DESIGN AND TOPOLOGY OPTIMIZATION OF LOW HEAD RADIAL GATE FOR SPILLWAY USING ANSYS

A DISSERTATION

Submitted in partial fulfillment of the

requirements for the award of the Degree

of

MASTER OF TECHNOLOGY

in

WATER RESOURCES DEVELOPMENT

By

ANUJ KUMAR

(16548003)

DEPARTMENT OF WATER RESOURCES DEVELOPMENT & MANAGEMENT

INDIAN INSTITUTE OF TECHNOLOGY, ROORKEE

ROORKEE – 247667 (INDIA)

MAY, 2018

CANDIDIATE'S DECLARATION

I hereby declare that the work carried out in this report entitled "DESIGN AND TOPOLOGY OPTIMIZATION OF LOW HEAD RADIAL GATE FOR SPILLWAYS USING ANSYS" is presented on behalf of the partial fulfillment of the requirement for the award of the degree of Master of Technology with specialization in Water Resources Development, submitted to the department of Water Resources Development and Management, Indian Institute of Technology, Roorkee, India, under the supervision and guidance of Er. S. K. Shukla, WRDM, IIT Roorkee, India.

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

Date: *04.05.2018* **……………………**

Place: *Roorkee* **(ANUJ KUMAR)**

CERTIFICATION

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

ACKNOWLEDGEMENT

I wish to express my deep sense of gratitude and sincere thanks to my supervisor **S. K. Shukla**, Adjunct Professor, Department of Water Resources Development & Management, IIT Roorkee, for guiding and encouraging me at every point throughout the study. In spite of his busy work schedule, his constant encouragement and co-operative attitude gave me confidence to complete the study within the time frame. I wish to thank him for his constant guidance and suggestions and his help and support towards my personal life.

I am also grateful to Er. Mayank Jain, THDC Rishikesh, for his valuable instruction & guidance in friendly way and made me complete my dissertation work successfully.

I am also thankful to all faculty and Staff of Department of Water Resources Development & Management who have directly or indirectly supported me in completion of the M. Tech dissertation work.

My special thanks to Mr. Gaurav Saini, PhD Scholar, AHEC, IIT Roorkee, for his helpful coordination in software learning for the study and guidance in preparation of the report.

My heartiest gratitude goes to my parents, brother and friends for constantly supporting me, which allowed me to concentrate on my work. I would like to dedicate this work to my family & friends.

....

(Anuj Kumar)

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

Abstract

Almost every project of water resource has a diversion work or reservoir to store the water for power generation, irrigation, industrial and domestic water supply. A spillway with control mechanism is provided in almost every project to release the water during excess inflow. Radial gates are commonly used as control mechanism for spillways. Radial gates are hinged gates with the skinplate in form of circular arc with the centre of curvature at the trunnion or hinge.

In this study a radial gate is designed for given technical parameters according to guidelines given in IS 4623. A 3D model of radial gate is developed on ANSYS using design modeler. This model is studied for different grade of structural steel and best grade is selected for further study. Topology optimization is performed on the model for best grade of structural steel.

According to IS 4623 material for main component of radial gate is structural steel. In this study three grades of structural steel ASTM A 36, ASTM A 992 and ASTM A 588 are selected for study. Study shows that ASTM A 992 structural steel is the best material for radial gates as it gives minimum values of stresses for same loading and boundary conditions. Yield strength of ASTM A 992 structural steel is more as compare to ASTM A 36, hence ASTM A 992 is used for topology optimization.

Topology optimization is performed using ANSYS to reduce weight and cost of radial gate. The weight of output model of topology optimization is 82% of the initial weight of model. This geometry is further modified using spaceclaim to perform design validation. This modification in geometry increases its weight to 85.85% of initial weight. Design validation is performed on modified geometry using ANSYS. Finally 14.15% reduction in weight is achieved and design validation shoes that stresses are within permissible limits.

Keywords: Radial gate, ANSYS, ASTM, Design modeler, Spaceclaim

CHAPTER 1: INTRODUCTION

The gate is one of the important components of hydraulic structures. It can close the opening of the structure as needed, also openings can be fully or partially opened to regulate upstream and downstream water levels and flow to obtain benefits such as flood protection, irrigation, drinking water supply, power generation, navigation etc. The main function of gates is to open and close the opening, flexibly and reliably to exert its benefits and maintain the safety of structure. Radial Gate also known as Trainter Gate after Jeremiah Burnham Tainter a structural engineer from Wisconsin USA.

1.1 Development of Radial Gates

The earliest radial gates were 8.75m wide and 1m high on the Seine river in Paris in 1853. Other early applications include 132 radial gate of 6m wide and 5.1m high on Nile river delta in 1860. The first German radial gate of 12m span and 1.87m high was installed in 1894/1895 near Berlin. Radial gates were again installed on Landwehr canal in Germany with a span of 5.56m and height 1.6m. In USA radial gate were used on Illinois-Missssissippi canal in 1895. In USA the radial gate were used for the first time, for lock aqueducts in New York barge canal in 1905. In Germany reversed radial gate was again used in 1953 at the Oberpeichning Dam. Because of good performance and high reliability 28 other radial gate were built up to 1976 in Germany. In 1902 double leaf metal gate of 5m wide and 5m high were used in Egypt [1].

Radial gates have also been widely used in India. 30 Radial Gate (25 main spillway gate and 5 auxiliary gate) was installed on Narayanpur Dam in 1985 and size of gate was 15m*10m. In 2006 Radial Gate was installed at Tehri dam in Uttarakhand. Size of radial gate at Tehri Dam is 15.5m*10.5m. Five years latter Radial Gate was installed at Koteshwar dam which is at downstream of Tehri dam. Four Radial Gate of 18m*16m was installed at Koteshwar dam.

1.2 Structural Features of Radial Gates

The Radial Gate have various component like skinplate, arm structure, support hinge or trunnion, and it is a complicated space structure system.

The gate is an important part of hydraulic structures, and its operation is related to the safety of the entire structure. There are many factors to be considered when designing the gate, and the structural stability of the gate is the most important consideration [2].

Increase the amount of material and the increase in the number of connecting parts between the components of the gate can achieve good structural stability, but will increase cost.

1.3 Unique advantages of radial gate

(1) A relatively large area of orifice can be closed.

(2) The required height and thickness of the pier are small as compare to vertical lift gate.

(3) Since the trunnion axis of the Radial Gate is generally arranged at the center of the curvature of the skinplate, water pressure passes through the trunnion axis. When the door is opened, it is only necessary to overcome the dead weight of the gate and the trunnion friction. Thus the radial gate opens and closes effortlessly, quickly, and operates reliably, reducing the capacity of the hoist. For high head gates the hoist capacity is a very important constraint.

(4) Operation of radial gate is quick because gate moves along circular arc.

(5) The lower portion of skinplate provide a converging passage when gate is lifted hence provide the effective discharge.

1.4 Structural Arrangement of Radial Gate

The Radial Gates are commonly used in water storage and hydropower projects. It has a simple structure and a small opening and closing force. The advantages, such as simple operation, good flow conditions etc. makes it suitable for use as spillway gate.

The Structural design must ensure that the structure has sufficient strength and overall stiffness, has a good processing technology, easy to manufacture, transport, install and corrosion resistance. The Radial Gate is mainly composed of the three structures skinplate, the support arm structure and the trunnion assembly.

Skinplate takes the water load and transmit it to stiffeners. Stiffener are in contact with horizontal girder and transmit load to these girders. The arm of the Radial Gate supports the horizontal girder and transmits its load to the support hinge or trunnion. The hinged support of the Radial Gate is the most important part of the entire gate.

There have been many accidents in the history of radial gates in the decades and some accidents have caused great hazards. The reason of these failure, in addition to manufacturing, installation and operation is mainly due to problems in the design.

In the past, the calculation of the gate was usually done by the planar system method. It failed to correctly reflect the radial gate because stress conditions is in complex spaces. Gates are often required for local opening applications. The high-speed flow at downstream of the gate can easily lead to negative pressure, when the pressure is low to the water vapour pressure it can cause cavitation and vibration, which can severely damage the structure.

Following are the problems in design of Radial Gates.

1. The calculation of the planar system algorithm of the gate design is to divide the entire structural system into individual components and the load is assigned to each component and each component is analyzed. This method ignores the structural integrity and the spatial structure characteristics of the radial gate. It is too conservative, and there are not enough safety margins in some key areas, resulting in the insecurity of the entire structure.
2. The error of the results is also large and cannot meet the computational requirements for high head radial gate structures. Currently in numerical analysis the widely used finite element method (FEM) is an efficient, and can more accurately reflect the coordination of the overall structure which consist of various components
3. The arm is the key component of the Radial Gate. In fact, the Radial Gate failure is also mostly manifested as arm destruction. The method to design the arm, consider arm a longitudinal main frame structure. The force is equally divided between two arms and this method does not consider the influence of the bending moment on the arm. It is considered that the arm only bears the axial force. But in fact, the force acting on the arm is based on the axial force and it also bears the effect of bending in both directions. Therefore, it is calculated according to the current planar system. The calculation method is not safe for the design of the arm.

The design of traditional Radial Gate generally uses the plane system calculation method, but in this method the calculation result exceeds 20 to 40% of the measured value in many places, and may be too small in some key areas. This method has some limitations. The topological optimization with proper modeling needs to be evaluated and validated to overcome the limitation. The finite element method that is widely used in numerical analysis is an efficient method [2].

1.5 Finite Element Method (FEM)

Finite element method is an effective numerical technique to analyze structures and to calculate stress, strain, deformation etc. In this method the given model is divided into small element known as finite element. In finite element method the result is generated for individual elements and then combined to generate the result for whole structure. In this method, digital computer generate and solve various simultaneous equations. The generated result is accurate and can be use engineering problems.

In this method finite element is generated by randomly oriented planes. Ideally to get exact solution number of element should be infinite. So while meshing a model a balance should be maintain between time required for calculation and accuracy. If variation in stress and deformation is less, then cores mesh can be used but if variation is more, then fine mesh is required.

1.5.1 Selection of displacement model

In FEM an initial approximation is required which is quite complicated, this is converted into relatively simpler polynomial for each element to make the computation easy.

For triangular element linear polynomial is approximated as:

$$\alpha = k1 + k2x + k3y$$

Where k_1, k_2 and K_3 are constant which depends upon α for these nodes.

For quadrilateral element bilinear function is approximated as:

$$\alpha = k1 + k2x + k3y + k4xy$$

So number of constant in polynomial equation is equal to number of node in the element. If mesh quality is good for an element and α_1 is exact then α is a good approximation [3].

1.5.2 Convergence requirements

In any numerical formation the solution must converge to exact solution of the problem. To convergence of solution two conditions must satisfy. Firstly displacement model should be continuous in the element and this is possible if displacement function is polynomial. Secondly displacement should be compatible in adjacent elements and this is possible when element deform without creating any openings, discontinuities or overlaps between them.

1.5.3 Nodal degree of freedom (DOF)

Degree of freedom is used to define the deformation of any element in finite element analysis. Degree of freedom include nodal displacements, nodal rotations and strains.

1.5.4 Nodal loads and forces

Concentrated loads are applied at nodal locations which have same location as applied load. Distributed loads action on an element are distributed among the nodes of that element according to minimum potential energy principle. If only gravitational force acting on a triangular element then they equally distributes among all three nodes of element.

1.5.5 Boundary conditions

In finite element analysis displacement is restricted at certain edges or points. These restriction also known as boundary condition. We need to define boundary condition for all solid mechanics problems. Boundary conditions are necessary for equilibrium of structure. Boundary conditions may be geometric or natural, but in finite element analysis we need to define only geometric boundary conditions. In some numerical methods, solution is obtain by iterative method so that boundary conditions are satisfied but in FEM boundary condition are define prior to analysis so no need of trial and directly solution is obtained.

1.5.6 Mathematical model

There are two major steps to study any physical phenomenon using Finite Element Method.

1. To form a mathematical model of physical process.
2. Numerical analysis of this mathematical model.

There are some assumptions about working of process, these assumptions helps in development of mathematical model. To evaluate mathematical model, numerical method and computers are used. Sometimes the solution of governing equation is not possible by direct method. In these cases, approximate method is used as alternative means to find the solution.

1.5.7 Unique features of FEM

1. Complex geometry is represented as group of simple sub-domain also known as finite element.
2. Approximation functions derived over every element of geometry using the idea that continuous functions can be represent by combination of polynomial.
3. Governing equations satisfies at each element to obtain algebraic relations between node values (undetermined coefficient).

Conventional method of designing the radial gate is lengthy and time consuming. In this method, to design a component its dimensions are assumed and checked for stresses. If stresses are within permissible limits then these dimensions are selected otherwise change the dimension and repeat the above process. With the help of modern computers finite element provides fast and accurate solution to engineering problems. Stress distribution provided by FEM helps to find the critical cross section and modify accordingly.

1.6 Objectives of the study

- In this study the objective is to design a radial gate for spillways for given technical parameters having accuracy in design, faster and simpler approach.
- Develop 3D model of this radial gate using ANSYS workbench. Perform the finite element analysis on this model for different grades of structural steel.
- Select the best grade of structural steel for radial gate and perform topology optimization to reduce the weight of gate.
- Finally validate the optimized design for same loading condition.

CHAPTER 2: LITERATURE REVIEW

2.1 Background

A literature review is conducted to understand the latest advancement in the design of radial gates. Various research papers published by authors are reviewed to understand the process of designing and modelling.

Following are some of the literature reviewed.

Kun cai et al. (2013) performed topology optimization on the arm of radial gate. Topology optimization was performed using CAD software. In this paper modified shape was suggested for the arm of radial gate. A radial gate was designed as per conventional method with inclined arm. Design of radial gate having inclined arm is complex as compare to straight arm. In designing the stiffness and strength must satisfy requirement of structural design. Conventional design had a total weight of 140T, maximum stress of 269MPa, maximum displacement of 34.3mm and natural frequency of 2.17Hz. Then a 3D model of this radial gate was developed using Hyperworks software. Topology optimization can be performed on various CAD or CAE software like Ansys, Hyperworks etc. Topology optimization was performed by using the gradient based method [4].

Topology optimization result in 24% reduction in weight of radial gate. The weight of radial gate after topology optimization became 105T. The stiffness of radial gate also improved after topology optimization and strength satisfied requirements of structural design. After topology optimization maximum stress increase to 291MPa but still within permissible limit. Maximum displacement decreased to 26.4mm and natural frequency increased to 3.51Hz. So he reduced the weight of radial gate to 76% and design was safe according to strength and rigidity criteria, through topology optimization.

G.Chandra Mohana Reddy et al.(2015) deigned a radial gate for given technical parameters and prepare a model of this radial gate using CATIA V5 R 20 software. For modeling the section radial gate was drawn and rotated about their respected pivoted point. This model was import to ANSYS software for analysis. This model was tested for different material to find the best material for radial gate. There are some materials in the ANSYS library and other materials can be defined in ANSYS by their properties like Young’s modulus of elasticity, poission’s ratio, yield strength and ultimate strength etc.

This model of radial gate was tested for high strength low alloy steel and low carbon steel. ANSYS performed finite element analysis and gave result for von mises (equivalent) stress, shear stress, elastic strain and total deformation. By analyzing these it was concluded that high strength low alloy steel is better material for radial gate as it gave less stresses and deformation. If stresses induced are less, then weight and cost of radial gate can be reduced [5].

K.Sarth Chandra et al. (2015) Performed vibration study on Folsom dam radial gate. The radial gate at Folsom dam failed in year 1995. A previous study shows that Folsom dam radial gate failed due to self-excited vibration. The weight of radial gate at Folsom dam radial gate was 87 ton. While designing the radial gate at Folsom dam two mode of vibration was considered, rotation of whole structure about trunnion pin and vibration of skinplate perpendicular to the direction of water load. But combination of these two vibrations was not considered in the designing state. If these two vibration mode combine together it may induce self-excited vibration and this is possible when hydrodynamic forces acts with inertia torque [6].

In this study, equations were derived for both mode of vibration based on previously derived equations pressure and addition of mass effect. Then an iterative method was used to solve these two equation coupled together. The solution of these equations gave the stability diagram for dynamic condition. The solution gave two possible regions which caused instability. These two cases was based on frequency of vibration of these two modes. When frequency of skinplate bending vibration is less than the frequency of rotation mode, in this case skinplate bending is the driving force and rotation of rigid body became responsive mode and this is most common unstable condition. In second case skinplate bending vibration is higher than the frequency of rotation mode. In this case rotation of rigid body is the driving force and bending of skinplate became responsive mode.

Robert T. Indri et al. (2015) studied the project of strengthening the radial gates at poe dam. Poe dam is located on Feather River in California. The dam was constructed in late 1950's. Poe dam has four spillway radial gates of size 15.24m*12.5m. Since the radial gates of dam is in working condition for very long time so a project to strengthening the radial gates is taken into consideration. A detail study of different part of radial gate was performed to find the critical component. The deflection of all four gate was calculated under normal working condition and trunnion friction was calculated for two radial gates. Test result show high value of coefficient of friction. The coefficient of friction at trunnion assembly was 0.46 which cause overstressing of different component of arm of radial gate. A separate study was performed to find the alternatives to strengthening the existing gates.

There were two recommendation based on the study. Firstly the trunnion bushing should be changed to reduce the friction and a radial gate with different material should be use to increase its strength. But the condition of contact surface was such that recommendation was made to change the entire yoke hub and trunnion assembly. Finally finite element analysis was performed to find the suitable material for the component that must be replaced by new component to increase the reliability of the gates. Finite element analysis suggest that the best material for yoke should be structural steel of grade ASTM A 106. The material for the pin of trunnion assembly was also changed. Study shows that the coefficient of friction at trunnion assembly must be less than 0.1 for that material change of composite bushing was recommended. No changes was recommended for the system geometry and hence load path remains same [7].

Rahul Sahu et al. (2017) used a combination of I section and hollow rectangular section instead conventional I section to reduce the weight. Technical specifications of Man dam was used to find the dimension of gate. A model redial gate was developed using SAP2000 software. Structural optimization was also performed using same software. Structural optimization also known as layout optimization because in this method layout of material distribution is changed to achieve optimization. A new designed of radial gate was proposed and this new design was tested for different material to find best material for radial gates. Two models was tested and compared one with conventional I section and other with unconventional hollow rectangular section and these models was analyzed for different materials [8].

In this study T shape vertical stiffener was replaced with rectangular hollow section. Reduction in weight was 64 kN, which is 23.3% of initial weight. ASTM A 653 grade of structural steel have minimum stresses, hence it is best material for radial gate out of all the tested materials. This study gave an effective method to design the radial gate because manual method of calculation is repetitive and highly time consuming, whereas SAP2000 is a productive, powerful and simple tool for analysis of structure.

Krzysztof Brusewicz (2017) conducted the study on a radial gate with finite element method using ABAQUS software. A radial gate was modeled and to simulate the effect of water, a water body was also modeled. In some cases like earthquake it is important to consider the weight of water. If water is modeled as body it will help in simulate effect of mass of water on radial gate. The level of water was changed during analysis and its effect was considered to find most vulnerable level of water. Most of the gates are designed for maximum water level but it is important to consider the effect of variation of water level when going for dynamic analysis. This model was tested for 13 different cases.

He found in his study that water level have a significant effect on frequencies of eigen mode. The study shows that as water level increases, frequency decreases and this decrease in frequency have substantial value. It was found that frequency at full reservoir level was 64% less as compare to the condition when there was no water. The study investigated susceptibility of radial gate for dynamic load. This study provided the base for further research on dynamic loads like earthquakes [9].

Liu Jiajia (2012) proposed a new technique to design a radial gate. The conventional method of designing the radial gate (plane system method) gives uneconomical design. Study shows that stress calculated by conventional method are 20% to 40% more than the actual stresses, but at some critical points it may be less than the actual value causing failure of radial gate. Finite Element Method gives accurate results but it is a complicated technique and modeling takes most of the time and efforts. So Liu Jiajia modified the conventional method to make it more useful for engineering applications and to reduce workload of the modeling.

There are two method for designing of radial gate. In first method bending of skinplate is considered about the skinplate and in second method skinplate is divided into small section fixed at its edges. These assumptions are not feasible so he performed a detail analysis on skinplate designing techniques [2].

He proposed a new frame structure model which composed of beams and plates, and applied this model for structural analysis with FEM on a radial gate. It was found that calculation result of frame structure model was comprehensive and reasonable as compare to the conventional method.

Ji Liang Liu et al. (2011) performed Simulations of Natural Vibration Characteristics of Large-size Radial Gate with Three Arms Considering Fluid-solid Coupling.

When orifice opening of flow under gate through spillways becomes large, large-size radial gates with three radial arms are normally recommended and have been used in various hydraulic engineering projects. However, radial gates with three arms have more chances to failure during its operation as compare to radial gates with two arms. They developed a finite element numerical model using ADINA software to simulate and analyze natural vibration characteristics of a radial gate with three arms at the Shuhe Hydropower Station using fluid-solid coupling theory. Simulation results of natural vibration characteristics for the gate were also developed without considering fluid-solid coupling. Simulated natural frequencies of the same mode are increased with the increase of openings, but the change amplitudes are not significant. This study provides useful information for structural dynamic design of large-size radial gates in order to avoid the natural frequency of radial gate close to the main pulsation frequency of water flow that may cause resonance [10].

Gorazd Novak et al. (2016) studied hydrodynamic forces during the operation of a model radial gate. They prepared a 1:17 scale physical hydraulic model of a radial gate and employed to determine forces during the lifting and lowering of the gate with rates from 0.1 to 1.1 m/min for prototype and to determine the combinations of discharge, gate opening and tailwater level that cause instability of the gate. Forces were measured in trunnion bearing and in both hoists without restricting the movement of the model. Physical hydraulic model was used to investigate forces in a box-designed radial gate at various modes of gate operation.

Tailwater levels covered a wide range, representing the changed conditions after the completion of the downstream dam. The study indicated certain trends regarding the instability of the gate, and showed that the current rate of raising the gates could be significantly increased, providing safer operation of the whole system during some specific operating conditions [11].

CHAPTER 3: METHODOLOGY

The aim of this study is to find the suitable grade of structural steel and to optimize the design to reduce the weight and hence cost of radial gates. IS code 4623 “Recommendation for structural design of radial gate” provides the guidelines to design the radial gates [12]. IS 4623 also suggest material for main components of radial gate. According to IS code structural steel is used for most component of radial gate like skinplate, horizontal girders, vertical stiffeners and arm. It is not mentioned in code that which grade of structural steel should be use for these component. Structural steel can broadly classified into four categories.

1. Carbon steel
2. High strength low alloy steel
3. Corrosion resistant high strength low alloy steel

These categories are further classified into various grades. In this study one grade is selected from each categories and best grade of structural steel is selected for radial gate. Finite element method is used for this study. A 3D model of radial gate in developed in ANSYS workbench using design modeler and analysis is performed on ANSYS mechanical. This model is tested for different grade of structural steel to find best grade for radial gates.

In this study topology optimization is also performed on the model of radial gate to reduce weight while stresses induced remain remains within permissible limits. Topology optimization is the technique in which some material is removed and shape is optimized so that stresses remain almost same. Topology optimization is performed on ANSYS software and it gave modified geometry which is quite irregular. This geometry should be modified to make it manufacturable and for further analysis.

3.1 Design of radial gate

A radial gate is designed for the given technical specifications and calculation is based on IS 4623. The technical parameters required to design the radial gate are defined in chapter 4. The size of the gate is specified as opening width and vertical height up to full reservoir level. Radial gate has a skinplate bent as an arc and its convex surface is in contact of water. Centre of curvature of skinplate always coincide with that of trunnion pin. Radial gate can rotate about trunnion pin. Stiffeners supports the skinplate and it may be horizontal or vertical or even both. In this study vertical stiffeners are used and to support them horizontal girders are used at suitable distance. The horizontal girders are supported by arms, which emanate from trunnion hub. Trunnion hub is located at axis of skinplate cylinder. Arm is the most important component of radial gate and transmit the water load from skinplate to trunnion assembly.

3.1.1 Design of skinplate and stiffeners

The radius of skinplate in the distance from centre of the trunnion to inner face of skinplate.

The radius of skinplate varies from H to 1.25H, depending upon trunnion location. Where, H is vertical height of gate. Minimum thickness of skinplate should be 8 cm.

There are two method to design the skinplate.

1. In this method bending of skinplate is considered about the stiffeners. If stiffeners are vertical then bending takes place in horizontal plane and vice versa.

   Bending stress = M/Z

   Where, M is bending moment and Z is section modulus.

   $M = wL^2/2$ (for end span)

   $M = wL^2/2$ (between two stiffeners)

   Where, w is load per unit length.

   If this bending stress is less than the allowable stress then design is safe otherwise increase the thickness of skinplate and repeat the above process.

2. In this method a formula is given in the code to calculate the bending stress. This calculated stress should be less than permissible value. Correlation is given as:

$$e = \frac{k}{100} * \frac{p * a * a}{S * S}$$

   Where, e = Bending stress. (N/mm^2)

   k = A factor whose value depends upon a & b. (Dimensionless)

   p = Water pressure (N/mm^2)

   a & b = Bay width. (mm)

   S = plate thickness. (mm)

   Values of all the constant is given in the code.

In this study first method is used to design the skinplate.

While designing vertical stiffeners and horizontal girders skinplate is considered coacting with them.

There are three correlation to calculate the coacting width of skinplate.

1. 40t + B

   Where, t is thickness of plate and B is width of stiffener in contact with skinplate.
2. 0.11*span
3. Centre to centre (c/c) distance of stiffeners.

Minimum of these three value should be selected for designing.

Combined stresses can be calculated with the help of following correlation:

$$\sigma = (\sigma_x^2 + \sigma_y^2 - \sigma_x\sigma_y + 3\tau_{xy}^2)^{1/2}$$

Where, σ = combined stresses.

σ_x = net stress in X-direction.

σ_y = net stress in Y-direction.

τ_{xy} = net shear stress in XY plane.

Calculated values of these stresses must be less than permissible values as specified in the code.

3.1.2 Design of horizontal girders

The number of horizontal girder depends upon height of gate. Number of horizontal girder should be minimum to simplify design, manufacturing and maintenance but stresses should be within permissible limit. As a general rule 3 horizontal girder shall be use for height less than 8.5m, for 8.5m to 12m number of horizontal girder shall be 3, and for more than 12m height 4 horizontal girder shall be use. In this study 3 horizontal girders are used.

Horizontal girders are designed as overhang beam supports by radial arms, but the supports that arm provides considered to be fixed or built-in support and not the hinge support. Hence beam become indeterminate and moment distribution method is used for analysis. Horizontal girders also checked for shear stress at the contact of horizontal girder and radial arm, this calculated shear stress should be less than permissible value.

3.1.3 Design of radial arms

Number of pair of arms is always equal to number of horizontal girder because two supports are required for each horizontal girder. Radial arm can be straight or inclined depending upon the site condition. In this study straight or parallel arms are used so there is no side thrust.

Radial arms are designed as columns under compression and total compressive stresses can be calculate according to code IS 800. After checking for bending and compressive stresses radial arm are check for unit formula.

$$\frac{fb}{Fb} + \frac{fc}{Fc} < 1$$

Where, fb = Induced bending stress.

Fb = Permissible bending stress.

Fc = permissible compressive stress.

fc = Induced compressive stress.

3.2 Modeling

ANSYS mechanical APDL has become a powerful tool for finite element analysis to solve enormous range of mechanical engineering applications. It has capabilities to solve problems ranging from simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis, in the fields of stress analysis, fluid and heat transfer, etc.

In this study a 3D model of radial gate is developed in ANSYS workbench design modeler. ANSYS toolbar contain various system templets. To start the project, drag the static structural templet into project schematic. This is also possible by double clicking at static structural templet. All items selected from toolbar are called system, and each system is divided into various cells. Static structural have following six cells.

1. Engineering data.
2. Geometry.
3. Model.
4. Setup.
5. Solution.
6. Results.

Engineering data cell is used to define a new material or select a material from ANSYS library for the analysis. It is possible to include more than one material at this stage and use them in analysis accordingly. New material is added to engineering data by defining its mechanical properties like yield and ultimate tensile and compressive strength, young's modulus of elasticity, poission's ratio etc.

Geometry cell is used for create, edit, import or update the model. Design modeler and space claim are two interface in ANSYS to develop a model. In this analysis design modeler is used for modelling. A 2D sketch is drawn using various curves and extruded to generate 3D solid. There are various tools in design modeler to generate 3D structure like Boolean, Pattern, Extrude etc.

Model cell have connections and meshing, and affects definition of geometry, mesh branches and coordinate systems. ANSYS provides various method of meshing like tetrahedral meshing, hexahedral meshing, surface meshing, beam meshing and mesh control may be local or global.

Setup cell is used to define loads and boundary conditions. The data from the previous steps is incorporated including connections. In this analysis water load and self-weight of radial gate is defined and supports are defined at trunnion and spillway crest.

Solution cell provides the access to solution branch of the application. It is also possible to share this solution data with downstream systems. Shearing of data become important when output for one analysis become the input for other analysis as in topology optimization.

Result cell shows the status and availability of the analysis result. It is not possible to share the data from this cell to any downstream system. This cell is used to read and print the results.

3.3 Topology optimization

ANSYS 18 have topology optimization system templet in its toolbox. Select this system templet and drag it to solution of static structural. It will import the geometry and results from previous analysis. Then define objective and apply controls so that manufacturing requirement are fulfilled. Right click on the result cell of topology optimization and select transfer to design validation system. The output of topology optimization become input for second static structural, where geometry modification and further analysis is performed.

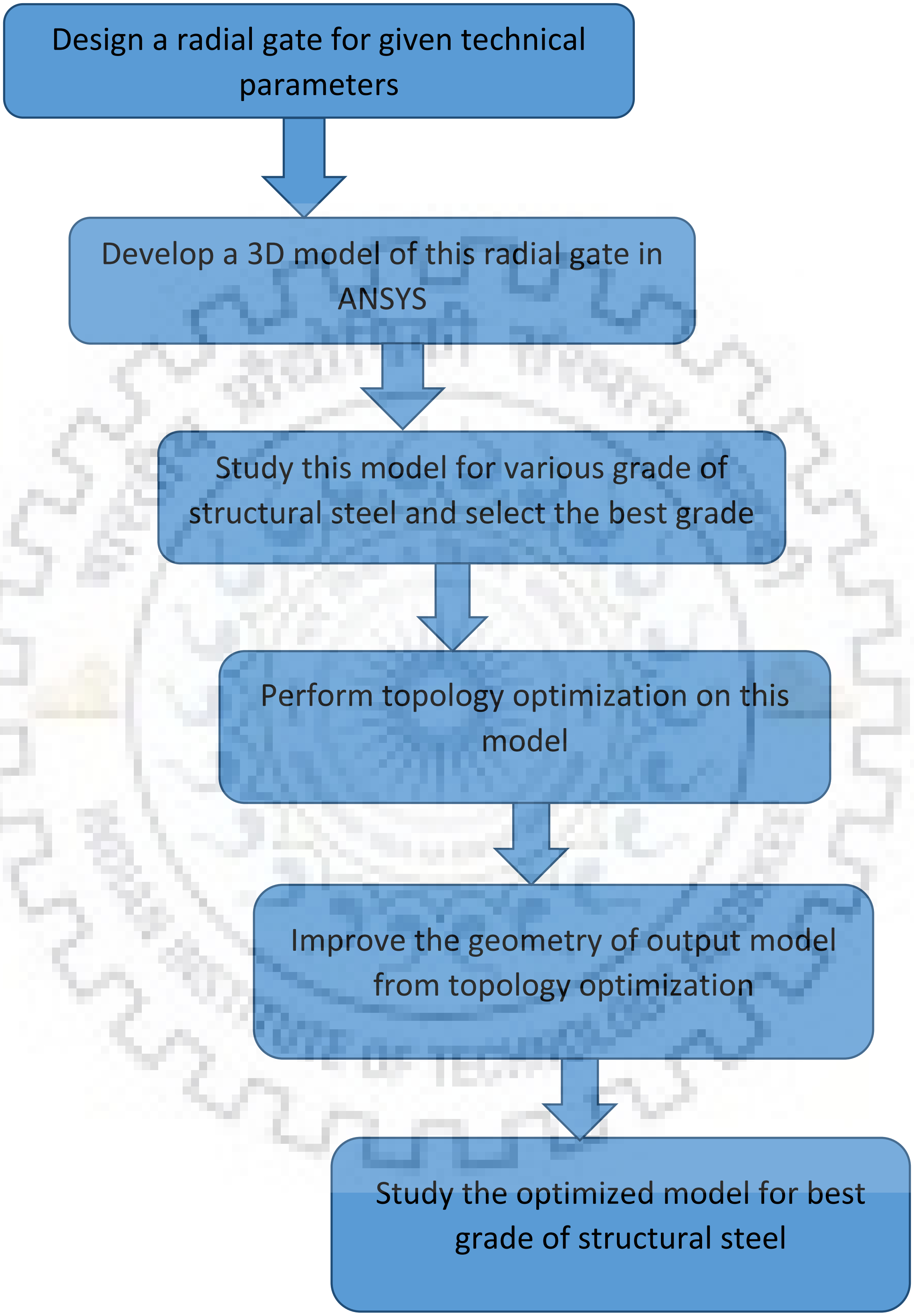

Figure 3.1 Methodology flow diagram

CHAPTER 4: DESIGN OF RADIAL GATE

A radial gate is designed as per conventional method according to guidelines provided in IS 4623. To design any component its cross section is assumed and stresses are calculated, these stresses should be less than permissible stresses given in code. If induced stresses exceeds the permissible stresses then modify the cross section and repeat the above process. Design data is taken from tehri dam.

In this analysis a radial gate is designed for following technical parameters.

Vent size =10.5m*15.5m

Radius of skin plate =18.75m

Full reservoir level =830m

Maximum water level =835m

Design head =15.25m

Trunnion level =823m

Accessibility - In-accessible

Water contact condition - Dry

4.1 Permissible stresses

As per IS 2026 material for skinplate, stiffeners, horizontal girder and arm girder is structural steel whose yield point (YP) is 250MPa or 2550 kg/cm^2.

As per IS 4623 the permissible stresses for dry and inaccessible condition can be calculate with the help of following correlations.

Bending stress (Tension and compression) = 0.45 YP = 1148 kg/cm^2

Direct stress (Tension and compression) = 0.45 YP = 1148 kg/cm^2

Shear stress =0.35 YP = 893 kg/cm^2

Combined stresses =0.60 YP = 1530 kg/cm^2

Bearing stresses =0.65 YP = 1658 kg/cm^2

4.2 Design of skinplate

The stresses in skinplate are calculated as continuous beam supported by vertical stiffeners for full reservoir level. Skinplate with variable thickness is used because height of gate in 15.5m and water load varies linearly with depth. The skinplat is considered having three different thickness. Arc length of bottom and middle portion is 5m each and arc length of top portion is 6m. Different thickness of skinplate for different portions is considered and tested for maximum load acting at that portion. In this study thickness of skinplate at bottom, middle and top portion are considered 14cm, 12cm and 10cm respectively. Corrosion resistance of 1.5mm is considered for skinplate.

Table 4.1 Design of skinplate

Skinplate thickness (cm)	14	12	10
Effective thickness (After considering corrosion allowances)	12.5	10.5	8.5
Section modulus (cm^3)	0.2604	0.1838	0.1204
Design head (m)	15.25	10.521	5.544
Load per cm length (kg)	1.525	1.0521	0.5544
Bending moment ($wl^2/12$) l=40.6 cm	209.47	144.52	76.15
Actual stress (kg/cm^2) (BM/section modulus)	804.416	786.28	632.47
Allowable stresses (kg/cm^2)	1148	1148	1148

Induced stresses at every section of skinplate is less than the permissible stresses, hence assumed thickness of skinplate are correct.

4.3 Design of vertical stiffeners

The vertical stiffeners are designed as a beam supported at horizontal girders. The section proposed for vertical stiffeners is T section. The loading is taken for 1m width of gate.

Sill reaction

When the gate is closed and resting on the sill there will be reaction sill due to self-weight of the gate in addition to water load.

Assuming gate weight = 80 T

Centre of gravity = 0.8 R

Where, R is the radius of skinplate.

Vertical reaction at sill = (80*0.8*18.75)/16.837

= 71.272 T

Sill reaction towards trunnion = 71.272*Cos(63.89)

= 31.36 T

Load per meter length = 31.36/10.5

= 2.987

Load per meter length is considered to be 3 T/m.

The load on vertical stiffener is calculated by adopting moment distribution method. There are two cases considered for designing of radial gate. Shear force diagram and bending moment diagram is drawn for both cases and maximum values are selected for designing.

1. The gate in just lifted position when water level is upto FRL.

   In this case vertical stiffener is considered as continuous beam supported by three horizontal girder. The sill reaction is zero in this case. Load diagram for this case is shown below.

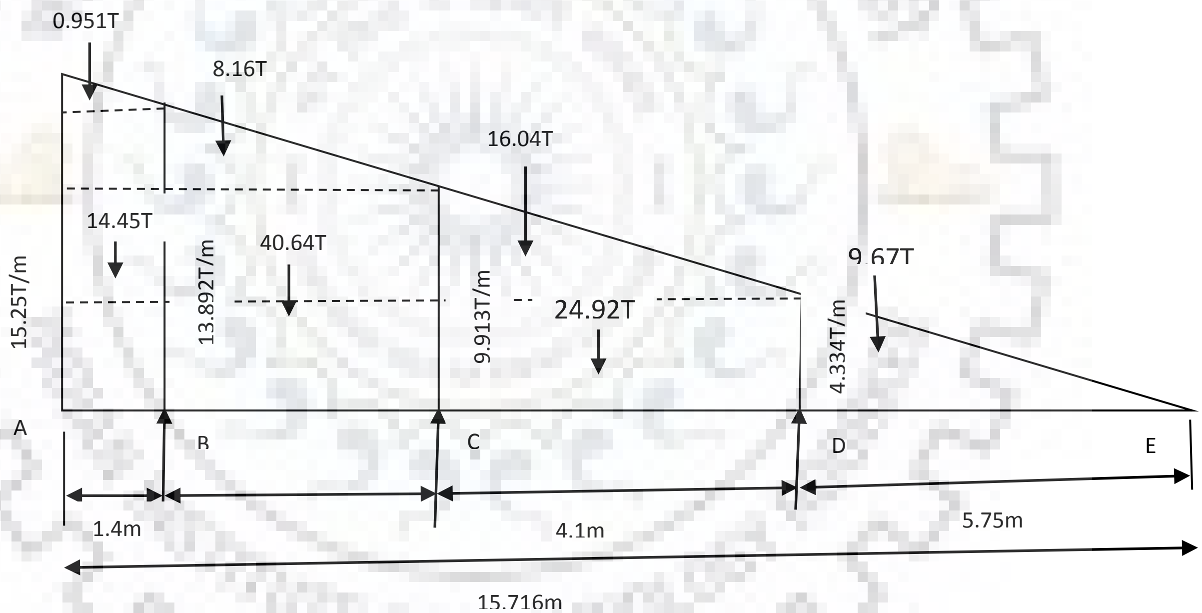

Figure 4.1 Load diagram

Vertical stiffeners acts like continuous beam with three supports. Continuous beams are indeterminate and reactions at supports can not be calculated by simple equation of equilibrium. Moment distribution method is applied to find out the reactions at supports. Shear force diagram is drawn for the vertical stiffener and maximum value of shear force is 24.555 T at support C.

Table 4.2 Moment distribution method Case 1

Member	BA	BC	CB	CD	DC	DE
Distribution factor	0	1	0.584	0.416	1	0
Fixed end moment	14.502	-17.231	16.116	-21.164	18.09	-14.407
Balance B & D	_ _	2.729	_ _	_ _	-3.683	_ _
Carry over	_ _	_ _	1.365	-1.842	_ _	_ _
Initial moments	14.502	-14.502	17.481	-23.006	14.407	-14.407
Distribute	_ _	_ _	3.227	2.298	_ _	_ _
Net moments	14.502	-14.502	20.708	-20.708	14.407	-14.407
Load due to moments	_ _	-1.514	1.514	1.096	-1.096	_ _
Reaction due to load	20.4	25.706	23.041	23.154	17.807	9.678
Load		44.646 T/m		48.805 T/m		26.389 T/m

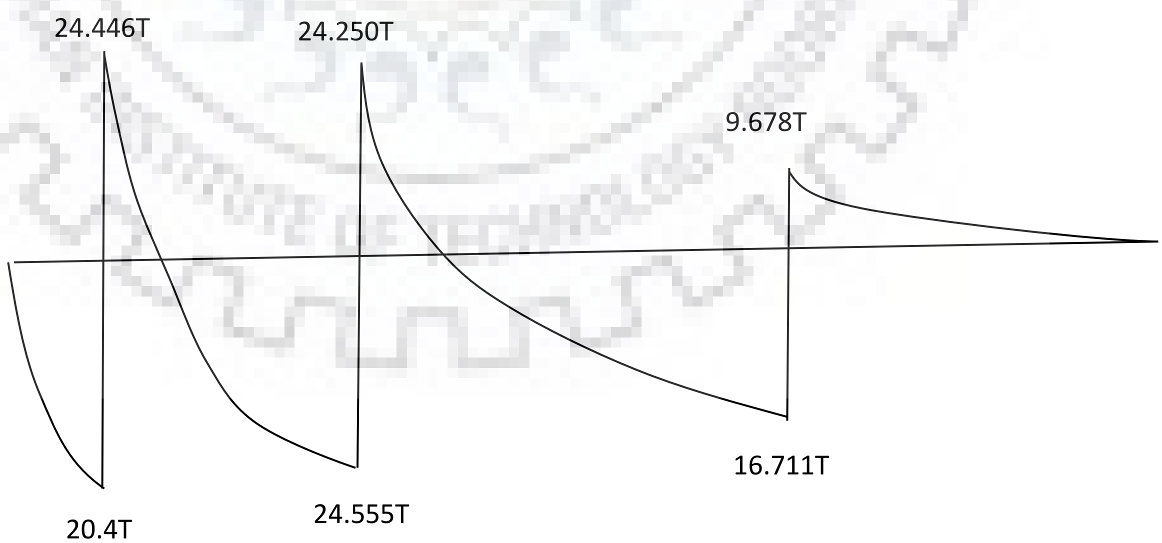

Figure 4.2 Shear force diagram for case 1

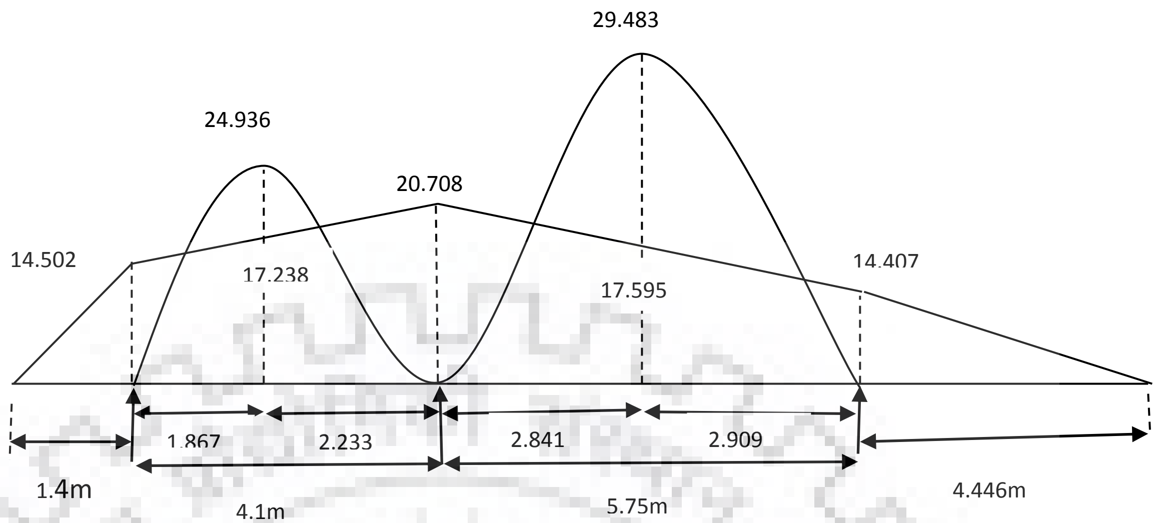

Figure 4.3 Bending moment diagram for case 1

2. The gate resting on the sill when water level is upto FRL

   When gate is resting on the sill it will provide a normal reaction and a component of this reaction will act towards trunnion.

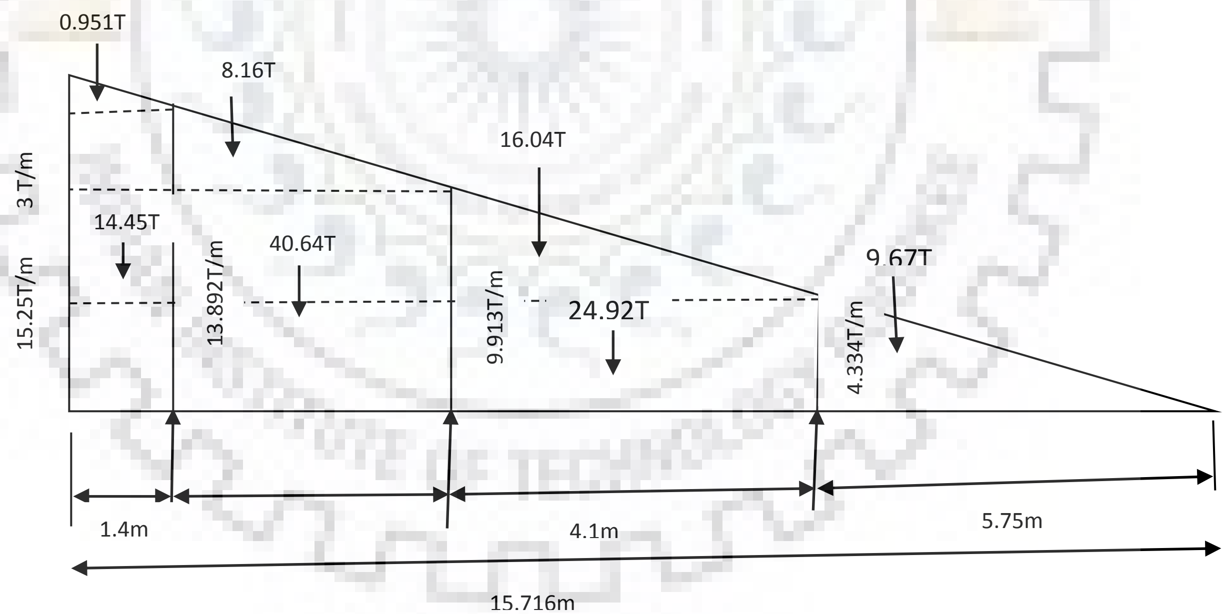

Figure 4.4 Load diagram for case 2

Moment distribution method is applied to calculate the reaction. Shear force diagram and bending moment diagram is drawn for case 2. Maximum values of shear force and bending moment is considered for designing.

Table 4.3 Moment distribution method for case 2

Member	BA	BC	CB	CD	DC	DE
Distribution factor	0	1	0.584	0.416	1	0
Fixed end moment	18.702	-17.231	16.116	-21.164	18.09	-14.407
Balance B & D	_ _	-1.471	_ _	_ _	-3.683	_ _
Carry over	_ _	_ _	-0.736	-1.842	_ _	_ _
Initial moments	18.702	-18.702	15.380	-23.006	14.407	-14.407
Distribute	_ _	_ _	4.454	3.172	_ _	_ _
Net moments	18.702	-18.702	19.834	-19.834	14.407	-14.407
Load due to moments	_ _	-0.276	0.276	0.944	-0.944	_ _
Reaction due to load	23.4	25.706	23.041	23.154	17.807	9.678
Load		48.884 T/m		47.415 T/m		26.541 T/m

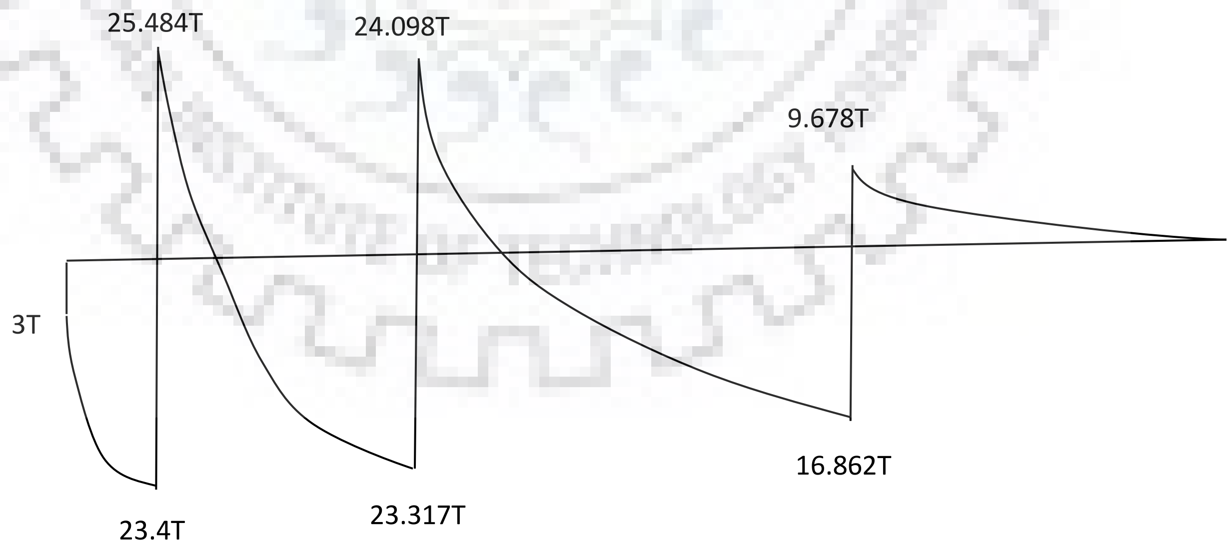

Figure 4.5 Shear force diagram for case 2

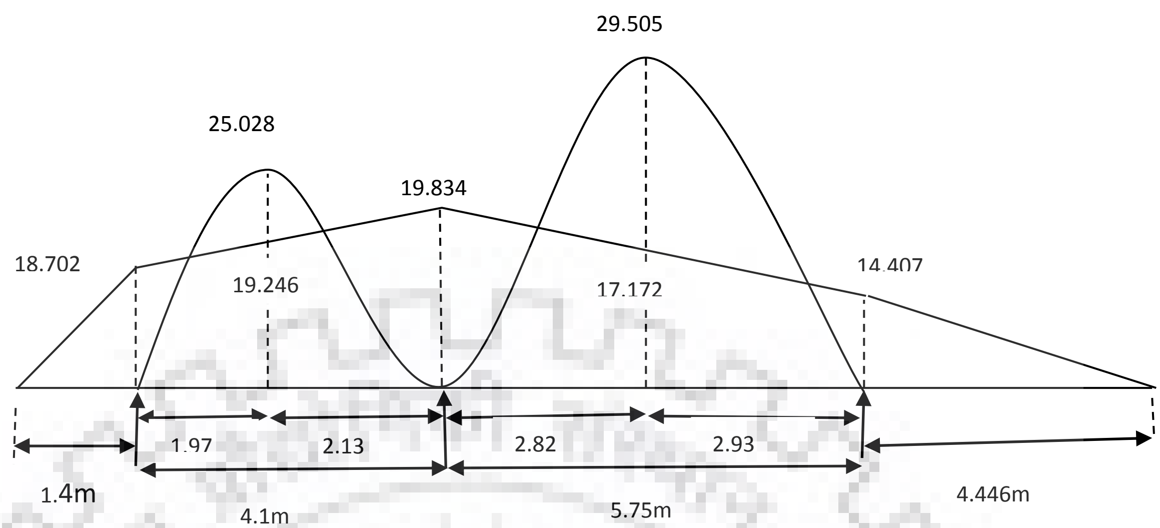

Figure 4.6 Bending moment diagram for case 2

Bending moment by water load

Maximum bending moment for stiffener with 14cm skinplate

= 18.702 Tm/m

Maximum bending moment for stiffener with 12cm skinplate

= 20.708 Tm/m

Maximum bending moment for stiffener with 10cm skinplate

= 14.407 Tm/m

Stiffener with 14 cm thick skinplate

Coacting width for stiffeners

1. 40t+B = (40*1.25)+1 = 51cm
2. 0.11 span = 0.11*406 = 44.66cm
3. c/c of girder = 17.5+(38.5/2) = 36.75cm

Adopt 30cm because minimum of these three must be selected.

Stiffener with 12 cm thick skinplate

Coacting width of stiffener

1. 40t+B = (40*1.05)+1 = 43cm
2. 0.11 span = 0.11*406 = 44.66cm
3. c/c of girder = 38.5cm

Adopt 38.5cm because minimum of these three must be selected.

Stiffener with 10 cm thick skinplate

Coacting width of stiffener

1. 40t+B = (40*.85)+1 = 35cm
2. 0.11 span = 0.11*406 = 44.66cm
3. c/c of girder = 38.5cm

Adopt 35cm because minimum of these three must be selected.

Vertical stiffener for large gate have T section.

A Cross section is assumed for vertical stiffener with following properties.

Area = 105.251 cm^2

I_{xx} = 14174.755 cm^4

Y_t = 9.785 cm

Y_c = 14.36 cm

Z_t = 1448.62 cm^3

Z_c = 987.01 cm^3

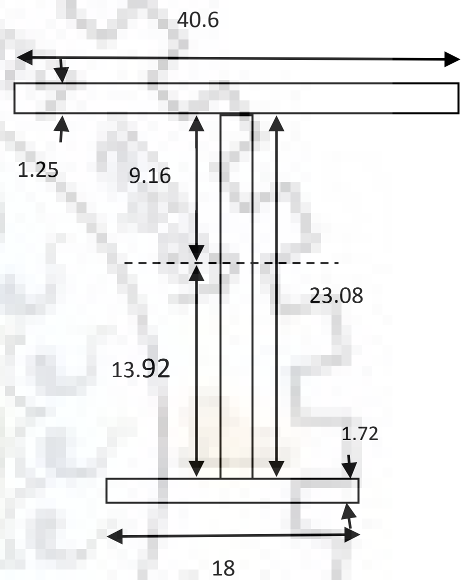

Figure4.7 Cross section of vertical stiffener

By water load

Maximum shear force for 14cm thick plate = 25.484 T/m

Maximum shear force for 12cm thick plate = 24.555 T/m

Maximum shear force for 10cm thick plate = 16.863 T/m

Maximum shear force = 11391 kg

Shear stress on web = shear force/area of web

= 11391/(23.08*1.02)

= 483.866 kg/cm^2

= 893 kg/cm^2

Maximum induced stress is less than the permissible stress of the material. Hence the design is safe and the assumed cross section is suitable for vertical stiffeners.

4.4 Design of horizontal girders

Horizontal girders are design as a beam having point loads at the location of stiffeners and supported by arms. Total 25 stiffeners are considered at uniform distance of 406mm between girders. Radial arm supports the horizontal girder and distance between two arms is 6300mm. The overhanging portion of girder have length of 2100mm on both sides. Load on horizontal girder is maximum when gate is resting on the sill.

Load on horizontal girder = 48.884 T/m

Load on each stiffener = 48884*0.385 kg

= 18820.34 kg

Total load on horizontal girder = 18820.34*25

= 470508.5 kg

Shear force will be zero at supports and at the mid-point of horizontal girder so bending moment will be maximum at one of these two locations.

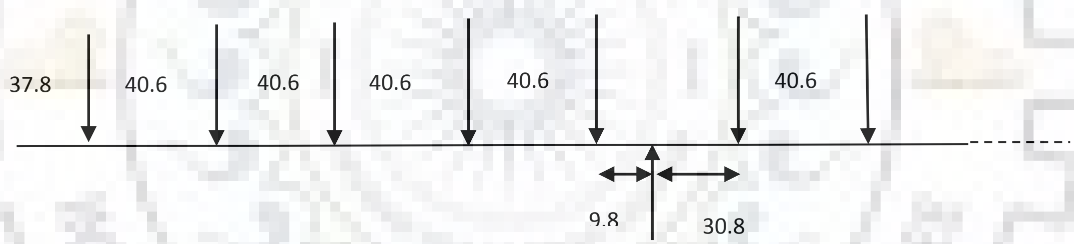

Figure 4.8 Load diagram for horizontal girders

Reaction at each support = 235254.25 kg

Bending moment at support = 18820.34*(172.2+131.6+91+50.4+9.8)

= 8563254.7 kg.cm

Bending moment at centre = 235254.25*315 – (18820.34*(487.2+446.6+406+356.4+324.8+284.2+243.6+203+162.4+121.8+81.2+40.6))

= 14674215.95 kg.cm

Maximum shear force = 18820.34*5

= 94101.7 kg

Proposed section for horizontal girder is shown in the figure.

Area = 412 cm^2

I_{xx} = 1175849 cm^4

Y = 75 cm

Z = 15678 cm^3

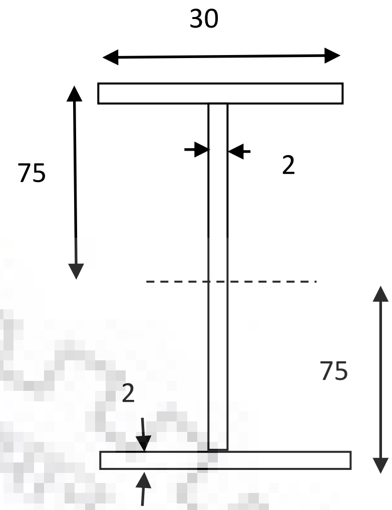

Figure 4.9 Cross section for horizontal girders

Assumed cross section is symmetric so bending stress at top and bottom portion are same.

Bending stresses (Tensile and compressive) = 14674215.95/15678

= 935.97 kg/cm^2

< 1148 kg/cm2

Hence, this cross section is safe for horizontal girders.

4.5 Design of radial arm

Length of radial arm = 1560 cm

Load on arm girder = 235254.25 kg

Anticlockwise cantilever moment at support = 8563254.7 kg.cm

Clockwise fixed end moment at support = $P(ab^2/L^2)$

$=(18820.34/630^2)$

$[(30.8*599.2^2)+(71.4*558.6^2)+(112*518^2)+(152.6*477.4^2)+(193.2*436.8^2)+(233.8*396.2^2)+(274.4*355.6^2)+(315*315^2)+(355.6*274.4^2)+(396.2*233.8^2)+(436.8*193.2^2)+(477.4*152.6^2)+(518*112^2)+(558.6*71.4^2)+(599.2*30.8^2)]$

= 323605711.8 kg.cm

= 153.448 T.m

The proposed section and its properties are as below:

Area = 334.2 cm^2

Ixx = 240251 cm^4

Y = 34.2 cm

Z = 7025 cm^3

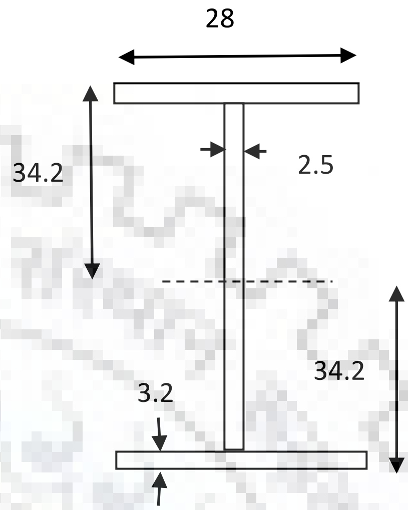

Figure 4.10 Cross section for radial arm

Stiffener factor = 3EI/L

In this case 3E is constant so stiffener factor is considered as I/L.

Stiffener factor for arm = 240251/1560 = 154.007

Stiffener factor for girder = 1175849/630 = 1866.43

Distribution factor for arm = 0.076

Distribution factor for girder = 0.923

Moment distribution method is applied to find clockwise moment on the arm.

Table 4.4 Moment distribution for radial arms

.076	.923	
11.662	-153.448	141.632
5.534	-72.816	65.363
2.487	-32.681	30.165
1.146	-15.082	13.92
0.529	-6.96	6.424

0.244	-3.212	2.964
0.133	-1.482	1.367
0.052	-0.683	0.630
0.024	-0.315	0.29
0.011	-0.145	0.134
.005	-0.067	0.061
.002	-0.035	0.028
.001	-0.014	.013
+21.83	-23.983	

Clockwise moment on arm = 23.983 Tm = 2398300 kg.cm

Anticlockwise cantilever moment = 8563254.7*.076

= 650807.35

Net moment = 1747492.634 kg.cm

Bending stresses = 1747492.634/7025

= 248.75 kg/cm^2

< 1102

Compressive stresses = 235254.25/334.2 = 703.93 < 1102

Check for unit formula

$\frac{fb}{Fb} + \frac{fc}{Fc}$ = (248.75/1102) + (703.93/1102) = 0.864 < 1

Hence the design is safe and assumed cross section is suitable for radial arm.

CHAPTER 5: FEM MODEL OF RADIAL GATE

Any simulation project in ANSYS starts with creation of geometric model. It is most time consuming process in 3D simulation. It is possible to generate a 3D complex geometry by applying boolean operations on simple 3D bodies. To generate any 3D body, a 2D sketch is drawn in a plane and then use tools like <Extrude>, <Sweep> and <Revolve>.

To start the project drag static structural from toolbox to project schematic. There are six cell in static structural. To edit any cell double click on that cell, and editing is possible at any stage of analysis. It is not possible to skip any cell of static structural.

5.1 Engineering data

Double click on engineering data cell to select or define the material for the analysis. In this analysis three grade of structural steel are defined for analysis.

ASTM A 36 – Carbon steel.

ASTM A 992 – High strength low alloy steel.

ASTM A 588 – Corrosion resistant high strength low alloy steel.

The grade of structural steel are defined with the help of various mechanical properties like density, strength, young's modulus of elasticity etc.

Table 5.1 Properties of different grades of structural steel

Properties	ASTM A 36	ASTM A 992	ASTM A 588
Density (kg/m^3)	7800	7850	7750
Young's modulus of elasticity (GPa)	200	200	205
Poission's ratio	0.26	0.3	0.28
Yield tensile strength (MPa)	250	345	345
Ultimate tensile strength (MPa)	450	450	483

5.2 Geometry

ANSYS provides two tools to generate a geometry, design modeler and spaceclaim. Right click on geometry cell and choose one of them. In this analysis design modeler is used to generate geometry. Design modeler is basic software for modeling and do not support many tool like copy and paste. On the other hand spaceclaim is advance modeler and have better graphics user interface (GUI) [13].

5.2.1 Skinplate

To draw the cross section of skinplate YY-Plane is selected and cross section is drawn with arc and lines. An arc of radius 18750mm and origin as centre is drawn corresponding to height of gate. The arc length of this arc is 16m and it makes an angle of 58.8922^0 at the origin. Then three arc of radius 18890mm, 18870mm and 18850mm are drawn from the same centre and arc length are 5000mm, 5000mm and 6000mm respectively. These arc are joined with straight line to form a close loop which is necessary to extrude a sketch. Now this sketch is extruded to get the width of skinplate.

Table 5.2 Details of extrude1

Geometry	sketch1
Operation	Add frozen
Direction vector	Normal
Direction	Both – symmetric
Extend type	Fixed
Depth	5250mm

5.2.2 Vertical stiffener

Vertical stiffener is a beam of T shape cross section bended in a circular arc. T section is generated by extruding sketc2 and sketch3 for depth of 5.1mm and 90mm respectively. Web of T section is obtained by extruding sletch2 and flange is obtain by extruding sketch3. Boolean1 is used to join web and flange and to generate the stiffeners.

Table 5.3 Details of extrude7

Geometry	sketch2
Operation	Add frozen
Direction vector	Normal
Direction	Both – symmetric
Extend type	Fixed
Depth	5.1mm

Table 5.4 Details of extrude9

Geometry	sketch3
Operation	Add frozen
Direction vector	Normal
Direction	Both – symmetric
Extend type	Fixed
Depth	90mm

Sketch2 and sketch3 are drawn in XY-Plane therefore this stiffener is at the centre of skinplate. To get other stiffener pattern1 and pattern2 is used on either side. Details of pattern1 and pattern2 are same but direction for pattern1 is +Z and for pattern2 the direction is –Z.

Table 5.5 Details of pattern1 and pattern2

Pattern type	Linear
Direction	Plane normal
Offset	406mm
Copies	12

5.2.3 Horizontal girders

Horizontal girder have I cross section which have two web and one flange. Sketch6 and sketch8 are extruded to generate flanges and sketch7 is extruded to generate web of I section beam.

Sketch6, sketch7 and sketch8 are extruded by extrude10, extrude11 and extrude13 respectively. Boolean2 is used for combining these three bodies and generate I shape bottom girder.

Table 5.6 Details of extrude10

Geometry	sketch6
Operation	Add frozen
Direction vector	Normal
Direction	Both – symmetric
Extend type	Fixed
Depth	5250mm

Table 5.7 Details of extrude11

Geometry	sketch7
Operation	Add frozen
Direction vector	Normal
Direction	Both – symmetric
Extend type	Fixed
Depth	5250mm

Table 5.8 Details of extrude13

Geometry	sketch8
Operation	Add frozen
Direction vector	Normal
Direction	Both – symmetric
Extend type	Fixed
Depth	5250mm

Pattern tool is used to generate middle and top girder. Pattern3 is used for middle girder and pattern5 for top girder. Centre bracing between top and bottom girder is generated by extruding sketch22. Sketch22 is extruded by extrude27. Pattern19 and pattern20 is used to generating bracing between middle and top girder. Bracing between bottom and middle girder is generated by extruding sketch23. Sketch23 is extruded by extrude28. Sketch23 is drawn in plane12.

Table 5.9 Details of pattern3

Pattern type	Circular
Axis	Plane normal
Angle	12.529^0
Copies	1

Table 5.10 Details of pattern5

Pattern type	Circular
Axis	Plane normal
Angle	17.521^0
Copies	1

Table 5.11 Details of extrude27

Geometry	sketch22
Operation	Add frozen
Direction vector	Normal
Direction	Both – symmetric
Extend type	Fixed
Depth	25mm

Table 5.12 Details of pattern19 and pattern20

Pattern type	Linear
Direction	Plane normal
Offset	2730mm
Copies	1

Table 5.13 Details of extrude28

Geometry	sketch23
Operation	Add frozen
Direction vector	Normal
Direction	Both – symmetric
Extend type	Fixed
Depth	25mm

Table 5.14 Details of plane12

Type	From plane
Base plane	XYPlane
Transform1	Offset Z
Value1	2730mm

5.2.4 Radial Arms

Radial arm have I section and have an offset 3150mm in +Z and –Z direction. A contact plate is the part of arm which is in direct contact with horizontal girder. A new plane is defined to draw the section of radial arm. Web and contact plate is drawn in plane7 and right side flange in plane8. Circular end and bracing of arm are also drawn in plane7. Sketch10 is extruded by extrude18 to generate contact plate. Sketch12 is extruded by extrude20 to get web of bottom arm. Sketch15 and sketch16 is extruded by extrude23 and extrude24 respectively. Sketch20 and sketch21 are extruded by extrude25 and extrude26 to generate bracing of arm. Sketch13 is extruded by extrude21 to get one flange of I section and another flange is generated by pattern7. Pattern8 and pattern9 is used to generate middle and top arm respectively. Pattern11 and pattern12 is used to generate, left side arm and its circular end. Pattern14 is used for bracing of left arm. Boolean17 and Boolean18 is used to finally generate right and left arm respectively.

Table 5.15 Details of plane7

Type	From plane
Base plane	XYPlane
Transform1	Rotation about Z
Value1	-23.054^0

Table 5.16 Details of plane8

Type	From plane
Base plane	Plane7
Transform1	Offset Z
Value1	326mm

Table 5.17 Details of extrude18

Geometry	sketch10
Operation	Add frozen
Direction vector	Normal
Direction	Both – symmetric
Extend type	Fixed
Depth	400mm

Table 5.18 Details of extrude20

Geometry	sketch12
Operation	Add frozen
Direction vector	Normal
Direction	Both – symmetric
Extend type	Fixed
Depth	310mm

Table 5.19 Details of extrude21

Geometry	sketch13
Operation	Add frozen
Direction vector	Normal
Direction	Both – symmetric
Extend type	Fixed
Depth	16mm

Table 5.20 Details of extrude23

Geometry	sketch15
Operation	Add frozen
Direction vector	Normal
Direction	Both – symmetric
Extend type	Fixed
Depth	400mm

Table 5.21 Details of extrude24

Geometry	sketch16
Operation	Add frozen
Direction vector	Normal
Direction	Both – symmetric
Extend type	Fixed
Depth	400mm

Table 5.22 Details of extrude25

Geometry	sketch20
Operation	Add frozen
Direction vector	Normal
Direction	Both – symmetric
Extend type	Fixed
Depth	250mm

Table 5.23 Details of extrude26

Geometry	sketch21
Operation	Add frozen
Direction vector	Normal
Direction	Both – symmetric
Extend type	Fixed
Depth	250mm

Table 5.24 Details of pattern7

Pattern type	Linear
Offset	652mm
Copies	1

Table 5.25 Details of pattern8

Pattern type	Circular
Angle	12.529^0
Copies	1

Table 5.26 Details of pattern9

Pattern type	Circular
Angle	17.571^0
Copies	1

Table 5.27 Details of pattern11, pattern12 and pattern14

Pattern type	Linear
Offset	6300mm
Copies	1

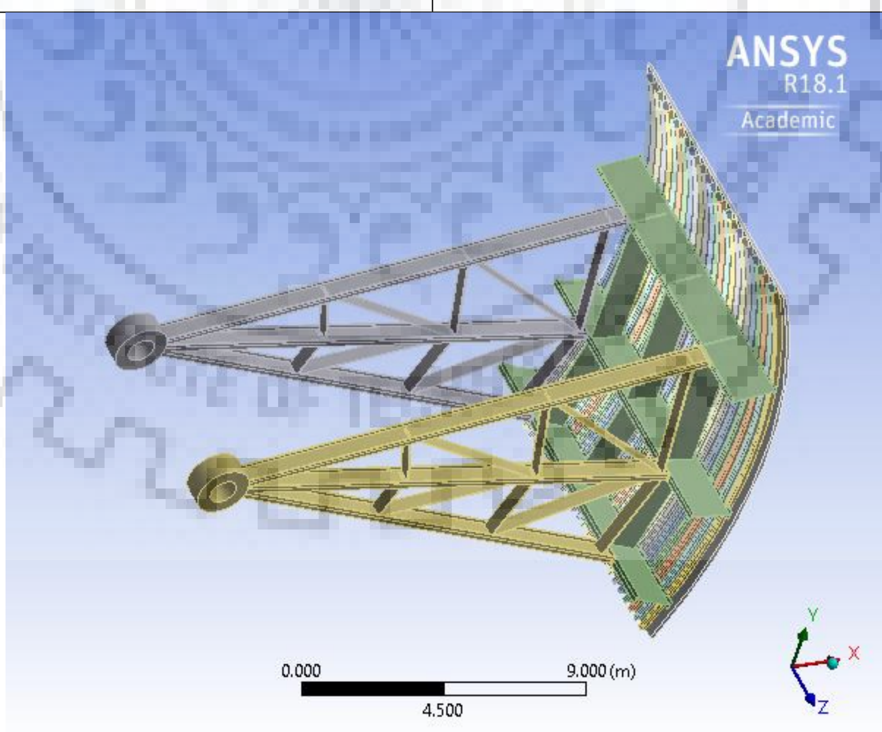

Figure 5.1 Model of radial gate

5.3 Model

Meshing is the process of divide the model into small finite element. In this analysis adaptive size function is used for meshing. Initially element size is default and latter sizing is inserted for each component. Element size for skinplate and vertical stiffener is 50mm and for horizontal girder is 75mm. The number of elements are 461450 and number of nodes are 1174643. Minimum aspect ratio is 1.1633, maximum aspect ratio is 81.353, average is 4.4165 and standard deviation is 3.8285. Transmission ratio is 0.272 and growth rate is 1.2.

5.4 Setup

In this step all the boundary conditions are defined. Standard earth gravity is defined for all bodies in –Y direction of global coordinate system. Frictionless support is provided at the bottom face of skinplate. Cylindrical support is provided at both circular end of radial arm with radial and axial component fixed and tangential component free. Water load is applied on skinplate with fluid density 1000kg/m^3, hydrostatic acceleration 9.81m/s^2 in Y direction and Y coordinate of free surface is 6.75m.

5.5 Solution

For this analysis normal stress, shear stress, von-mises stress or equivalent stress, normal elastic strain and total deformation is inserted into solution. Von-mises stress is widely used in model testing. Von-mises stress is based on maximum distortion energy theory and its value should be less than yield strength of material.

5.6 Topology optimization

In topology or topological optimization, physics of problem and FEM combined together to get optimum shape for given loads and constraints. ANSYS provides tools for mass, stress and strain optimization. Topology optimization provides the shape which are optimal than other engineering method with trial and error method. Generally topology optimization provides irregular shapes which are difficult to manufacture, but advancement in additive manufacturing allows to manufacture these complex shapes.

In this analysis topology optimization is performed on the model to reduce its weight. ANSYS do not support topology optimization for inertial and fluid solid interface loads. ANSYS performs mass optimization and not the weight optimization. For topology optimization standard gravity is deleted from the model. Effect of weight is considered in next step that is design validation. Topology optimization is performed with a convergence accuracy of 0.1% and optimality criteria solver type. All the regions, where boundary conditions are applied are excluded from topology optimization.

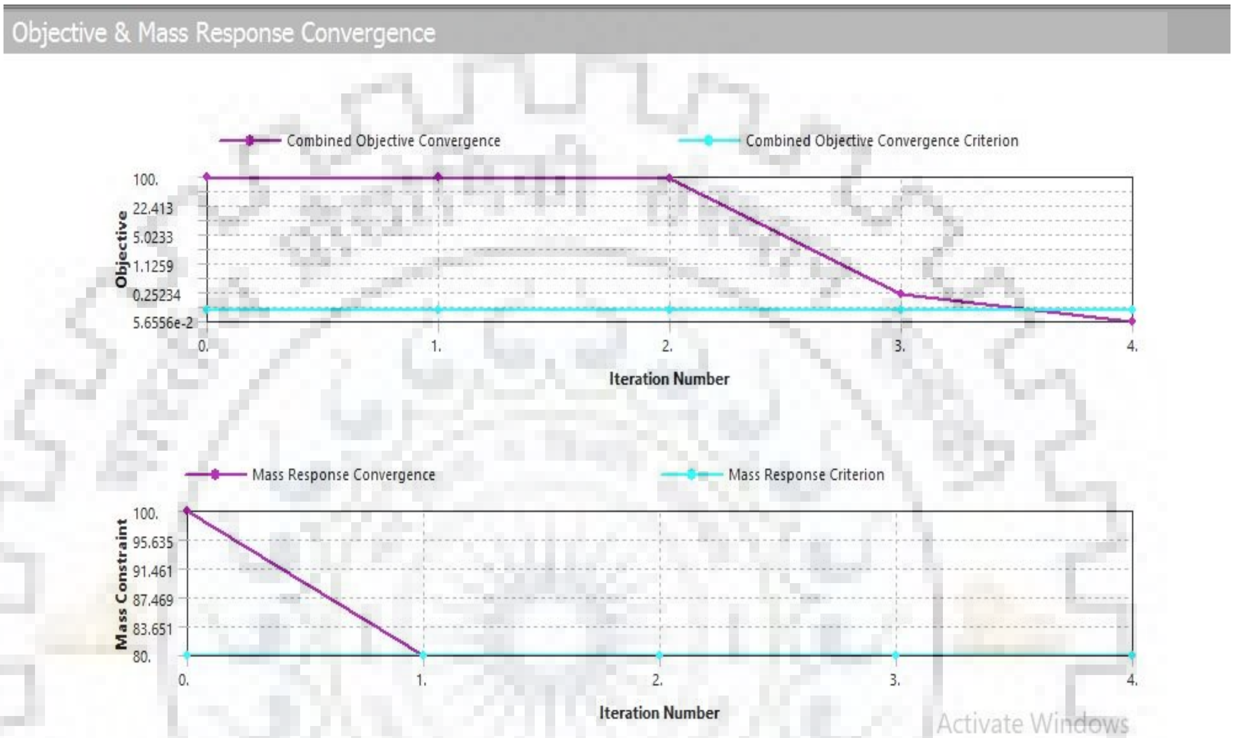

Figure 5.2 Objective and mass response convergence

The shape provided by topology optimization is irregular and difficult to manufacture. The result of topology optimization is transferred to design validation system. Right click on result cell and click on “Transfer to design validation system” this opens a new static structural. To transfer the result from topology optimization to new static structural update result of topology optimization and geometry of new static structural.

This geometry is modified using space claim as it provides better tools to edit a 3D geometry than design modeler. Standard gravity again inserted to consider the effect of self-weight of structure for further analysis. Design validation is done for this model with same loading and boundary conditions. A comparison is drawn between the models before and after the topology optimization for ASTM A992 grade of structural steel.

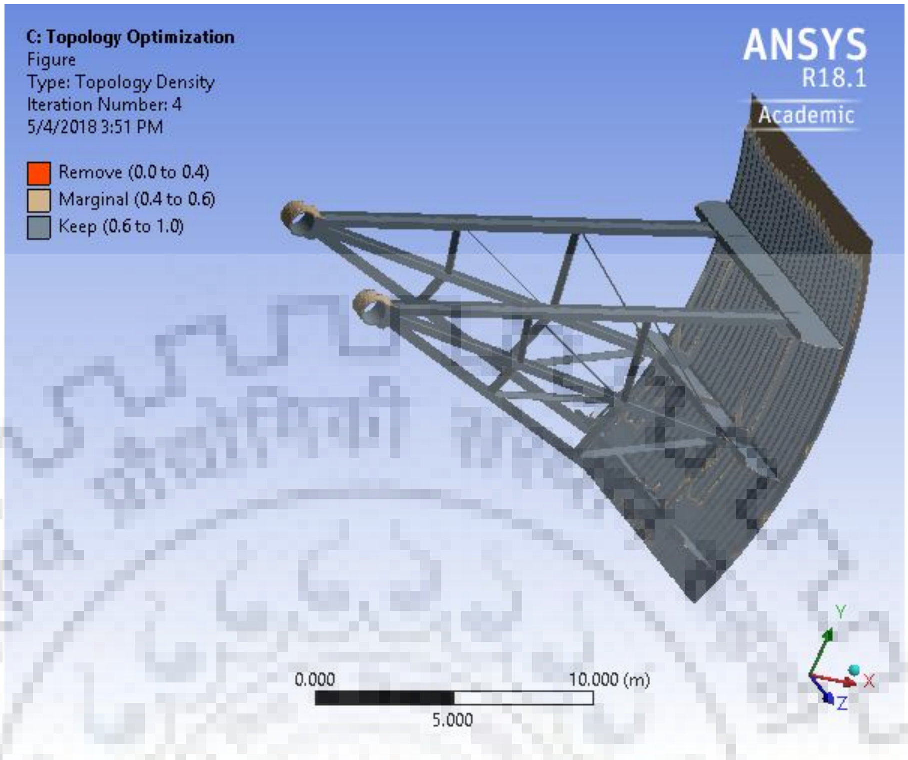

Figure 5.3 Geometry after topology optimization

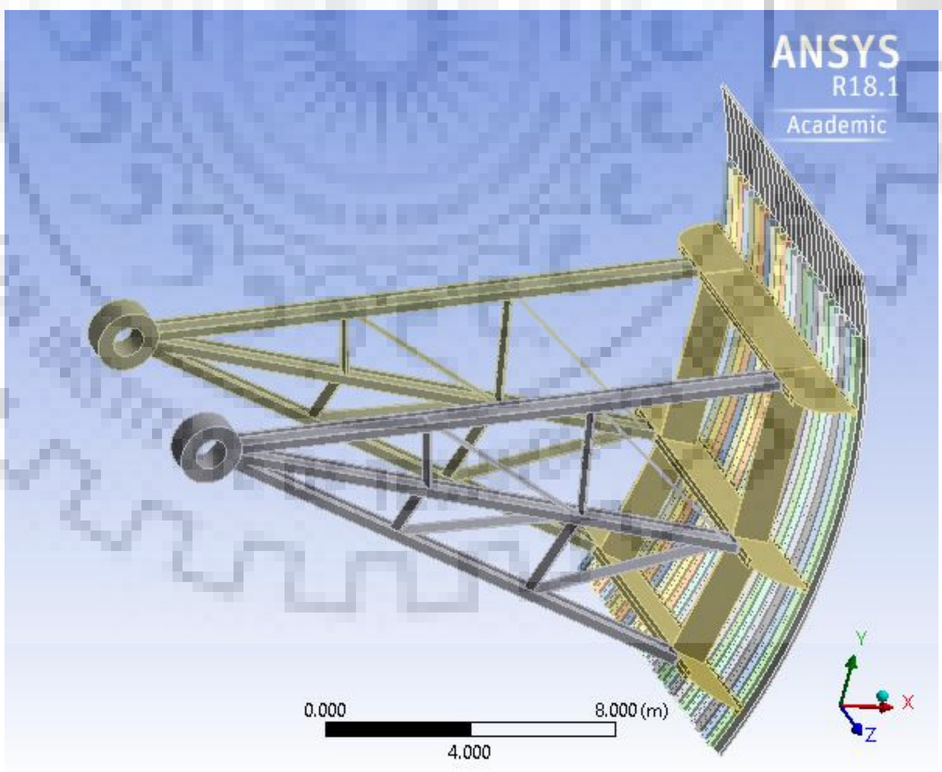

Figure 5.4 Modified geometry after topology optimization

CHAPTER 6: RESULTS AND DISCUSSIONS

A radial gate is designed for given technical parameters. A 3D model of this radial gate is developed design modeler of ANSYS workbench. Finite element analysis is performed on the model of radial gate using ANSYS. The study is performed on three grades of structural steel. ANSYS provides values of normal stress, shear stress, equivalent stress, normal elastic strain and total deformation for each grade of structural steel. Analysis shows that ASTM A992 is best grade of structural steel for radial gates. Maximum and minimum values of various parameters provided by ANSYS are given in the table 6.1 and in figure 6.1 to 6.15.

Table 6.1 Results for various grades of structural steel

		ASTM A 36	ASTM A 992	ASTM A 588
Normal stress(MPa)	Maximum	62.313	72.313	66.196
	Minimum	-129.73	-140.39	-133.97
Shear stress(MPa)	Maximum	84.378	84.1	84.22
	Minimum	-29.37	-29.05	-29.25
Von-mises stress(MPa)	Maximum	203.86	199.61	201.89
	Minimum	.000143	.000763	.000381
Normal elastic strain	Maximum	.00026	.000255	.0002541
Total deformation(mm)	Maximum	29.95	29.59	28.83
Yield strength (MPa)		250	345	340

Von-mises stresses are minimum for ASTM A 992. Though there is not much difference in the stresses induced for different grades of structural steel but yield strength for ASTM A 992 is 345 MPa which is much higher as compare to that of ASTM A 36 (250MPa). So ASTM A 992 is selected for topology optimization. Topology optimization is performed on the same model using ANSYS. After topology optimization weight of gate is reduced by 14.15% and stresses are within permissible limits. The results of topology optimization are shown in the table 6.2 and shown in figure 6.15 to 6.20.

Table 6.2 Results of topology optimization

		Before topology optimization	After topology optimization
Volume (m3)		32.281	27.712
Mass (kg)		$2.5341*10^5$	$2.1754*10^5$
Normal stress (MPa)	Maximum	73.313	133.25
	Minimum	-140.39	-240.44
Shear stress (MPa)	Maximum	84.1	93.36
	Minimum	-29.05	-37.46
Von-mises stress (MPa)	Maximum	199.61	272.37
	Minimum	.000763	.00707
Normal elastic strain	Maximum	.000255	.000596
Total deformation(mm)	Maximum	29.59	33.262

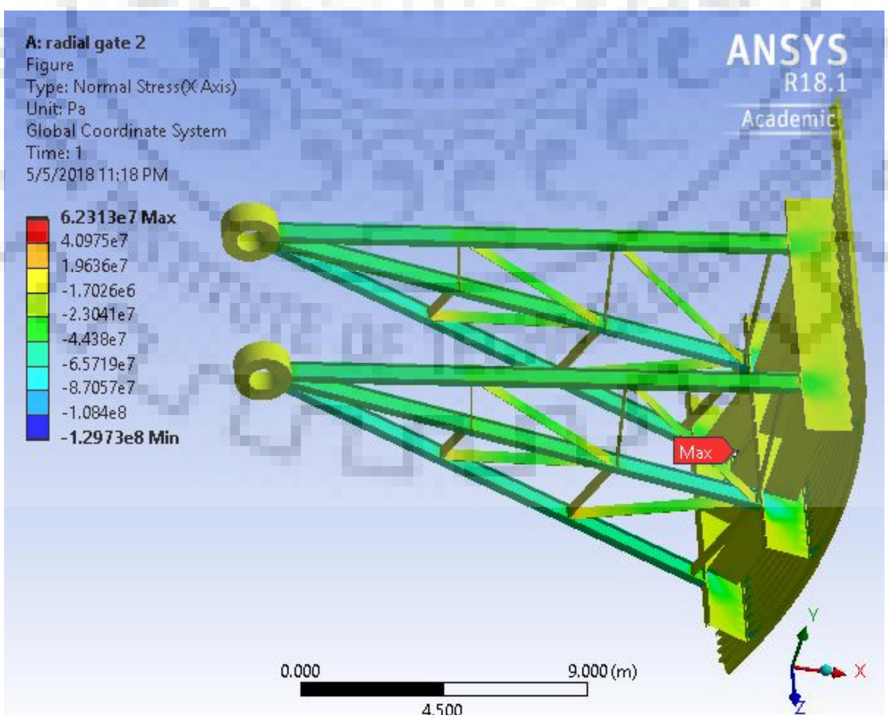

Figure 6.1 Normal stress for ASTM A 36

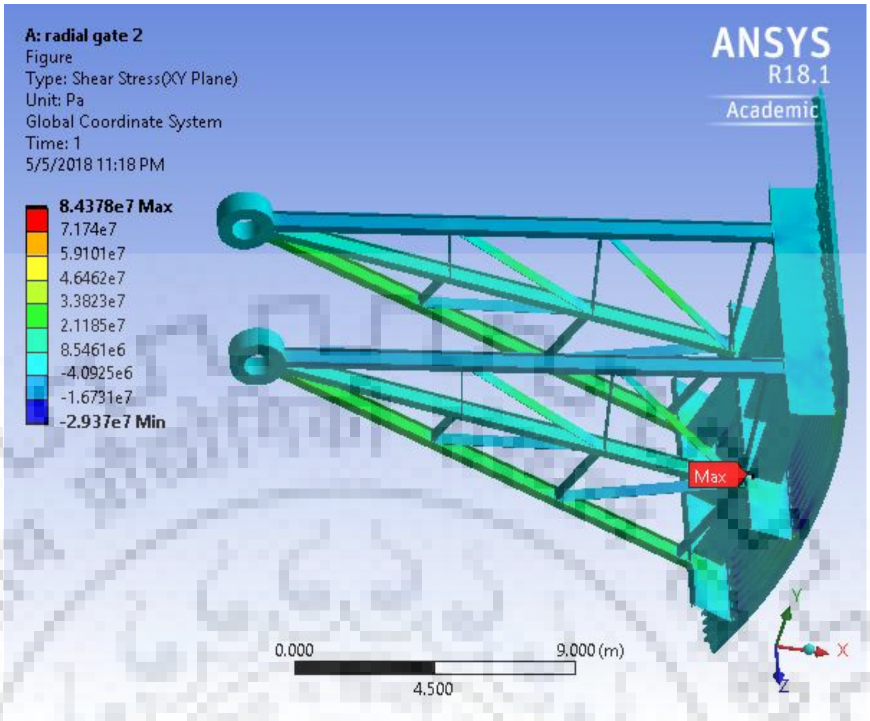

Figure 6.2 Shear stress for ASTM A 36

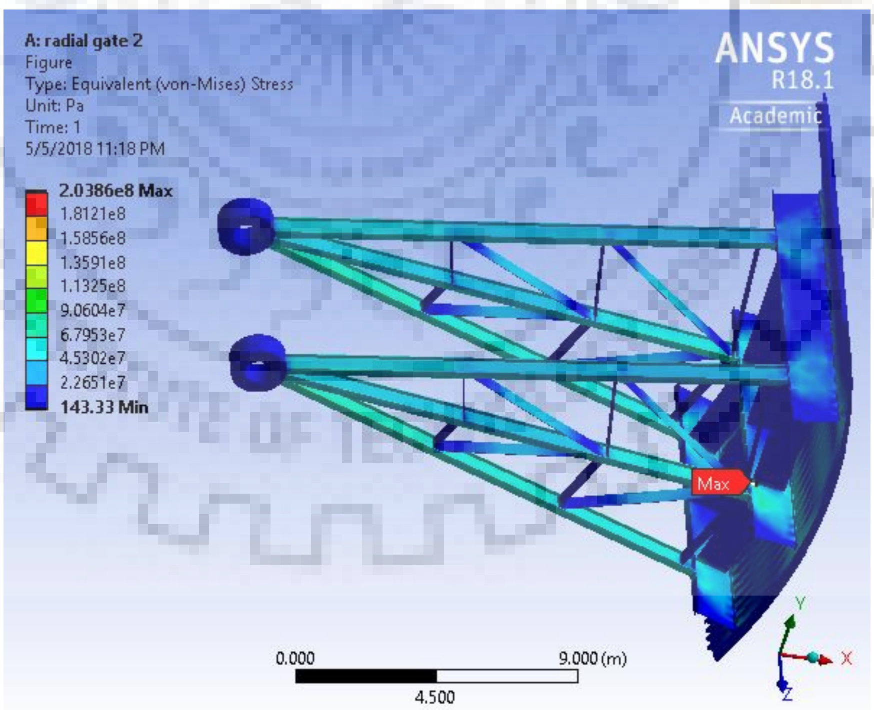

Figure 6.3 Von-mises stress for ASTM A 36

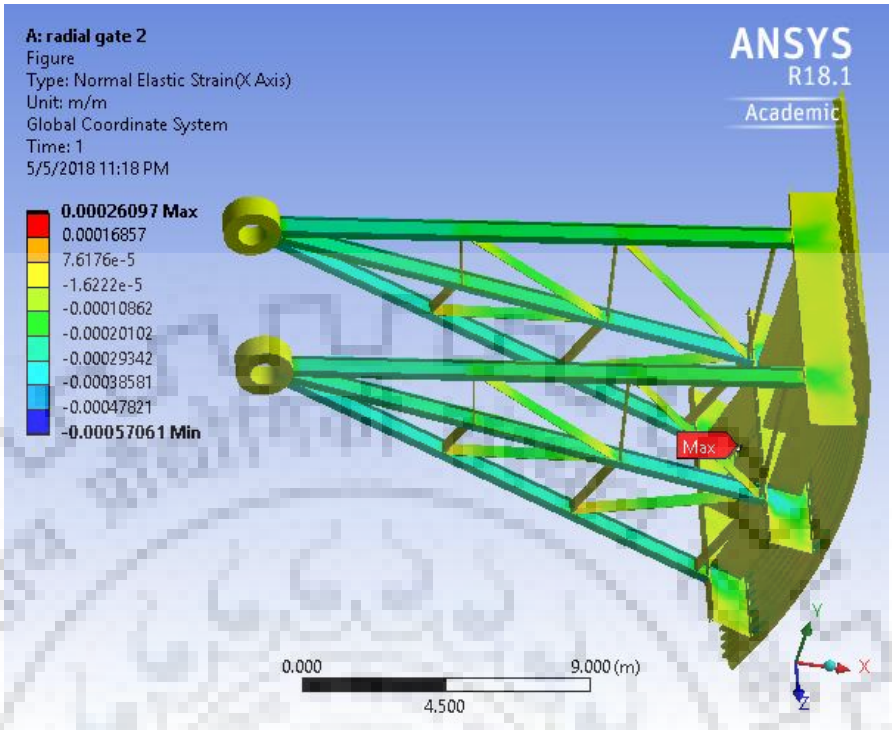

Figure 6.4 Normal elastic strain for ASTM A 36

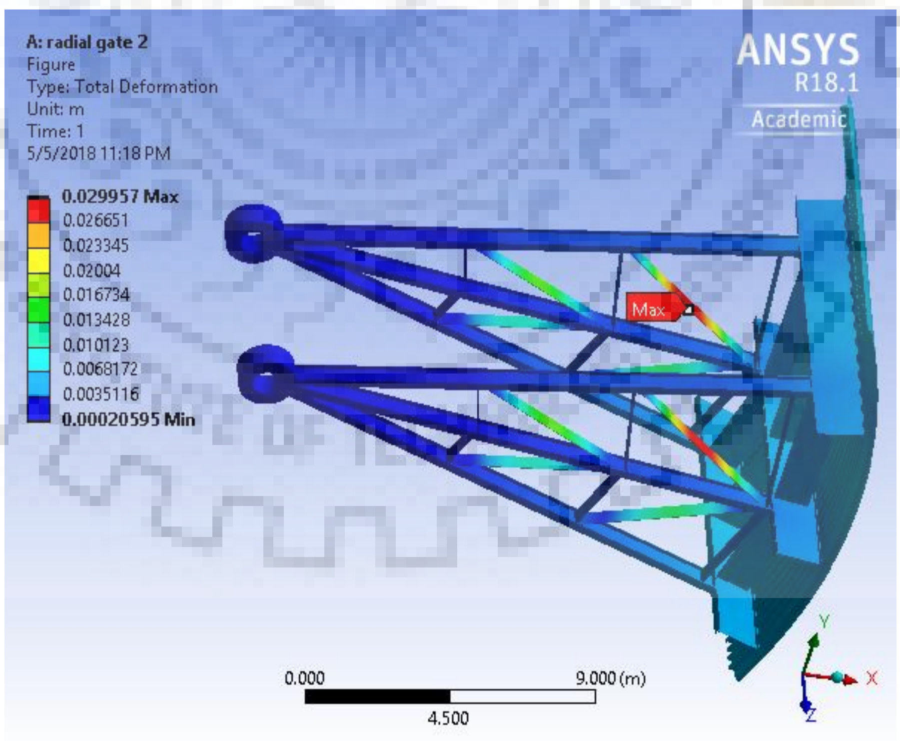

Figure 6.5 Total deformation for ASTM A 36

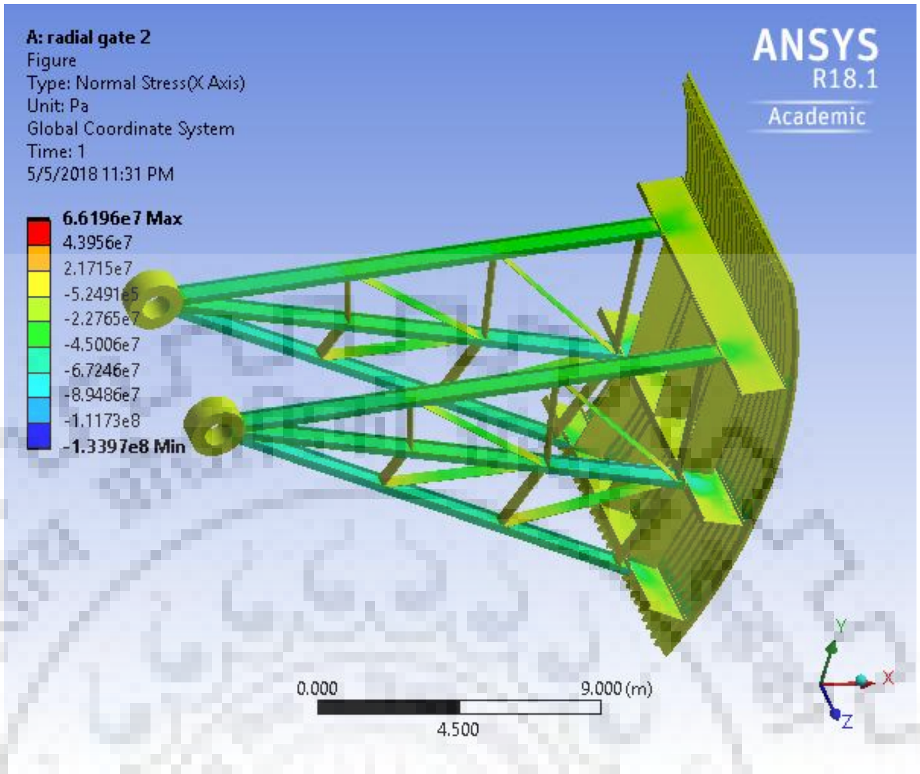

Figure 6.6 Normal stress for ASTM A 588

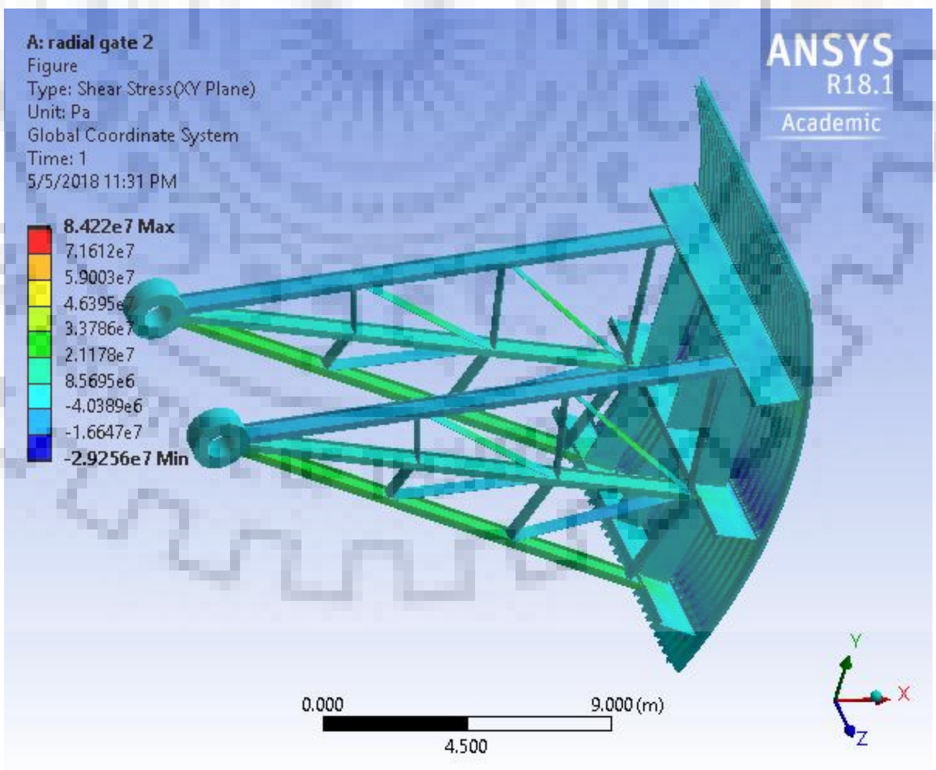

Figure 6.7 Shear Stress for ASTM A 588

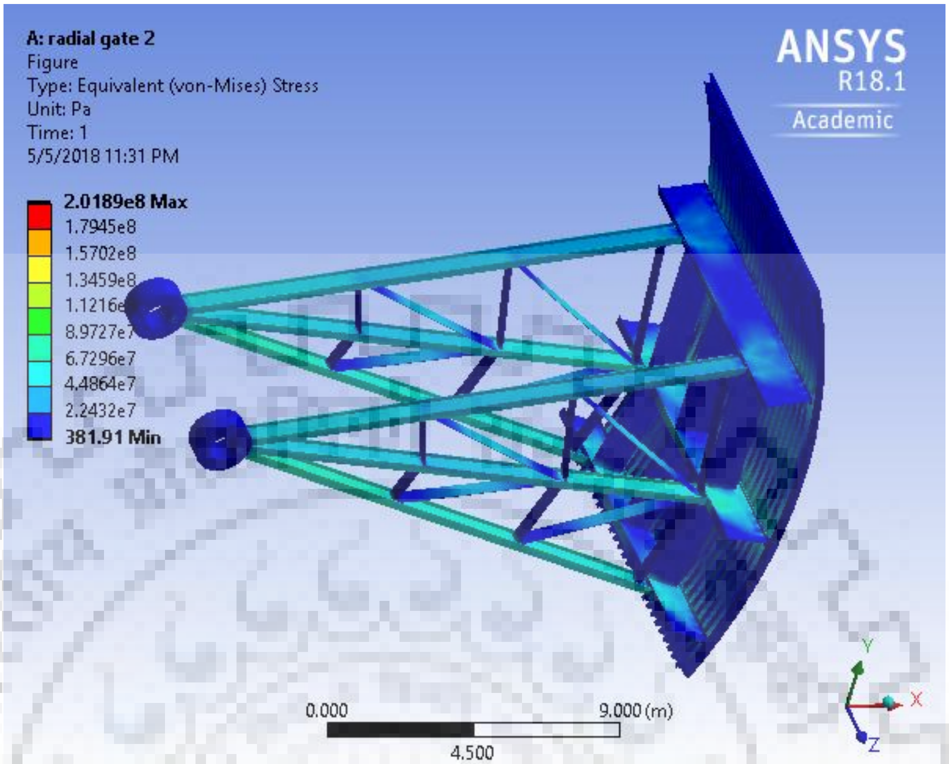

Figure 6.8 Von-mises stress for ASTM A 588

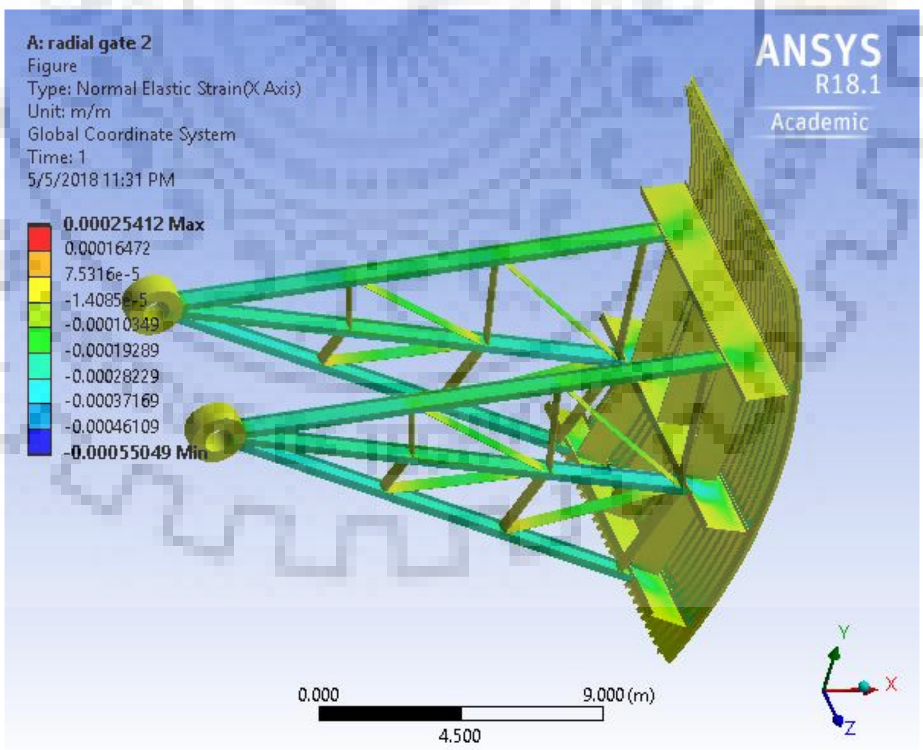

Figure 6.9 Normal elastic strain for ASTM 588

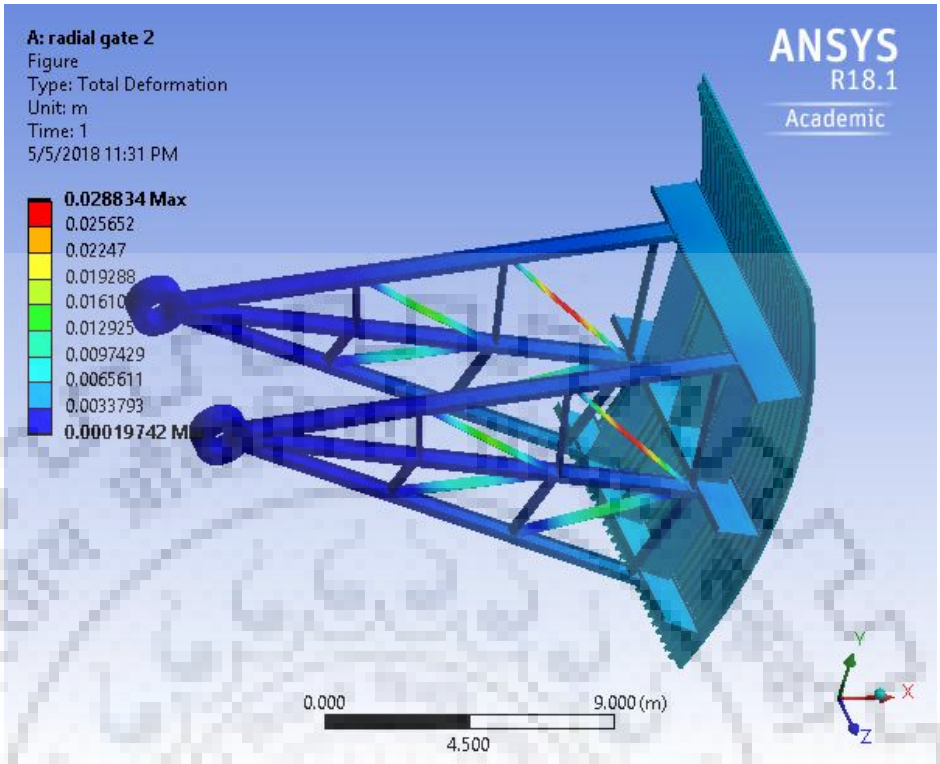

Figure 6.10 Total deformation for ASTM A 588

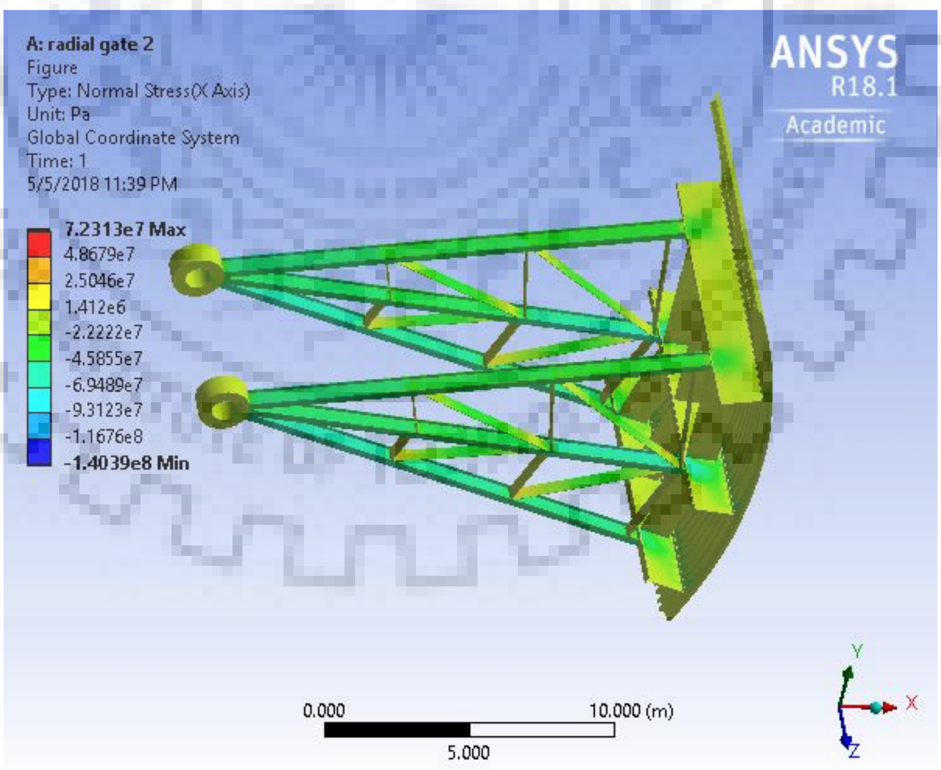

Figure 6.11 Normal stress for ASTM A 992

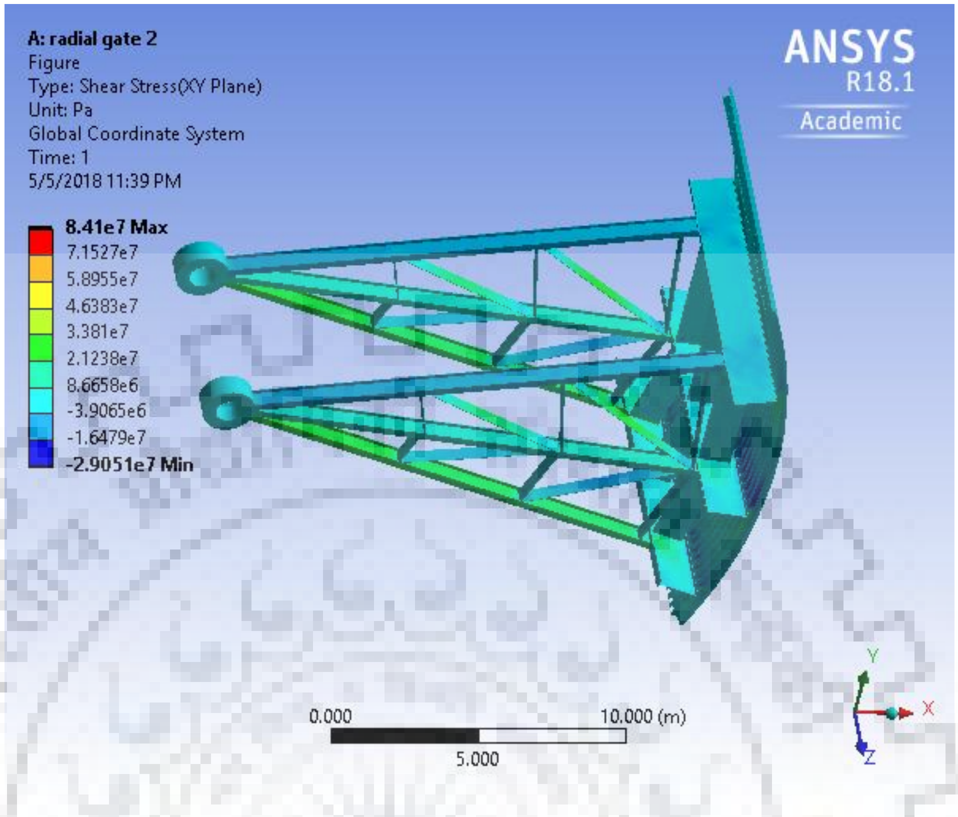

Figure 6.12 Shear stress for ASTM A 992

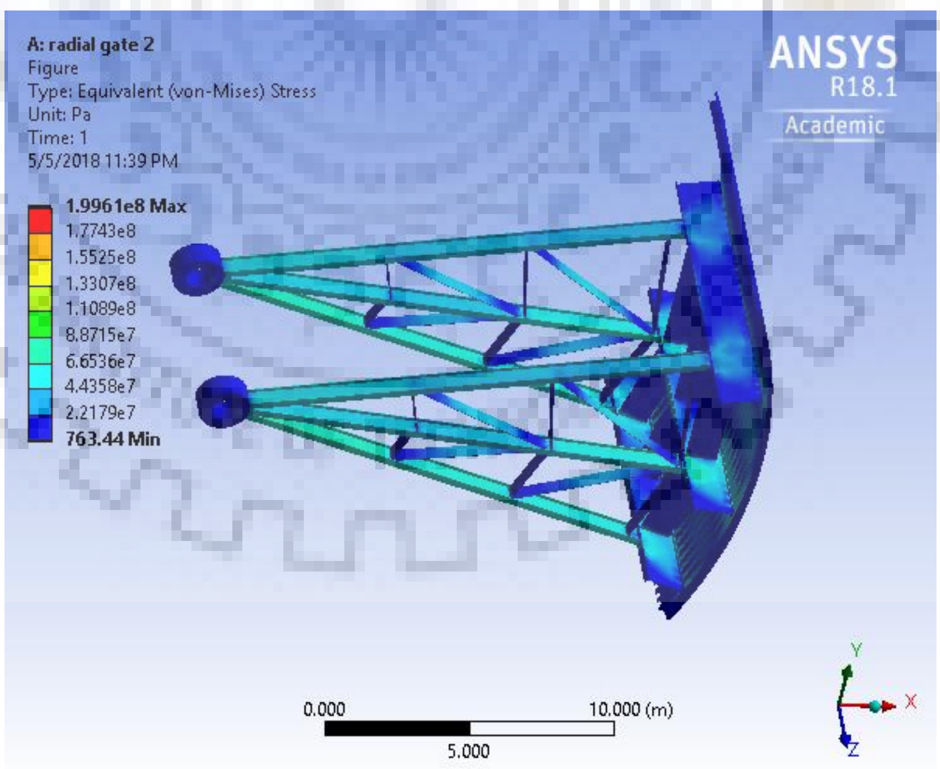

Figure 6.13 Von-mises stress for ASTM A 992

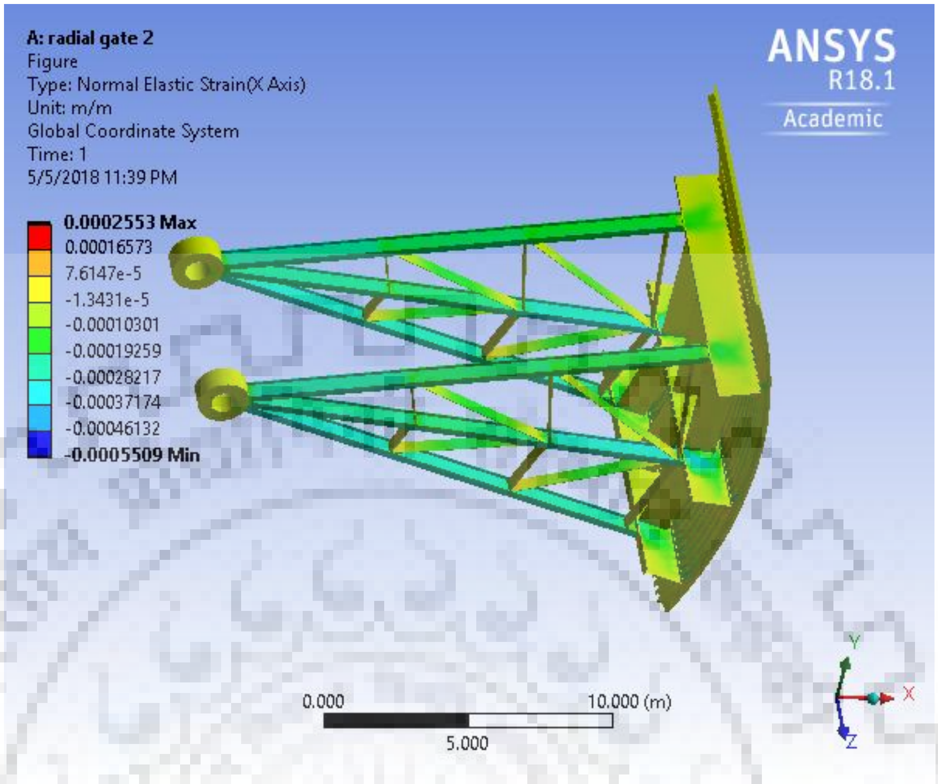

Figure 6.14 Normal elastic strain for ASTM A 992

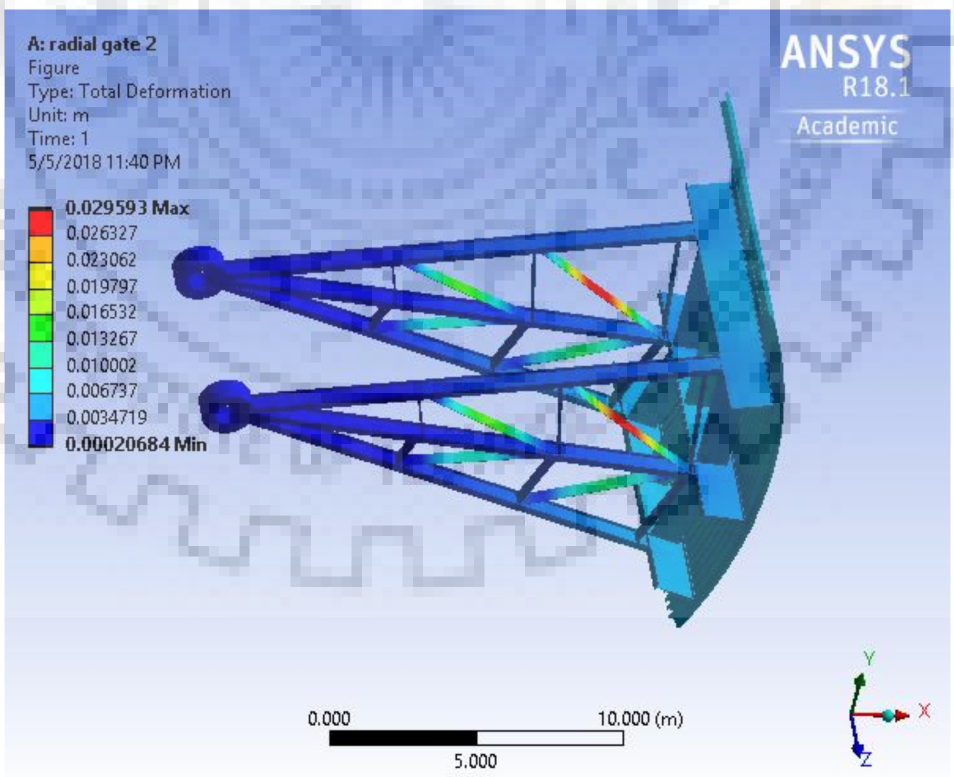

Figure 6.15 Total deformation for ASTM A 992

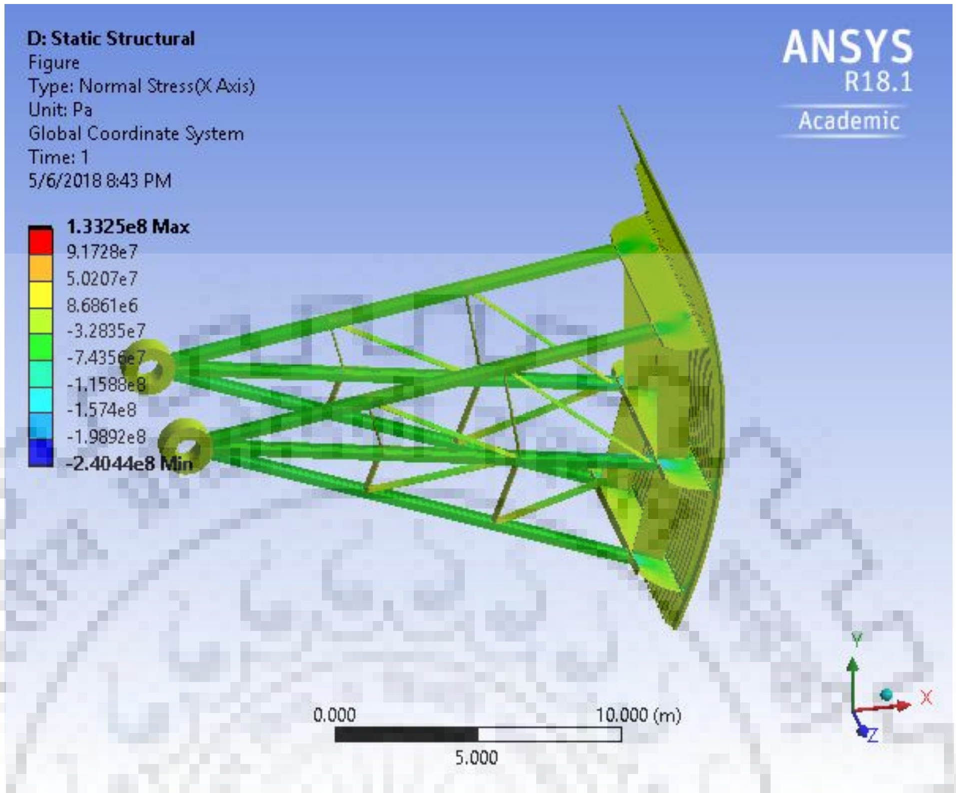

Figure 6.16 Normal stress for optimized design

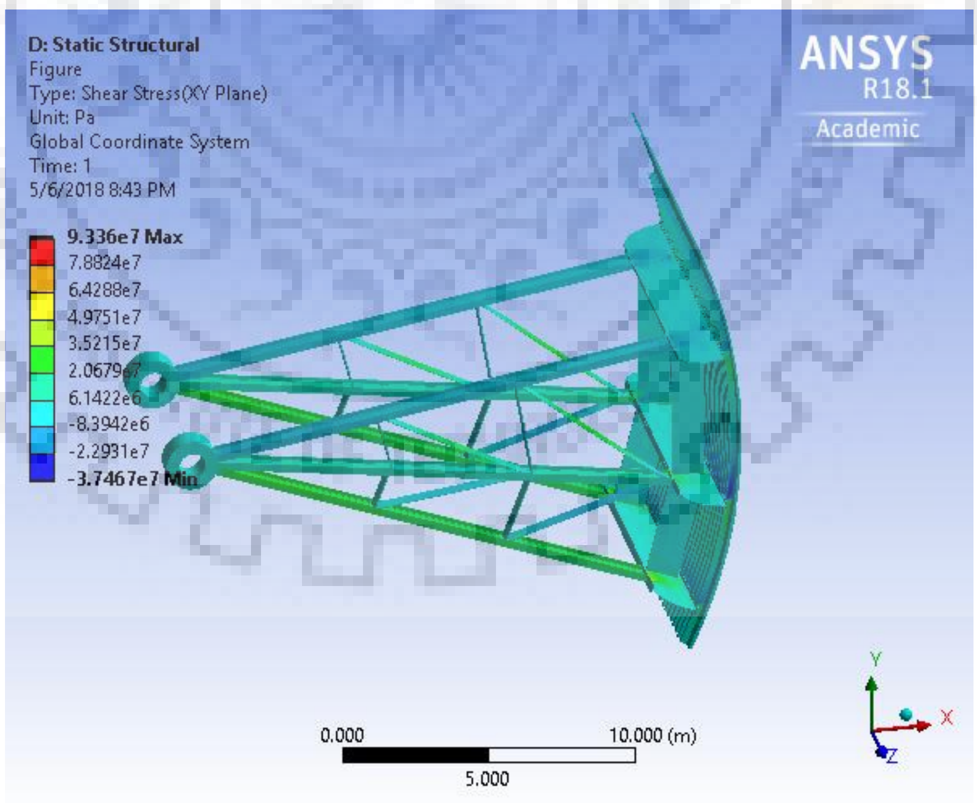

Figure 6.17 Shear stress for optimized design

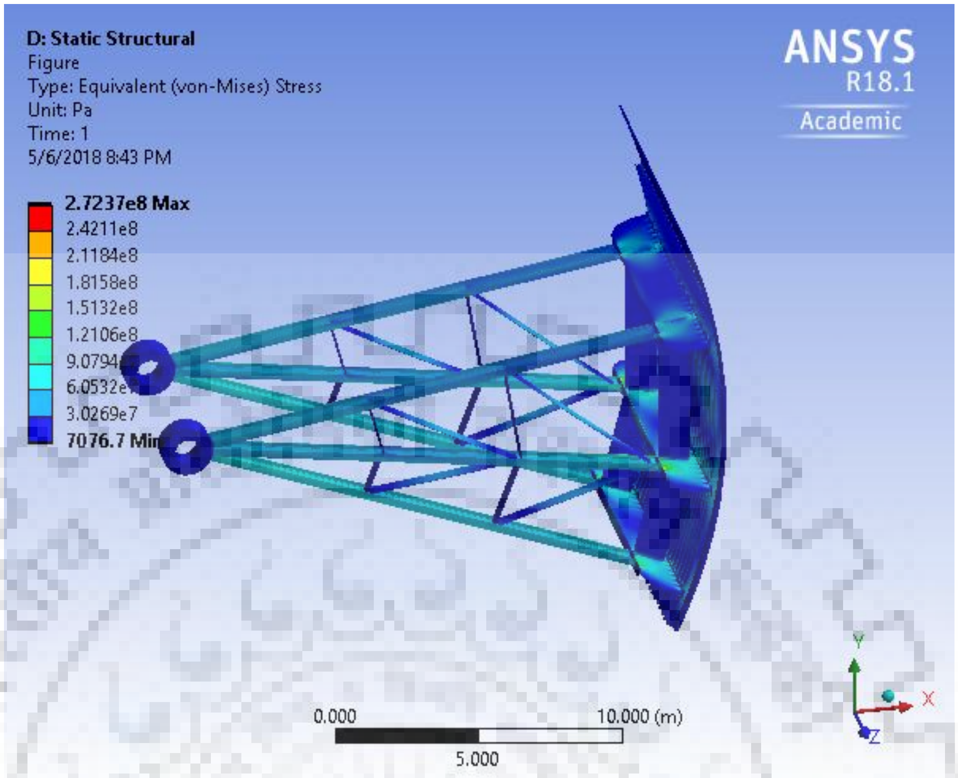

Figure 6.18 Von-mises stress for optimized design

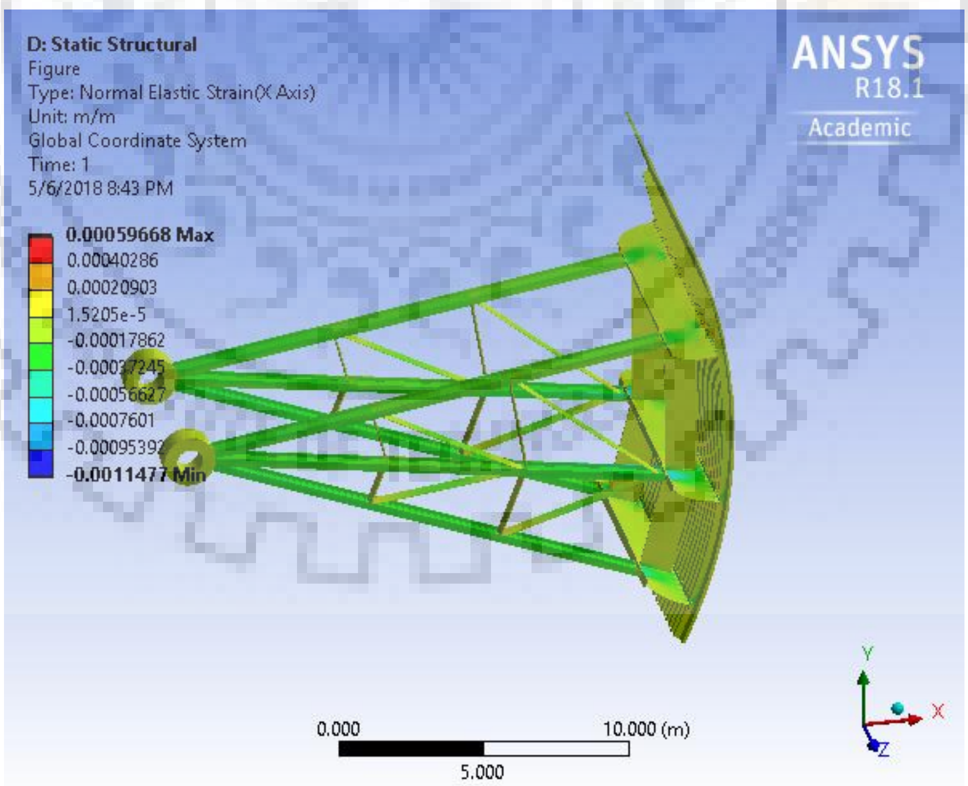

Figure 6.19 Normal elastic strain for optimized design

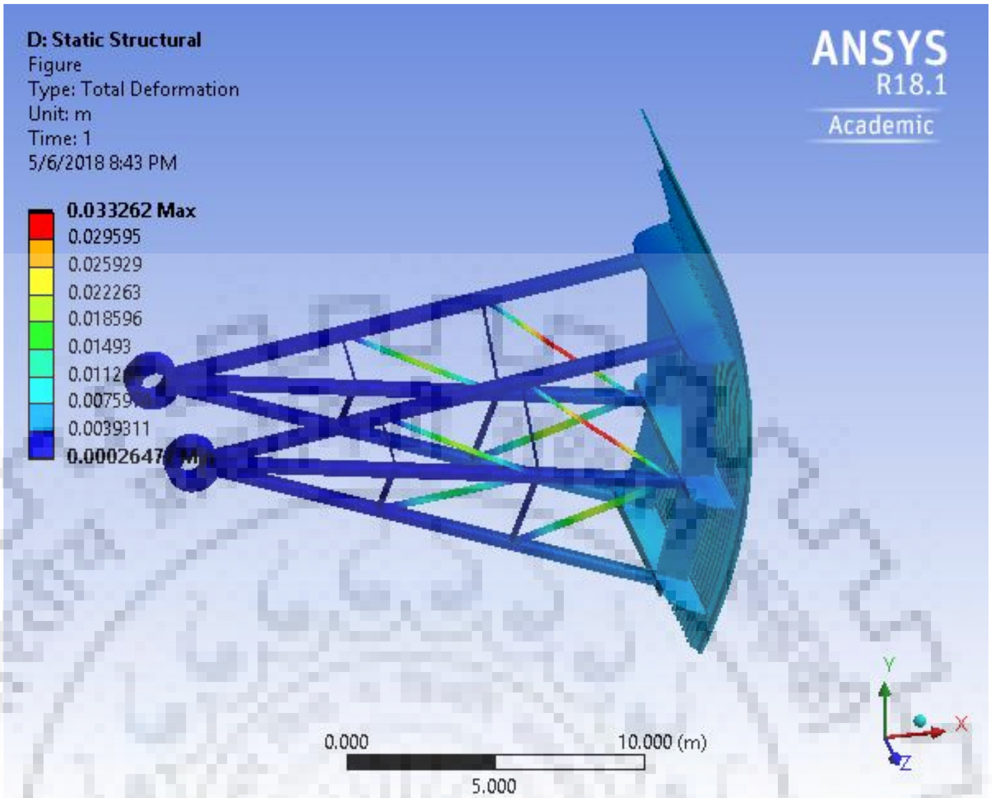

Figure 6.20 Total deformation for optimized design

CHAPTER 7: CONCLUSION

A radial gate is designed by conventional method with the help of IS 4623. This radial gate is modeled on ANSYS Workbench and analysis is performed for various grade of structural steel. ANSYS Mechanical is a FEA tool for structural analysis including linear and nonlinear studies. ANSYS supports material models for wide range of mechanical problems. Three grade of structural steel are selected, one from each carbon steel (ASTM A 36), high strength low alloy steel (ASTM A 992) and corrosion resistant high strength low alloy steel (ASTM A 588). A comparison is drawn for stresses and strain among these grades of structural steel.

Study shows that ASTM A 992 is best grade of structural steel for radial gates as stresses induced are minimum (199.61 MPa) as compare to other selected grades. Sustainability of ASTM A 992 is also good as it has minimal effect on environment. Almost 80% of ASTM A 992 structural steel members fabricated today comes from recycled materials.

Most commonly used grade of structural steel is ASTM A 36. This study shows that induced stresses are maximum for ASTM A 36, minimum for ASTM A 992 and stresses for ASTM A 588 are in between them. Though difference in stresses induced for different grades is marginal but yield strength for ASTM A 992 is considerably high (345 MPa) as compare to ASTM A 36 (250 MPa). Factor of safety increases as ASTM A 992 structural steel is used for radial gates. Hence it is possible to reduce the weight of radial gate if ASTM A 992 grade of structural steel is used.

Topology optimization is performed on the model of radial gate for ASTM A 992 structural steel. Topology optimization is performed to retain 80% mass but ANSYS retains 82% mass. Output geometry of topology optimization is quite irregular and to perform design validation, some modifications in geometry are necessary. These modifications are performed using SPACECLAIM and further increases the mass of model. This modified model have 85.85% mass of initial value. Design validation of this modified model is performed using ANSYS and these results are compared with the results before topology optimization.

Study shows that topology optimization reduces the mass of radial gate by 14.15% and maximum stresses are within permissible limits of ASTM A 992 structural steel. After topology optimization maximum von-mises stress increase by 36.41% and minimum von-mises stress increase by

826.6%. This shows that stresses after topology optimization are distributed uniformly and stress concentration is rescued. Maximum von-mises stress of modified design exceeds the yield strength of ASTM A 36 structural steel, hence this topology optimization is not possible using this material.

Study shows that ASTM A 992 structural steel is best material for radial gates and 14.15% reduction in weight is possible through topology optimization by using this grade of structural steel.

In this study only three grade of structural steel are considered, it is possible to perform similar analysis on other grades of structural steel and find better techno-economical material. This study consider only four main component of radial gate but similar study is possible for other component and different material can be considered for different component with scope for optimization for topology.

References

[1] Paulo C.F. Erbisti (2014). *Design of hydraulic gates*. EH leiden, CRC press/Balkema.

[2] Liu, J. J. (2007). *Research on calculation method of design for radial steel gate.* Master's thesis, Department of hydraulic structure engineering, Hohai University.

[3] Sharma, M. K. (2004). *Three dimensional stress analysis of radial gate using FEM (ANSYS Software).* Master's thesis, Department of water resources development and management, IIT Roorkee.

[4] Kun cai, Hongyang He, Xinhuan Li & Yan Li. (2013). *A new design of a hydraulic steel radial gate with two oblique arm by topology optimization.* Advanced materials result, 2906-2912.

[5] Reddy G. C. M. & Subramanyam. B. (2015). *Design and analysis of spillway radial gate*. International journal of scientific research and education, 6, 3878-3893.

[6] K.Sarth Chandra, C.Chandramohan & Ayub Ashwak. (2015). *Theory of coupled-mode self-excited vibration of tainter gate.* International refereed journal of engineering and science, 4, 64-87.

[7] Robert T., Mohammad A., Chris Cornell, Mohammadreza M., Keith De Lapp &Stephen Hom. (2014). *Poe dam radial gate strengthening project: challenge accepted.* Annual USSD conference San Francisco, California.

[8] Sahu R., Ajmera S. (2017). *Design of radial gate using rectangular and I section – A case study.* International journal of advances in science research and engineering, 3.

[9] Krzysztof Brusewicz, Witold Sterpejkowicz-Wersocki & Robert Jankowski. (2017*). Modal analysis of a steel radial gate exposed to different water levels.* Archives of hydro-engineering and environmental mechanics, 64, 37-47.

[10] Ji Liang Liu, Xing Fang, Zheng Zhong Wang & Dan Xia Sun. (2011). *Simulation of natural vibration characteristics of large size radial gate with three arms considering fluid solid coupling*. Applied mechanics and materials, 50-51, 200-204.

[11] Gorazd N., Jure M., Martin B., & Stasa V. (2016). *Hydrodynamic forces during the operation of a model radial gate*. Journal of applied water engineering and research, 5, 70-77.

[12] IS 4623 (2000): *Recommendations for Structural Design of Radial Gates*. Bureau 0f Indian Standards [WRD 12: Hydraulic Gates and Valves].

[13] Hardwick, J.D., Attari, J., Lewin, J. (2000). *Flow-induced vibration of torrumbarry weir gates*. Conference on Flow-induced vibration, Lucerne, Switzerland, Jun, Balkema, 219–224.

[14] H. KhaliliShayan., J. Farhoudi. And R. Roshan (2014*). Estimation of flow discharge under the sluice and radial gates based on contraction coefficient*. IJST, Transactions of Civil Engineering, 38, 449-463