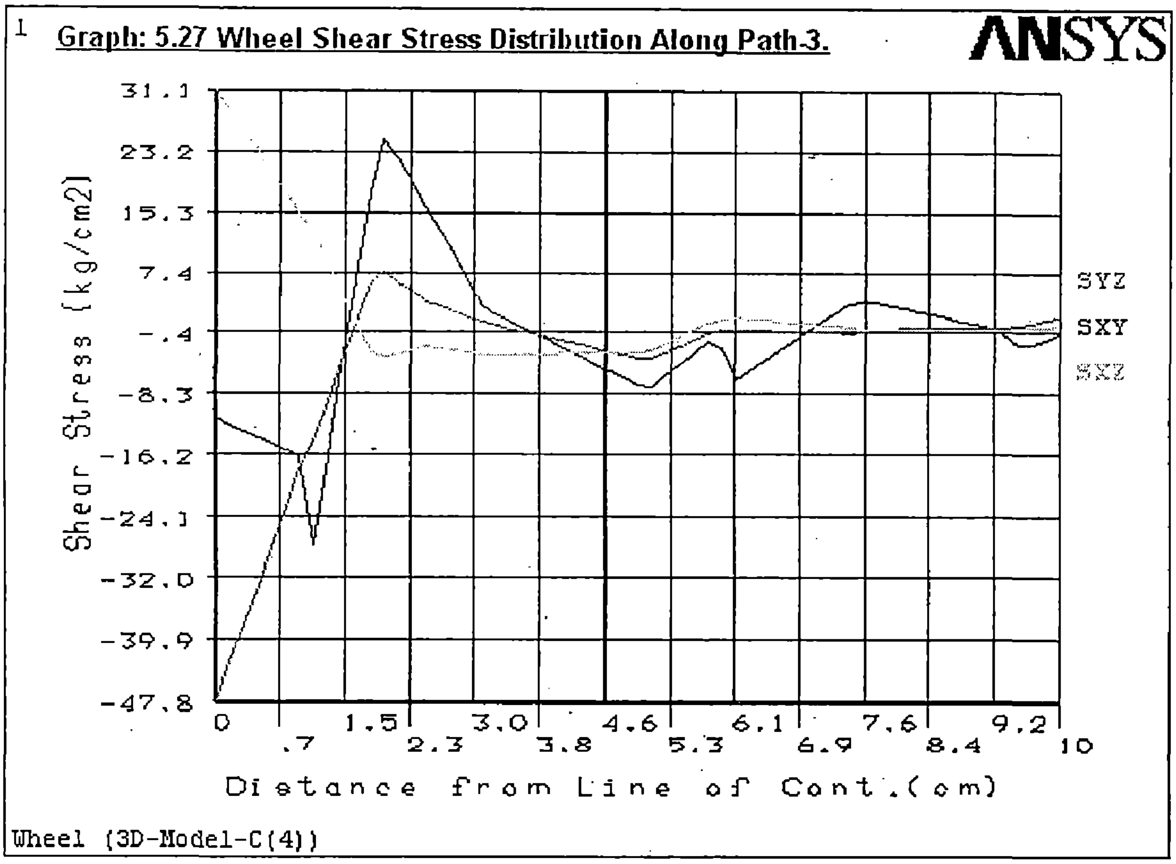


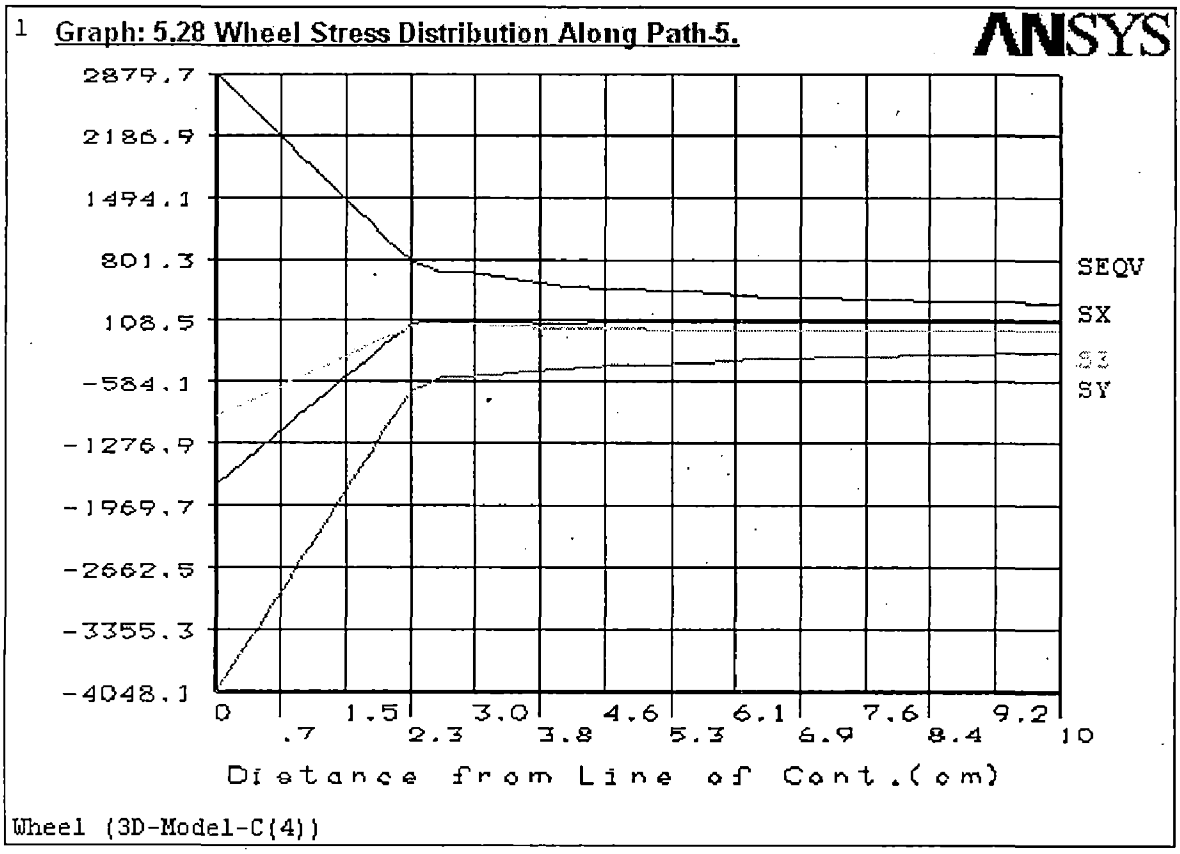


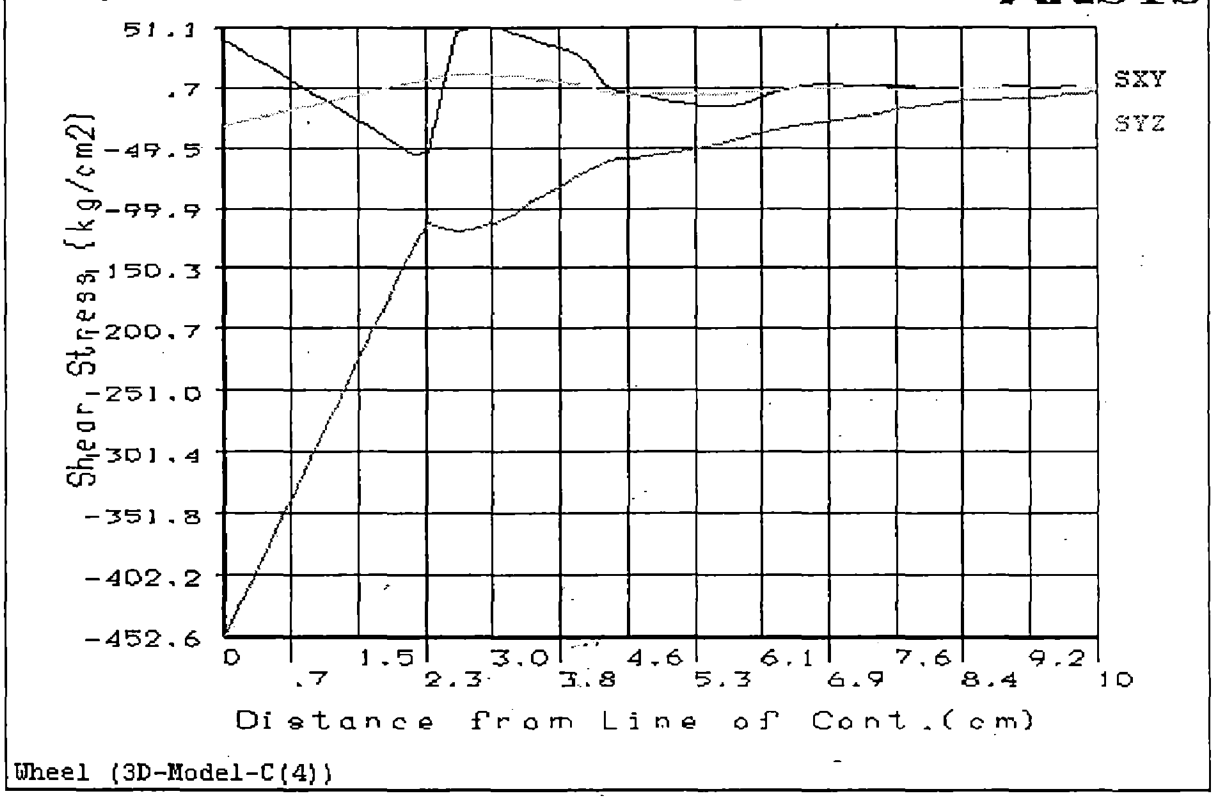




Note: Since, it is observed at time of analysis that, stress distribution is almost same for the paths 2, 3 & 4 so, mid-path's results are presented here only.


1
Graph: 5.27 Wheel Shear Stress Distribution Along Path-3.
ANSYS
31.1
23.2
15.3
7.4
-.4
-8.3
-16.2
-24.1
-32.0
-39.9
-47.8
Shear Stress (kg/cm2)
0
.7
1.5
2.3
3.0
3.8
4.6
5.3
6.1
6.9
7.6
8.4
9.2
10
Distance from Line of Cont.(cm)
SYZ
SXY
SXZ
Wheel (3D-Model-C(4))


1
Graph: 5.28 Wheel Stress Distribution Along Path-5.
ANSYS
2879.7
2186.9
1494.1
801.3
108.5
-584.1
-1276.9
-1969.7
-2662.5
-3355.3
-4048.1
0
.7
1.5
2.3
3.0
3.8
4.6
5.3
6.1
6.9
7.6
8.4
9.2
10
Distance from Line of Cont.(cm)
SEQV
SX
S3
SY
Wheel (3D-Model-C(4))



1 Graph: 5.29 Wheel Shear Stress Distribution Along Path-5.

From the Graph: 5.24 to 5.29 it is observed that, the depth of influence of the maximum stresses are varying from the contact line to a depth: 1.5 – 2.3 cm.

5.3.2 Validation of Wheel FEM Result

It is obvious from the Fig. 5.24 - 5.30 and Graph: 5.23 that, maximum **SY** and **SXY** Stresses are taking place with contact line, which also varying between nodal and element solutions: contact stress SY= -4728 kg/cm^2 (NS) & **-9070** kg/cm^2 (ES) and shear stress: SXY= -634.021 kg/cm^2 (NS) & **3142** kg/cm^2 (ES), and Von Mises Seqv stress: Seqv = 3024 kg/cm^2 (NS) and **7687** kg/cm^2 (ES). For clear understanding, a comparison between design and maximum FEM results is given in Table: 5.6.

Table: 5.6 Comparison between DESIGN and FEM results of the Wheel.

Name and Location of maximum value	**Design results**	**FEM**		**Variation (%)**
		Results	**Ref. fig/gr**	
Contact stress on contact line: SY (kg/cm^2)	7532.93	-4728 (NS) **-9070** (ES) **7687** (Eqv)	Fig: 5.25, Gr: 5.23 Fig: 5.30	-37.23 +20.40 +2.00
Shear stress on contact line: SXY (kg/cm^2)	2290.01	-634 (NS) **3142** (ES)	Fig: 5.27	-72.31 +37.20

From the Table: 5.6 it appear that, FEM maximum contact stress (SY) varying from –37.23% to +20.40% and the upper limit is less than the permissible contact stress 9800 kg/cm^2, and maximum shear stress (SXY) varying from –72.31% to +37.20% and it also well below the permissible shear stress 4900 kg/cm^2. However, these variations might be for some assumptions made in design consideration as well as some FEM iteration errors and as such, may be considered within acceptable margin.

5.4 SUMMARY OF COMPARISON OF THE DESIGN & FEM RESULTS

For better understanding between the variation of DESIGN and FEM results a summary comparison for the **significant** stresses/deflection only is shown in Table: 5.7.

Table: 5.7 Summary of comparison of the DESIGN and FEM results:

Location & direction of results		Design results (kg/cm^2)	FEM		Variation (%)
			Results (kg/cm^2)	Ref. graph /figure	
a) Skin plate interior panel stresses in kg/cm^2:					
Mid-point with vertical stiffeners	SX	529.14	454.13	Gr: 5.4, 5.9-5.15	-14.18
	SY	158.74	152.88	„	-3.69
Mid-point with central horizontal girder	SX	126.81	113.30	Gr: 5.5, 5.16-5.21	-10.65
	SY	422.70	390.72	„	-7.57
b) Central horizontal girder bending stresses at the centre of span in kg/cm^2:					
Top flange (skin plate)		637.54	668.79	Fig: 5.10, ET: 5.1	+8.04
Bottom flange		-978.20	-1026.30	„	+4.92
Deflection (cm)		0.54	0.568701	Fig: 5.9	+5.31
c) Top/Bottom horizontal girder bending stresses at the centre of span in kg/cm^2:					
Top flange (skin plate)		728.03	706.21	Fig: 5.12, ET: 5.2	-3.00
Bottom flange		-1019.18	-988.62	„	-3.00
Deflection (cm)		0.54	0.576006	Fig: 5.11	+6.67
d) Vertical end girder bending stresses at wheel support points in kg/cm^2:					
Top flange (skin plate)		-431.75	-401.91	Fig: 5.14, ET: 5.3	-6.91
Bottom flange		230.49	214.58	„	-6.90
e) Wheel stresses on contact line in kg/cm^2:					
Contact stress (SY)		7532.93	7687 (Eqv)	Fig: 5.30	+2.00
Shear stress (SXY)		2290.01	3142 (ES)	Fig: 5.27	+37.20

CONCLUSION AND SUGGESTION

6.1 CONCLUSION

Three-dimensional analysis by Finite Element Method (FEM) through ANSYS Program on a 'Fixed Wheel Vertical Lift Gate' and its comparison with conventionally design results are presented in this study. This study reveals that three-dimensional displacement and stress analyses are most suitable for predicting deflection and nature of stress distribution in a composite indeterminate structure that is otherwise not possible through two-dimensional approach.

The following points/ remarks may be made in view of the results and discussion of this analysis:

- It is necessary to mention here that, initially two-dimensional design results are checked by two-dimensional FEM component-wise for horizontal girders, vertical end girder, vertical stiffener and for skin plate's (three-dimensional) interior & exterior panel-wise with same loads & support conditions as in design consideration. It is found that, for two-dimensional line elements, FEM results match in to-to with design results except for the deflections in horizontal girders which are almost same as found in three-dimensional analysis, and for skin plate (three-dimensional shell elements as two-dimensional shell facility is not available in Ansys 5.4) ANSYS significant stresses are 3% less in interior panel & 160% more (far below the permissible stress) in exterior panel than the design stresses. It shows that two-dimensional design approaches for beams are OK and for skin plates it is conservative.

- Significant stresses obtained by three-dimensional analysis in skin plate taking place either with horizontal girders or vertical stiffeners are less than the two-dimensional design stresses, which indicates that, the two-dimensional approach for skin plate design is conservative.

- In two-dimensional design approach for tapered section girders like horizontal girders in this study, there is no direct method for determination of deflection other than graphical approach, which may lead to erroneous result but three-dimensional FEM analysis gives results close to the exact result. In this study, design deflection was 0.54 cm and the same by three-dimensional FEM are 0.56 cm in Central Horizontal Girder & 0.57 cm in Top Horizontal Girder/Bottom Horizontal Girder, which might be treated as exact deflections.

- In two-dimensional approach, though methods are available for spacing the horizontal girders with equal loads but sometimes, it may be necessary to rearrange the spacing from easy/feasible manufacturing point of view. In that case equal distribution of loads for space between two girders as assumed in design consideration may deviate. This deviation i.e. actual load behaviors might be modeled through three-dimensional FEM e.g. in this study, it is observed that, actually the Central Horizontal Girder is sharing more loads whereas the Top Horizontal Girder/Bottom Horizontal Girder is sharing less loads than the design consideration.

- One most interesting thing " the loading nature on the two vertical end girders are exactly same as assumed in two-dimensional design consideration that, concentrated loads come to it (VEGs) though horizontal girders" reveals from FEM analysis. As a result, shear force and bending moment diagrams are exactly same in two-dimensional as well as in three-dimensional FEM with little variation of the magnitude.

- From the FEM shear force and bending moment diagrams it is obvious that, the each vertical stiffeners set is acted as a continuous beam loaded uniformly and supported on horizontal girders, which is same as two-dimensional design concept, 'simply supported beam between two horizontal girders'. Loads shared by them are less than the design concept. So, two-dimensional design approach is conservative.

- From the three-dimensional wheel analysis it is noticed that, contact line is penetrating inside the model that is not similar to the real case as rest on a flat surface (track plate). The variations of stresses between the nodal and element solutions are quite significant. The element solution results are very much close to the two-dimensional design results. The depth of influence of the contact stresses inside the wheel body is moderately clear in three-dimensional analysis whereas it was not so clear in two-dimensional design.

- Overall, it is obvious from this three-dimensional FEM ANSYS study that, the two-dimensional design of the fixed wheel vertical lift gate for tailrace channel of the Dharasu Power House is quite safe and satisfactory.

- Therefore, it is obvious that, though two-dimensional design approach is much more conservative, still for a complicated, expensive and durable structure three-dimensional FEM analysis should be carried out for realization of actual load behaviors prior to manufacturing and installation.

6.2 SUGGESTION FOR FURTHER STUDY

- With different sizes gates a study may be carried out for finding an economical gate size.

1. ANSYS 5.4 Basic Analysis Procedures Guide

2. ANSYS 5.4 Elements Manual

3. ANSYS 5.4 Modeling and Meshing Guide

4. ANSYS 5.4 Structural Analysis Guide

5. ANSYS 5.4 Theory Manual

6. ANSYS 5.4 Workbook Examples

7. ANSYS5.4 Advanced Analysis Techniques

8. ANSYS5.4 Commands Manual

9. ANSYS5.4 Operations Guide

10. ANSYS5.4 Verification Manual

11. Arya A S "Structural Design in steel, Masonry and Timber" Neemchand and Brothers, Roorkee, UA, India.

12. Bairagi N K "A Text Book of Plate Analysis" Khanna Publishers (1986).

13. Chandrakant S Desai and John F Abel "Introduction to Finite Element Method" Van Nostrand Reinhold Company.

14. Chandrupatala T R and Belgunda A (2000) " Introduction to Finite Elements in Engineering" Prentice Hill, Delhi, India.

15. Chauhan, Gopal, "Hydro-mechanical Equipment", Class works session-2003-04 for M. Tech. degree, WRDTC, IIT Roorkee, UA, India.

16. Creager, W.P. and Justin, J.D, "Hydroelectric Handbook", 2nd. Edition-1963, John Wiley & Sons, Inc. London.

17. Davis, C.V., "Handbook of Applied Hydraulics", 3rd. Edition-1969, McGRAW-HILL Publishing Company Ltd.

18. Grishin, M.M., "Hydraulic Structures", Vol-2, Translated from Russian, V. Kolykhamatov, Mir Publishers, Moscow, in-1982.

19. IS – 2062 – 1982: Indian Standard Specification for structural steel for fusion welding quality.

20. IS – 226 – 1975: Indian Standard Specification of structural steel standard quality.

21. IS – 4622 – 1992: Indian Standard Fixed Wheel Gates Structural Design Recommendations (2^{nd} Revision).

22. IS – 800 – 1984: Indian Standard Code of practice for use of structural steel in general building construction.

23. IS – 823 – 1964: Indian Standard Code of practice for manual metal arc welding of mild steel.

24. Jack Lewin, "Gates and Valves", in free surface flow and submerged outlets, 1^{st}. Edition-1995, Thomas Telford, Great Britain.

25. Nigam, P.S., "Handbook of Hydro-Electric Engineering", 2^{nd}. Edition-1985, Nem Chand and Bros, Roorkee, UA, India.

26. Power House Design Unit –II, "Design Calculation and Drawings of VL Fixed Wheel Gate for Tailrace Channel of Dharasu Power House" IDO, Roorkee, UA, India.

27. Ramamurthan S and Narayan R " Strength of Materials" Dhanapat Rai and Sons.

28. Rao S S "The Finite Elements Method in Engineering" Pergamon Press Oxford (1982).

29. Reddy J N "An Introduction to the Finite Element Method" 2^{nd}. Edition, McGRAW-HILL Publishing Company Ltd.

30. Timoshenko, S "Strength of Materials" 1965, D Van Nostrand Company Inc.

31. Timoshenko, S and J N Goodier "Theory of Elasticity" McGRAW-HILL Publishing Company Ltd.

32. Varshney, R.S., "Gates and Valves", 1^{st}. Edition-2002, Nem Chand and Bros, Roorkee, UA, India.

33. William A Nash "Theory and Problems of Strength of Materials" 1957. Schaum Publishing Co. New York.

34. Zienkiewicz O C "The Finite Element Method in Engineering science" McGRAW-HILL Publishing Company. Ltd. LONDON.

Design of Fixed Wheel Vertical Lift Gates at Tail Race Channel of Dharasu Power House

1.0 Design Data Available:

1)	Sill level	:	EL 820.10 m
2)	Bottom of Brest wall	:	EL 824.00 m
3)	Top of pier	:	EL 638.84 m
4)	Clear opening (height)	:	3.90 m
5)	Clear opening (span)	:	6.75 m
6)	No. of Bays	:	4.00 nos.
7)	Width of Pier	:	1.90 m
8)	Design Head	:	16.40 m
9)	Type of Proposed Gates	:	Fixed wheel gate.
10)	No. of Gates required	:	4.00 nos.

2.0 Permissible Stresses:

The stresses in different members under normal conditions due to dynamic loads and due to static loads after gate as closed will not in any case exceed the maximum permissible stresses specifired as below with reference to IS 4622 -1992 for Indian standard steel (Annex-A).

Strength of materials:

	Yield stress y_p (kg/cm^2)	Tensile Strength UTS (kg/cm^2)	Modulus of Elasticity E (Kg/cm^2)
a) Structural steel:(As per IS 2062 -1980)			
for thickness : 6 - 20 mm	2600.00	4200.00	2.01E+06
for thickness : >20 - 40 mm	2400.00	4200.00	
b) Carbon steel, Class-3 (IS 2004-1970)	2750.00	5000.00	
c) Brass		4200.00	

As per IS 4622 : 1992 (ANNEX B)

		Wet condition: Accessible	Wet condition: Inaccessible
a) Structural Steel:			
i) Direct compresson and compresson in bending (thickness >20-40mm)	:	0.45 y_p= 1080.00	0.40 y_p= 960.00
ii) Direct tension and tension in bending (thickness >20-40mm)	:	0.45 y_p= 1080.00	0.40 y_p= 960.00
iii) Shear in webs (thickness : 6 - 20 mm)	:	0.35 y_p= 840.00	0.30 y_p= 720.00
iv) Combined stresses			
for thickness : 6 - 20 mm	:	0.60 y_p= 1560.00	0.50 y_p= 1300.00
for thickness : >20 - 40 mm	:	0.60 y_p= 1440.00	0.50 y_p= 1200.00
v) Bearing stress (thickness >20-40mm)		0.65 y_p= 1560.00	0.55 y_p= 1320.00
b) Other materials:			
i) Bearing stress for Bronze or Brass	:	0.035UTS= 147.00	0.030UTS= 126.00
ii) Bearing stress for Concrete (M-20)	:		40.00
iii) Bearing stress for Concrete (M-25)	:		50.00

Note: In emergency condition the permissible stresses will be increased by 13.5% of the values specified for normal conditions as above. Gate leaf has been designed for wet-accessible conditions and embeded parts for wet-inaccessible conditions.

3.0 Data Assumed:

a) The gate is divided into two equal parts to facilitate transportation while errection splicing by both side, plate with rubber seal, bolts & nuts fit arrangement is to be done. Each gate is provided with two wheels on either side and are positioned in such a way that they share the water load almost equally. Calculation have been done for bottom most gate unit and the same is used for the other top position too.

b) The rubber seal have to be provided on river side of brest wall. Thus skin plate has to be provided on the same side.

The following data are assumed:

i)	Height of the unit gate, h =	205 cm
	No. of unit gates per Bay, n =	2
	Total height covered, H = n.h =	410.00 cm
	No. of girder provided per unit	3
	C/C spacing of the girder	75 cm
	End spacing i.e. from c/l of girder to end of the unit	27.5 cm
	C/C spacing of the vertical stiffeners	75 cm
	No. of wheels provided per unit on each side	2
ii)	C/C of side seals (load span), L_{span} =	690.00 cm
iii)	C/C of tracks (wheel span), L_w =	725 cm
iv)	Edge to edge of gate grooves	775 cm
v)	Height of top seal's c/l from the sill level	390 cm
vi)	Edge to edge of plates of groove, L_p =	7.58 m

All anchorage in 1st. stage concreting should be embeded in with M-25 grade concrete and 2nd. stage concrete comprises of M-25 grade.

4.0 Design of Lower Unit of the Gate:

4.1 Water Load Diagram and Total Water Load on the Gates:

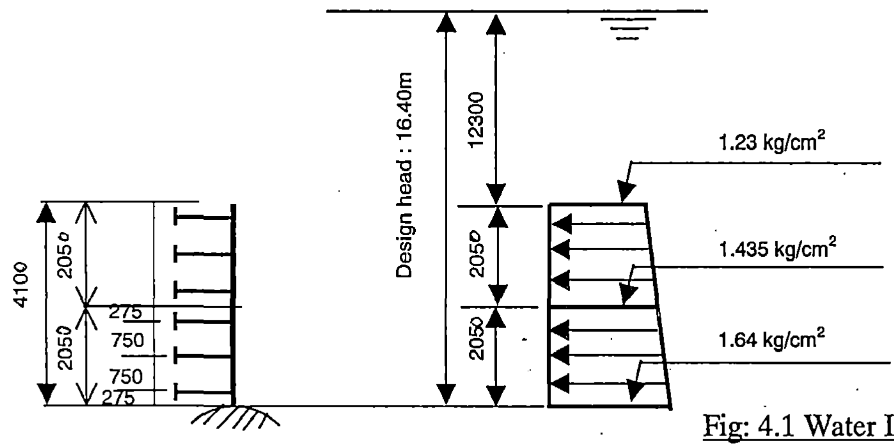


Fig: 4.1 Water Load Diagram

4.2 Water Load on the Bottom Unit (unit-I)

C to C of side seal, L_s = 690.000 cm

Height of gate, h_1 = 205.00 cm

Water pressure at top of the gate, P_{t1} = 1.435 kg/cm^2

(Water Pr in kg/cm^2 = depth of water in metre/10;)

Water pressure at bottom of the gate, P_{b1} = 1.640 kg/cm^2

Total water load on bottom unit (unit-I), F_{w1} $\{(P_{t1}+P_{b1})/2\}.L_s.h_1$

= 217479.4 kg

4.3 Water Load on the Top Unit (Unit-II):

C to C of side seal, L_s = 690.000 cm

Height of gate, h_2 = 205.00 cm

Water pressure at top of the gate, P_{t2} = 1.230 kg/cm^2

Water pressure at bottom of the gate, P_{b2} = 1.435 kg/cm^2

Total water load on top unit (unit-II), F_{w2} = $\{(P_{t2}+P_{b2})/2\}.L_s.h_2$

= 188482.1 kg

Total water load on the entire gate, $F_w = (F_{w1}+F_{w2})$ = 405961.5 kg

4.4 Check

Total Area of the Trapezium ie. load for unit span, $A_w = \{(P_t+P_b)/2\}.(h_1+h_2)$

= 588.4 kg/cm

Total water load, $F_w = A_w.L_s$

= 405962 kg

= 405.96 mt

The variation of water pressure with respect to gate height is very small. So, it can be taken as 1.64 kg/cm^2 over the complete gate unit height. The design will therefore, be more in safe side.

So, Design Water Pressure, P = 1.64 kg/cm^2

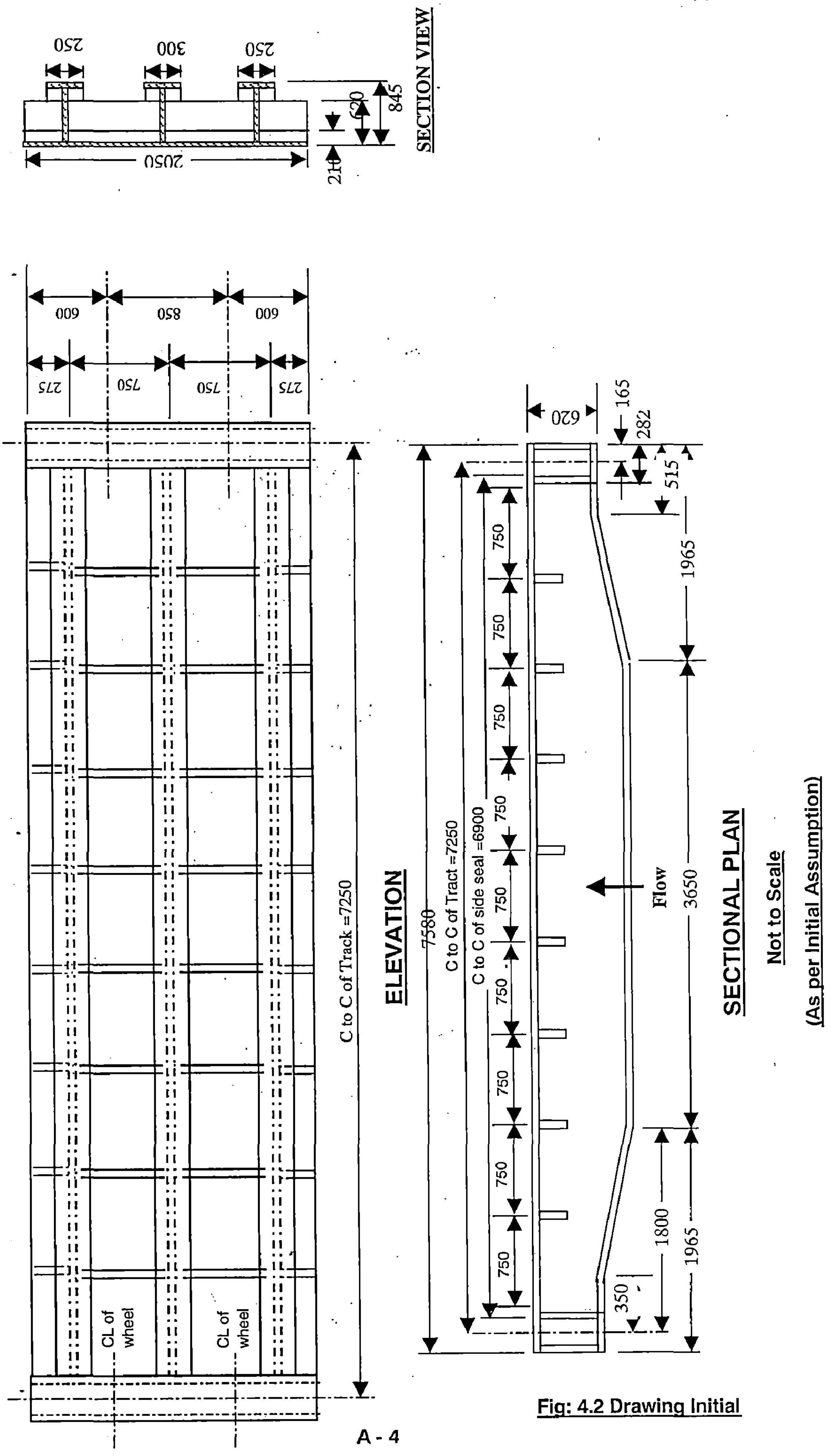


Fig: 4.2 Drawing Initial

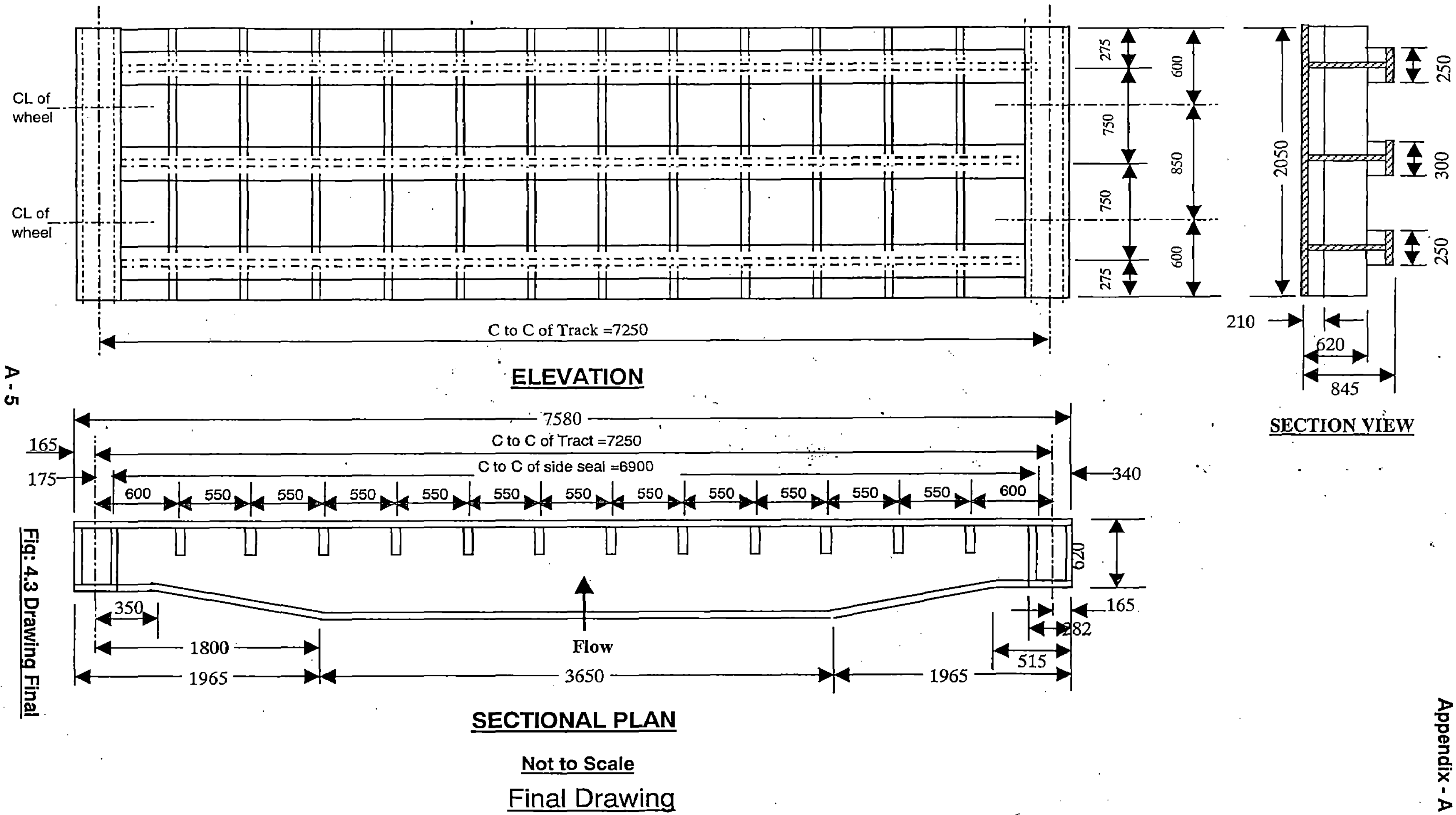


Fig: 4.3 Drawing Final

5.0 Design of Skin Plate:

The variation of water pressure with respect to gate height is very small. So, it can be taken as 1.64 kg/cm2 for over the complete gate unit-I height. The design will therefore, be more in safe side.

So, design pressure, P = 1.64 kg/cm^2

5.1 Check for Interior Panel:

Maximum panel size, a = 750 mm and b = 750 mm

Assume thickness of the skin plate, t = 20 mm

Gate to be treated as painted with epoxy paint. Hence, taking no corrosion allowance into consideration.

So, effective thickness of skin plate, t = 20 mm

Now, with reference to IS 4622 : 1992, page 12, fig. 5, Annex-C, Clause 5.2.3(a). We have the equation: (Note: symbols of stress & thickness have been changed.)

$$S = \frac{K}{100} x \frac{Pxa^2}{t^2} \quad \text{------------------------ (I)}$$

Where,

S = bending stress in flat plate in N/mm^2 or kg/cm^2

K = non dimensional factor depends on the values of a & b.

P = water pressure in N/mm^2 or kg/cm^2 (relative to the plate centre).

a, b = bay width in mm or cm as shown in figure below.

t = plate thickness in mm or cm,

Here,

P = 0.16088 N/mm^2 = 1.64 kg/cm^2

a = 750 mm = 75 cm

b = 750 mm = 75 cm

t = 20 mm = 2 cm

b/a = 1.00

Value of K is taken from table - 2 as below:

b/a	+/- S_{2x}	+/- S_{2y}	+/- S_{3x}	+/- S_{4y}
1.00	13.7	13.7	30.9	30.9

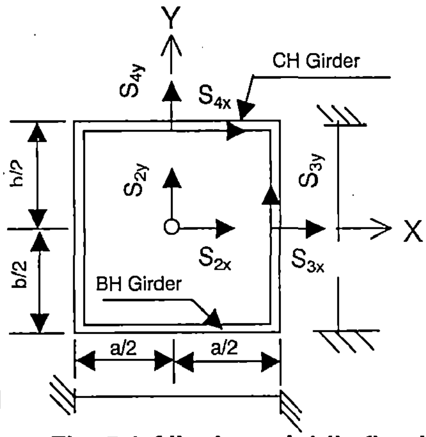


Fig. 5.1 All edges rigidly fixed

Now, putting the above values in equation (I) we have,

+/-' S_{2x} = 315.956 kg/cm^2

+/-' S_{2y} = 315.956 kg/cm^3

+/-' S_{3x} = 712.631 kg/cm^2 (Max stress) <1080 kg/cm^2

+/-' $S_{3y} = 0.3xS_{3x}$ = 213.789 kg/cm^2

+/-' S_{4y} = 712.631 kg/cm^2 (Max stress) <1080 kg/cm^2

+/-' $S_{4x} = 0.3xS_{4y}$ = 213.789 kg/cm^2

Hence the plate thickness adopted is OK.

5.2 Check for Bottom Cantilever Portion of the Skin Plate:

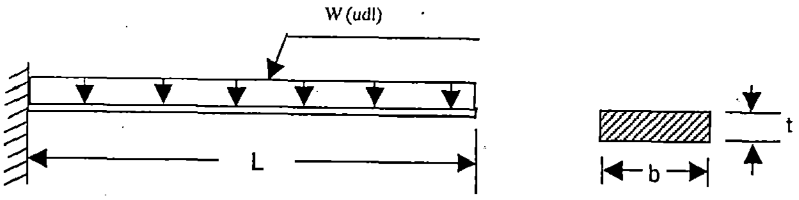


Figure: 5.2

Here, W = 1.64 kg/cm
L = 27.5 cm

Here,
b = 1 cm
t = 2 cm

M_{max} per cm width = $WxL^2/2$ = 620.125 kg-cm
Z for skin plate = $(bxt^2)/6$ = 0.66667 cm^3
f_{max} for cantilever portion = M/Z = 930.188 kg/cm^2 <1080 kg/cm^2

5.3 Check for Corrosion Allowance Taking into Consideration:

Consider corrosion allowance = 1.5 mm
So, effective thickness of skin plate, t = 18.5 mm or 1.85 cm
Now, putting the above values in equation (I) we have,
Max stress, S_{3x} = 832.878 kg/cm^2 <1080 kg/cm^2, Hence safe.

5.4 Check for Bottom Panel (two short & one long edges fixed and one long edge simply supported) :

Here,

P = 0.16088 N/mm^2 1.64 kg/cm^2
a = 275 mm = 27.50 cm
b = 750 mm = 75.00 cm
t = 20 mm = 2.00 cm
b/a = 2.73

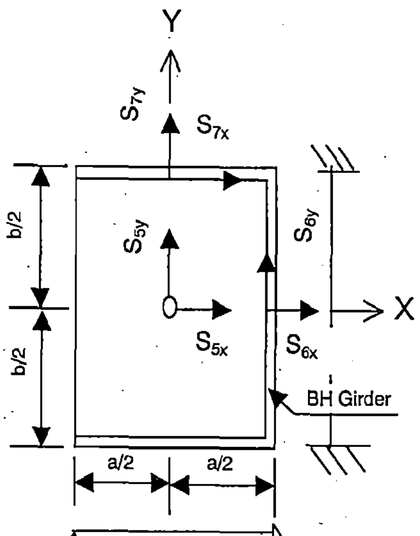


Fig. 5.3

Value of K is taken from table - 2 as below:

b/a	+/- S_{5x}	+/- S_{5y}	+/- S_{6x}	+/- S_{7y}
3	37.4	12.0	74.0	47.1
2.5	36.6	13.3	73.2	47.0
2.73	36.96	12.71	73.56	47.05

Here the value of K is maximum for stress, S_{6x}.
Now, putting the corresponding values in equation (I) we have,

+/-' S_{5x} = 114.61 kg/cm^2
+/-' S_{5y} = 39.41 kg/cm^3
+/-' S_{6x} = 228.09 kg/cm^2 (Max stress) <1080 kg/cm^2
+/-' $S_{6y} = 0.3xS_{6x}$ = 68.43 kg/cm^2
+/-' S_{7y} = 145.87 kg/cm^2
+/-' $S_{7x} = 0.3xS_{7y}$ = 43.76 kg/cm^2

Hence the plate thickness adopted is OK.

5.5 Check for Corrosion Allowance (bottom panel) :

Consider corrosion allowance = 1.5 *mm*

So, effective thickness of skin plate, t = 18.5 mm or 1.85 cm

Now, putting the above values in equation (I) we have,

Max stress, S_{7y} = 170.682 kg/cm^2 <1080 kg/cm^2 , Hence safe.

After Check with Combined Stresses, Revise Calculation :

5.6 Check for Interior Panel:

The distance between vertical stiffeners is proposed as, 550 mm instead of 750 mm C/C.

Maximum panel sixe, a x b = 550 750 mm

So, effective thickness of skin plate, t = 20 mm

Here,

P =	0.16088 N/mm^2	1.64 kg/cm^2
a =	550 mm	55 cm
b =	750 mm	75 cm
t =	20 mm	2 cm
b/a =	1.36	

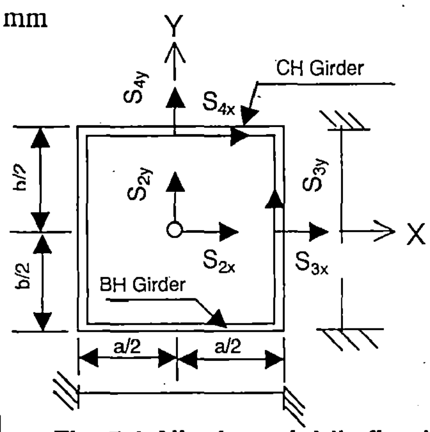


Fig. 5.1 All edges rigidly fixed

Value of K is taken from table - 2 as below:

b/a	+/- S_{2x}	+/- S_{2y}	+/- S_{3x}	+/- S_{4y}
1.5	22.1	12.2	45.5	34.3
1.25	18.8	13.5	40.3	33.9
1.36	20.30	12.91	42.66	34.08

Now, putting the above values in equation (I) we have,

+/- S_{2x} = 251.771 kg/cm^2

+/- S_{2y} = 160.105 kg/cm^2

+/- S_{3x} = 529.136 kg/cm^2 (Max stress) <1080 kg/cm^2

+/- S_{3y} =0.3xS_{3x} = 158.741 kg/cm^2

+/- S_{4y} = 422.7 kg/cm^2 (Max stress) <1080 kg/cm^2

+/- S_{4x} =0.3xS_{4y} = 126.81 kg/cm^2

Hence the plate thickness adopted is OK.

5.7 Check for Bottom Panel (two short & one long edges fixed and one long edge simply supported) :

Here,

P =	0.16088	N/mm^2	1.64	kg/cm^2
a =	275	mm	27.50	cm
b =	550	mm	55.00	cm
t =	20	mm	2.00	cm
b/a =	2.00			

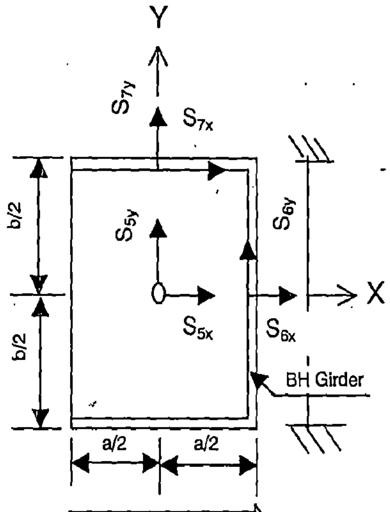


Fig. 5.3

Value of K is taken from table - 2 as below:

b/a	+/- S_{5x}	+/- S_{5y}	+/- S_{6x}	+/- S_{7y}
2.00	33.80	15.50	. 68.30	- 47.00

Here the value of K is maximum for stress, S_{6x}.

Now, putting the corresponding values in equation (I) we have,

+/-' S_{5x} = 104.80 kg/cm^2

+/-' S_{5y} = 48.06 kg/cm^3

+/-' S_{6x} = 211.77 kg/cm^2 (Max stress) <1080 kg/cm^2

+/-' $S_{6y} = 0.3 x S_{6x}$ = 63.53 kg/cm^2

+/-' S_{7y} = 145.729 kg/cm^2

+/-' $S_{7x} = 0.3 x S_{7y}$ = 43.72 kg/cm^2

Hence the plate thickness adopted is OK.

6.0 Design of Vertical Stiffener

6.1 Loading Arrangement :

The vertical stiffeners are designed as simply supported beam on horizontal girders.

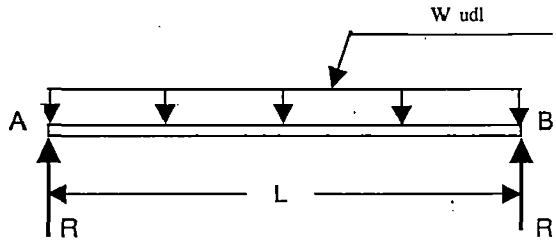


Figure : 6.1

Here,

L = 750 mm or 75 cm

P = 1.64 kg/cm^2

Load bearing width, b= 750 mm or 75 cm

The load on the vertical stiffener (udl), W =Pxb= 123.00 kg/cm

Then the reaction force at A & B, $R_A = R_B$ =WxL/2= 4612.5 kg

Maximum Bending Moment, M=WxL^2/8= 86484.4 kg-cm

6.2 Coacting Width of Skin Plate :

As per ANNEX- 'D' page 15 of IS 4622 : 1992, we have, L_l = 750 mm

So, B=L_l/2 = 375 mm

L_l/B = 2.00

From Fig. 11, page - 15 of IS, corresponding to L_l/B, we have V_l = 0.38

So, coacting width of skin plate = 2xV_l xB = 285 mm

As per IS 4622 : 1992, page 4, art. 5.2.4, we have, coacting widths are given by:

a) 40 t + B ; where, t = thickness of skin plate, and

B = width of stiffener flange in contact with the skin plate;

b) 0.11 span; Here, span = 750 mm

c) centre to centre of stiffeners or girders;

Here, t = 20 mm B = 10 mm

a) coacting width = 40 t + B = 810.00 mm

b) coacting width = 0.11x span= 82.50 mm

c) coacting width = centre to centre of stiffeners = 750.00 mm

Hence, coacting width calculated above, choosen as 82.5 mm and figured section:

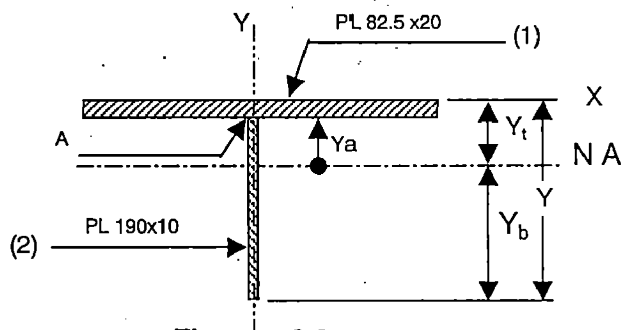


Figure : 6.2

6.3 Calculation of Neutral Axis and Section Modulus :

Table: 6.1

Item No.	Item	Size		A (cm^2)	Y (cm)	AxY	AY^2	I_{self} (cm^4)
		b (cm)	h (cm)	(b x h)	dist from X	(cm^3)	(cm^4)	$bh^3/12$
(1)	skin plate	8.25	2.00	16.50	1.00	16.50	16.50	5.50
(2)	web	1.00	19.00	19.00	11.50	218.50	2512.75	571.58
Sum:			21.00	35.50		235.00	2529.25	577.08

So, Y_t = sum(AY)/sum(A) = 6.62 cm $Y = h_1 + h_2 =$ 21.00 cm

$Y_b = (Y - Y_t) =$ 14.38 cm

$Y_a = (Y_t - h_1) =$ 4.62 cm

Then, I_{NA} = sum(AY^2) + sum(I_{self}) - sum(A)xY_t^2 : 1550.7 cm^4

$Z_t = I_{NA} / Y_t =$ 234.25 cm^3

$Z_b = I_{NA} / Y_b =$ 107.84 cm^3

$Za = I_{NA} / Y_a =$ 335.67 cm^3

6.4 Check for Bending stresses:

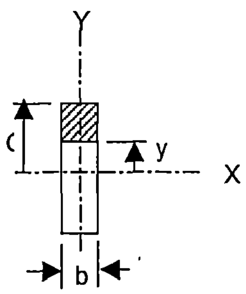
$f_t = M/Z_t =$ 369.19 kg/cm^2 (Tension) <1080 kg/cm^2,

$f_b = M/Z_b =$ 802.01 kg/cm^2 (Compression) <1080 kg/cm^2, Hence safe.

$f_a = M/Z_a =$ 257.65 kg/cm^2 (Tension)

6.5 Check for Shear stresses:

Maximum shear force, $V = R_A =$ 4612.5 kg



We have, shear stress,

$$S_s = \frac{V}{I.b}\int_{Y_0}^{Y_c} Y dA = \frac{V.A.\overline{Y}}{I.b} \quad \text{--- (II)}$$

Where,

A.Y(bar)=$b_1 x h_1 x (h_1/2 + Y_a) + b_2 x Y_a^2/2 =$ 103.396 cm^3

I = 1550.7 cm^4

$b = b_2 =$ 1.00 cm

Putting the values in equation (II), we have max. shear stress at NA,

$f_{s(max)} =$ 307.55 kg/cm^2

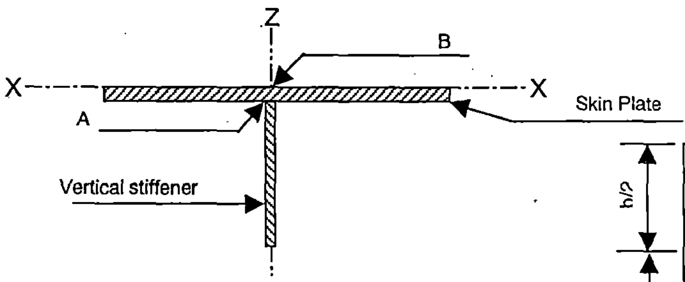
<u>Shear stress at point 'A',</u> < 840.00 kg/cm^2

A.Y(bar)=$b_1 x h_1 x (h_1/2 + Y_a) =$ 92.7254 cm^3

Putting the values in equation (II), we have max. shear stress at point 'A',

$f_{s(A)} =$ 275.81 kg/cm^2

6.6 Check for Combined Stresses for Skin plate with Vertical Stiffeners:



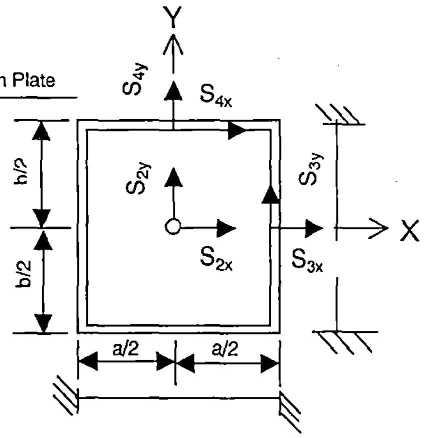


Figure : 6.3

Stresses at point 'B'

Total stress in 'Y' direction:

$f_{BY} = f_{t(stiff)} + S_{3y(skin\ plate)} =$ 582.98 kg/cm^2 (Tension)

Total stress in 'X' direction:

$f_{BX} = S_{3x(skin\ plate)} =$ 712.63 kg/cm^2 (Tension)

Shear stress, $f_{sB} =$ 0 kg/cm^2

So, combined stress:

$fc = sqrt(f_{BY}^2 + f_{BX}^2 - f_{BY} \times f_{BX} + 3S_s^2) =$ 657.464 kg/cm^2 < 1560.00 kg/cm^2

Stresses at point 'A'

Total stress in 'Y' direction:

$f_{AY} = f_{A(stiff)(ten)} + S_{3y(skin\ plate)(comp)} =$ 43.86 kg/cm^2 (Tension)

Total stress in 'X' direction:

$f_{AX} = S_{3x(skin\ plate)} =$ -712.63 kg/cm^2 (Compression)

Shear stress, $f_{sA} =$ 275.81 kg/cm^2

So, combined stress:

$fc = sqrt(f_{BY}^2 + f_{BX}^2 - f_{BY} \times f_{BX} + 3f_{sA}^2) =$ 877.059 kg/cm^2 < 1560.00 kg/cm^2

Hence, the section shoosen is safe.

After Check with Combined Stresses Revise Calculation (ref: sec-13):

6.7 Coacting Width of Skin Plate :

As per ANNEX- 'D' page 15 of IS 4622 : 1992, we have, L_{span} = 750 mm

So, $B=L_{span}/2$ = 375 mm, $L_I = 0.577L_{span}$= 432.75 mm

$L_{II} = 0.423L_{span}$= 317.25 mm

So, L_I/B = 1.15 and L_{II}/B = 0.846

From Fig. 11, page - 15, corresponding L_I/B & L_{II}/B, we have

V_I = 0.22 and V_{II}= 0.12

So, coacting width of skin plate at the centre = $2V_I B$ = 165 mm

and coacting width of skin plate at the support = $2V_{II}B$ = 90 mm

As per IS 4622 : 1992, page 4, art. 5.2.4, we have, coacting widths are given by:

a) 40 t + B ; where, t = thickness of skin plate, and

B = width of stiffener flange in contact with the skin plate;

b) 0.11 span; Here, span = 750 mm

c) centre to centre of stiffeners or firders;

Here, t = 20 mm B = 10 mm

a) coacting width = 40 t + B = 810.00 mm

b) coacting width = 0.11x span= 82.50 mm

c) coacting width = centre to centre of stiffeners = 550.00 mm

Hence, the least value among the above coacting widths is 82.5 mm and figured section:

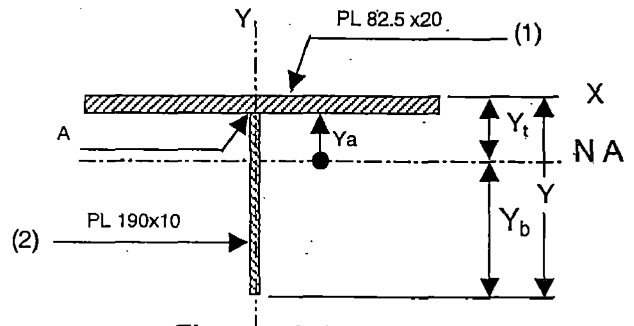


Figure : 6.4

6.8 Calculation of Neutral Axis and Modulus of Section :

Table: 6.2

Item No.	Item	Size		A (cm²)	Y (cm)	AxY	AY²	I_{self} (cm⁴)
		b (cm)	h (cm)	(b x h)	dist from X	(cm³)	(cm⁴)	bh³/12
(1)	skin plate	8.25	2.00	16.50	1.00	16.50	16.50	5.50
(2)	web	1.00	19.00	19.00	11.50	218.50	2512.75	571.58
Sum:			21.00	35.50		235.00	2529.25	577.08

So, Y_t = sum(AY)/sum(A) = 6.62 cm $Y = h_1 + h_2$ = 21.00 cm

$Y_b = (Y - Y_t)$ = 14.38 cm

$Y_a = (Y_t - h_1)$ = 4.62 cm .

Then, I_{NA} = sum(AY^2) + sum(I_{self}) - sum(A)xY_t^2 = 1550.7 cm⁴

$Z_t = I_{NA}/Y_t$ = 234.25 cm³

$Z_b = I_{NA}/Y_b$ = 107.84 cm³

$Za = I_{NA}/Y_a$ = 335.67 cm³

6.9 Check for Bending stresses:

Here,

L = 750 mm or 75 cm

P = 1.64 kg/cm^2

Load bearing width, b= 550 mm or 55 cm

The load on the vertical stiffener (udl), W =Pxb= 90.20 kg/cm

Then the reaction force at A & B, $R_A = R_B$ =WxL/2= 3382.5 kg

Maximum Bending Moment, $M=WxL^2/8=$ 63421.9 kg-cm

$f_t = M/Z_t =$ 270.74 kg/cm^2 (Tension) <1080 kg/cm^2 ,

$f_b = M/Z_b =$ 588.14 kg/cm^2 (Compression) <1080 kg/cm^2 , Hence safe.

$f_a = M/Z_a =$ 188.94 kg/cm^2 (Tension)

6.10 Check for Shear stresses:

Maximum shear force, $V = R_A =$ 3382.5 kg

We have, shear stress, $$S_s = \frac{V}{I.b}\int_{Y_0}^{Y_c} Y dA = \frac{V.A.\overline{Y}}{I.b} \quad \text{--- (II)}$$

Where,

A.Y(bar)=$b_1xh_1x(h_1/2 + Y_a) + b_2xY_a^2/2 =$ 103.396 cm^3

I = 1550.7 cm^4

$b = b_2 =$ 1.00 cm

Putting the values in equation (II), we have max. shear stress at NA,

$f_{s(max)} =$ 225.54 kg/cm^2

< 840.00 kg/cm^2

<u>Shear stress at point 'A',</u>

A.Y(bar)=$b_1xh_1x(h_1/2 + Y_a) =$ 92.7254 cm^3

Putting the values in equation (II), we have max. shear stress at point 'A',

$f_{s(A)} =$ 202.26 kg/cm^2

6.11 Check for Combined Stresses for Skin plate with Vertical Stiffeners:

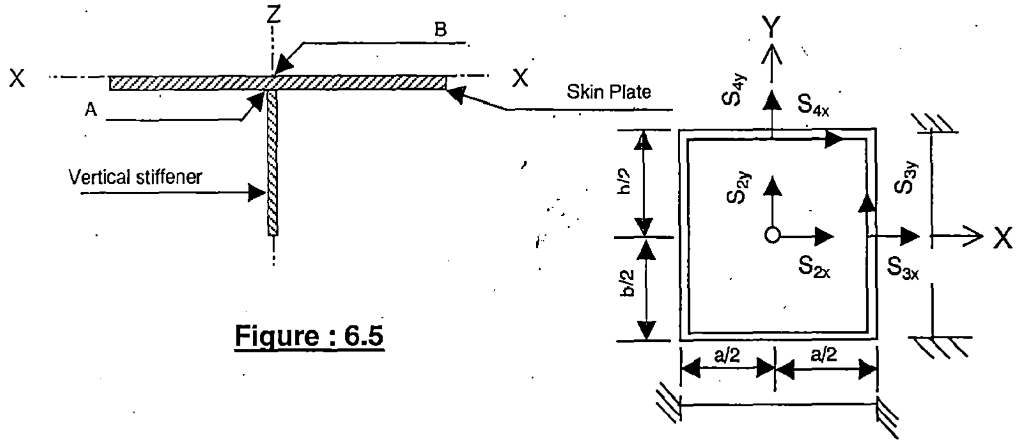


Figure : 6.5

Stresses at point 'B'

Total stress in 'Y' direction:

$f_{BY} = f_{t(stiff)} + S_{3y(skin\ plate)} =$ 429.48 kg/cm^2 (Tension)

Total stress in 'X' direction:

$f_{BX} = S_{3x(skin\ plate)} =$ 529.14 kg/cm^2 (Tension)

Shear stress, $f_{sB} =$ 0 kg/cm^2

So, combined stress:

$fc = \mathrm{sqrt}(f_{BY}^2 + f_{BX}^2 - f_{BY} \times f_{BX} + 3S_s^2) =$ 487.016 kg/cm^2 < 1560.00 kg/cm^2

Stresses at point 'A'

Total stress in 'Y' direction:

$f_{AY} = f_{A(stiff)(ten)} + S_{3y(skin\ plate)(comp)} =$ 30.20 kg/cm^2

Total stress in 'X' direction:

$f_{AX} = S_{3x(skin\ plate)} =$ -529.14 kg/cm^2 (Compression)

Shear stress, $f_{sA} =$ 202.26 kg/cm^2

So, combined stress:

$fc = \mathrm{sqrt}(f_{BY}^2 + f_{BX}^2 - f_{BY} \times f_{BX} + 3f_{sA}^2) =$ 647.768 kg/cm^2 < 1560.00 kg/cm^2

Hence, the section shoosen is safe.

7.0 Design for Section of Horizontal Girder :

7.1 Girder Arrangement :

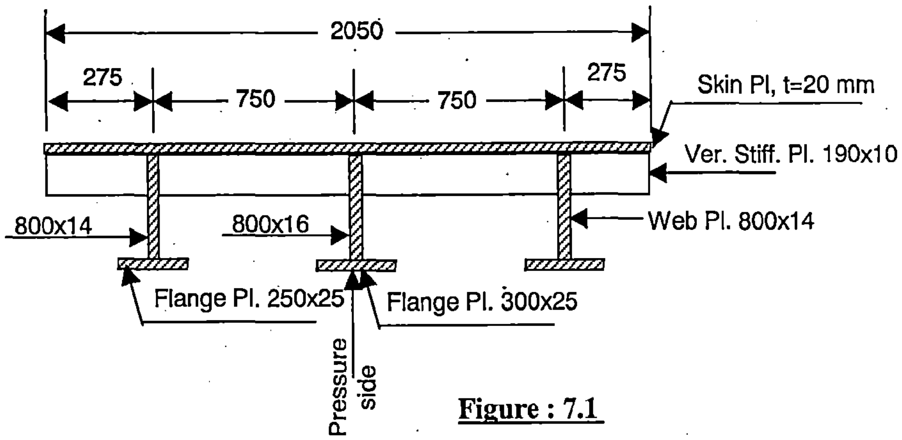


Figure : 7.1

7.2 Design for Central Horizontal Girder :

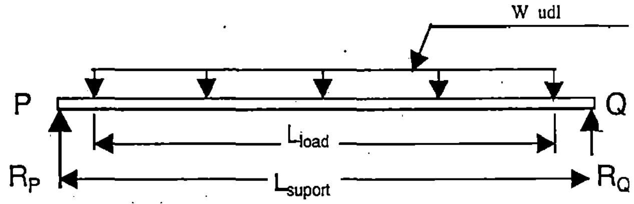


Figure : 7.2

Here,	Water pressure, P =	1.64 kg/cm^2		
	Width of water pressure on beam, B =	75.00 cm		
	Water load per unit length (udl), W =PxB=	123.00 kg/cm		
	Length of load span on the beam, L_{load} =	6900.00 mm	or	690.00 cm
	Centre to centre of wheel, L_{suport} =	7250.00 mm	or	725.00 cm
	Reaction force, $R_p = R_Q = WxL_{load}/2$ =	42435.0 kg		
	Max B.M, $M = R_P x L_{suport}/2 - (WxL^2_{load}/8)$=	8062650 kg-cm		

7.3 Coacting width of Skin Plate with Horizontal Girder :

As per IS 4622 : 1992, page 4, art. 5.2.4, we have, coacting widths are given by:

a) 40 t + B ; where, t = thickness of skin plate, and
B = width of stiffener flange in contact with the skin plate;

b) 0.11 span; Here, span = 7250 mm

c) centre to centre of stiffeners or girders;

Here, t = 20 mm B = 16 mm

a) coacting width = 40 t + B = 816.00 mm

b) coacting width = 0.11x span= 797.50 mm

c) coacting width = centre to centre of Girders = 750.00 mm

Hence, coacting width calculated above, choosen as 750 mm (least).

7.4 Calculation of Neutral Axis and Section Modulus :

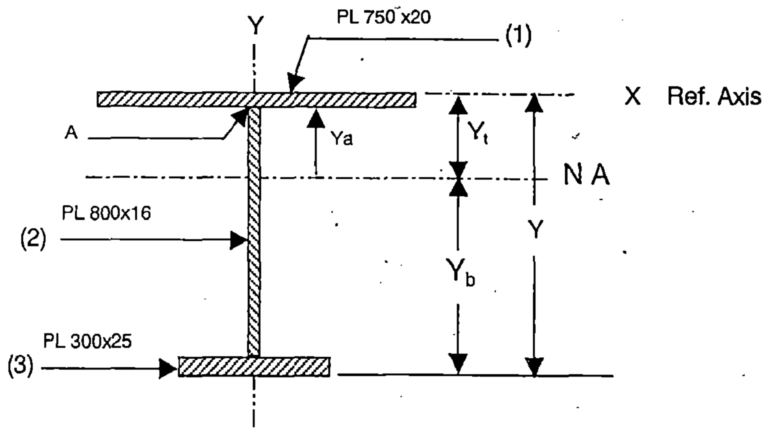


Figure : 7.3

Table: 7.1

Item No.	Item	Size		A (cm^2)	Y (cm)	AxY	AY^2	I_{self} (cm^4)
		b (cm)	h (cm)	(b x h)	dist from X	(cm^3)	(cm^4)	$bh^3/12$
(1)	skin plate	75.00	2.00	150.00	1.00	150.00	150.00	50.0
(2)	web	1.60	80.00	128.00	42.00	5376.00	225792	68266.7
(3)	Flange	30.00	2.50	75.00	83.25	6243.75	519792	39.1
Sum:			84.50	353.00		11769.8	745734	68355.7

So, Y_t = sum(AY)/sum(A) = 33.34 cm $Y = h_1 + h_2 + h_3 =$ 84.50 cm

$Y_b = (Y - Y_t) =$ 51.16 cm

$Y_a = (Y_t - h_1) =$ 31.34 cm

Then, $I_{NA} = \text{sum}(AY^2) + \text{sum}(I_{self}) - \text{sum}(A) \times Y_t^2 =$ 421662 cm^4

$Z_t = I_{NA} / Y_t =$ 12646.5 cm^3

$Z_b = I_{NA} / Y_b =$ 8242.36 cm^3

$Za = I_{NA} / Y_a =$ 13453.6 cm^3

7.5 Check for Bending stresses:

$f_t = M/Z_t =$ 637.54 kg/cm^2 (Tension) <1080 kg/cm^2 ,

$f_b = M/Z_b =$ 978.20 kg/cm^2 (Compression) <1080 kg/cm^2 , Hence safe.

$f_a = M/Z_a =$ 599.30 kg/cm^2 (Tension)

7.6 Check for Combined Stresses for Horizontal Girder with Skin plate :

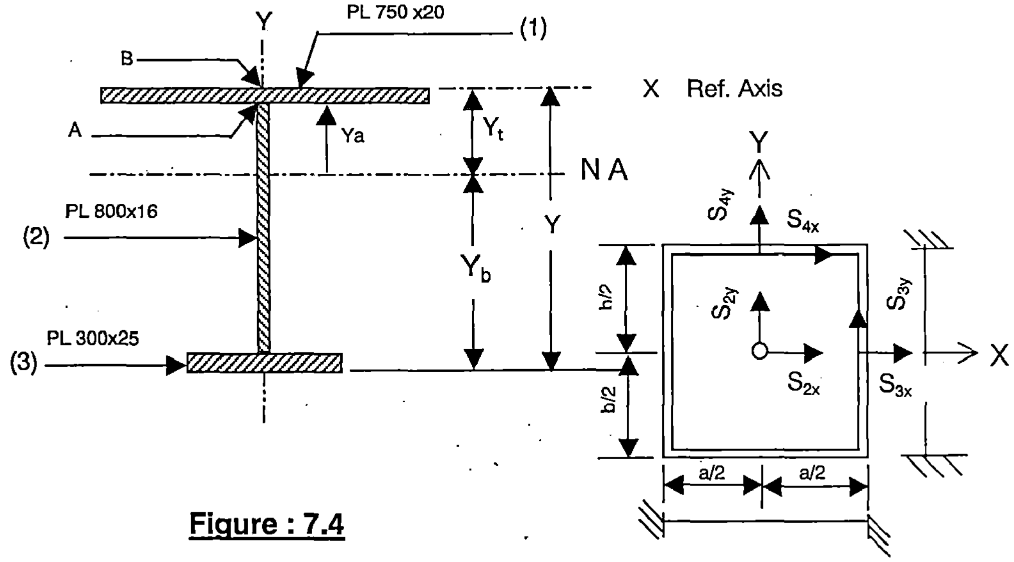


Figure : 7.4

Stresses at point 'B'

Total stress in 'X' direction:

$f_{BX} = f_{t(beam)} + S_{4x(skin\ plate)} =$ 851.33 kg/cm^2 (Tension)

Total stress in 'Y' direction:

$f_{BY} = S_{4y(skin\ plate)} =$ 712.63 kg/cm^2 (Tension)

Shear stress, $f_{sB} =$ 0 kg/cm^2

So, combined stress:

$fc = sqrt(f_{BY}^2 + f_{BX}^2 - f_{BY} \times f_{BX} + 3S_s^2) =$ 791.15 kg/cm^2 < 1560.00 kg/cm^2

Hence safe.

Stresses at point 'A'

Total stress in 'X' direction:

$f_{AX} = f_{A(beam)(ten)} + S_{4x(skin\ plate)(comp)} =$ 385.51 kg/cm^2 (Tension)

Total stress in 'Y' direction:

$f_{AY} = S_{4y(skin\ plate)} =$ -712.63 kg/cm^2 (Compression)

Shear stress, $f_{sA} = \frac{VAY}{Ib} =$ 0.00 kg/cm^2 (since, at a pt of max BM, V=0)

So, combined stress:

$fc = sqrt(f_{AX}^2 + f_{AY}^2 - f_{AX} \times f_{AY} + 3f_{sA}^2) =$ 964.978 kg/cm^2 < 1560.00 kg/cm^2

Hence, the section shoosen is safe.

7.7 Check for Deflection of Centre Horizontal Girder :

Allowable deflection for centre girder, say d = L/800

Here, span, L = 7250 mm Then, d = 9.06 mm, or 0.90625 cm

And, actual deflection, $d_{actual} = \frac{5}{384} \times \frac{WL^4}{EI}$ ----------------- (III

Here, W = 123.00 kg/cm E = 2.01E+06 kg/cm^2

L = 725.00 cm I = 421662 cm^4

Now, putting the values in equation (III) we have,

$d_{actual} =$ 0.52208 cm < 0.90625 cm

Hence safe.

7.8 Curtailment of Centre Horizontal Girder :

Size of the Horizontal Girder required adjoining the End Girder will be as below:

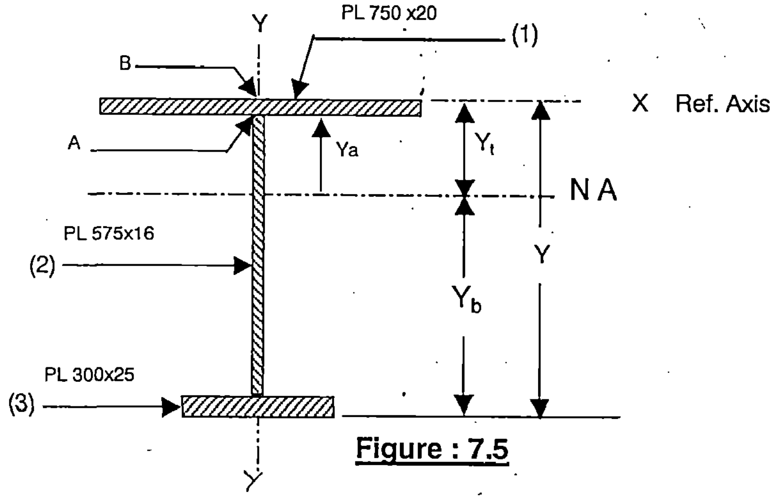


Figure : 7.5

7.8.1 **Calculation of Neutral Axis and Modulus of Section :**

Table: 7.2

Item No.	Item	Size		A (cm^2)	Y (cm)	AxY	AY^2	I_{self} (cm^4)
		b (cm)	h (cm)	(b x h)	dist from X	(cm^3)	(cm^4)	$bh^3/12$
(1)	skin plate	75.00	2.00	150.00	1.00	150.00	150.00	50.0
(2)	web	1.60	57.50	92.00	30.75	2829.00	86992	25347.9
(3)	Flange	30.00	2.50	75.00	60.75	4556.25	276792	39.1
Sum:			62.00	317.00		7535.3	363934	25437.0

So, Y_t = sum(AY)/sum(A) = 23.77 cm $\quad$ $Y = h_1 + h_2 + h_3 =$ 62.00 cm

$Y_b = (Y - Y_t) =$ 38.23 cm

$Y_a = (Y_t - h_1) =$ 21.77 cm

Then, $I_{NA} = \text{sum}(AY^2) + \text{sum}(I_{self}) - \text{sum}(A) x Y_t^2 =$ 210254 cm^4

$Z_t = I_{NA} / Y_t =$ 8845.2 cm^3

$Z_b = I_{NA} / Y_b =$ 5499.79 cm^3

$Za = I_{NA} / Y_a =$ 9657.8 cm^3

7.8.2 **Calculation of Bending Moment :**

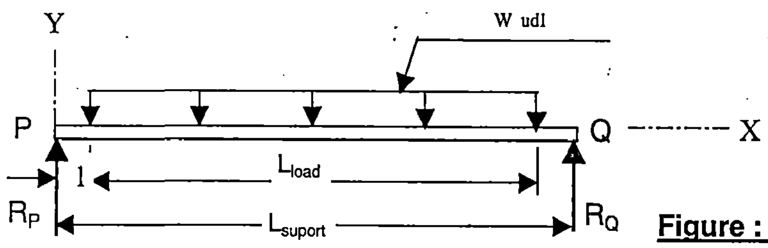


Figure : 7.6

Here, Water pressure, P = 1.64 kg/cm^2
Width of water pressure on beam, B = 75.00 cm
Water load per unit length (udl), W =PxB= 123.00 kg/cm
Length of load on beam, L_{load} = 6900.00 mm or 690.00 cm
Centre to centre of wheel, L_{suport} = 7250.00 mm or 725.00 cm
Reaction force, $R_p = R_Q = WxL_{load}/2$ = 42435.0 kg
Distance of the point in X direction , x = 90 cm (middle of taper sec)
Offset of load from point P, l = 17.5 cm
So, B.M at a distance x , $M = R_P.x - (W.(x-l)^2/2)$ = 3495891 kg-cm

7.8.3 Check for Bending stresses:

$f_t = M/Z_t$ = 395.23 kg/cm^2 (Tension)
$f_b = M/Z_b$ = 635.64 kg/cm^2 (Comp.) < 1080.00 kg/cm^2
$f_a = M/Z_a$ = 361.98 kg/cm^2 (Tension) **Hence safe.**

7.8.4 Check for Combined Stresses for Curtail Central H Girder with Skin plate :

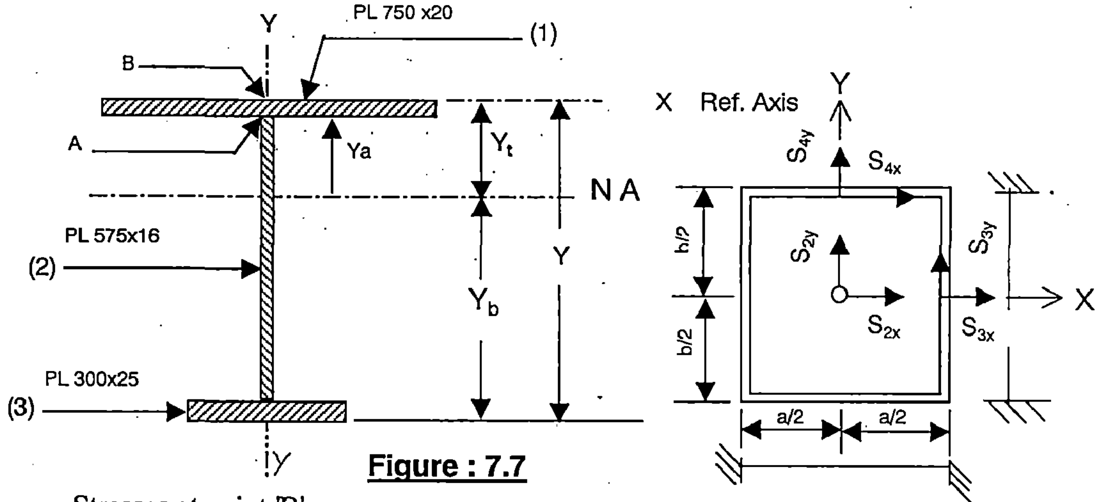


Figure : 7.7

Stresses at point 'B'

Total stress in 'X' direction:

$f_{BX} = f_{t(beam)} + S_{4x(skin\ plate)}$ = 609.02 kg/cm^2 (Tension)

Total stress in 'Y' direction:

$f_{BY} = S_{4y(skin\ plate)}$ = 712.63 kg/cm^2 (Tension)

Shear stress, f_{BS} = 0 kg/cm^2

So, combined stress at pt B:

$fc = \text{sqrt}(f_{BY}^2 + f_{BX}^2 - f_{BY} \times f_{BX} + 3f_{Bs}^2)$ = 666.89 kg/cm^2 < 1560.00 kg/cm^2

Hence safe.

Stresses at point 'A'

Total stress in 'X' direction:

$f_{AX} = f_{A(beam)(ten)} + S_{4x(skin\ plate)(comp)}$ = 148.19 kg/cm^2 (Tension)

Total stress in 'Y' direction:

$f_{AY} = S_{4y(skin\ plate)}$ = -712.63 kg/cm^2 (Compression)

Shear force at a distance x, $V= R_P - W.(x-l) =$ 33517.5 kg

$A.Y(bar)=b_1xh_1x(h_1/2 +Y_a)+b_2xY_a^2/2 =$ 3794.74 cm^3

I = 210254 cm^4 $b = b_2 =$ 1.60 cm

Shear stress, $f_{AS} = \frac{V\overline{AY}}{Ib} =$ 378.08 kg/cm^2

So, combined stress at pt A:

$fc = sqrt(f_{AY}^2 + f_{AX}^2 - f_{AY} x f_{AX} +3f_{AS}^2) =$ 1031.63 kg/cm^2 < 1560.00 kg/cm^2

Hence, the section shoosen is safe.

7.8.5 Check for Deflection of Curtail Centre Horizontal Girder :

Allowable deflection for centre girder, say d = L/800

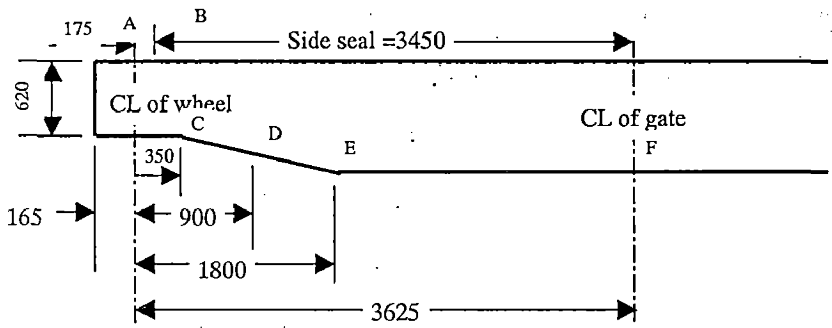
Here, span, L = 7250 mm Then, d = 9.06 mm, or 0.90625 cm

Actual deflection:

The mathematical determination of deflection becomes tedious when the section is not uniform through out the length as here due to curtailment of girder's section. Graphical determination technique may be conveniently applied. So with reference to Design of Machine Members by Alex Vallance & Vention Levy Donghtie's book, page-194, we have that texts on machines show that the second derivative of the deflection equation of any shaft is expressd by the relation.

$$\frac{d^2y}{dx^2} = \frac{M}{IE} \quad \text{(IV)}$$

Where, y = deflection at a distance x from one end of the shaft in cm;
M = bending moment at section x in kg-cm;
E = modulus of elasticity of material in kg/cm^2
I = rectangular moment of inertia if shaft area at the same section in cm^4



Here, web size is 800x16 mm from pt. E - F & 575x16 mm A - C as shown in above fig.

Here, Water pressure, P = 1.64 kg/cm^2

Width of water pressure on beam, B = 75.00 cm

Water load per unit length (udl), W =PxB= 123.00 kg/cm

Length of load span on the beam, L_{load} = 6900.00 mm or 690.00 cm

Centre to centre of wheel, L_{suport} = 7250.00 mm or 725.00 cm

Reaction force, $R_A = WxL_{load}/2 =$ 42435.0 kg

x =Distance from centre of wheel to pt. in X direction,

Offset of load from point P, l = 17.5 cm

So, B.M at different pt at a distance x , $M = R_P.x - (W.(x-l)^2/2)$ kg-cm

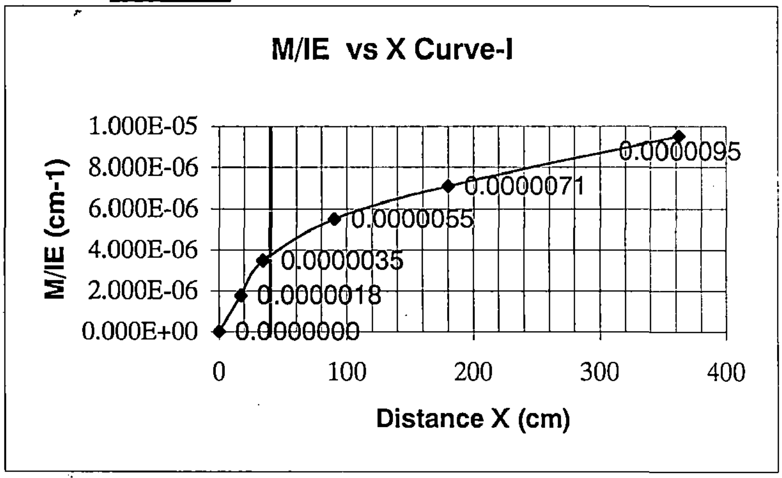
Table: 7.3 E = 2.01E+06 kg-cm^2

Point	Size of web		Size of flange		Dist from centre of wheel x (cm)	Bending moment M(kg-cm)	Moment of inertia I (cm4)	M/IE (cm-1)
	Height h (cm)	Thickness t_w (cm)	Top bxt (cm)	Bottom bxt (cm)				
A	57.50	1.60	75.0x2.0	30.0x2.5	0.00	0.0	210254.2	0.000E+00
B	57.50	1.60	75.0x2.0	30.0x2.5	17.50	742612.5	210254.2	1.757E-06
C	57.50	1.60	75.0x2.0	30.0x2.5	35.00	1466391	210254.2	3.470E-06
D	68.75	1.60	75.0x2.0	30.0x2.5	90.00	3495891	315958.2	5.505E-06
E	80.00	1.60	75.0x2.0	30.0x2.5	180.00	6014316	421662.1	7.096E-06
F	80.00	1.60	75.0x2.0	30.0x2.5	362.50	8062650.0	421662.1	9.513E-06

Now, by ploting the above values of X & M/IE, we get the following curve -I :

Graph: 7.1

X (cm)	M/IE (cm^{-1})
0.00	0.000E+00
17.50	1.757E-06
35.00	3.470E-06
90.00	5.505E-06
180.00	7.096E-06
362.50	9.513E-06



One small area = 0.00004

Now, find the integration ie. area under the curve-I;

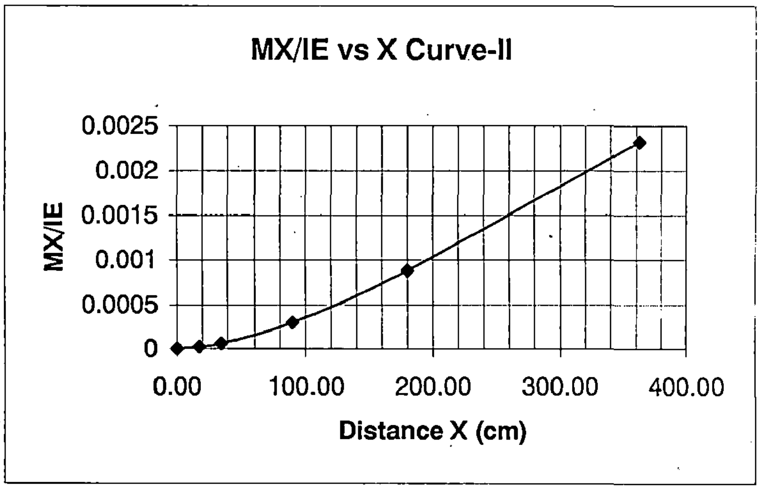
			X(cm)	(MX/IE)
up to 1st. Point no. of small area =	0		0.00	0
up to 2nd. Point no. of small area =	0.50	= area =	17.50	2.00E-05
up to 3rd. Point no. of small area =	1.5		35.00	6.00E-05
up to 4th. Point no. of small area =	7.5		90.00	3.00E-04
up to 5th Point no. of small area =	22		180.00	8.80E-04
up to 6th Point no. of small area =	58		362.50	2.32E-03

Now, by ploting the above values of X & MX/IE, we get the following curve -II :

One small area = 0.01 cm

Total no. of small area above the curve-II = 54

Deflection, d_{actual} = area = 0.54 cm



Graph: 7.2

8.0 Design of Bottom Horizontal Girder :

8.1 Calculation of Bending Moment :

As the water load acting on the bottom horizontal girder will be less than the central horizontal girder. Hence it has to design seperately as below.

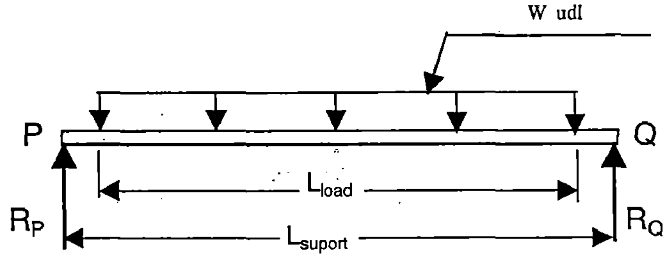


Figure : 8.1

Here, Design Water pressure, P =	1.64 kg/cm^2		
Distance between centre & bottom H girder, d =	75.00 cm		
Distance bet. BH girder & edge of skin plate, e =	27.50 cm		
Width of water pressure on beam, B = d/2 +e =	65.00 cm		
Water load per unit length (udl), W =PxB=	106.60 kg/cm		
Length of load on beam, L_{load} =	6900.00 mm	or	690.00 cm
Centre to centre of wheel, L_{suport} =	7250.00 mm	or	725.00 cm
Reaction force, $R_p = R_Q = WxL_{load}/2$ =	36777.0 kg		
Max B.M, $M = R_PxL_{suport}/2 - (WxL^2_{load}/8)$=	6987630 kg-cm		

8.2 Coacting width of Skin Plate with Bottom Horizontal Girder :

As per ANNEX- 'D' page 15 of IS 4622 : 1992, we have, L_I = 750 mm

So, $B=L_I/2$ = 375 mm

L_I/B = 2.00

From Fig. 11, page - 15, corresponding to L_I/B, we have V_I = 0.38

So, coacting width of skin plate = $2xV_I\,xB$ = 285 mm

As per IS 4622 : 1992, page 4, art. 5.2.4, we have, coacting widths are also given by:

a) 40 t + B ; where, t = thickness of skin plate, and
B = width of stiffener flange in contact with the skin plate;

b) 0.11 span; Here, span = 7250 mm

c) centre to centre of stiffeners or girders;

Here, t = 20 mm B = 16 mm

a) coacting width = 40 t + B = 816.00 mm

b) coacting width = 0.11x span= 797.50 mm

c) coacting width = double of the end distance i.e. 2e = 550.00 mm

Hence, coacting width calculated above, choosen as 550 mm (least).

8.3 Calculation of Neutral Axis and Section Modulus :

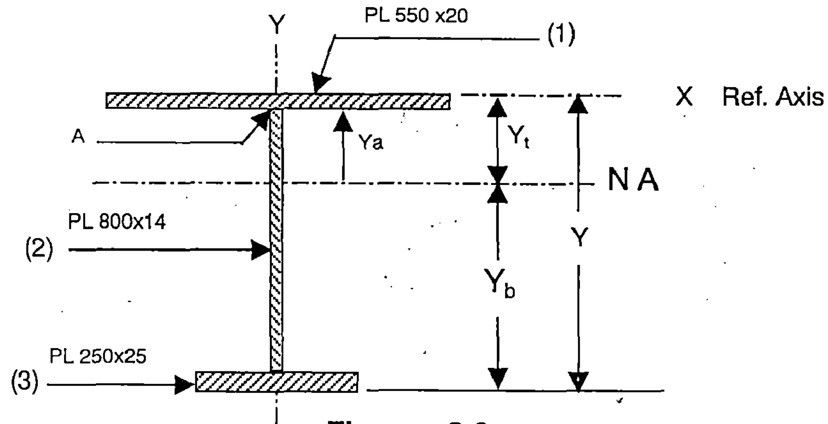


Figure : 8.2

Table: 8.1

Item No.	Item	Size		A (cm^2)	Y (cm)	AxY	AY^2	I_{self} (cm^4)
		b (cm)	h (cm)	(b x h)	dist from X	(cm^3)	(cm^4)	$bh^3/12$
(1)	skin plate	55.00	2.00	110.00	1.00	110.00	110.00	36.67
(2)	web	1.40	80.00	112.00	42.00	4704.00	197568	59733
(3)	Flange	25.00	2.50	62.50	83.25	5203.13	433160	32.55
Sum:			84.50	284.50		10017.1	630838	59803

So, Y_t = sum(AY)/sum(A) = 35.21 cm $\quad$ $Y = h_1 + h_2 + h_3 =$ 84.50 cm

$Y_b = (Y - Y_t) =$ 49.29 cm

$Y_a = (Y_t - h_1) =$ 33.21 cm

Then, $I_{NA} = \text{sum}(AY^2) + \text{sum}(I_{self}) - \text{sum}(A) \times Y_t^2 =$ 337942 cm^4

$Z_t = I_{NA}/Y_t =$ 9598.0 cm^3

$Z_b = I_{NA}/Y_b =$ 6856.14 cm^3

$Za = I_{NA}/Y_a =$ 10176.0 cm^3

8.4 Check for Bending stresses:

$f_t = M/Z_t =$ 728.03 kg/cm^2 (Tension) <1080 kg/cm^2 ,

$f_b = M/Z_b =$ 1019.18 kg/cm^2 (Compression) <1080 kg/cm^2 , Hence safe.

$f_a = M/Z_a =$ 686.67 kg/cm^2 (Tension)

8.5 Check for Combined Stresses for Bottom H. Girder with Skin plate :

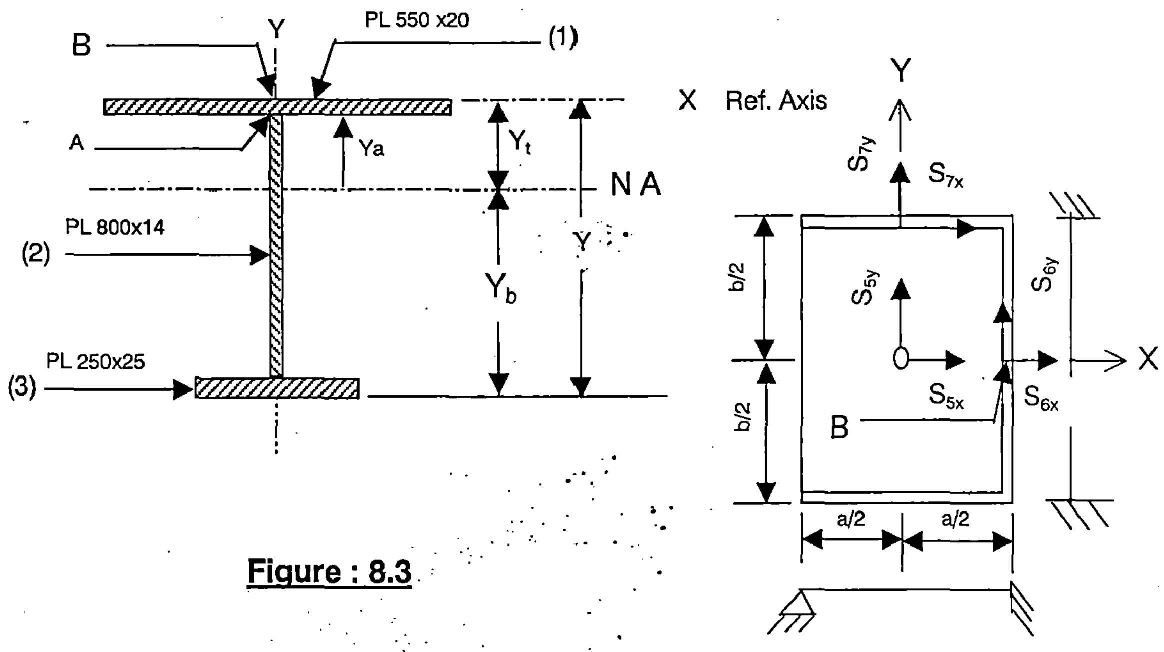


Figure : 8.3

Stresses at point 'B'

Total stress in 'Y' direction:

$f_{BY} = f_{t(beam)} + S_{6y(skin\ plate)} =$ 796.46 kg/cm^2 (Tension)

Total stress in 'X' direction:

$f_{BX} = S_{6X(skin\ plate)} =$ 228.09 kg/cm^2 (Tension)

Shear stress, $f_{sB} =$ 0 kg/cm^2

So, combined stress:

$fc = sqrt(f_{BY}^2 + f_{BX}^2 - f_{BY} \times f_{BX} + 3S_s^2) =$ 710.42 kg/cm^2 < 1560.00 kg/cm^2

Hence safe.

Stresses at point 'A'

Total stress in 'Y' direction:

$f_{AY} = f_{A(beam)(ten)} + S_{6Y(skin\ plate)(comp)} =$ 618.25 kg/cm^2 (Tension)

Total stress in 'X' direction:

$f_{AX} = S_{6X(skin\ plate)} =$ -228.09 kg/cm^2 (Compression)

Shear stress, $f_{sA} = \frac{VA\bar{Y}}{Ib} =$ 0.00 kg/cm^2

So, combined stress:

$fc = sqrt(f_{AY}^2 + f_{AX}^2 - f_{AY} \times f_{AX} + 3f_{sA}^2) =$ 758.47 kg/cm^2 < 1560.00 kg/cm^2

Hence, the section shoosen is safe.

8.6 Curtailment of Bottom Horizontal Girder :

Size of the Horizontal Girder required adjoining the End Girder will be as below:

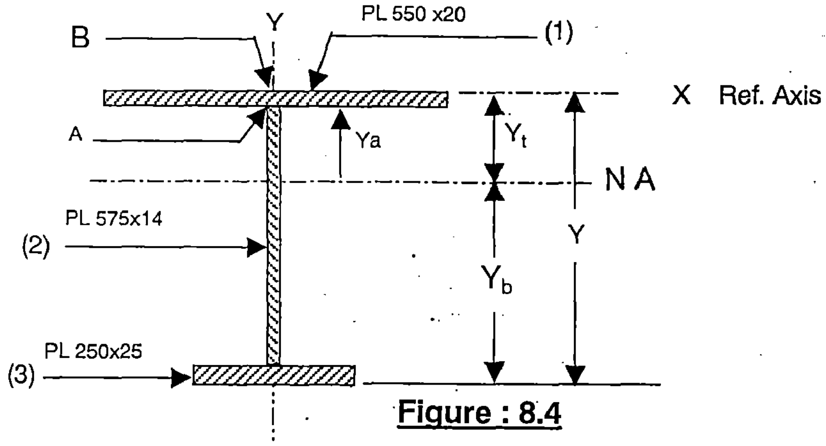


Figure : 8.4

8.6.1 **Calculation of Neutral Axis and Modulus of Section of Curtail BH Girder :**

Table: 8.2

Item No.	Item	Size		A (cm^2)	Y (cm)	AxY	AY^2	I_{self} (cm^4)
		b (cm)	h (cm)	(b x h)	dist from X	(cm^3)	(cm^4)	$bh^3/12$
(1)	skin plate	55.00	2.00	110.00	1.00	110.00	110.00	36.7
(2)	web	1.40	57.50	80.50	30.75	2475.38	76118	22179.4
(3)	Flange	25.00	2.50	62.50	60.75	3796.88	230660	32.6
Sum:			62.00	253.00		6382.3	306888	22248.6

So, Y_t = sum(AY)/sum(A) = 25.23 cm $Y = h_1 + h_2 + h_3 =$ 62.00 cm

$Y_b = (Y - Y_t) =$ 36.77 cm

$Y_a = (Y_t - h_1) =$ 23.23 cm

Then, $I_{NA} = \text{sum}(AY^2) + \text{sum}(I_{self}) - \text{sum}(A) x Y_t^2 =$ 168136 cm^4

$Z_t = I_{NA} / Y_t =$ 6665.12 cm^3

$Z_b = I_{NA} / Y_b =$ 4572.18 cm^3

$Za = I_{NA} / Y_a =$ 7239.05 cm^3

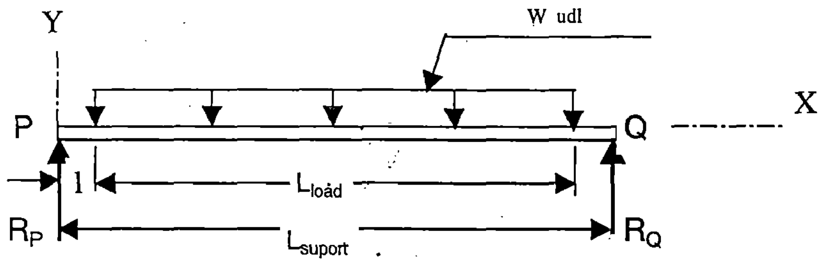


Figure : 8.5

Here,

Water pressure, P =	1.64 kg/cm^2			
Width of water pressure on beam, B =	65.00 cm			
Water load per unit length (udl), W =PxB=	106.60 kg/cm			
Length of load on beam, L_{load} =	6900.00 mm	or	690.00 cm
Centre to centre of wheel, L_{suport} =	7250.00 mm	or	725.00 cm
Reaction force, $R_p = R_Q = W x L_{load}/2$ =	36777.0 kg			

Distance of the point in X direction , x = 90.0 cm (middle of taper sec)

Offset of load from point P, l = 17.5 cm

So, B.M at a distance x , $M = R_P.x - (W.(x-l)^2/2) =$ 3029772 kg-cm

8.6.2 Check for Bending stresses of Curtail BH Girder:

$f_t = M/Z_t =$ 454.57 kg/cm^2 (Tension)

$f_b = M/Z_b =$ 662.65 kg/cm^2 (Comp.) < 1080.00 kg/cm^2

$f_a = M/Z_a =$ 418.53 kg/cm^2 (Tension) **Hence safe.**

8.6.3 Check for Combined Stresses for Curtail BH Girder with Skin plate :

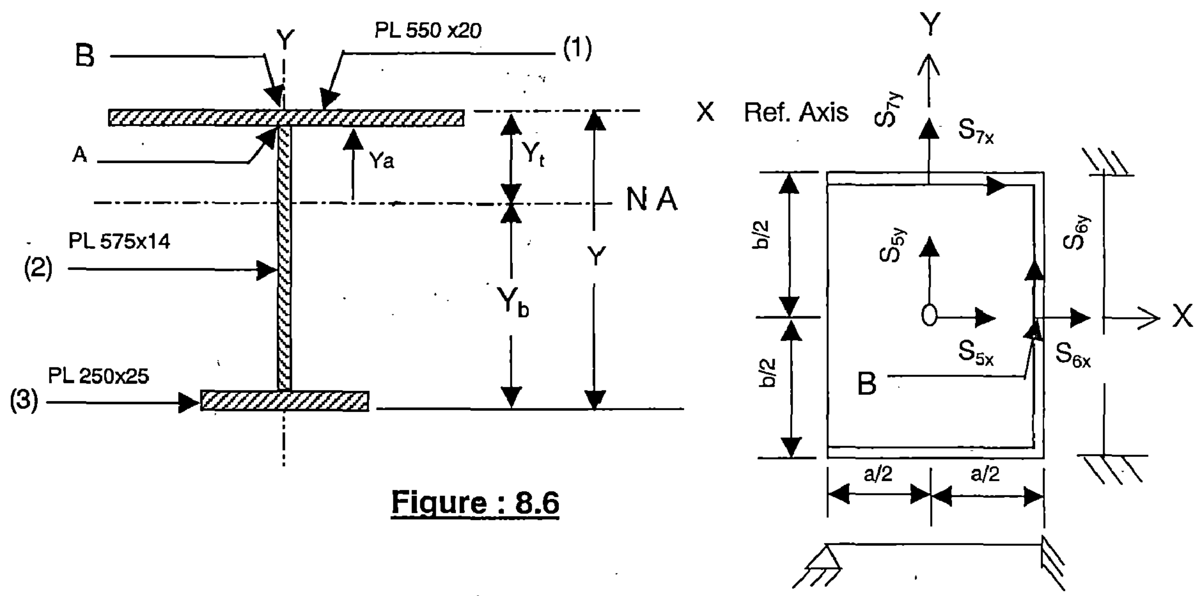


Figure : 8.6

Stresses at point 'B'

Total stress in 'Y' direction:

$f_{BY} = f_{t(beam)} + S_{6y(skin\ plate)} =$ 523.00 kg/cm^2 (Tension)

Total stress in 'X' direction:

$f_{BX} = S_{6X(skin\ plate)} =$ 228.09 kg/cm^2 (Tension)

Shear stress, $f_{sB} =$ 0 kg/cm^2

So, combined stress:

$fc = sqrt(f_{BY}^2 + f_{BX}^2 - f_{BY} x f_{BX} + 3S_s^2) =$ 454.16 kg/cm^2 < 1560.00 kg/cm^2

Hence safe.

Stresses at point 'A'

Total stress in 'Y' direction:

$f_{AY} = f_{A(beam)(ten)} + S_{6Y(skin\ plate)(comp)} =$ 350.10 kg/cm^2 (Tension)

Total stress in 'X' direction:

$f_{AX} = S_{6X(skin\ plate)} =$ -228.09 kg/cm^2 (Compression)

Shear force at distance x, $V = R_P - W.(x-l) =$ 29048.5 kg

$A.Y(bar) = b_1 x h_1 x (h_1/2 + Y_a) + b_2 x Y_a^2/2 =$ 3042.51 cm^3

I = 168136 cm^4 $b = b_2 =$ 1.40 cm

Shear stress, $f_{AS} = \frac{VA\overline{Y}}{Ib} =$ 375.46 kg/cm^2

So, combined stress:

$fc = sqrt(f_{AY}^2 + f_{AX}^2 - f_{AY} \times f_{AX} + 3f_{sA}^2) =$ 823.03 $kg/cm^2 <$ 1560.00 kg/cm^2

Hence, the section choosen is safe.

8.6.4 Check for Deflection of Curtail Bottom Horizontal Girder :

Allowable deflection for centre girder, say, d = L/800

Here, span, L = 7250 mm Then, d = 9.06 mm, or 0.90625 cm

Actual deflection:

The mathematical determination of deflection becomes tedious when the section is not uniform through out the length as here due to curtailment of girder's section. Graphical determination technique may be conveniently applied. So with reference to Design of Machine Members by Alex Vallance & Vention Levy Donghtie's book, page-194, we have that texts on machines show that the second derivative of the deflection equation of any shaft is expressd by the relation.

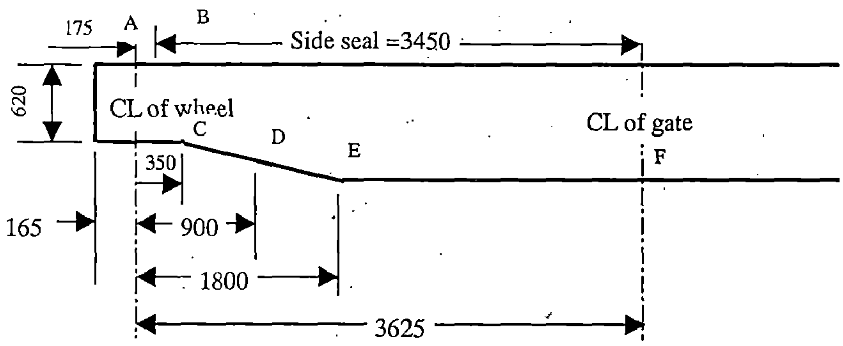
$$\frac{d^2y}{dx^2} = \frac{M}{IE} \quad \text{------------------ (IV)}$$

Where, y = deflection at a distance x from one end of the shaft in cm;

M = bending moment at section x in kg-cm;

E = modulus of elasticity of material in kg/cm^2

I = rectangular moment of inertia if shaft area at the same section in cm^4



Here, web size is 800x14 mm from pt. E - F & 575x14 mm A - C as shown in above fig.

Here, Design Water Pressure, P = 1.64 kg/cm^2

Width of water pressure on beam, B = 65.00 cm

Water load per unit length (udl), W =PxB= 106.60 kg/cm

Length of load on beam, L_{load} = 6900.00 mm or 690.00 cm

Centre to centre of wheel, L_{suport} = 7250.00 mm or 725.00 cm

Reaction force, $R_A = W x L_{load}/2 =$ 36777.0 kg

x =Distance from centre of wheel to pt. in X direction,

Offset of load from point P, l = 17.5 cm

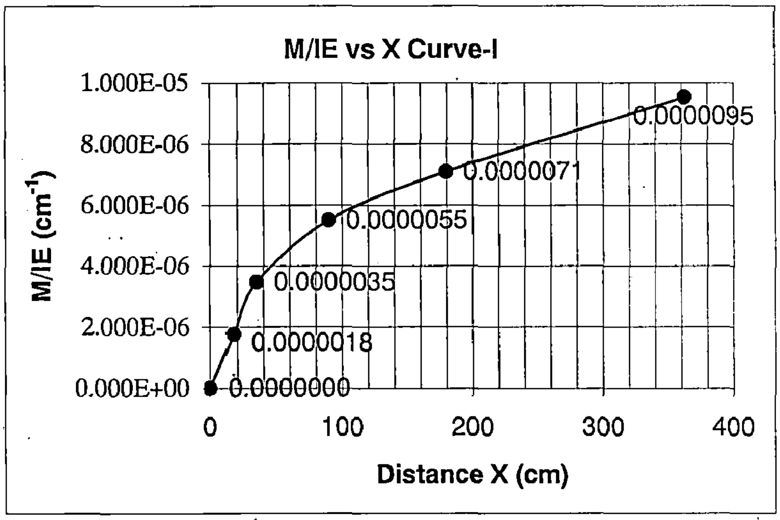
So, B.M at different pt at a distance x , $M = R_P.x - (W.(x-l)^2/2)$ kg-cm

Table: 8.3 E = 2.01E+06 kg-cm²

Point	Size of web		Size of flange		Dist from centre of wheel x (cm)	Bending moment M(kg-cm)	Moment of inertia I (cm4)	M/IE (cm-1)
	Height h (cm)	Thickness t_w (cm)	Top bxt (cm)	Bottom bxt (cm)				
A	57.50	1.40	55.0x2.0	25.0x2.5	0.00	0.0	168136.1	0.000E+00
B	57.50	1.40	55.0x2.0	25.0x2.5	17.50	643597.5	168136.1	1.904E-06
C	57.50	1.40	55.0x2.0	25.0x2.5	35.00	1270871.9	168136.1	3.760E-06
D	68.75	1.40	55.0x2.0	25.0x2.5	90.00	3029771.9	253039.0	5.957E-06
E	80.00	1.40	55.0x2.0	25.0x2.5	180.00	5212406.9	337942.0	7.674E-06
F	80.00	1.40	55.0x2.0	25.0x2.5	362.50	6987630.0	337942.0	1.029E-05

Now, by ploting the above values of X & M/IE, we get the following curve -I :

X (cm)	M/IE (cm⁻¹)
0.00	0.000E+00
17.50	1.904E-06
35.00	3.760E-06
90.00	5.957E-06
180.00	7.674E-06
362.50	1.029E-05



Graph: 8.1

One small area = 0.00004

Now, find the integration ie. area under the curve-I;

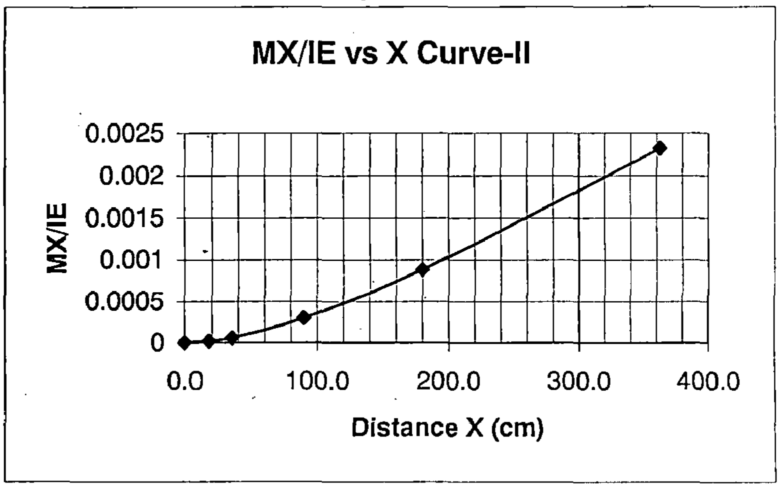
			X(cm)	(M/IE).X
up to 1st. Point no. of small area =	0		0.0	0
up to 2nd. Point no. of small area =	0.50	= area =	17.50	0.00002
up to 3rd. Point no. of small area =	1.5		35.00	0.00006
up to 4th. Point no. of small area =	7.5		90.00	0.0003
up to 5th Point no. of small area =	22		180.00	0.00088
up to 6th Point no. of small area =	58		362.50	0.00232

Now, by ploting the above values of X & MX/IE, we get the following curve -II :

One small area = 0.01 cm

Total no. of small area above the curve-II = 54

Deflection, d_{actual} = area = 0.54 cm



Graph: 8.2

9.0 Design for Vertical End Girder:

9.1 Load Diagram

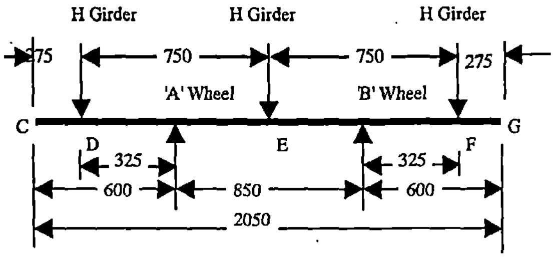


Figure: 9.1

Load from H Girders are,

P_D = 36777.0 kg

P_E = 42435.0 kg

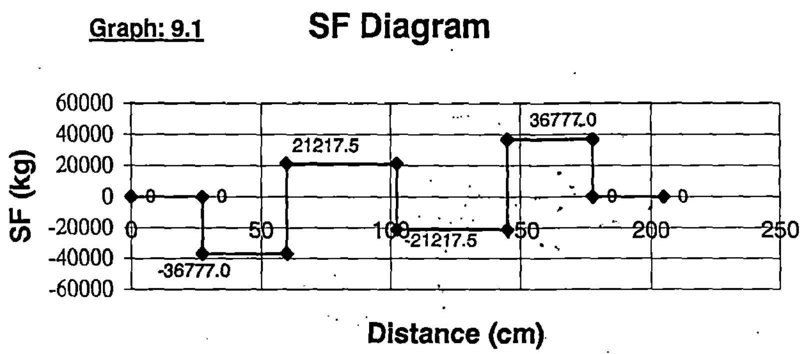
P_F = 36777.0 kg

Reaction forces at wheels are,

R_A = 57994.5 kg

R_B = 57994.5 kg

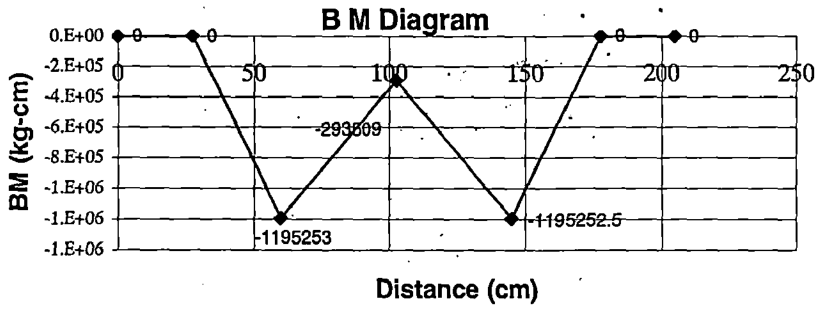
Graph: 9.1



Dist. (cm)	SF (kg)
0.0	0
27.5	0
27.5	-36777.0
60.0	-36777.0
60.0	21217.5
102.5	21217.5
102.5	-21217.5
145.0	-21217.5
145.0	36777.0
177.5	36777.0
177.5	0
205.0	0

Maximum Shear Force, V = 36777.0 kg.

Graph: 9.2



Dist (cm)	M(kg-cm)
0.0	0
27.5	0
60.0	-1195253
102.5	-293509
145.0	-1195253
177.5	0
205.0	0

Bending moment at point 'A' M_A = -1195253 kg-cm

Bending moment at point 'E' M_E = -293509 kg-cm

From the above diagram it is obvious that support moment will govern.

9.2 Check for Section when there is no Wheel Axle Hole:

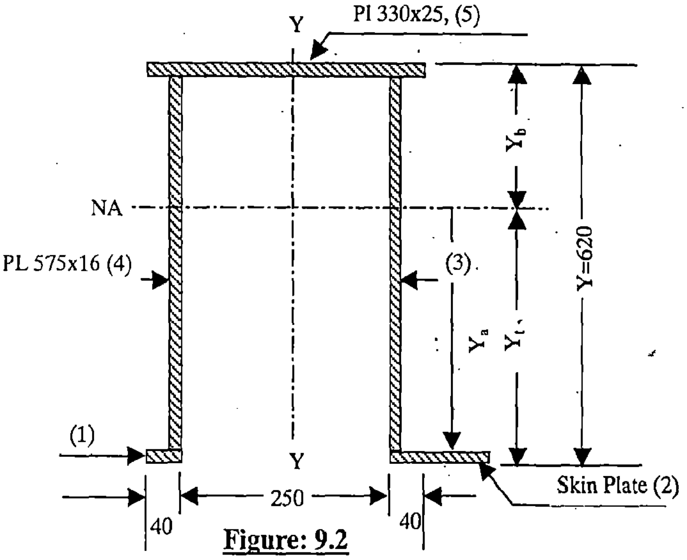


Figure: 9.2

9.3 Calculation of Neutral Axis and Modulus of Section of Vertical End Girder :

Table: 9.1

Item No.	Item	Size		A (cm^2)	Y (cm)	AxY	AY^2	I_{self} (cm^4)
		b (cm)	h (cm)	(b x h)	dist from X	(cm^3)	(cm^4)	$bh^3/12$
(1)	Skin pl. _	4.00	2.00	8.00	1.00	8.00	8.0	2.7
(2)	flange	4.00	2.00	8.00	1.00	8.00	8.0	2.7
(3)	Web _	1.60	57.50	92.00	30.75	2829.00	86991.8	25347.9
(4)	plate	1.60	57.50	92.00	30.75	2829.00	86991.8	25347.9
(5)	Flange pl.	33.00	2.50	82.50	60.75	5011.88	304471.4	43.0
Sum:			62.00	282.5		10685.9	478470.9	50744.1

So, Y_t = sum(AY)/sum(A) = 37.83 cm $Y = h_1 + h_3 + h_5 =$ 62.00 cm

$Y_b = (Y - Y_t) =$ 24.17 cm

$Y_a = (Y_t - h_1) =$ 35.83 cm

Then, $I_{NA} = \text{sum}(AY^2) + \text{sum}(I_{self}) - \text{sum}(A) \times Y_t^2 =$ 125010 cm^4

$Z_t = I_{NA} / Y_t =$ 3304.86 cm^3

$Z_b = I_{NA} / Y_b =$ 5171.28 cm^3

9.4 Check for Bending stresses of Vertical End Girder:

$f_t = M/Z_t =$ 361.67 kg/cm^2 (Comp.)   < 1080.00 kg/cm^2

$f_b = M/Z_b =$ 231.13 kg/cm^2 (Tension)   **Hence safe.**

9.5 Check for Shear stresses of Vertical End Girder:

Here, Max shear force, V = 36777.0 kg

We have, $$Shear - Stress = f_s = \frac{VA\overline{Y}}{Ib} =$$

For max shear stress (at NA)

Here, $A\overline{Y} = b_1 x h_1 x (h_1/2 + Y_a) x2 + (b_3 x Y_a^2/2) x2 =$ 2642.83 cm^3

I = 125010 cm^4 $b = (b_3 + b_4) =$ 3.20 cm

Now, putting values in the above equation, $f_{s(max)} =$ 242.97 kg/cm^2,< 840.00 kg/cm^2

9.6 Check for Section when there are Wheel Axle Holes:

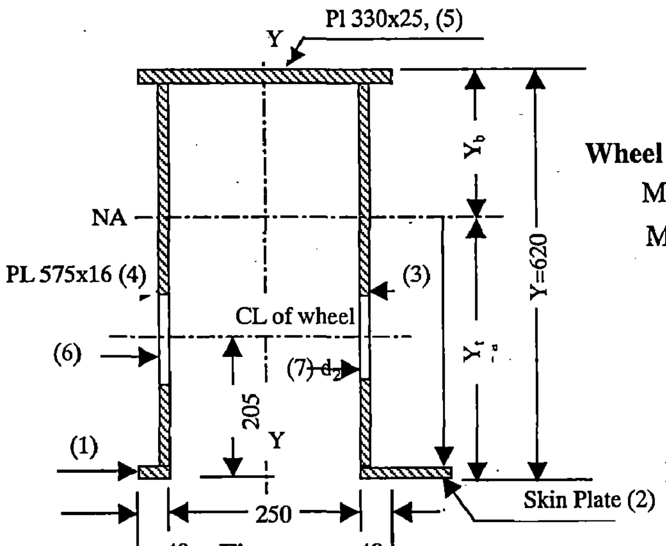


Figure:

Wheel Axel Dimension

Max. dia, d_1 = 120 mm

Min. dia, d_2 = 110 mm

9.7 Calculation of Neutral Axis and Modulus of Section of V.End Girder with Hole :

Table: 9.2

Item No.	Item	Size		A (cm^2)	Y (cm)	AxY	AY^2	I_{self} (cm^4)
		b (cm)	h (cm)	(b x h)	dist from X	(cm^3)	(cm^4)	$bh^3/12$
(1)	skin plate	4.00	2.00	8.00	1.00	8.00	8.0	2.7
(2)	flange	4.00	2.00	8.00	1.00	8.00	8.0	2.7
(3)	Web	1.60	57.50	92.00	30.75	2829.00	86991.8	25347.9
(4)	plate	1.60	57.50	92.00	30.75	2829.00	86991.8	25347.9
(5)	Flange pl.	33.00	2.50	82.50	60.75	5011.88	304471.4	43.0
(6)	Hole (d_1)	1.60	12.00	-19.20	20.50	-393.60	-8068.8	-230.4
(7)	Hole (d_2)	1.60	11.00	-17.60	20.50	-360.80	-7396.4	-177.5
Sum:			62.00	245.7		9931.5	463005.7	50336.3

So, Y_t = sum(AY)/sum(A) = 40.42 cm $Y = h_1 + h_3 + h_5$ = 62.00 cm

$Y_b = (Y - Y_t)$ = 21.58 cm   CL of hole, Y_h = 20.5 cm

$Y_a = (Y_t - h_1)$ = 38.42 cm

Then, $I_{NA} = \text{sum}(AY^2) + \text{sum}(I_{self}) - \text{sum}(A) \times Y_t^2$ = 111900 cm^4

$Z_t = I_{NA} / Y_t$ = 2768.36 cm^3

$Z_b = I_{NA} / Y_b$ = 5185.65 cm^3

9.8 Check for Bending stresses of Vertical End Girder with Hole:

$f_t = M/Z_t$ = 431.75 kg/cm^2 (Comp.)   < 1080.00 kg/cm^2

$f_b = M/Z_b$ = 230.49 kg/cm^2 (Tension)   **Hence safe.**

9.9 Check for Shear stresses of Vertical End Girder with Hole:

Here, Max shear force, V = 36777.0 kg

We have, $$Shear - Stress = f_s = \frac{VA\bar{Y}}{Ib} =$$

For max shear stress (at NA)

Here, $A\bar{Y} = b_1 x h_1 x (h_1/2 + Y_a) x 2 + (b_3 x Y_a^2/2) x 2$ = 2259.54 cm^3

I = 111900 cm^4 $b = (b_3 + b_4)$ = 3.20 cm

Now, putting values in the above equation, $f_{s(max)}$ = 232.07 kg/cm^2,< 840.00 kg/cm^2

Hence the adopted Vertical End Girder is OK.

10.0 Design of Wheels :

10.1 Selection of Material & Properties :

Cast steel to IS - 1030-1998, Grade C, Designation IV - 27.54,

Yield Strength, Y_p= 2700 kg/cm^2 E = 2.10E+06 kg/cm^2

U.T.S = 5400 kg/cm^2

10.2 Permissible Stresses :

Adopting flat wheel with line contact, (ref. IS 4622 : 1992, page-18, Annex-F)

Permissible Contact Stress at the surface of wheel, f_c= 1.4xUTS = 7560 kg/cm^2

Desiging the wheel with line contact, the contact stresses between the wheel and the track shall be calculated in accordance with the following formula:

$$f_C = 0.418\sqrt{\frac{PE}{rl}} \text{ ----------------- (V)}$$

Where,

f_C = contact stress in N/mm^2 or kg/cm^2 ;

P = wheel load in N or kg ;

E = modulus of elasticity N/mm^2 or kg/cm^2 ;

r = radious of wheel in mm or cm and

l = tread width of wheel in mm or cm.

Here,

f_c = 7560.0 kg/cm^2 r = 25.00 cm (assumed)

P = 57994.5 kg E = 2.1E+06 kg/cm^2

Rearranging equation (V) we have,

$$l = \frac{0.418^2 \times PE}{r \cdot f_c^2} = 14.8928 \text{ cm}$$

15.00 cm (say)

Hence, wheel size:

Dia, D_w = **50.00** cm , Tread width, l_w = **15.00** cm

11.0 Design of Wheel Axles (Shafts) :

Ref. IS 1570(part-v) 1998, Table-3, Page-6

11.1 Selection of Material & Properties :

Stainless Steel, Hardened & Tempered,

Steel designation - 12Cr12;

Tensile strength, UTS= 7000 kg/cm^2

Yield strength, Y_p = 4100 kg/cm^2 (0.2% proof stress)

% Elongation (bar : 5 - 100 mm) = 16

11.2 Permissible Stresses :

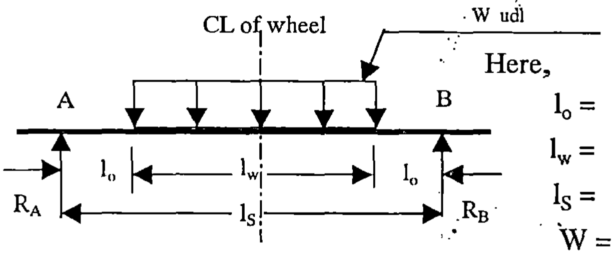
Bending stress, S_{pb} =UTS/3= 2333.33 kg/cm^2

Shear stress, S_{ps} =0.6xS_b = 1400.00 kg/cm^2

11.3 Calcalulation of Diameter of the Axle :

Wheel load, P_w = 57994.5 kg

This wheel load is distributed over the axle of length, l_w = 15.00 mm



Here,

l_o = 5.8 cm

l_w = 15 cm

l_S = 26.6 cm (C-C of web of VEG).

$W = P_w / l_x$ = 3866.3 kg/cm

Reaction forces are, R_A = 28997.3 kg R_B = 28997.3 kg

Maximum bending moment, $M = Ra \times l_s/2 - W \times l_w^2/8$ = 276924 kg-cm

Maximum shear force, $V = R_A$ = 28997.3 kg

Permissible Section Modulus, $Z_{per} = M_{max}/S_{pb}$ = 118.682 cm^3

For shaft, moment of inertia, $I = (pi)D^4/64$, and $Z = I/(D/2) = (pi)D^3/32 = Z_{per}$;

Rearranging the above equation we have,

$D = \{Z_{per} \times 32/(pi)\}^{1/3}$ = 10.6545 cm

= 12.00 cm (say)

12.0 Design / Selection of Wheel Bearing :

12.1 Friction Forces :

Ref. IS 4622 : 1992, page-5, clause -5.4.5.2;

Wheel friction forces for bush & roller bearing's are given by the formula:

$$F = \frac{P}{R}(f_a \times r + f_r) \text{ ---------------- (VI)}$$

Where,

F = total wheel friction in N or kg,

P = total hydro-static load in N or kg

R = wheel radious in mm or cm;

f_a = coefficient of axle friction (sliding);

f_r = coefficient of rolling friction in mm or cm and

r = effective radious of the bearing in mm or cm.

Here, P = 57994.5 kg

R = 25.00 cm

f_a = 0.20 (at starting with bronze bush)

f_r = 0.10 cm (average)

For bronze bushing :

Thickness of bushing is given by:

t = (0.08D+0.3)

Where, t = thickness of bush in cm; Here, D = 12.00 cm

D = diameter of the axle in cm;

So, t = 1.26 cm And therefore, r = 7.26 cm

Now, putting values in equation (IV), we have, F= 3600.3 kg

12.2 Tentative Weight of the Gate : **Table: 12.1**

Item No	Item		Size (cm)			Volume	Unit wt	Weight
	Description & (No.)		length	breath	thickness	(cm^3)	(kg/cm^3)	(kg)
1	Skin plate (1)		765.00	205.00	2.00	313650.0	0.00785	2462.15
2	Vertical stiffeners (8)		205.00	19.00	1.00	31160.0	0.00785	244.61
3	Horizontal girders	web (3)	732.00	80.00	1.47	257664.0	0.00785	2022.66
		flange (3)	732.00	26.67	2.50	146400.0	0.00785	1149.24
4	V End girders	web (4)	205.00	57.50	1.60	75440.0	0.00785	592.20
		flange (2)	205.00	33.00	2.50	33825.0	0.00785	265.53
5	Wheels (4)		dia. =	50.00	15.00	117750.0	0.00785	924.34
6	Axles (4)		dia. =	12.00	33.00	14921.3	0.00785	117.13
	Sum:					990810.28		7777.86

Therefore, weight of the gate including 10% welding wt. = 8555.65 kg

Now, if bush bearings are used then

total friction forces in 4 bearings, Ft = Fx4 = 14401.2 kg

which is greater than the weight of the gate. Hence, bush bearings can not be used.

12.3 Selection of Spherical Roller Bearing :

Total wheel load, P = 57994.5 kg

Ref. SKF General Catalogue - 3200E, Reg. 47 12 000. 1981-09;

Adopt 2 nos. of SKF Spherical Roller bearings of bearing No: 24024 CC, for each wheel.

Detail of each bearing:

Weight = 5.4 kg

Bore dia, d = 120 mm

Outer dia, D = 180 mm

Bearing width, B = 60 mm

Inner race outer dia, d_1 = 132 mm

Outer race inner dia, D_1 = 160 mm

Static load capacity, C_o = 360000 N, or 36000 kg

Dynamic load capacity, C = 374000 N, or 37400 kg

Therefore, static load capacity of two bearing, F_{co}= 72000 kg

Now, effective radious of the bearing, $r = (d_1 + D_1)/4$ = 7.30 cm

f_a = 0.02 (roller bearing)

f_r = 0.10 cm (average)

Now, putting the corresponding values in equation (IV), we have, F= 485.994 kg

So, for 4 wheels, total friction forces, F_{tw} = Fx4 = 1943.98 kg , < 8555.65 kg (gate)

Hence, OK.

12.4 Check for Bending Stresses at the Support of Axle :

The axle diameter varies from 14.5 cm at one support to 11 cm at the other support with 12 cm at the centre under the wheel bearings.

Maximum bearing stress at the support is given by, $S_{br} = R_B/(d_2.t)$ =

Here, R_B = 28997.3 kg

d_2 = 11.00 cm

t = 1.60 cm

So, $S_{br} = P/(d.t)$ = 1647.57 kg/cm^2, > 1560.00 kg/cm^2 & unsafe.

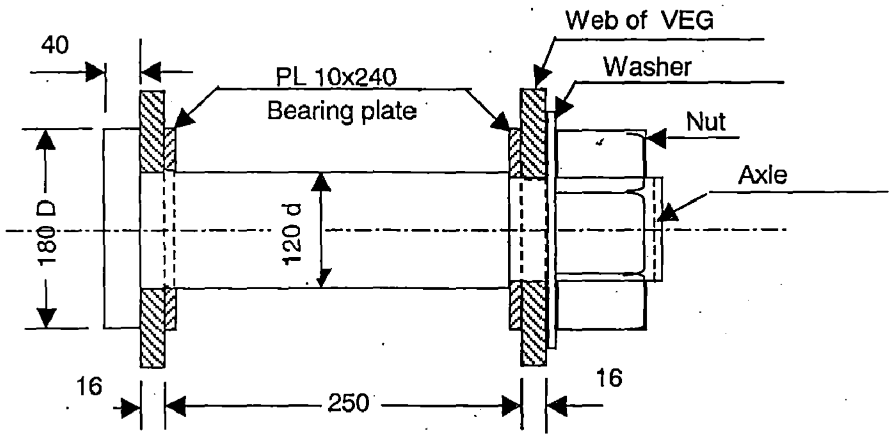
Providing 10 mm thick & 240 mm L bearing plate on both sides of the wheel inside the webs of vertical end girder as shown in figure below.

Then, t = 2.60 cm

Therefore, $S_{br} = P/(d.t)$ = 1013.89 kg/cm^2, < 1560.00 kg/cm^2

Hence, safe.

12.5 Final Arrangement of the Axle :



12.6 Check for Hardness of Wheel :

a) Wheel load,

Normal, P_n = 57994.5 kg

Maximum, $P_{max} = P_n \times 1.5$ = 86991.8 kg (when one wheel not in contact with the track)

b) Wheel Dia, D = 50.00 cm

c) Tread width, B = 15.00 cm

d) Wheel material:

Steel with UTS = 5400 kg/cm^2

e) Hardness of the material is given by the relation,

UTS = 35 BHN kg/cm^2

Therefore, BHN= 154.286

Ref. IS 4622 : 1992, Annex-E, page-15;

Critical stress of projected area, Sc =0.169BHN - 15.174 N/mm^2

which is equivalent to, S_c = 1.69BHN - 151.74 kg/cm^2 (taking, g =10;)

Then projected area of the wheel, A_W =DxB= 750.00 cm^2

Hence, stress for maximum load, $S_{max} = P_{max}/A_w$ = 115.989 kg/cm^2

Taking factor of safty, sf = 2

Required critical stress for design, $S_{cd} = S_{max} \times sf$ = 231.978 kg/cm^2

Therefore, required BHN = $(S_{cd} + 151.74)/1.69$= 227.05

= 230.00 (say)

After Check Revise Calculation :

10.3 Selection of Material & Properties :

Cast steel to IS - 2644 - 1979, Table -1, Grade -2, CS700,

Yield Strength, Y_p= 5600 kg/cm^2

U.T.S = 7000 kg/cm^2 BHN= 207

E = 2.10E+06 kg/cm^2

10.4 Permissible Stresses :

Adopting flat wheel with line contact, (ref. IS 4622 : 1992, page-18, Annex-F)

Permissible Contact Stress at the surface of wheel,

f_{pc}= 1.4xUTS = 9800 kg/cm^2

Also as per ref. IS 4622 : 1992, Annex-E, page-15; we have,

Critical stress of projected area, Sc =0.169BHN - 15.174 N/mm^2

which is equivalent to, S_c = 1.69BHN - 151.74 kg/cm^2 (taking, g =10;)

= 198.08 kg/cm^2

Let, safty factor, sf = 2

So, allowable critical stress, $S_{ac}=S_c/sf$ = 99.04 kg/cm^2

10.5 Design of Wheel :

Projected area of the wheel = wheel load/ critical stress

ie. $A_{pa} = P/S_{ac}$ = 585.566 cm^2

Now, for wheel diameter, D_w = 50.00 cm

Net tread width, $l_{nw} = A_{pa}/D_w$ = 11.71 cm

= 12.00 cm (say)

Taking corner rounding on each side, f_o = 5.00 mm

So, wheel tread becomes, $l_w = l_{nw} + 2f_o$ = 13.00 cm

10.6 Check for Line Contact Condition :

We have, line contact stress is given by - $f_c = 0.418\sqrt{\frac{PE}{rl}}$ ------------------ (V)

Here, P = 57994.5 kg E = 2.1E+06 kg/cm^2

$r = D_w/2$ = 25.0 cm $l = l_w$ = 13.00 cm

Now, putting values in equation (V), we have,

f_c = 8091.66 kg/cm^2 < 9800 kg/cm^2 **Hence, safe.**

Therefore, the above wheel size is safe; but same size as per earlier calculation is being adopted.

ie. D_w = **50.00** cm l_w = **15.00** cm

10.7 Check for Shear Stress :

With reference to Theory of Elasticity by Timmoshanko & IS 4622 - 1992, for line contact design condition:

We have, Maximum shear stress, $f_{sm} = 0.304f_c$;

And allowable shear stress (least of),

S_{sa} = 0.7 UTS or 24BHN = 4900 or 4968 kg/cm^2

Now, putting the respective values in equation (V), we have,

f_c = 7532.93 kg/cm^2

So, f_{sm} = 2290.01 kg/cm^2 < 4900 kg/cm^2 (max.)

Both the allowable shear stresses are much greater than the actual calculated value and hence safe.

10.8 Check for Depth to the point of max. Shear Stress :

We have, $b = 1.55\ sqrt(P.r/l_w.E)$

and depth, Z = 0.786 b;

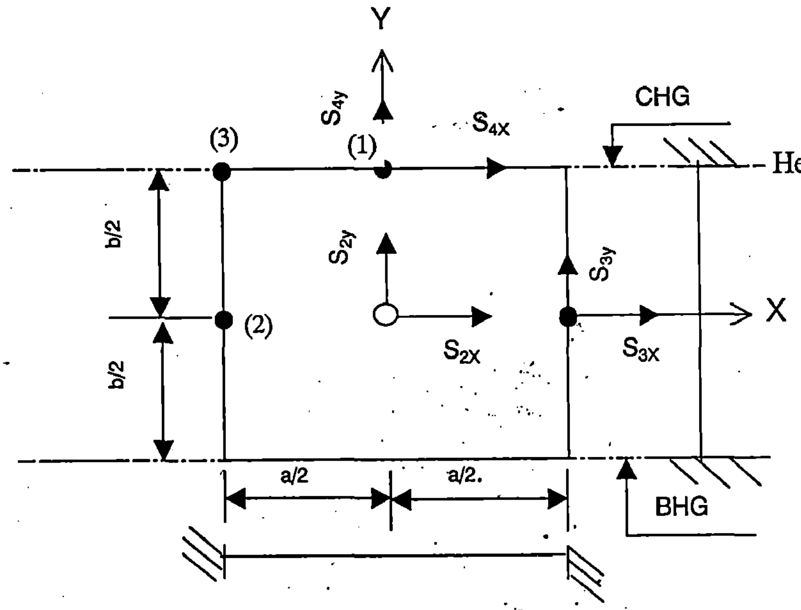
So, b = 3.3E-01 cm

and depth, Z = 0.26137 cm

Therefore, depth of hardening >= 2Z = 0.52275 cm

13.0 Check for Combined Stresses :

13.1 Check for combination of stresses in skin plate with central horizontal girder & vertical stiffener at the centre of the central horizontal girder :



Here,
a = 750 mm
b = 750 mm

Fig. 13.1 All edges rigidly fixed

(I) Stresses in CHG (ref. art.7.4) :

Stress in skin plate, $f_{hs} = f_{tg}$ = 637.54 kg/cm^2

Stress in flange (bottom), $f_{hf} = f_{bg}$ = 978.20 kg/cm^2

Stress in web at point A, f_a = 599.30 kg/cm^2

(II) Stresses in skin plate (ref. art. 5.1) :

+/- S_{3X} = 712.631 kg/cm^2 +/- S_{4X} = 213.789 kg/cm^2

+/- S_{3Y} = 213.789 kg/cm^2 +/- S_{4Y} = 712.631 kg/cm^2

Shear Stress in Skin Plate :

Design pressure, P = 1.64 kg/cm^2

C/C spacing of vertical stiffeners, Ls = 75.00 cm

Thickness of skin plate, t = 2.00 cm

Shear stress in skin plate is given by, $S_s = PxL_s/(t.t)$;

So, Shear stress, S_s = 30.75 kg/cm^2

(III) Stresses in vertical stiffener (ref. Art. 6.3 & 6.4) :

$S_{s.s} = f_t$ = 369.19 kg/cm^2

$S_{s.w} = f_a$ = 257.65 kg/cm^2

$S_{s.f} = f_b$ = -802.01 kg/cm^2 (comprèsion)

Shear stress, $S_{s(NA)}$ = 307.55 kg/cm^2

Shear stress, $S_{s(A)}$ = 275.81 kg/cm^2

13.2 Check for combined stresses at point (3) :

Combined stress is given by the equation,

$$S_c = \sqrt{S_x^2 + S_y^2 - S_x \times S_y + 3S_s^2} \text{ ------------ (VII)}$$

Total stresses in X direction, $S_x = f_{hs}$;

Total stresses in Y direction, $S_y = +/- S_{s.s}$;

Now, for: $S_x =$ 637.54 kg/cm^2 ; $S_y =$ 369.19 kg/cm^2

And, $S_s =$ 307.55 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 768.857 kg/cm^2 < 1560.00 kg/cm^2

Now, for: $S_x =$ 637.54 kg/cm^2 ; $S_y =$ -369.19 kg/cm^2 (comp.)

And, $S_s =$ 307.55 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 1030.48 kg/cm^2 < 1560.00 kg/cm^2

13.3 Check for combined stresses at point (2) :

Total stresses in X direction, $S_x = f_{hs} + (+/- S_{3x}) =$ 1350.17 kg/cm^2 or,

= -75.09 kg/cm^2 (comp.)

Total stresses in Y direction, $S_y = +/- (S_{s.s} + S_{3y}) =$ 582.98 kg/cm^2

Now, for: $S_x =$ 1350.17 kg/cm^2 ; $S_y =$ 582.98 kg/cm^2

And, $S_s =$ 30.75 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 1174.11 kg/cm^2 < 1560.00 kg/cm^2

Now, for: $S_x =$ 1350.17 kg/cm^2 ; $S_y =$ -582.98 kg/cm^2 (comp.)

And, $S_s =$ 30.75 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 1718.36 kg/cm^2 > 1560.00 kg/cm^2

13.4 Check for combined stresses at point (1) :

Total stresses in X direction, $S_x = f_{hs} + (+/- S_{4x}) =$ 851.33 kg/cm^2 or,

= 423.75 kg/cm^2

Total stresses in Y direction, $S_y = +/- S_{4y} =$ 712.63 kg/cm^2

Now, for: $S_x =$ 851.33 kg/cm^2 ; $S_y =$ 712.63 kg/cm^2

And, $S_s =$ 30.75 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 792.941 kg/cm^2 < 1560.00 kg/cm^2

Now, for: $S_x =$ 851.33 kg/cm^2 ; $S_y =$ -712.63 kg/cm^2 (comp.)

And, $S_s =$ 30.75 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 1357.25 kg/cm^2 < 1560.00 kg/cm^2

13.5 Checking for Shear Stress :

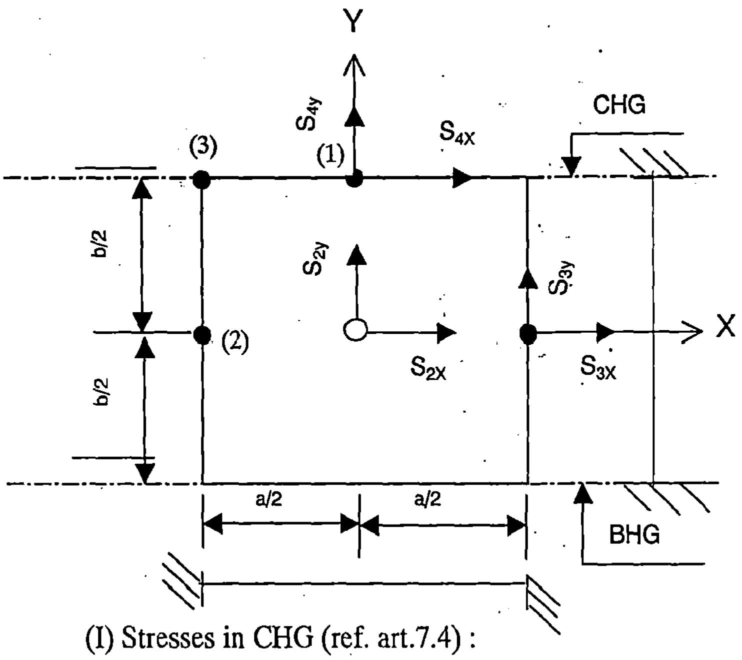
Shear stress at the centre of the BHG will be zero because shear force at the centre is zero. Therefore, shear stress at the top of the skin plate is zero and at the joint of skin plate with web of the BHG is also zero.

Remarks:

As it is clear from the above calculations that the arrangement adopted is coming out to be unsafe for the present conditions. Hence, to reduce the stresses in skin plate the placement of vertical stiffeners is being proposed closer than adopted already i.e. to be fixed at a distance 550 mm in place of distance 750 mm C to C.

After Redesign Revise Check

13.6 Check for combination of stresses in skin plate with central horizontal girder & vertical stiffener at the centre of the central horizontal girder :



Here,

a = 550 mm

b = 750 mm

(I) Stresses in CHG (ref. art.7.4) :

Stress in skin plate, $f_{hs} = f_{tg} =$ 637.54 kg/cm^2

Stress in flange (bottom), $f_{hf} = f_{bg} =$ 978.20 kg/cm^2

Stress in web at point A, $f_a =$ 599.30 kg/cm^2

(II) Stresses in skin plate (ref. art. 5.6) :

$\pm S_{3X} =$ 529.136 kg/cm^2 $\pm S_{4X} =$ 126.81 kg/cm^2

$\pm S_{3Y} =$ 158.741 kg/cm^2 $\pm S_{4Y} =$ 422.7 kg/cm^2

Shear Stress in Skin Plate :

Design pressure, P = 1.64 kg/cm^2

C/C spacing of vertical stiffeners, Ls = 55 cm

Thickness of skin plate, t = 2 cm

Shear stress in skin plate is given by, $S_s = PxL_s/(t.t)$;

So, Shear stress, $S_s =$ 22.55 kg/cm^2

(III) Stresses in vertical stiffener (ref. Art. 6.8 & 6.9) :

$S_{s.s} = f_t =$ 270.74 kg/cm^2

$S_{s.f} = f_b =$ -588.14 kg/cm^2 (compresion)

$S_{s.w} = f_a =$ 188.94 kg/cm^2

Shear stress, $S_{s(NA)} =$ 225.54 kg/cm^2

Shear stress, $S_{s(A)} =$ 202.26 kg/cm^2

13.7 Check for combined stresses at point (3) :

Combined stress is given by the equation,

$$S_c = \sqrt{S_x^2 + S_y^2 - S_x \times S_y + 3S_s^2} \text{ ------------ (VII)}$$

Total stresses in X direction, $S_x = f_{hs}$;

Total stresses in Y direction, $S_y = +/- S_{s.s}$;

Now, for: $S_x =$ 637.54 kg/cm^2 ; $S_y =$ 270.74 kg/cm^2

And, $S_s =$ 225.54 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 678.046 kg/cm^2 < 1560.00 kg/cm^2

Now, for: $S_x =$ 637.54 kg/cm^2 ; $S_y =$ -270.74 kg/cm^2 (comp.)

And, $S_s =$ 225.54 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 897.195 kg/cm^2 < 1560.00 kg/cm^2

13.8 Check for combined stresses at point (2) :

Total stresses in X direction, $S_x = f_{hs} + (+/-S_{3x}) =$ 1166.67 kg/cm^2

= 108.40 kg/cm^2

Total stresses in Y direction, $S_y = +/- (S_{s.s} + S_{3y}) =$ 429.48 kg/cm^2

Now, for: $S_x =$ 1166.67 kg/cm^2 ; $S_y =$ 429.48 kg/cm^2

And, $S_s =$ 22.55 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 1022.76 kg/cm^2 < 1560.00 kg/cm^2

Now, for: $S_x =$ 1166.67 kg/cm^2 ; $S_y =$ -429.48 kg/cm^2 (comp.)

And, $S_s =$ 22.55 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 1431.14 kg/cm^2 < 1560.00 kg/cm^2

13.9 Check for combined stresses at point (1) :

Total stresses in X direction, $S_x = f_{hs} + (+/-S_{4x}) =$ 764.35 kg/cm^2

= 510.73 kg/cm^2

Total stresses in Y direction, $S_y = +/- S_{4y} =$ 422.70 kg/cm^2

Now, for:	S_x =	764.35 kg/cm^2 ;		S_y =	422.70 kg/cm^2	
			And,	S_s =	22.55 kg/cm^2	

Putting these values in equation (V), we have,

S_c = 664.333 kg/cm^2 < 1560.00 kg/cm^2

Now, for:	S_x =	764.35 kg/cm^2 ;		S_y =	-422.70 kg/cm^2	(comp.)
			And,	S_s =	22.55 kg/cm^2	

Putting these values in equation (V), we have,

S_c = 1042.84 kg/cm^2 < 1560.00 kg/cm^2

13.10 Check for combination of stresses in skin plate with bottom horizontal girder & vertical stiffener at the centre of the bottom horizontal girder :

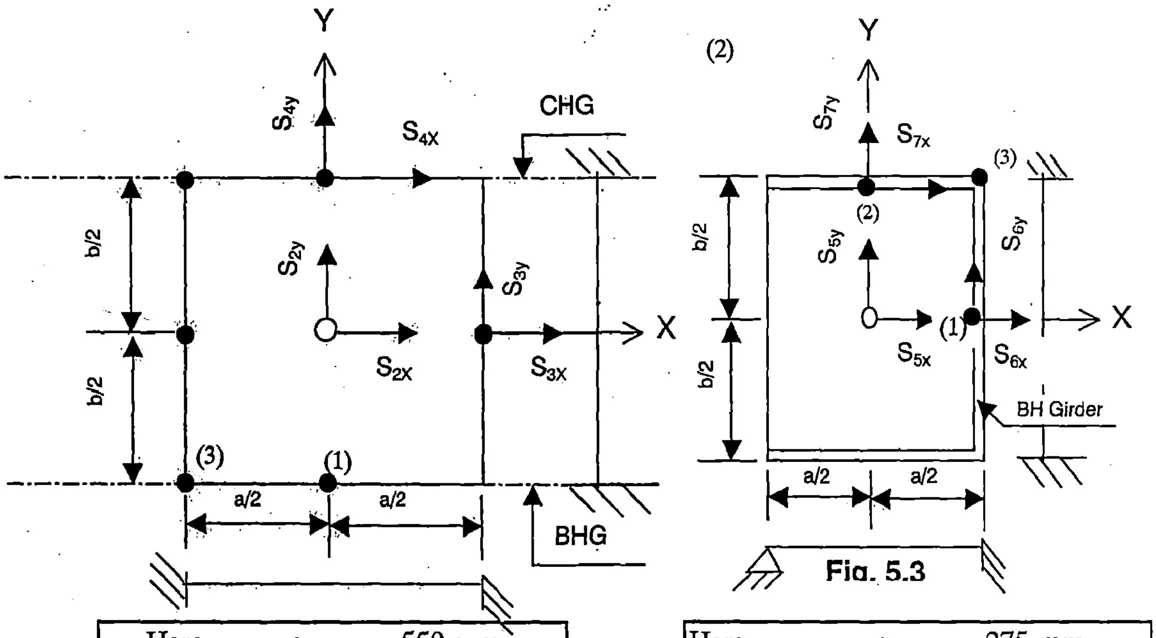


Fig. 5.3

Here,	a =	550 mm
	b =	750 mm

Here,	a =	275 mm
	b =	550 mm

(I) Stresses in BHG (ref. art.8.3) :

Stress in skin plate, $f_{hs} = f_{tg}$ = 728.03 kg/cm^2

Stress in flange (bottom), $f_{hf} = f_{bg}$ = 1019.18 kg/cm^2

Stress in web at point A, f_a = 686.67 kg/cm^2

(II) Stresses in skin plate (ref. art. 5.6 & 5.7) :

+/- S_{3X} =	529.14 kg/cm^2	+/- S_{6X} =	211.77 kg/cm^2
+/- S_{3Y} =	158.74 kg/cm^2	+/- S_{6Y} =	63.53 kg/cm^2
+/- S_{4X} =	126.81 kg/cm^2	+/- S_{7X} =	43.72 kg/cm^2
+/- S_{4Y} =	422.7 kg/cm^2	+/- S_{7Y} =	145.73 kg/cm^2

Shear stress, S_s = 22.55 kg/cm^2

(III) Stresses in vertical stiffener (ref. Art. 6.8 & 6.9) :

$S_{s.s} = f_t =$ 270.74 kg/cm^2

$S_{s.f} = f_b =$ 588.14 kg/cm^2 (compresion)

$S_{s.w} = f_a =$ 188.94 kg/cm^2

Shear stress, $S_{s(NA)} =$ 225.54 kg/cm^2

Shear stress, $S_{s(A)} =$ 202.26 kg/cm^2

13.11 Check for combined stresses at point (3) :

Combined stress is given by the equation,

$$S_c = \sqrt{S_x^2 + S_y^2 - S_x \times S_y + 3S_s^2} \text{ ------------- (VII)}$$

Total stresses in X direction, $S_x = f_{hs}$;

Total stresses in Y direction, $S_y = +/- S_{s.s}$;

Now, for: $S_x =$ 728.03 kg/cm^2 ; $S_y =$ 270.74 kg/cm^2

And, $S_{s(NA)} =$ 225.54 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 747.542 kg/cm^2 < 1560.00 kg/cm^2

Now, for: $S_x =$ 728.03 kg/cm^2 ; $S_y =$ -270.74 kg/cm^2 (comp.)

And, $S_s =$ 225.54 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 976.233 kg/cm^2 < 1560.00 kg/cm^2

13.12 Check for combined stresses at point (2) :

Total stresses in X direction, $S_x = f_{hs} + (+/-S_{7y}) =$ 873.76 kg/cm^2

= 582.30 kg/cm^2

Total stresses in Y direction, $S_y = +/- (S_{s.s} + S_{7X}) =$ 314.46 kg/cm^2

Now, for: $S_x =$ 873.76 kg/cm^2 ; $S_y =$ 314.46 kg/cm^2

And, $S_s =$ 22.55 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 767.53 kg/cm^2 < 1560.00 kg/cm^2

Now, for: $S_x =$ 873.76 kg/cm^2 ; $S_y =$ -314.46 kg/cm^2 (comp.)

And, $S_s =$ 22.55 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 1067.06 kg/cm^2 < 1560.00 kg/cm^2

13.13 Check for combined stresses at point (1) :

Total stresses in X direction, $S_x = f_{hs} + (+/-S_{4x}) =$ 854.84 kg/cm^2

= 601.22 kg/cm^2

Total stresses in Y direction, $S_y = +/- S_{4y} =$ 422.70 kg/cm^2

Now, for: $S_x =$ 854.84 kg/cm^2 ; $S_y =$ 422.70 kg/cm^2

And, $S_s =$ 22.55 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 741.357 kg/cm^2 < 1560.00 kg/cm^2

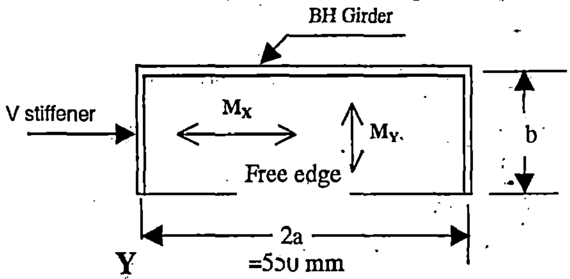
Now, for: $S_x =$ 854.84 kg/cm^2 ; $S_y =$ -422.70 kg/cm^2 (comp.)

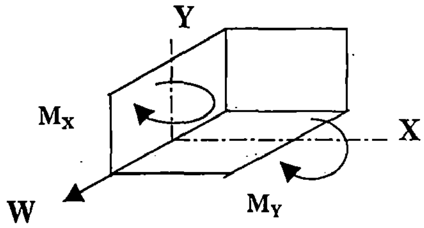
And, $S_s =$ 22.55 kg/cm^2

Putting these values in equation (V), we have,

$S_c =$ 1127.96 kg/cm^2 < 1560.00 kg/cm^2

13.14 Check for combined stresses at point (1) below Bottom Horizontal Girder :





Sign convention

Here, $a =$ 27.5 cm

$b =$ 27.5 cm

With reference to fig-1 of EMNO 27 of USBR, we have,

Moment, $M = (\text{coefficient} = C_m) x (P.b^2)$

and Reaction, $R = (\text{coefficient} = C_r) x (P.b)$;

For plate fixed along three edges as shown in above fig, moment & reaction coefficients for load-I, uniform load.

Here, Design water load per unit width, $P =$ 1.64 kg/cm

and, $a/b =$ 1.0

Maximum coefficient for moment in X direction ie. at the end, $C_{mx} =$ 0.2613

Maximum coefficient for moment in Y direction ie. at the centre of BHG, $C_{my} =$ 0.2043

Maximum coefficient for reaction in X direction , $C_{rx} =$ 1.2115

Maximum coefficient for moment in Y direction , $C_{ry} =$ 0.6725

Now, putting the respective values in the above equation, we have,

Maximum bending moment in X direction, $M_x =$ 324.077 kg-cm

Maximum bending moment in Y direction, $M_y =$ 253.383 kg-cm

Maximum shear force in X direction, $R_x =$ 54.6387 kg

Maximum shear force in Y direction, $R_y =$ 30.3298 kg

Bending stress in cantilever portion of the bottom panel, $S_{can} = M/Z$;

Where, $Z = bh^2/6$; Here, for unit load width, b = 1 cm

Thickness, h = 2 cm

So, Z = 0.6667 cm^3

Therefore, bending stress in Y direction, S_{can} = 380.07 kg/cm^2

$S_{x(can)} = 0.3S_{can}$ = 114.02 kg/cm^2

Hence, total stresses in X direction,

$S_x = f_{hs}$ +/- (max. of S_{6Y} or $S_{x(can)}$)= 842.05 kg/cm^2 < 1080.00 kg/cm^2

= 614.01 kg/cm^2 < 1080.00 kg/cm^2

Total stresses in Y direction, S_y = +/- (max. of S_{6X} or S_{can})= 380.07 kg/cm^2

Now, for: S_x = 842.05 kg/cm^2 ; S_y = 380.07 kg/cm^2

And shear stress, S_s = 22.55 kg/cm^2

Putting these values in equation (V), we have,

S_c = 731.43 kg/cm^2 < 1560.00 kg/cm^2

Now, for: S_x = 842.05 kg/cm^2 ; S_y = -380.07 kg/cm^2 (comp.)

And, S_s = 22.55 kg/cm^2

Putting these values in equation (V), we have,

S_c = 1084.01 kg/cm^2 < 1560.00 kg/cm^2

Hence, it clear from the above calculations at points 1, 2 & 3 of the middle panel and bottom panel of the gate that the sections are safe in combined stresses also.

14.0 Design for Embedded Parts :

14.1 Design of Track :

14.1.1 Selection of Material:

Track :

Stainless Steel confirming to IS 1570-1972 (part-v), Table-3 :
Steel designation = 15 Cr 16 Ni -2;
Surface of track plate to be hardened to BHN = 280
Depth of hardness from top, d_h = 3.00 mm

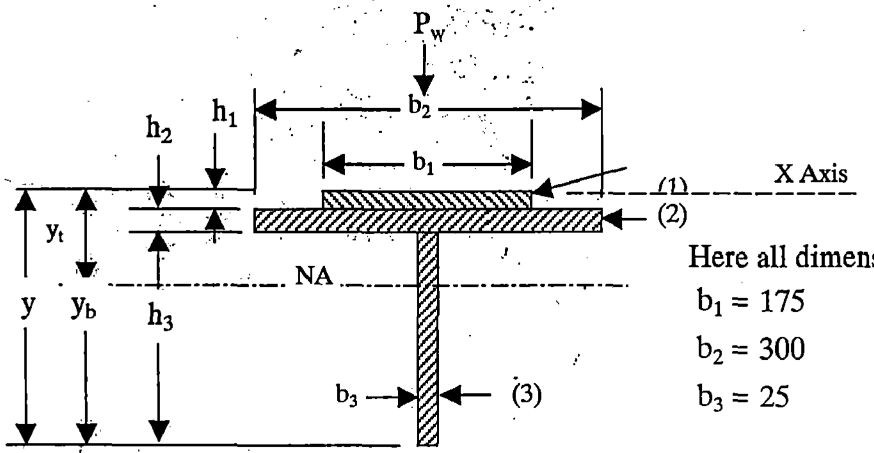
UTS = 8300 to 10300 kg/cm^2
Maximum allowable shear stress, S_{sa} = 0.7 UTS = 5810 kg/cm^2 (min.)
or, 24.1BHN = 6748 kg/cm^2

Track Base :

Structural Steel confirming to IS 2062 - 1980;

Note : The Track Plate to be welded to the top flange of the Track Base.

14.1.2 Selection of Section :



Here all dimensions are in mm

b_1 = 175 h_1 = 20
b_2 = 300 h_2 = 36
b_3 = 25 h_3 = 300

Figure : 14.1

14.1.3 Calculation of Neutral Axis and Modulus of Section :

Item No.	Item	Size		A (cm^2)	Y (cm)	AxY	AY^2	I_{self} (cm^4)
		b (cm)	h (cm)	(b x h)	dist from X	(cm^3)	(cm^4)	bh^3/12
(1)	Track	17.50	2.00	35.00	1.00	35.00	35.00	11.7
(2)	Track base	30.00	3.60	108.00	3.80	410.40	1560	116.6
(3)	Web	2.50	30.00	75.00	20.60	1545.00	31827	5625.0
Sum:			35.60	218.00		1990.4	33422	5753.3

So, Y_t = sum(AY)/sum(A) = 9.13 cm $Y = h_1 + h_2 + h_3$ = 35.60 cm

$Y_b = (Y - Y_t)$ = 26.47 cm

Then, I_{NA} = sum(AY^2) + sum(I_{self}) - sum(A)xY_t^2 = 21002 cm^4

$Z_t = I_{NA} / Y_t$ = 2300.3 cm^3

$Z_b = I_{NA} / Y_b$ = 793.43 cm^3

14.1.4 **Check for Line Contact Condition :**

We have, from equation (V) line contact stress is given by -

$$f_C = 0.418\sqrt{\frac{PE}{rl}} \text{ ----------------- (V)}$$

Here,

Maximum design wheel load, $P_w = P =$ 57994.5 kg E = 2.1E+06 kg/cm^2

$r = D_w/2 =$ 25.0 cm $l = l_w =$ 15.00 cm

Now, putting values in equation (V), we have,

$f_c =$ 7532.93 kg/cm^2 < 0 kg/cm^2 **Hence, safe.**

14.1.5 **Determination of Area of contact :**

Ref: IS 4622 : 1992 (page-7, art. 5.7.5.2)

We have,

$$L = \frac{3}{2} \times \frac{P}{w \times p_c} \text{ ———————————— (VIII)}$$

Where,

L = Length of influence under track base in mm or cm

P = total wheel load in N or kg = 57994.5 kg

w = width of track in contact with concrete in mm or cm

p_c = stress in concrete in N/mm^2 or kg/cm^2 = 7532.93 kg/cm^2

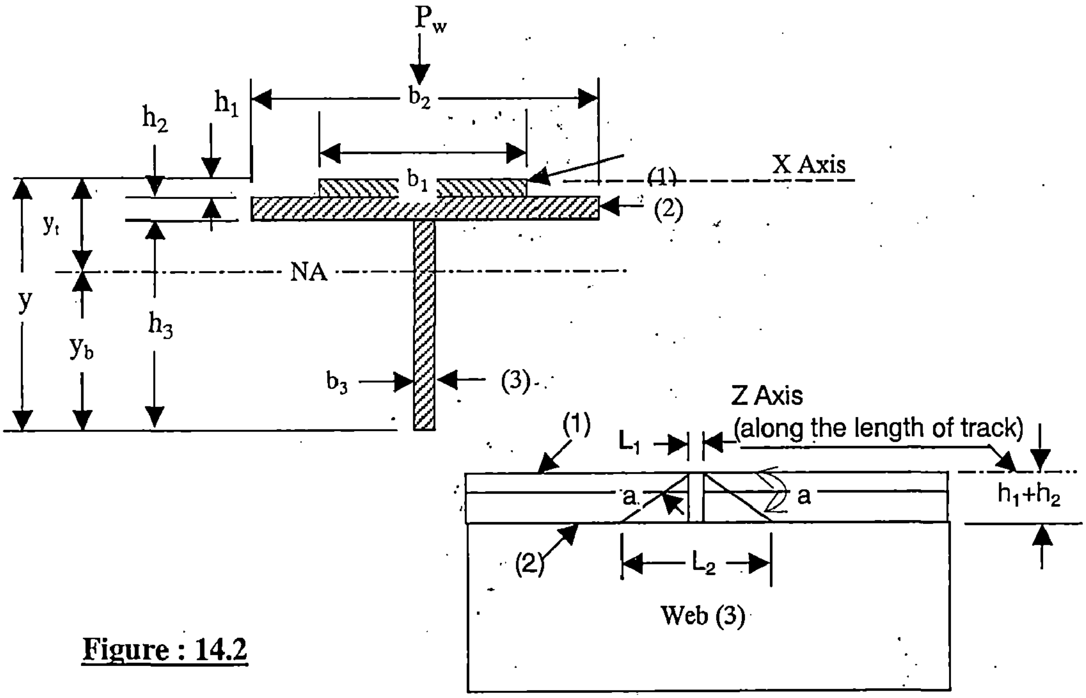


Figure : 14.2

Now, rearranging equation (VI) we have,

Area of contact, say, $A = L.w = 3P/2p_c =$ 11.5482 cm^2

Let, $w = l_w =$ 15.00 cm

So, $L_1 = A/w =$ 0.76988 cm (length on top of the track)

Angle of influence,say, a = 30 degree (within steel)

Length of influence at the bottom of the track base flange,

say, $L_2 = 2(h_1+h_2).Cot(a) + L_1 =$ 20.1688 cm

14.1.6 **Bearing Stress on Web,**

$S_b = P/(L_2 .b_3) =$ 1150.18 kg/cm^2 < 1320.00 kg/cm^2

(And also as per ref. IS 800-1984, clause-6.3),

Permissible bearing stress, $S_p = 0.75Y_p =$ 1800 kg/cm^2

Then bearing stress in Concrete,

Ref: IS 4622 : 1992 (page-7, art. 5.7.5)

Bearing stresses in concrete shall be found from the following formula,

$$p = 0.2813 \times P\left(\frac{E_c}{E_s \times I \times w^2}\right)^{\frac{1}{3}} \text{------------------ (IX)}$$

Where,

p = bearing stress in concrete in N/mm^2 or kg/cm^2 ;

P = total wheel load in N or kg ;

E_c = modulus of elasticity of concrete in N/mm^2 or kg/cm^2 ;

E_s = modulus of elasticity of steel in N/mm^2 or kg/cm^2 ;

I = moment of inertia of the track base in mm^4 or cm^4 and

w = width of the track base in contact with concrete mm or cm ;

Here, P = 57994.5 kg E_s = 2.1E+06 kg/cm^2

E_c = 1.4E+05 kg/cm^2 w = 30.00 cm

Moment of inertia of Track base :

Item No.	Item	Size		A (cm^2)	Y (cm)	AxY	AY^2	I_{self} (cm^4)
		b (cm)	h (cm)	(b x h)	dist from X	(cm^3)	(cm^4)	$bh^3/12$
(2)	Flange	30.00	3.60	108.00	1.80	194.40	349.92	116.6
(3)	Web	2.50	30.00	75.00	18.60	1395.00	25947	5625.0
Sum:			33.60	183.00		1589.4	26297	5741.6

So, Y_t = sum(AY)/sum(A) = 8.69 cm $Y = h_2 + h_3 =$ 33.60 cm

$Y_b = (Y - Y_t) =$ 24.91 cm

Then, $I_{NA} = \text{sum}(AY^2) + \text{sum}(I_{self}) - \text{sum}(A) x Y_t^2 =$ 18234 cm^4

$Z_t = I_{NA}/Y_t =$ 2099.4 cm^3 $Z_b = I_{NA}/Y_b =$ 731.86 cm^3 (min.)

Now, putting all the above values in equation (VII), we have,

p = 26.0305 kg/cm^2 < 50.00 kg/cm^2
(M-25)

14.1.7 **Bending of Track :**

Ref: IS 4622 : 1992 (page-7, art. 5.7.5.4)

Bending stresses in track base shall be found from the following formula,

$$f_b = 0.5 \times \frac{P}{Z}\left(\frac{E_s}{E_c} \times \frac{I}{w}\right)^{\frac{1}{3}} \text{------------------ (X)}$$

Here, $P =$ 57994.5 kg $I =$ 21002 cm^4

$Z = Z_{min} =$ 793.43 cm^3 $w =$ 30.00 cm

Now, putting the corresponding values in equation (VIII), we have,

$f_b =$ 800.31 kg/cm^2 < 960.00 kg/cm^2

14.1.8 **Bending of Flange :**

We have, maximum bending moment, $M_{max} = W.L^2/2$

Here, treating one side of the flange as cantilever ie. $L = b_2/2 =$ 15.00 cm

Load bearing width, b = 1.00 cm

Load per unit length for unit width, W = p.b = 26.0305 kg/cm

So, M = 2928.43 kg-cm

Section modulus for unit width, $Z = b.(h_1+h_2)^2/6=$ 5.22667 cm^3

Then bending stress, $f_b = M/Z=$ 560.287 kg/cm^2 < 960.00 kg/cm^2

14.1.9 **Check for Overlapping of Bearing Stress or Check of Concrete Steel due to Overlapping :**

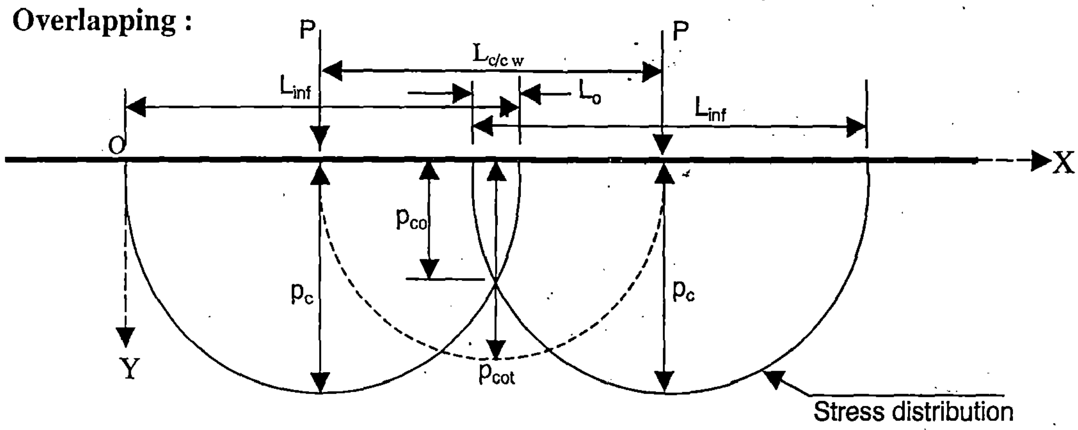


Figure : 14.3

Here, Distance between C/C of wheels, $L_{c/c\,w} =$ 85.00 cm

Bearing stress in concrete, $p_c =$ 26.0305 kg/cm^2

Design wheel load, P = 57994.5 kg

Width of track base flange in contact with concrete, w = 30.00 cm

The length of influence 'L_{inf}' in concrete is found by putting the corresponding values inequation (VI)

$L_{inf} =$ 111.397 cm

The parabolic stress distribution equation is given by-

$x.(L_{inf} - x) = k.y$;

At the max. stress point , $x = L_{inf}/2 =$ 55.70 cm and $y = p_c$;

So, constant of the equation, $k = x.(L_{inf} - x)/p_c =$ 119.181 cm^4/kg

At the point of intersection, $x = L_{inf}/2 + L_{c/c\,w}/2 =$ 98.1986 cm

Then, $y = p_{co} = x.(L_{inf} - x)/k =$ 10.8749 cm^2/kg

So, max height of the envoloping parabola, $p_{cot} = 2.p_{co} =$ 21.7499 cm^2/kg

which is < 50.00 cm^2/kg

Length of overlap, $L_o = L_{inf} - L_{c/c\,w} =$ 26.40 cm

14.1.10 **Depth of Maximum Shear Stress & Thickness of Track :**

With reference to the Theory of Elasticity by Timoshanko & Goodier, we have,

$$b = \sqrt{\frac{8P.r(1-n^2)}{\pi El}} \text{ ——————— (XI)}$$

$$f_c = \sqrt{\frac{PE}{2\pi(1-n^2)rl}} = \frac{2P}{\pi b.l} \text{ ——————— (XII)}$$

Where,	Here,	
P = wheel load in N or kg;	= 57994.5 kg	
f_c = contact stress in N/mm² or kg/cm² ;	= 8091.66 cm²/kg	(from atr.10.4)
E =modulus of elasticity in N/mm² or kg/cm² ;		
l = tread width in mm or cm ;	= 12.00 cm	(from atr.10.3)
b = semi-minor axis of ellips of contact in mm or cm ;		
n = poission's ratio = 0.27		
r = radious of the wheel in mm or cm ;		

Now, from equation (XII) we have, b =2.P/(pi).l.f_c = 0.3802 cm

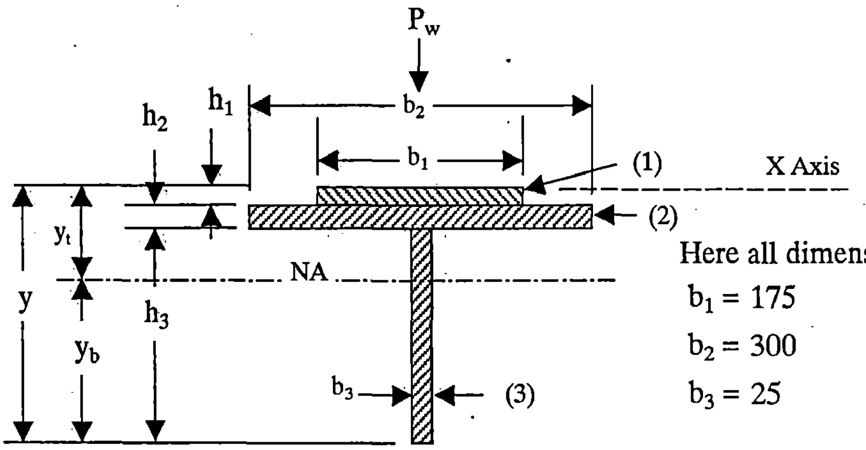
Depth of maximum shear stress, d_s = 0.78 b = 0.29658 cm

Thickness of the track, h_t = 3 d_s = 0.88974 cm

= 0.9 cm (say)

Maximum shear stress, S_s = 0.304f_c = 2459.86 kg/cm²

14.1.11 **Reselection of Section instead of the above :**



Here all dimensions are in mm

b_1 = 175	h_1 = 16
b_2 = 300	h_2 = 40
b_3 = 25	h_3 = 304

Figure : 14.4

14.1.12 **Calculation of Neutral Axis and Modulus of Section :**

Item No.	Item	Size		A (cm²)	Y (cm)	AxY	AY²	I_{self} (cm⁴)
		b (cm)	h (cm)	(b x h)	dist from X	(cm³)	(cm⁴)	bh³/12
(1)	Track	17.50	1.60	28.00	0.80	22.40	17.92	6.0
(2)	Track base	30.00	4.00	120.00	3.60	432.00	1555	160.0
(3)	Web	2.50	30.40	76.00	20.80	1580.80	32881	5853.0
Sum:			36.00	224.00		2035.2	34454	6019.0

So, Y_t = sum(AY)/sum(A) = 9.09 cm $\quad$ Y = $h_1 + h_2 + h_3$ = 36.00 cm

$Y_b = (Y - Y_t)$ = 26.91 cm

Then, I_{NA} = sum(AY^2) + sum(I_{self}) - sum(A)xY_t^2 = 21982 cm^4

$Z_t = I_{NA}/Y_t$ = 2419.3 cm^3

$Z_b = I_{NA}/Y_b$ = 816.72 cm^3

14.1.13 **Check for Line Contact Condition :**

We have, line contact stress is given by - $\quad f_c = 0.418\sqrt{\frac{PE}{rl}}$ ---------------- (V)

Here,

Maximum design wheel load, P_w = P = 57994.5 kg $\quad$ E = 2.1E+06 kg/cm^2

r = $D_w/2$ = 25.0 cm $\quad$ l = l_w = 13.00 cm

Now, putting values in equation (III), we have,

f_c = 8091.66 kg/cm^2 < 0 kg/cm^2 **Hence, safe.**

14.1.14 **Area of contact :**

Ref: IS 4622 : 1992 (page-7, art. 5.7.5.2)

We have, $\quad L = \frac{3}{2} \times \frac{P}{w \times p_c}$ ———————— (VIII)

Where,

L = Length of influence under track base in mm or cm

P = total wheel load in N or kg = 57994.5 kg

w = width of track in contact with concrete in mm or cm

p_c = stress in concrete in N/mm^2 or kg/cm^2 = 8091.66 kg/cm^2

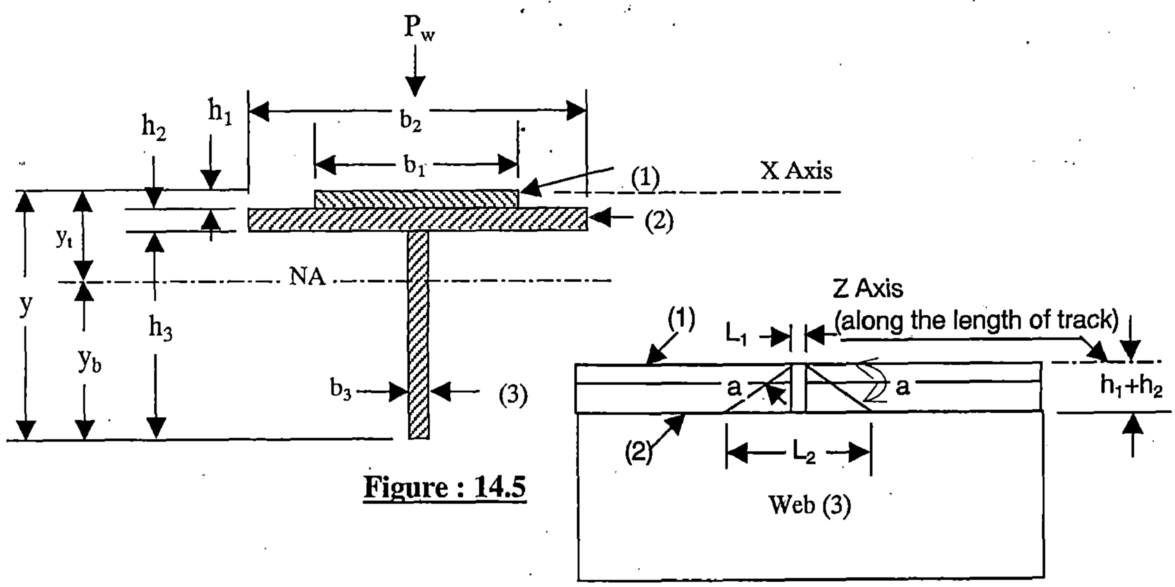


Figure : 14.5

Now, rearranging equation (VI) we have,

Area of contact, say, A = L.w = 3P/2p_c= 10.7508 cm^2

Let, w = l_w = 13.00 cm

So, L_1 = A/w = 0.82698 cm $\quad$ (length on top of the track)

Angle of influence,say, a = 30 degree (within steel)

Length of influence at the bottom of the track base flange,

say, $L_2 = 2(h_1+h_2).Cot(a) + L_1 =$ 20.226 cm

14.1.15 **Bearing Stress on Web,**

$S_b = P/(L_2 .b_3) =$ 1146.93 kg/cm^2 < 1320 kg/cm^2

(And also as per ref. IS 800-1984, clause-6.3),

Permissible bearing stress, $S_p = 0.75Y_p =$ 1800 kg/cm^2

Then bearing stress in Concrete,

Ref: IS 4622 : 1992 (page-7, art. 5.7.5)

Bearing stresses in concrete shall be found from the following formula,

$$p = 0.2813 \times P\left(\frac{E_c}{E_s \times I \times w^2}\right)^{\frac{1}{3}} \text{------------------ (IX)}$$

Where,

p = bearing stress in concrete in N/mm^2 or kg/cm^2 ;

P = total wheel load in N or kg ;

E_c = modulus of elasticity of concrete in N/mm^2 or kg/cm^2 ;

E_s = modulus of elasticity of steel in N/mm^2 or kg/cm^2 ;

I = moment of inertia of the track base in mm^4 or cm^4 and

w = width of the track base in contact with concrete mm or cm ;

Here, P = 57994.5 kg E_s = 2.1E+06 kg/cm^2

E_c = 1.4E+05 kg/cm^2 w = 30.00 cm

14.1.16 **Moment of inertia of Track base :**

Item No.	Item	Size		A (cm^2)	Y (cm)	AxY	AY^2	I_{self} (cm^4)
		b (cm)	h (cm)	(b x h)	dist from X	(cm^3)	(cm^4)	bh^3/12
(2)	Flange	30.00	4.00	120.00	2.00	240.00	480.00	160.0
(3)	Web	2.50	30.40	76.00	19.20	1459.20	28017	5853.0
Sum:			34.40	196.00		1699.2	28497	6013.0

So, Y_t = sum(AY)/sum(A) = 8.67 cm $Y = h_2 + h_3 =$ 34.40 cm

$Y_b = (Y - Y_t) =$ 25.73 cm

Then, I_{NA} = sum(AY^2) + sum(I_{self}) - sum(A)xY_t^2 = 19779 cm^4

$Z_t = I_{NA} / Y_t =$ 2281.4 cm^3 $Z_b = I_{NA} / Y_b =$ 768.68 cm^3 (min.)

Now, putting all the above values in equation (VII), we have,

p = 25.3346 kg/cm^2 < 50.00 kg/cm^2

(M-25)

14.1.17 **Bending of Track :**

Ref: IS 4622 : 1992 (page-7, art. 5.7.5.4)

Bending stresses in track base shall be found from the following formula,

$$f_b = 0.5 \times \frac{P}{Z}\left(\frac{E_s}{E_c} \times \frac{I}{w}\right)^{\frac{1}{3}} \text{-------------------} (X)$$

Here, P = 57994.5 kg I = 21982 cm^4

$Z = Z_{min}$ = 816.72 cm^3 w = 30.00 cm

Now, putting the corresponding values in equation (VIII), we have,

f_b = 789.39 kg/cm^2 < 960.00 kg/cm^2

14.1.18 **Bending of Flange :**

We have, maximum bending moment, $M_{max} = W.L^2/2$

Here, treating one side of the flange as cantilever ie. $L = b_2/2$ = 15.00 cm

Load bearing width, b = 1.00 cm

Load per unit length for unit width, W = p.b = 25.3346 kg/cm

So, M = 2850.14 kg-cm

Section modulus for unit width, $Z = b.(h_1+h_2)^2/6$= 5.22667 cm^3

Then bending stress, f_b =M/Z= 545.307 kg/cm^2 < 960.00 kg/cm^2

Hence Safe.

14.2 Design of Sill Beam :

The distance between centre to centre of anchor bolts are provided in gate groove for embedding sill beam is, say, L_a = 50 cm

Length of sill beam, L_{sil} = 675.0 cm

Total weight of the gate, W_{gt} = 2 .unit wt. = 17111.3 kg (say), 20000 (art. 12.2)

So, design load, $P_{sil} = 1.5\ W_{gt}$ = 30000 kg

This load will act as UDL over the length of the sill beam,

So, udl on the sill beam, $W = P_{sil}/L_{sil}$ = 44.44 kg/cm

Due to impact load on the sill beam by the total weight of the at the time of closing ;

Therefore, considered design udl on the sill beam, W_d =2.W= 88.89 kg

Assume, sill beam to be simply supported between the anchor bolts;

So, maximum bending moment is given by, $M=W_d.L_a^2/8$ = 27777.8 kg-cm

Allowable bending stress, S_a = 960.00 kg/cm^2

Required section modulus, $Z = M/S_a$ = 28.94 cm^3

14.2.1 Adopting rolled beam section, ISMB 200 as below :

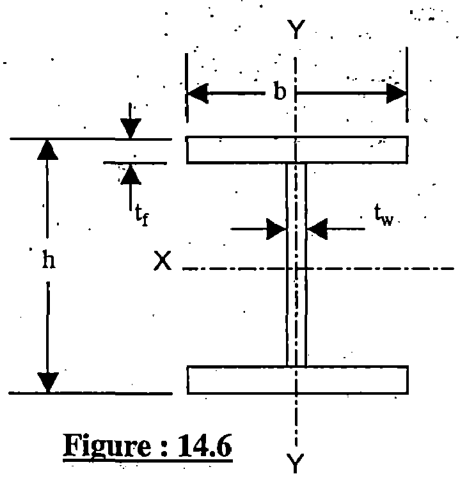


Figure : 14.6

Dimension and other properties of the section:

h =	20.00 cm	w =	0.254 kg/cm
b =	10.00 cm	I_{xx} =	2235.40 cm^4
t_f =	1.08 cm	I_{yy} =	150.00 cm^4
t_w =	0.57 cm	Z_{xx} =	223.50 cm^3
a =	32.33 cm^2	Z_{yy} =	30.00 cm^3
radius (root), r_1 =	1.10 cm		
radius (top), r_2 =	0.55 cm		

Slope of the flange, D = 98 degrees

Therefore, adoping Z_a = 223.50 cm^3

So, actual bending stress, $f_b = M/Z_a$ = 124.285 kg/cm^2 < 960.00 kg/cm^2

14.3 Design of Base Plate for Main Sill Beam :

Centre to centre distance of anchor bolts, L_a = 50 cm

Let, width of the plate, b = 30 cm

and thickness of the plate, h = 1.2 cm

Thus, total load bearing area, $A=L_a \times b$ = 1500 cm^2

For M-25 grade concrete, permissible bearing stress, S_p = 50.00 kg/cm^2

Therefore, the load that can be supported by M-25 concrete,

$P = A.S_p$ = 75000.0 kg

The required load capacity is, $P = W_d.L_a$ = 4444.44 kg

Hence Safe.

Check for Shear:

Shearing area per cm length of the plate, $A_s = 2.h$ = 2.4 cm^2

Then shear stress, $f_s = W_d/A_s$= 37.037 kg/cm^2

< 720.00 kg/cm^2

Hence Safe.