SRM VALLIAMMAI ENGINEERING COLLEGE

(An Autonomous Institution)

SRM Nagar, Kattankulathur – 603 203

DEPARTMENT OF CIVIL ENGINEERING

II SEMESTER

M.E STRUCTURAL ENGINEERING

ST3265- STRUCTURAL DESIGN STUDIO

Regulation – 2023

Academic Year 2023– 24

Prepared by

Mr.G.R.Iyappan

Assistant Professor (O.G)/ CIVIL

Vision

To produce competent and quality engineers by imparting knowledge, excellence and global perspectives in Civil Engineering to our students and to make them ethically strong professional engineers to build our nation.

Mission

- To produce outstanding graduates with high technical knowledge to serve the nation.
- To impart value based education
- To provide solution to the challenges in the field of Civil Engineering.

Program Educational Objectives (PEOs)

PEO:1 - Gain knowledge and skills in structural engineering which willen able them to have a career and professional accomplishment in the public or private sector organizations.

PEO:2 - Become consultants in Structural Engineering and solve complex real-life issues related to the analysis, design and maintenance of structures under various environmental conditions.

PEO:3 - Contribute to the enhancement of knowledge in Structural Engineering by performing quality research in institutions of international repute or Research organizations or Academia.

PEO:4 - Practice their profession with good communication, leadership, ethics and social responsibility and formulate solutions that are technically sound, economically feasible, and Socially acceptable.

PEO:5 - Graduates will function in multi disciplinary teams and adapt to evolving technologies through life-long learning and innovation

PROGRAMME OUTCOMES (POs)

PO:1 - An ability to independently carry out research/investigation and development work to solve practical problems.

PO:2- An ability to write and present a substantial technical report/document.

PO:3 -Students should be able to demonstrate a degree of mastery over the area as per the specialization of the program. The mastery should be at a level higher than the requirements in the appropriate bachelor's program.

PREFACE

This instruction manual has been prepared by the Department of Civil Engineering to facilitate instruction doing practical classes and further to be used as a reference manual by the M.E Structural Engineering students of this college. This manual covers explanation of experiments included in the syllabus according to Regulation 2023 in ST3265- Structural Design Studio Laboratory for the M.E (Structural) degree course.

Prepared By
Mr.G.R.Iyappan
Assistant Professor
Lab Incharge

GENERAL INSTRUCTIONS

The following instructions should be strictly followed by students in the Studio **Lab:**

1. Students should wear lab coat in Studio lab.
2. Students are advised to enter the Studio lab WITH FORMAL SHOES ONLY.
3. They are not supposed to move the systems and monitors.
4. They should enter in the login name and password assigned to each student.
5. Students are advised to complete their record work before the next class.
6. Students are asked to logout from their area and switch off the computers before leaving the lab.
7. Students can access the printers through lab technician.
8. Students have free access to use the computers and software available in the lab.
9. During the laboratory hours, accessing the internet is strictly prohibited.
10. Computer games are strictly prohibited in the Studio lab.

ST3265 STRUCTURAL DESIGN STUDIO

L	T	P	C
0	0	4	2

COURSE OBJECTIVES:

- To design a structure using modern software tools available like ETABS, STAAD, STRAP, etc. and present it in the form of a complete detailed drawing. Students have to work individually with standard codes, computational tools and software packages for analyzing, designing and detailing a structure. A detailed report on the work done shall be submitted by individual students in the form of a report and presentation.

TOTAL: 45 PERIODS

COURSE OUTCOMES:

- On completion of the course, the student is expected to be able to

1. Understand the requirements of a structure and model it accordingly using computer software.
2. Analyze the structure for various loads and load combinations according to the relevant IS codes.
3. Design and detail structures using computer software/tools and check the correctness using manual approximate methods.
4. Prepare the complete structural drawings using computer software.
5. Observe the flow of forces in a structure and its response to it.

ST3265	PO			PSO		
	1	2	3	1	2	3
CO1	3	-	2	3	3	3
CO2	3	3	1	2	3	2
CO3	2	-	3	2	3	2
CO4	3	3	2	3	-	1
CO5	3	1	3	3	3	3

List of Experiments

1. Introduction about Stadd Pro V8i
2. Analysis and Design Of Cantilever And Counterfort Retaining Wall
3. Analysis and Design of Circular Water Tank Fixed Base
4. Analysis and Design of Circular Water Tank Flexible Base
5. Rectangular Water Tank with Fixed Base
6. Rectangular Water Tank with Flexible Base
7. Hemispherical Steel Water Tank
8. R.C.C Slab Culvert
9. Tee Beam Bridge
10. Analysis and Design Of Truss Girder
11. Deck Type Plate Girder Bridges

Topic beyond Syllabus

1. Analysis of Beam
2. Analysis and Design of Multi Story Building
3. Analysis and Design of Circular Slab

INDEX

Name: ______________ Branch: ____________Sec: ______ REG No: ___________

SI.No.	Date	Title	Teacher's Sign / Remarks

Expt.No:01 **INTRODUCTION**

Aim:

To study the introduction about STAAD Pro

Overview:

STAAD- Standard Analyze and Design

Company name - Bentley

Version- STAADV8i.

STAAD or (STAAD Pro) is a structural analysis and design computer program originally developed by Research Engineers International in Yorba Linda, CA. In late 2005, Research Engineer International was bought by Bentley Systems. An older version called STAAD-III for windows is used by Iowa State University for educational purposes for civil and structural engineers. The commercial version STAAD Pro is one of the most widely used structural analysis and design software. It supports several steel, concrete and timber design codes. It can make use of various forms of analysis from the traditional 1st order static analysis, 2nd order p-delta analysis, geometric nonlinear analysis or a buckling analysis. It can also make use of various forms of dynamic analysis from modal extraction to time history and response spectrum analysis.

In recent years it has become part of integrated structural analysis and design solutions mainly using an exposed API called Open STAAD to access and drive the program using a VB macro system included in the application or other by including open STAAD functionality in applications that themselves include suitable programmable macro systems. Additionally STAAD Pro has added direct links to applications such as RAM Connection and STAAD Foundation to provide engineers working with those applications which handle design post processing not handled by STAAD Pro itself.

Advantages:

1. Easy to use interface,
2. Conformation with the Indian Standard Codes,
3. Versatile nature of solving any type of problem,
4. Accuracy of the solution.

STAAD Pro features a state-of-the-art user interface, visualization tools, powerful analysis and design engines with advanced finite element and dynamic analysis capabilities. From model generation, analysis and design to visualization and result verification, STAAD Pro is the professional’s choice for steel, concrete, timber, aluminum and cold-formed steel design of low and high-rise buildings, culverts, petrochemical plants, tunnels, bridges, piles and much more.

STAAD Pro consists of the following:

The STAAD Pro Graphical User Interface:

It is used to generate the model, which can then be analyzed using the STAAD engine. After analysis and design is completed, the GUI can also be used to view the results graphically.

The STAAD analysis and design engine:

It is a general-purpose calculation engine for structural analysis and integrated Steel, Concrete, timber and aluminum design.

To start with we have solved some sample problems using STAAD Pro and checked the accuracy of the results with manual calculations. The results were to satisfaction and were accurate. In the initial phase of our project we have done calculations regarding loadings on buildings and also considered seismic and wind loads.

Structural analysis comprises the set of physical laws and mathematics required to study and predicts the behavior of structures. Structural analysis can be viewed more abstractly as a method to drive the engineering design process or prove the soundness of a design without a dependence on directly testing it.

To perform an accurate analysis a structural engineer must determine such information as structural loads, geometry, support conditions, and materials properties. The results of such an analysis typically include support reactions, stresses and displacements. This information is then compared to criteria that indicate the conditions of failure. Advanced structural analysis may examine dynamic response, stability and non-linear behavior.

The aim of design is the achievement of an acceptable probability that structures being designed will perform satisfactorily during their intended life. With an appropriate degree of safety, they should sustain all the loads and deformations of normal construction and use and have adequate durability and adequate resistance to the effects of seismic and wind. Structure and structural elements shall normally be designed by Limit State Method. Account should be taken of accepted theories, experiment and experience and the need to design for durability. Design, including design for durability, construction and use in service should be considered as a whole. The realization of design objectives requires compliance with clearly defined standards for materials, production, workmanship and also maintenance and use of structure in service.

The design of the building is dependent upon the minimum requirements as prescribed in the Indian Standard

Codes. The minimum requirements pertaining to the structural safety of buildings are being covered by way of laying down minimum design loads which have to be assumed for dead loads, imposed loads, and other external loads, the structure would be required to bear. Strict conformity to loading standards recommended in this code, it is hoped, will not only ensure the structural safety of the buildings which are being designed.

Structural engineering is the branch of civil engineering that deals with the analysis and Design of structure that will support or resist the stresses, forces and loads. Throughout the history, Civil engineers have made an important contribution to improve the environment by designing and Construction major structures and facilities. The profession requires a lot of accuracy because most of the time; the final result of any project will directly or indirectly affect people's lives and safety. As a Result, software usage in this industry reduces the complexity of loss of challenging projects.

Many changes happened to the requirements of structural analysis due to the introduction of Computer software. In the past, the required skill for analysis was the capability to simplify the Complex structural systems so they could be analyzed by hand. This meant that an analyst would have a number of methods which might be applied to a specific problem. The choice of which Method to use was very important, since the incorrect choice could mean that the solution was Impossible by hand due to the requirement of large number of equations to be solved. On the other Hand, analysis process recently has an array of powerful computer programs which can handle most.

Structural problems that an engineer is likely to face:

Nowadays, software of structural design is used mostly for the analytical purpose in the detailed design phase, and becoming more versatile. They are being applied to all phases of the Design process, from the generation of design conceptions (design topologies, or layouts), through Preliminary design (design shape specification), and finally in the detailed process (sizing of Structure members).

Therefore, structural software can be defined as software which employed mostly by civil Engineers to prototype and analyze architectural designs to meet their professional objectives. Structural software gives the opportunity to visualize different structure's components on screen. These components such as: footings columns, beams, joists, rafters and the like. Calculation and data Analysis for a wide range of building elements are the most important part of these programs. Also Engineers required to know what type of loads are the components weighing. By applying this Software, structural engineers accomplish more than they ever could before, both in quantity and Quality STAAD program:

STAAD (Structural Analysis and Design Program) is a general structural and design program With applications primarily in the building industry-commercial buildings, bridges and highway Structures, industrial structures, chemical plant structures, dams, retaining walls, turbine foundations, Culverts and other embedded structures. STAAD supports several steel, concrete and timber Design codes. It offers various analysis methods such as the traditional 1st order static analysis, 2nd Order P-delta analysis, geometric nonlinear analysis and the buckling analysis. It can also analyze Various type of dynamic analysis starts from model extraction to time history and response spectrum analysis. STAAD is fast, efficient, easy to use and accurate platform, conformation with many Standard codes and qualified to solve any type of problem. These features had made the STAAD the Professional's choice. Today, STAAD is one of the most popular structural engineering software Products.

Methods in STAAD Pro:

- Snap and grid method
- Structural wizard
- Excel method
- DXF method
- STAAD editor method

Snap and grid method:

Open STAAD Pro> snap node/ beam > edit. Plane X-Y

Angle of plane:

Y-Y Grid origin

X	Y	Z
0	0	0

In this you can change the values and spacing of the coordinates. Snap and grid method>snap node beam> to create.

Structural wizard:

Geometry>run structural wizard>model type(frame model)> bay frame> transfer> yes> close.

Excel sheet method:

To create node and member

DXF method:

To import the file from AutoCAD format to STAAD format

a) Draw in AutoCAD> save> file type: AutoCAD 2007/ LT 2007 DXF*(dxf)> save> close.

b) Open STAAD Pro> file> import> select required file> ok.

Translation repeat:

To create building from a single member

Translation repeat>choose global direction >no.of steps required> default step spacing> link steps>open base> ok.

Circular repeat:

Select the member > geometry> circular repeat> choose the axis of rotation (Y)> total angle 360^0>type the required no.of steps>select node> link steps> open base> ok.

Mirror:

Select the member> go to mirror> select mirror plane (X-Y OR X-Z)> select node > ok.

Rotate:

Go to rotate> type the required angle> select node> represent the node in the diagram> copy/move> ok.

Geometry:

Add beam> from point to point> between midpoints>by perpendicular direction> add curve beam.

Insert node:

Select member> add new point>Midpoint/ add n points>type the required distance> ok.

Result:

Thus the introduction about STAAD Pro is studied successfully.

Outcome:

At the end of this experiment, student acquires knowledge about the STAAD Pro.

Expt.No: 02	**Analysis of Beam**
Analysis a simply supported beam with udl load of 4KN/m throughout the beam a length of 5m.	

STEP 1: MODELLING

@ To select snap/node option and connect two nodal point.
@ Enter the nodal name A and B
@ By using render view to identify the assigned member

STEP 2: SUPPORT SPECIFICATION

= Using command option, add support as hinged at both the point A and B

STEP 3: MATERIAL PROPERTIES

= Add a material property for selected members
= A selected member is considering as beam.
= For a beam of cross section 300mm×300mm.
= To assign the respective element by using assign command.
= To change the view (i.e) 3D rendered view for the purpose of material assigned or not.

STEP 4: LOADING DETAILS

= Add a dead load, live load.
= Assign a self-weight of -1.
= Assume a L.L of 4KN/m as given in examples.

STEP 5: LOAD COMBINATION

Design the categories of load combination such as 1.5 D.L + 1.5 L.L.

STEP 6: PERFORM ANALYSIS

= Select the perform analysis, run analysis to create a post processing mode.
= Therefore B.M, S.F & Deflection of beams are determined.

STEP 7: DESIGN PROCEDURE

= Change the concrete code IS 456:2000.
= (Max. main) Selected parameters.
= Assign command. To design beam.
To design element /node.
Take up.

Assign the above option.

STEP 8: DESIGN DETAILING

= To select perform analysis, click all parameters and analyze.
= Select the run analysis, to find overall result.

Result:

Therefore the determination of shear force, bending moment and design of reinforced concrete, and quantities of steel, concrete, reinforcement needed are formed by using STAAD.PRO.

Outcome:

At the end of this experiment, student acquires knowledge in creating and analyzing beam using STAAD Pro.

Expt.No:03 ANALYSIS AND DESIGN OF MULTI STORY BUILDING

Aim:

To create and analyze the multi storey building by using STAAD Pro

Software required:

STAAD PRO V8i

Procedure in STAAD Pro:

Preprocessing:

1. *Create model*

   a) New project> plane> length units as m> force units as KN>type the file name as multi storey building>next> add beam.

   b) Geometry> run structural wizard> frame model> bay frame>type the length as 12.6, no.of bays along length as 3, height as 12.6, no. of bays along height as 4> apply & transfer the model to STAAD Pro screen.

2. *Property*

   a) General> property> define> rectangular>0.35x 0.23> add> close

   b) Select rectangular and assign toview.

3. *Support*

   Support> fixed support> select the bottom nodes> assign selected nodes.

4. *Loads*

   a) Load and definition> load case detail> add>change the title as DL>add> then change LL> add> WL> add> close.

   b) Select DL> add> member load> uniform load> type W1 as -40KN/m < add.

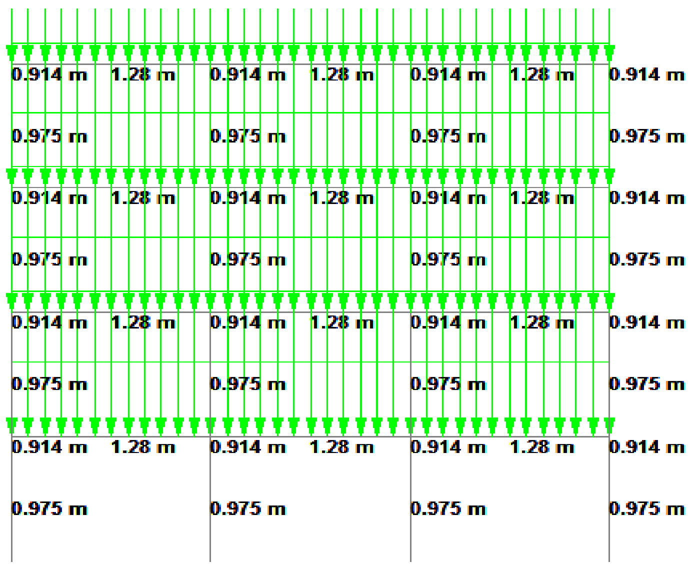
   c) Select LL > add> member load> uniform load> type W1 as -20KN/m < add.

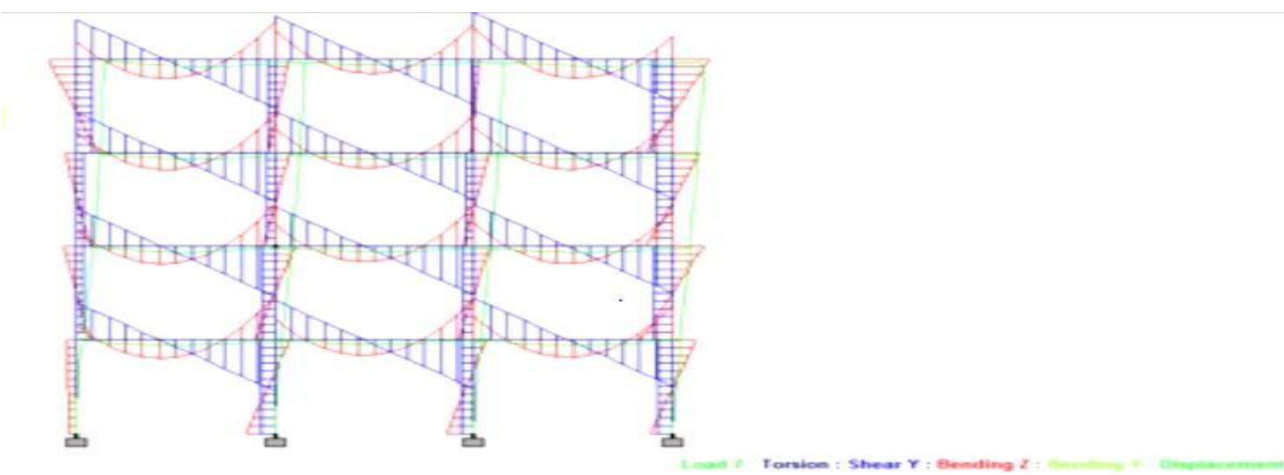
   d) Select WL > add> nodal load> type X as 25> add> type X as 20> add> type X as 15> add> type X as 10> add>close>Select DL> click select > member parallel to x direction> assign selected beams.

   e) Similarly, click select > member parallel to x direction> assign selected beams.

   f) WL> select node cursor> select the particular node> assign selected node.

   g) For load combinations, go to load case details> add> define combination> type name as DL+LL> FOS as 1.5>in available load case, select DL & LL and send it other side> add. Select DL & LL and send it other side> add.


0.914 m 1.28 m 0.914 m 1.28 m 0.914 m 1.28 m 0.914 m
0.975 m 0.975 m 0.975 m 0.975 m
0.914 m 1.28 m 0.914 m 1.28 m 0.914 m 1.28 m 0.914 m
0.975 m 0.975 m 0.975 m 0.975 m
0.914 m 1.28 m 0.914 m 1.28 m 0.914 m 1.28 m 0.914 m
0.975 m 0.975 m 0.975 m 0.975 m
0.914 m 1.28 m 0.914 m 1.28 m 0.914 m 1.28 m 0.914 m
0.975 m 0.975 m 0.975 m 0.975 m


Torsion : Shear Y : Bending Z :

a) Then similarly type name as DL+LL+WL> FOS as 1.5>in available load case, select DL, LL & WL and send it other side> add. Select DL, LL& WL and send it other side> add.

b) Then similarly type name as DL+LL-WL> FOS as 1.5>in available load case, select DL, LL & WL and send it other side> add. Select DL & LL and send it other side> add. After that select WL and send it other side> add.

c) Then similarly type name as DL+LL+WL> FOS as 1.2>in available load case, select DL, LL & WL and send it other side> add. Select DL, LL& WL and send it other side> add.

5. *Design parameters*

a) Design concrete> IS 456>define parameter> give F_C as 20000 KN/m^2 for M20 concrete >F_C as 30000 KN/m^2 for M30 concrete>add> F_Y as 415000 KN/m^2> add> select> ratio as 3>add.

b) Then go to commands>design beam> add>design column> add>take off > close.

c) Select F_C> select entire structure using cursor> assign it to selected beams> similarly do it for F_Y and ratio then assign it.

d) For beams > select the beams parallel to x direction and for columns column parallel to y direction and then assign it.

6. *Analysis*

Analysis& print> add> close>analyze> run analyze> save> post processing> done.

Post-processing:

a) In post-processing you have to see the deflection, bending moment, shear, axial forces, stresses and everything by turn on the particular specified icons for all the load combinations already given.

b) Results> view value> beam results> maximum displacement> activate.

c) You can also see the animations of displacement in it.

Result:

Thus the multi storey building is created and analyzed by using STAAD Pro.

Outcome:

At the end of this experiment, student acquires knowledge in creating and analyzing multi storey building using STAAD Pro

Expt.No:04 ANALYSIS AND DESIGN OF CANTILEVER AND COUNTERFORT RETAINING WALL

Aim:

To create and analyze the cantilever or counterfort retaining wall by using STAAD Pro.

Software Required:

STAAD PRO V8i

Procedure:

Preprocessing:

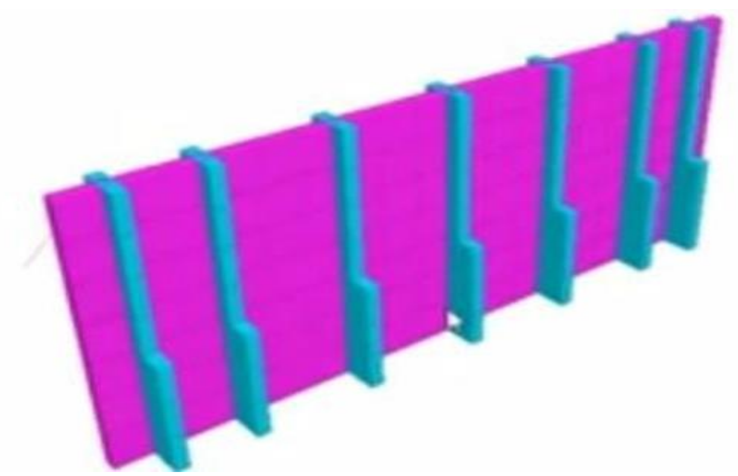
1. *Create model*
   a) New project> plane> length units as m> force units as KN>type the file name as cantilever or counter fort retaining wall> next> add beam.
   b) Snap node/ beam > edit> X in right, X spacing, Y in right, Y spacing> ok.
   c) Now draw the retaining wall model by using snap node/ beam.
2. *Property*
   a) General> property> define> rectangular>0.35x 0.23> add> close
   b) Select rectangular and assign to view.
3. *Support*

   Support> fixed support> select the bottom nodes> assign selected nodes.
4. *Loads*
   a) Load and definition> load case detail> add>change the title as DL>add> then change LL> add> WL> add> close.
   b) Select DL> add> member load> uniform load> type W1 as -40KN/m> add.
   c) Select LL> add> give the soil pressure acting on the retaining wall.
   d) Assign the given loads on the retaining wall.
   e) For load combinations, go to load case details> add> define combination> type name as DL+LL> FOS as 1.5>in available load case, select DL & LL and send it other side> add. Select DL & LL and send it other side> add.
5. *Design parameters*

   a) Design concrete> IS 456>define parameter> give F_c as 20000 KN/m^2 for M20 concrete >F_c as

30000 KN/m^2 for M30 concrete>add> FY as 415000 KN/m^2> add> select> ratio as 3>add.

b) Then go to commands>design beam> add>design column> add>take off > close.



c) Select Fc> select entire structure using cursor> assign it to selected beams> similarly do it for FY and ratio then assign it.

d) For beams > select the beams parallel to x direction and for columns column parallel to y direction and then assign it.

Analysis:

Analysis& print> add> close>analyze> run analyze> save> post processing> done

Post-processing:

a) In post-processing you have to see the deflection, bending moment, shear, axial forces, stresses and everything by turn on the particular specified icons for all the load combinations already given.

b) Results> view value> beam results> maximum displacement> activate.

You can also see the animations of displacement in it.

Result:

Thus the retaining wall is created and analyzed by using STAAD Pro.

Outcome:

At the end of this experiment, student acquires knowledge to create and analyze the cantilever or counterfort retaining wall by using STAAD Pro.

Exp.No:05 **ANALYSIS AND DESIGN OF CIRCULAR SLAB**

Aim:

To create and analyze the circular slab by using STAAD Pro

Software required

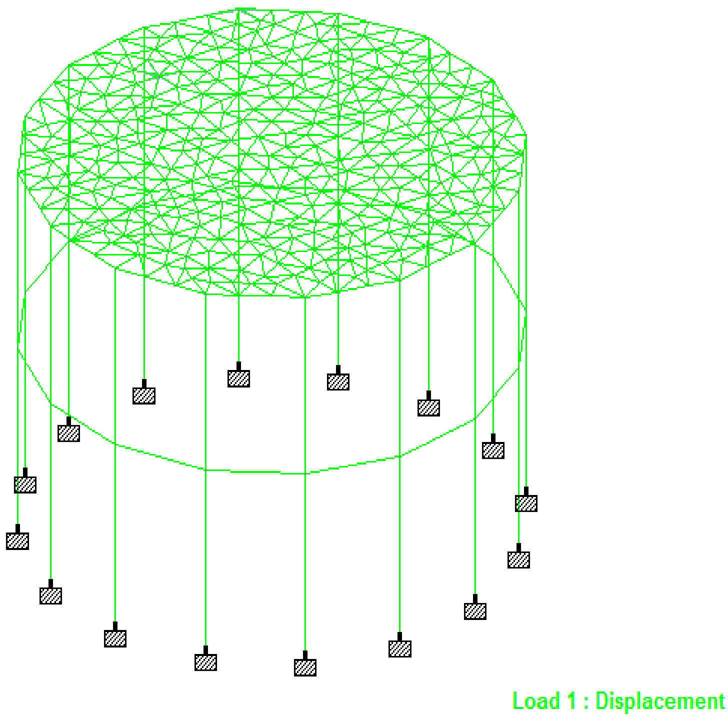
STAAD PRO V8i

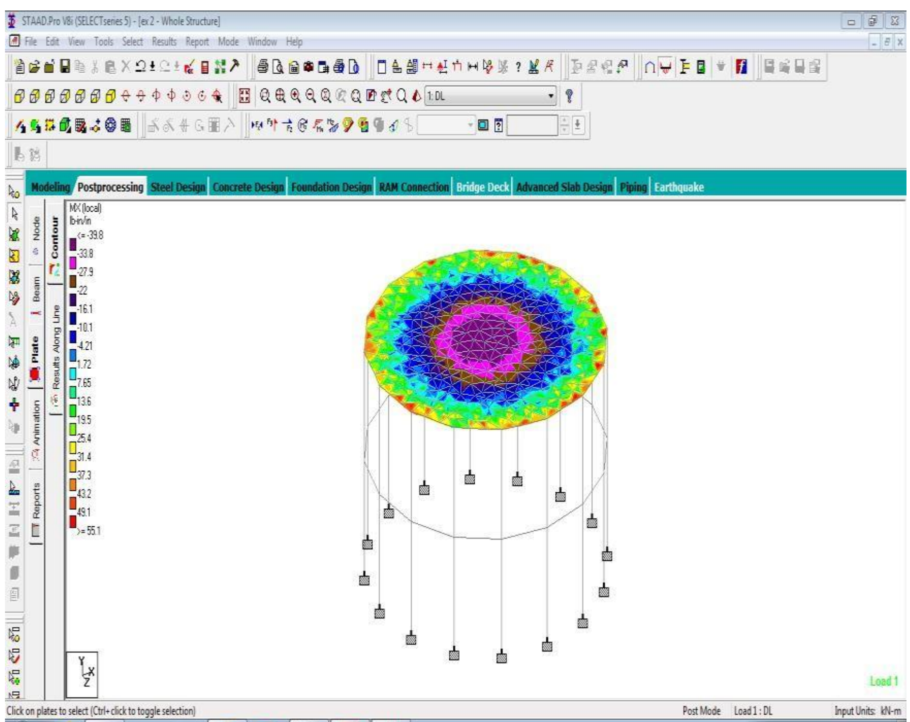
Procedure in STAAD Pro:

Preprocessing:

1. *Create model*
   a) New project> plane> length units as m> force units as KN>type the file name as circular slab >next> add beam.
   b) Geometry> run structural wizard> frame model> reverse cylindrical model >type the length as 7m, no.of bays along length as 2, radius as 5, no.of bays along periphery as 16> apply & transfer the model to STAAD Pro screen.
   c) Remove the bottom member by selecting and deleting it.
2. *Create plate*
   a) Select the top horizontal member of the model> right click the cursor> new view> isometric view.
   b) Select the surface mesh and create the mesh by connecting all nodes. Give the division as 3 (i.e., along the periphery you will get 16 x 3 + 48 elements).
   c) Then display the whole structure by using the specified icon.
3. *Property*
   a) General> property> define> rectangular>0.38 x 0.23> add> close.
   b) Circular>0.45> add> close.
   c) Go to thickness> plate element thickness> 0.15.
   d) Select rectangular and assign the horizontal beams, Select circular and assign the vertical columns, select the plate by using plate cursor and assign it.
4. *Support*

   Support> fixed support> select the bottom nodes> assign selected nodes.
5. *Loads*
   a) Load and definition> load case detail> add>change the title as DL> add.
   b) Again change the title as plate load> add> close.
   c) Select DL> add>plate load as -1KN/m > add.


Load 1 : Displacement


STAAD.Pro V8i (SELECTseries 5) - [ex2 - Whole Structure]
File Edit View Tools Select Results Report Mode Window Help
Modeling
Postprocessing
Steel Design
Concrete Design
Foundation Design
RAM Connection
Bridge Deck
Advanced Slab Design
Piping
Earthquake
Node
Beam
Plate
Animation
Reports
Contour
Results Along Line
MX (local)
lb-in/in
<= -39.8
-33.8
-27.9
-22
-16.1
-10.1
-4.21
1.72
7.65
13.6
19.5
25.4
31.4
37.3
43.2
49.1
>= 55.1
Load 1
Click on plates to select (Ctrl+click to toggle selection)
Post Mode
Load 1 : DL
Input Units: kN-m

d) Select the plate by using plate cursor>assign to selected plate> yes.

6. *Design parameters*

a) Design concrete> IS 456>define parameter> give F_c as 20000 KN/m^2 for M20 concrete > F_Y as 415000 KN/m^2> add> select> ratio as 3>add.

b) Then go to commands>design beam> add>design column> add>take off > close.

c) Select F_c> select entire structure using cursor> assign it to selected beams> similarly do it for F_Y and ratio then assign it.

d) For beams > select the beams parallel to x direction and for columns column parallel to y direction and then assign it.

7. *Analysis*

Analysis& print> add> close>analyze> run analyze> save> post processing> done.

Post-processing:

a) In post-processing you have to see the deflection, bending moment, shear, axial forces, stresses and everything by turn on the particular specified icons for all the load combinations already given.

b) Results> plate rest contour> stress type M_X> apply> close.

c) You can also see the animations of displacement in it

Result:

Thus the circular slab is created and analyzed by using STAAD Pro.

Outcome:

At the end of this experiment, student acquires knowledge in creating and analyzing the circular slab using STAAD Pro.

Exp.No:06 ANALYSIS AND DESIGN OF CIRCULAR WATER TANK FIXED BASE

Aim:

To create and analyze the circular water tank with fixed base by using STAAD Pro

Software required

STAAD PRO V8i

Procedure in STAAD Pro:

Preprocessing:

1. *Create model*

a) New project> plane> length units as m> force units as KN>type the file name as circular water tank>next> add beam.

b) Geometry> run structural wizard> frame model> reverse cylindrical model >type the length as 10m, no.of bays along length as 3, change the property in length as 4,4,2>radius as 5m no.of bays along periphery as 18> apply & transfer the model to STAAD Pro screen.

c) Remove the bottom member by selecting and deleting it.

d) Select front view and select the top horizontal beams of the tank> right click> new view> ok.

e) Isometric view> add beam> create beam in its diameter> select beam > insert node> add midpoint> ok.

f) Select node> copy > paste>change y distance as 0.5m> ok.

g) Delete the created bean in its diameter

h) Geometry> add beam> add the beam by cursor in between the centre and edge of circle.

i) Select beam> select insert node> give n= 5> add n points> ok.

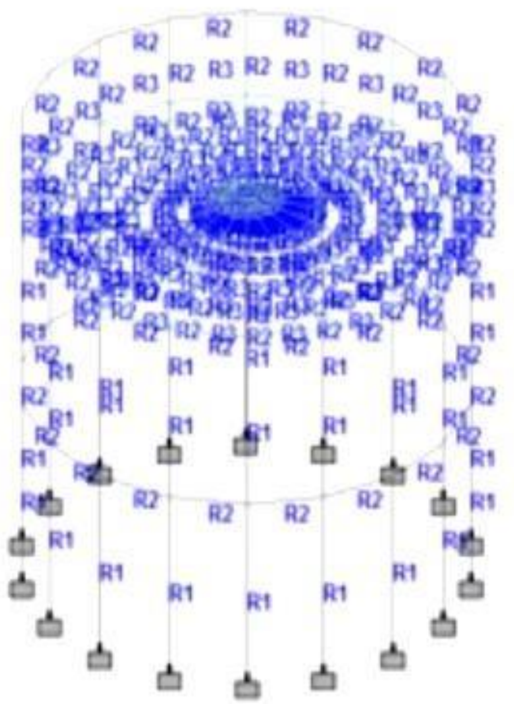
j) Then select the entire beam>circular repeat>, select direction as Y, no.of steps= 18, link steps, open base>select the center node of the circle> ok.

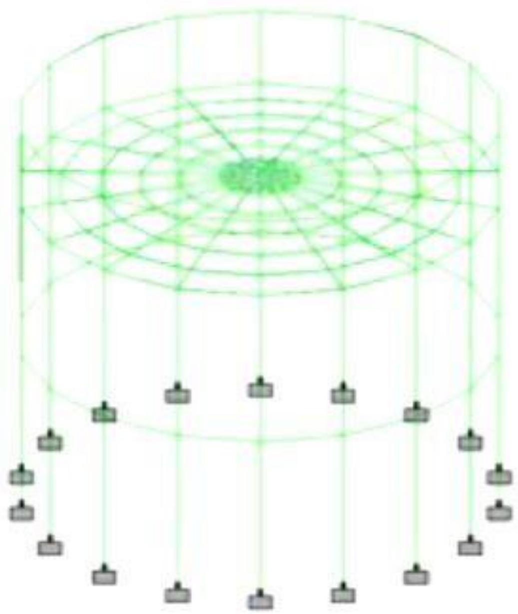
k) Top view> select the beams > delete.

l) Create mesh at the center and give the division as 1> ok.

m) Select all the beams> select fill floor grid with plates> thus the plates are created>select all beams in that and delete it.

n) Select 4 node plates, create the plate for the side walls> fill floor grid with plates and make it as plate.





1. *Property*
   a) General> property> define> rectangular>0.3 x 0.23> add> close
   b) circular>0.3> add> close
   c) gotothickness>plateelementthickness>0.15.
   d) Select rectangular and assign the horizontal beams, Select circular and assign the vertical columns, select the plate by using plate cursor and assign it.
2. *Support*

Support> fixed support> select the bottom nodes> assign selected nodes.

3. *Loads*

o) Load case detail> add>DL>add> close DL> add> pressure on full plate> global direction y>W1 = -20 KN/m^2> add> close.

p) Select the bottom plate using plate cursor and assign it.

q) DL> add> plate load> hydrostatic >select plate < select the specified plate by using cursor>done> W1= -20 KN/m^2 (depending on the z- axis direction) and W2 =0.001>global y axis& local Z axis> close.

2. *Design parameters*

a) Design concrete> IS 456>define parameter> give F_c as 20000 KN/m^2 for M20 concrete > F_Y as 415000 KN/m^2> add> close..

b) Then go to commands>design beam> add>design column> add>take off > close.

c) Select F_c> select entire structure using cursor> assign it to selected beams> similarly do it for F_Y and ratio then assign it.

d) For beams > select the beams parallel to x direction and for columns column parallel to y direction and then assign it.

3. *Analysis*

Analysis& print> add> close>analyze> run analyze> save> post processing> done.

Post-processing:

a) In post-processing you have to see the deflection, bending moment, shear, axial forces, stresses and everything by turn on the particular specified icons for all the load combinations already given.

b) Results> plate rest contour> stress type M_X> apply> close.

c) You can also see the animations of displacement in it.

Result:

Thus the circular water tank with fixed base is created and analyzed by using STAAD Pro.

Outcome:

At the end of this experiment, student acquires knowledge in creating and analyzing the circular water tank with fixed base using STAAD Pro.

Exp.No:07 ANALYSIS AND DESIGN OF CIRCULAR WATER TANK FLEXIBLE BASE

Aim:

To create and analyze the circular water tank with Flexible base by using STAAD Pro

Software required

STAAD PRO V8i

Procedure in STAAD Pro:

Preprocessing:

4. *Create model*

a) New project> plane> length units as m> force units as KN>type the file name as circular water tank>next> add beam.

b) Geometry> run structural wizard> frame model> reverse cylindrical model >type the length as 10m, no.of bays along length as 3, change the property in length as 4,4,2>radius as 5m no.of bays along periphery as 18> apply & transfer the model to STAAD Pro screen.

c) Remove the bottom member by selecting and deleting it.

d) Select front view and select the top horizontal beams of the tank> right click> new view> ok.

e) Isometric view> add beam> create beam in its diameter> select beam > insert node> add midpoint> ok.

f) Select node> copy > paste>change y distance as 0.5m> ok.

g) Delete the created bean in its diameter

h) Geometry> add beam> add the beam by cursor in between the centre and edge of circle.

i) Select beam> select insert node> give n= 5> add n points> ok.

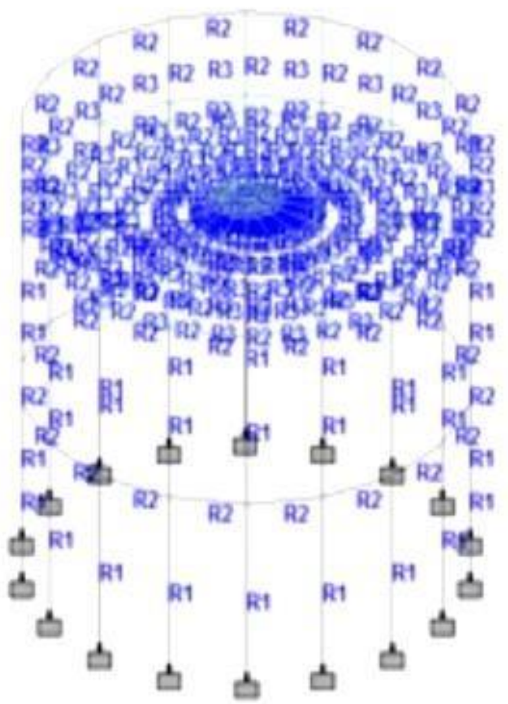
j) Then select the entire beam>circular repeat>, select direction as Y, no.of steps= 18, link steps, open base>select the center node of the circle> ok.

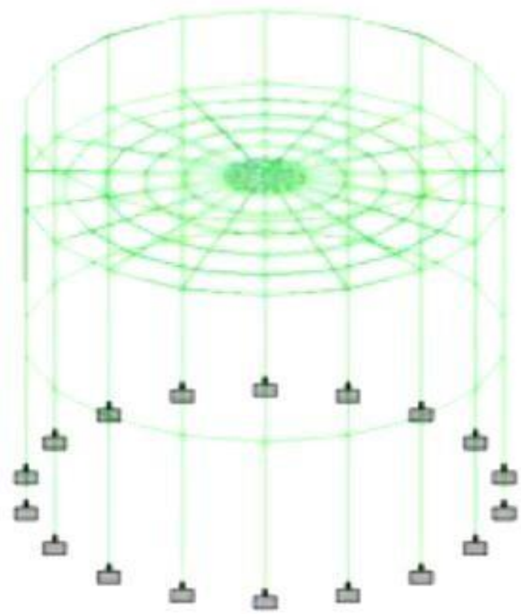
k) Top view> select the beams > delete.

l) Create mesh at the center and give the division as 1> ok.

m) Select all the beams> select fill floor grid with plates> thus the plates are created>select all beams in that and delete it.

n) Select 4 node plates, create the plate for the side walls> fill floor grid with plates and make it as plate.





1. *Property*

   a) General> property> define> rectangular>0.3 x 0.23> add> close

   b) circular>0.3> add> close

   c) gotothickness>plateelementthickness>0.15.

   d) Select rectangular and assign the horizontal beams, Select circular and assign the vertical columns, select the plate by using plate cursor and assign it.

4. *Support*

Support> fixed support> select the bottom nodes> assign selected nodes.

5. *Loads*

o) Load case detail> add>DL>add> close DL> add> pressure on full plate> global direction y>W1 = -20 KN/m^2> add> close.

p) Select the bottom plate using plate cursor and assign it.

q) DL> add> plate load> hydrostatic >select plate < select the specified plate by using cursor>done> W1= -20 KN/m^2 (depending on the z- axis direction) and W2 =0.001>global y axis& local Z axis> close.

5. *Design parameters*

a) Design concrete> IS 456>define parameter> give F_C as 20000 KN/m^2 for M20 concrete > F_Y as 415000 KN/m^2> add> close..

b) Then go to commands>design beam> add>design column> add>take off > close.

c) Select F_C> select entire structure using cursor> assign it to selected beams> similarly do it for F_Y and ratio then assign it.

d) For beams > select the beams parallel to x direction and for columns column parallel to y direction and then assign it.

6. *Analysis*

Analysis& print> add> close>analyze> run analyze> save> post processing> done.

Post-processing:

d) In post-processing you have to see the deflection, bending moment, shear, axial forces, stresses and everything by turn on the particular specified icons for all the load combinations already given.

e) Results> plate rest contour> stress type M_X> apply> close.

f) You can also see the animations of displacement in it.

<u>Result:</u>

Thus the circular water tank with flexible base is created and analyzed by using STAAD Pro.

<u>Outcome:</u>

At the end of this experiment, student acquires knowledge in creating and analyzing the circular water tank with flexible base using STAAD Pro.

Exp.No:08 **RECTANGULAR WATER TANK WITH FIXED BASE**

Aim:

To create and analyze the rectangular water tank with fixed base by using STAAD Pro

Software required:

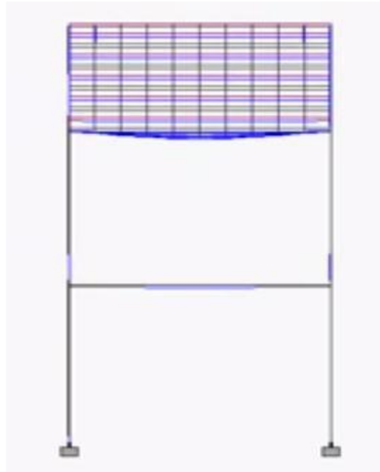
STAAD PRO V8i

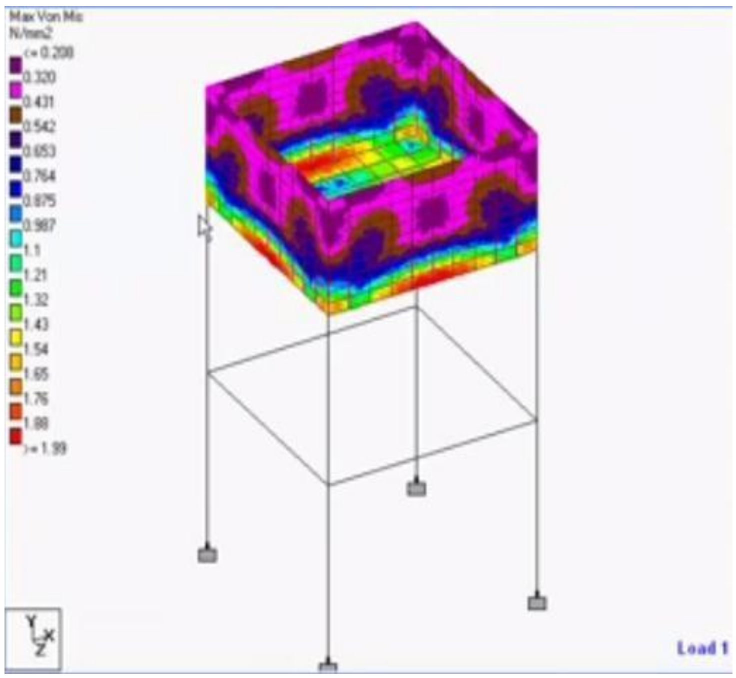
Procedure in STAAD Pro:

Preprocessing:

1. *Create model*
   a) New project> plane> length units as m> force units as KN>type the file name as rectangular water tank>next> add beam.
   b) Geometry> run structural wizard> frame model> bay frame>type the length as 4, no.of bays along length as 1, height as 10, no.of bays along height as 3>width as 4m >no.of bays along width ad 1 apply & transfer the model to STAAD Pro screen.
   c) Then change the characteristics bay along height as 4,4, 2
   d) Now apply and transfer the model to STAAD screen.
2. *Create plate*
   a) Mesh> create mesh for plates> quadrilateral meshing> ok > apply.
   b) Similarly create mesh for all side walls.
   c) Then delete all the beam element in the water tank.
3. *Property*
   a) General> property> define> rectangular>0.23 x 0.23> add> close
   b) Go to thickness> plate element thickness> 0.15.
   c) Select rectangular and assign to view, Select the plate by using plate cursor and assign it.
4. *Support*

   Support> fixed support> select the bottom nodes> assign selected nodes.
5. *Loads*
   a) Load case detail> add>DL>add> close.
   b) DL> add> pressure on full plate> global direction y>W1 = -20 KN/m^2> add> close.





c) Select the bottom plate using plate cursor and assign it.

d) DL> add> plate load> hydrostatic >select plate > select the specified plate by using cursor>done> W1= -20 KN/m^2 (depending on the z- axis direction) and W2 =0.001>global y axis& local Z axis> close.

e) Similarly give the load on the basis of z axis direction for all the side walls of the tanks.

6. *Design parameters*

a) Design concrete> IS 456>define parameter> give F_C as 20000 KN/m^2 for M20 concrete > F_Y as 415000 KN/m^2> add> close.

b) Then go to commands>design beam> add>design column> add>take off > close.

c) Select Fc> select entire structure using cursor> assign it to selected beams> similarly do it for FY and ratio then assign it.

d) For beams > select the beams parallel to x direction and for columns column parallel to y direction and then assign it.

7. *Analysis*

Analysis& print> add> close>analyze> run analyze> save> post processing> done.

Post-processing:

a) In post-processing you have to see the deflection, bending moment, shear, axial forces, stresses and everything by turn on the particular specified icons for all the load combinations already given.

b) Results> plate rest contour> stress type MX> apply> close.

c) Deflection and stresses acting in the water tank

d) You can also see the animations of displacement in it.

Result:

Therefore the determination of shear force, bending moment and design of reinforced concrete, rectangular water tank and quantities of steel, concrete, reinforcement needed are formed by using STAAD.PRO.

Outcome:

At the end of this experiment, student acquires knowledge in creating and analyzing the rectangular water with fixed base by using STAAD Pro.

Exp.No:09 **RECTANGULAR WATER TANK WITH FLEXIBLE BASE**

Aim:

To create and analyze the rectangular water tank with flexible base by using STAAD Pro.

Software required:

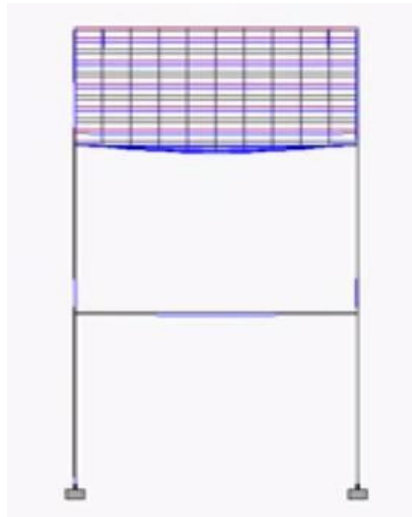
STAAD PRO V8i

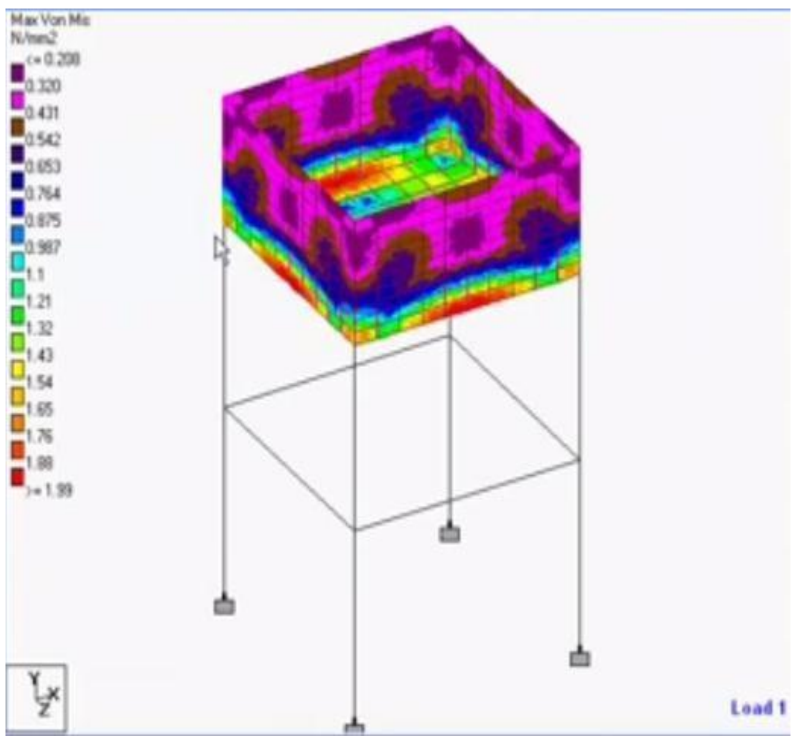
Procedure in STAAD Pro:

Preprocessing:

1. *Create model*
   a) New project> plane> length units as m> force units as KN>type the file name as rectangular water tank>next> add beam.
   b) Geometry> run structural wizard> frame model> bay frame>type the length as 4, no.of bays along length as 1, height as 10, no.of bays along height as 3>width as 4m >no.of bays along width ad 1 apply & transfer the model to STAAD Pro screen.
   c) Then change the characteristics bay along height as 4,4, 2
   d) Now apply and transfer the model to STAAD screen.
2. *Create plate*
   a) Mesh> create mesh for plates> quadrilateral meshing> ok > apply.
   b) Similarly create mesh for all side walls.
   c) Then delete all the beam element in the water tank.
3. *Property*
   a) General> property> define> rectangular>0.23 x 0.23> add> close
   b) Go to thickness> plate element thickness> 0.15.
   c) Select rectangular and assign to view, Select the plate by using plate cursor and assign it.
4. *Support*

   Support> hinged support> select the bottom nodes> assign selected nodes.
5. *Loads*
   a) Load case detail> add>DL>add> close.
   b) DL> add> pressure on full plate> global direction y>W1 = -20 KN/m^2> add> close.




Max Von Mis
N/mm2
<= 0.208
0.320
0.431
0.542
0.653
0.764
0.875
0.987
1.1
1.21
1.32
1.43
1.54
1.65
1.76
1.88
>= 1.99
Load 1

a) Select the bottom plate using plate cursor and assign it.

b) DL> add> plate load> hydrostatic >select plate > select the specified plate by using cursor>done> W1= -20 KN/m^2 (depending on the z- axis direction) and W2 =0.001>global y axis& local Z axis> close.

c) Similarly give the load on the basis of z axis direction for all the side walls of the tanks.

8. *Design parameters*

a) Design concrete> IS 456>define parameter> give F_C as 20000 KN/m^2 for M20 concrete > F_Y as 415000 KN/m^2> add> close.

b) Then go to commands>design beam> add>design column> add>take off > close.

c) Select F_C> select entire structure using cursor> assign it to selected beams> similarly do it for F_Y and ratio then assign it.

d) For beams > select the beams parallel to x direction and for columns column parallel to y direction and then assign it.

9. *Analysis*

Analysis& print> add> close>analyze> run analyze> save> post processing> done.

Post-processing:

a) In post-processing you have to see the deflection, bending moment, shear, axial forces, stresses and everything by turn on the particular specified icons for all the load combinations already given.

b) Results> plate rest contour> stress type M_X> apply> close.

c) Deflection and stresses acting in the water tank

d) You can also see the animations of displacement in it.

Result:

Therefore the determination of shear force, bending moment and design of reinforced concrete, rectangular water tank and quantities of steel, concrete, reinforcement needed are formed by using STAAD.PRO.

Outcome:

At the end of this experiment, student acquires knowledge in creating and analyzing the rectangular water with flexible base by using STAAD Pro.

Exp.No :10 **HEMISPHERICAL STEEL WATER TANK**

STEP 1: MODELLING

1. To create a base slab of tank using radial command.
2. Enter the radius of base slab as 6.15m.
3. By using snap/ beam mode version, click on translational repeat, circular repeat to create a 360◦ sweep circular tank.
4. To create a dome of tank. Select run structure wizard, click surface model select spherical cap from spherical surface.
5. Assume the dia. of sphere greater than base dia. top dia. as 0, latitude as 4 , longitude as 36.
6. Import structure to STAAD- select reference point x=z=0.

STEP 2: SUPPORT SPECIFICATION

1. Using command option, add support as fixed.
2. Change the view from +x.
3. Select the bottom nodal point of elevated W.T and then assign to selected nodes.

STEP 3: MATERIAL PROPERTIES

1. Add a material property for all plate and beam.
2. Assume the plate thickness as 200mm.
3. For a beam of cross section 250mm×250mm.
4. To assign the respective element by using assign command.
5. To change the view (i.e) 3D rendered view for the purpose of material assigned or not.

STEP 4: LOADING DETAILS

Add a dead load, live load and load combination (D.L, L.L).

STEP 5: PERFORM ANALYSIS

1. Select the perform analysis, run analysis to create a post processing mode.
2. Therefore B.M, S.F & Deflection of beams are determined.

STEP 7: DESIGN PROCEDURE

1. Change the concrete code IS 456:2000.
2. Selected parameters.
3. To assign click on command and assign.

STEP 8: DESIGN DETAILING

1. To select perform analysis, click all parameters and analyze.
2. Select the run analysis, to find overall result.

Result:

Therefore the determination of shear force, bending moment and design of reinforced concrete, rectangular water tank and quantities of steel, concrete, reinforcement needed are formed by using STAAD.PRO.

Outcome:

At the end of this experiment, student acquires knowledge in creating and analyzing the hemispherical water tank by using STAAD Pro.

Exp. No: 11 **R.C.C SLAB CULVERT**

Analysis and design a Rectangular Slab Culvert.

STEP 1: MODELLING

1. Create a slab using linear command.
2. Enter the co-ordinates for the slab of size 7m×7.5m.

STEP 2: SUPPORT SPECIFICATION

1 Using command option, add support as fixed.
2 Select the bottom nodal point and then assign to selected nodes.

STEP 3: MATERIAL PROPERTIES

1 Add a material property for all plate and beam.
2 Assume the plate thickness as 250mm.
3 For a beam of cross section 250mm×250mm.
4 To assign the respective element by using assign command.
5 To change the view (i.e) 3D rendered view for the purpose of material assigned or not.

STEP 4: LOADING DETAILS

1. Add a dead load, live load.
2. Assign a self weight factor as -1.5.
3. Add vehicle load using vehicle definition.
4. Then through load generation, generate the load.
5. When generating a load set x=y=0 and z= 3.5m.
6. Set load increment as 1.5 in x and z=y=0.

STEP 5: LOAD COMBINATION

Design the categories of load combination such as 1.5 D.L + 1.5 L.L.

STEP 6: PERFORM ANALYSIS

1. Select the perform analysis, run analysis to create a post processing mode.
2. Therefore B.M, S.F & Deflection of beams are determined.

STEP 7: DESIGN PROCEDURE

1. Change the concrete code IS 456:2000.
2. Select the required parameters and define the selected parameters.
3. Assign the defined parameters.
4. Click on command to design beam, element, take off and assign selected options.

STEP 8: DESIGN DETAILING

1. To select perform analysis, click all parameters and analyze.
2. Select the run analysis, to find overall result.

Result:

Therefore the determination of shear force, bending moment and design of reinforced concrete of rectangular slab culvert and quantities of steel, concrete, reinforcement needed are formed by using STAAD.PRO.

Outcome:

At the end of this experiment, student acquires knowledge in creating and analyzing the R C C Slab Culvert by using STAAD Pro.

Exp.No:12 **TEE BEAM BRIDGE**

Aim:

To create and analyze tee- beam bridge by using STAAD Pro

Software required:

STAAD PRO V8i

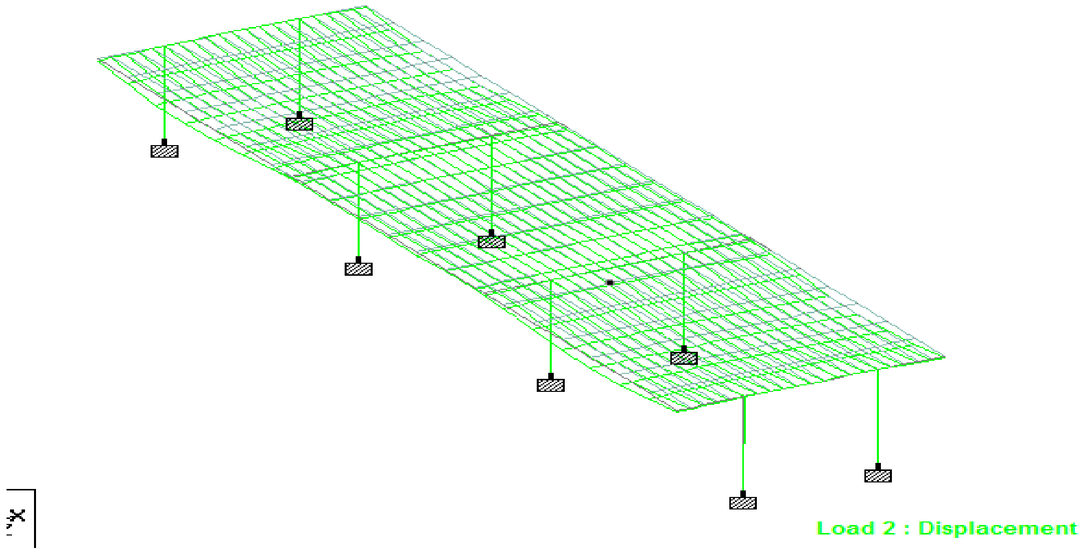
Procedure in STAAD Pro:

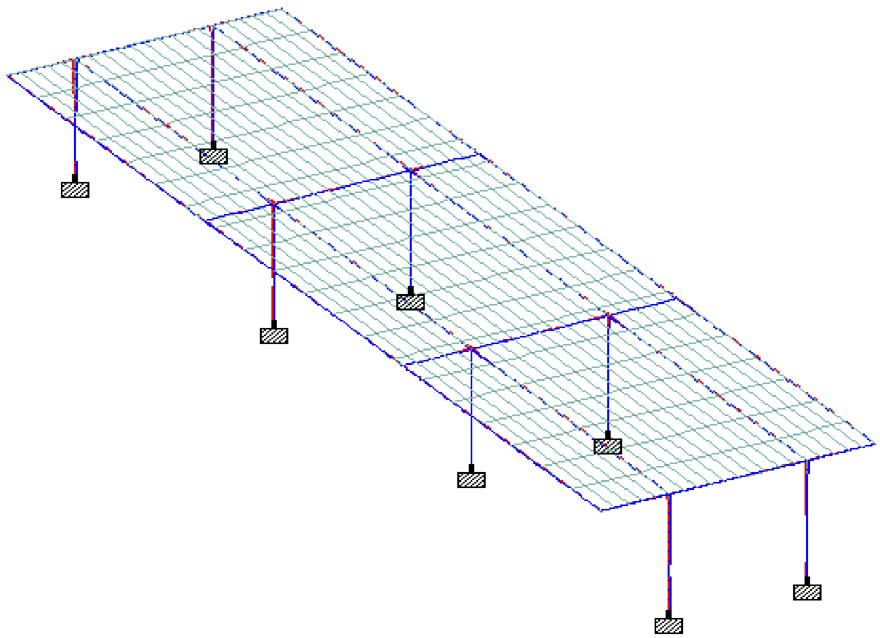
Preprocessing:

1. *Create model*
   a) New project> space > length units as m> force units as KN>type the file name as tee beam bridge >next>add beam.
   b) Snap node/ beam> edit and create a beam for 20 m in x- axis and two 10 m column vertically in the span of the beam at a distance of 5m from the edge of the horizontal beam
   c) Translation repeat> global direction Z, no.of steps=3, spacing = 25m, link steps and open the base> ok.
2. *Create plate*

   Select mesh> create mesh by connecting the four corners of the slab of the bridge.
3. *Property*
   a) General> property> define> rectangular>0.3 x 0.23> add> close
   b) Circular>0.3> add> close
   c) Go to thickness> plate element thickness> 0.15.
   d) Select rectangular and assign the horizontal beams, Select circular and assign the vertical columns, select the plate by using plate cursor and assign it.
4. *Support*

   Support> fixed support> select the bottom nodes> assign selected nodes.
5. *Loads*
   a) Definition> vehicle definition>add>close.
   b) Load case 1> add>self weight> factor -1> add> close> assign to view> assign.
   c) Load case detail> add> load generation>no.of loads to be generated= 20, preferred load to be added=1>add>close.





Bending stresses

d) Generate load> add> x coordinate= 10m > y- coordinate>10m>x increment 0> z coordinate 3m, z increment> 30 m> close.

6. *Design parameters*

a) Design concrete> IS 456>define parameter> give F_c as 20000 KN/m^2 for M20 concrete > F_Y as 415000 KN/m^2> add> close.

b) Then go to commands>design beam> add>design column> add>take off > close.

c) Select Fc> select entire structure using cursor> assign it to selected beams> similarly do it for FY and ratio then assign it.

d) For beams > select the beams parallel to x direction and for columns column parallel to y direction and then assign it.

7. *Analysis*

Analysis& print> add> close>analyze> run analyze> save> post processing> done.

Post-processing:

a) In post-processing you have to see the deflection, bending moment, shear, axial forces, stresses and everything by turn on the particular specified icons for all the load combinations already given.

b) Results> plate rest contour> stress type MX> apply> close.

c) You can also see the animations of displacement in it.

Result:

Thus the tee beam girder is created and analyzed by using STAAD Pro.

Outcome:

At the end of this experiment, student acquires knowledge in creating and analyzing the tee beam girder using STAAD Pro.

Expt.No:13 **ANALYIS AND DESIGN OF TRUSS GIRDER**

Aim:

To create and analyze the truss by using STAAD Pro

Software required:

STAAD PRO V8i

Procedure in STAAD Pro:

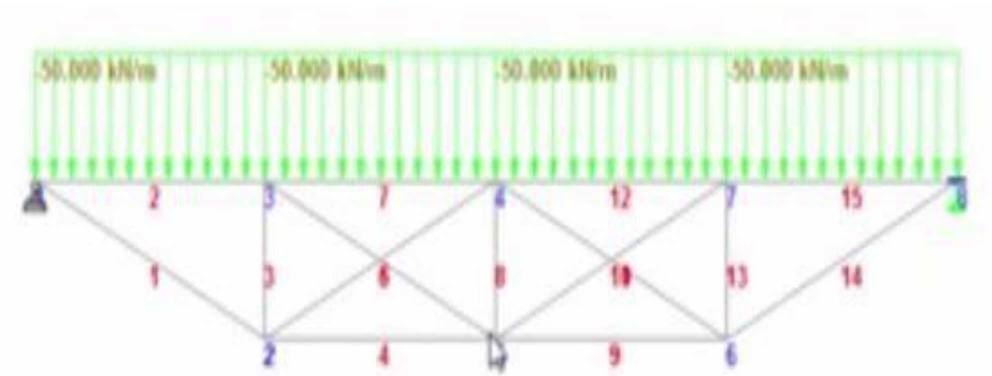
Preprocessing:

1. *Create model*
   a) New project> plane> length units as m> force units as KN>type the file name as truss> next> add beam.
   b) Snap node/ beam > edit> X in right = 4, X spacing 6 m, Y in right=1, Y spacing=3> ok.
   c) Now draw the truss model by using snap node/ beam.
2. *Property*
   a) General> property>section database> Indian> channel> IS MC 125> add
   b) Select ISMC 125>assign to view>yes.
3. *Support*

   Support> create> fixed> release the displacement in X- axis by selecting FX, MX, MY, MZ.> add>

   use cursor to sign > assign.
4. *Load*
   a) Load and definition> load case detail> add>change the title as DL> add.
   b) DL>self weight> uniform forces> W1 = -50 KN/m> add> close.
   c) Assign the load in the top the beam by using the cursor.
5. Analysis

   Analysis& print> add> close>analyze> run analyze> save> post processing> done.



Loads and supports in truss

Post-processing:

a) In post-processing you have to see the deflection, bending moment, shear, axial forces, stresses and everything by turn on the particular specified icons for all the load combinations already given.

b) Results> plate rest contour> stress type MX> apply> close.

You can also see the animations of displacement in it.

Result:

Thus the truss is created and analyzed by using STAAD Pro.

Outcome:

At the end of this experiment, student acquires knowledge in creating and analyzing the truss by using STAAD Pro.

Expt. No.14 **DECK TYPE PLATE GIRDER BRIDGES**

Design a deck type plate girder railway bridge for single tract B.G. main line loading for the following data:

1. Effective span : 24 m.
2. Spacing of plate girder : 1.9 m c/c
3. Weight of stock rails : 440 N/m (90 lb/yard rails)
4. Weight of guard rails : 260 N/m
5. Weight of fastenings etc: 280 N/m of track
6. Timber sleepers : 250 mm x 150 mm x 2.8 m @0.4 m c/c
7. Density of timber : 7.4 KN/m^3

Take permissible stresses as per Railway Steel Bridge Code.

Solution:

Step 1. Computation of loads, B.M. and S.F. in Plate girder:

Weight of main rails per meter run of track = 0.44 x 2 = 0.88 KN/m Weight of guard rails per meter run of track = 0.26 x 2 = 0.52 KN/m Weight of fastening per meter run of track @ 0.28 KN/m = 0.25 KN/m

Weight of sleeper per meter run of track = = 1.94 KN/m

Total Load = 3.62 KN/m

Self weight of girder for B.G. main line loading is w = 0.78 L = 18.72 KN/m

Total dead load per meter run of track = 3.62 + 18.72 = 22.34 KN/m

Total Dead Load = 22.34 x 24 = 536.2 KN

Dead load per girder = 0.5 x 536.2 = 268.1 KN

EUDL for B.M., for BG (revised) for loaded length L = span = 24 m is 2034 KN Impact load (I.L.) for B.M., = 2034 x 0.417 = 848.2 KN

Hence total load per track = 536.2 + 2034 + 848.2 = 3418.4 Hence load per girder, for B.M. = 0.5 x 3418.4 = 1709.2 KN M_{max} = (WL/8) = (1709.2 x 24) / 8 = 5127.6 KN/m

This occurs at the midspan of each girder. B.M. at distance x from the midspan is for end sections, L = 24 m

Hence EUDL for shear = 2231 KN

Impact load, = 2231 x 0.417 = 930.3 KN

Total (L.L + I.L) = 2231 + 930.3 = 3161.3 KN per track

Hence, (L.L + I.L) = 0.5 x 3161.3 = 1580.6 KN

S.F. due to (L.L + I.L) = 0.5 x 1580.6 = 790.3KN

Dead load per girder, W_d = 268.1 KN

Dead load shear per girder = 0.5 x 268.1 = 134.1 KN Total S.F. at each end of a girder =790.3+134.1=924.4KN For any section x from one end, L = l – x

Let W_e be the total L.L. + I.L. one length L for each girder

$F_x = R_A = (W_e (l-x))/2l$ For section at 2 m from end,

L = 24 – 2 = 22 m EUDL for shear = 2102 KN

(L.L + I.L) = 2102 (1 + 0.436) = 3018.4 KN

For both girders, (L.L + I.L) for each girder

W_e = 0.5 x 3018.4 = 1509.2 KN

F_2 = (1509.2 (24 – 2)) / (2 x 24) = 691.7 KN

Also, S.F. due to dead load = $((W_d)/2)-((W_d)/l)x$ = (268.1/2)-(268.1/24)x =134.05-11.17x At x = 2m, F = 134.05 – 111.17 x 2 = 111.7 KN

Total shear =691.7+111.7=803.4KN

Step 2. Design of central section of plate girder:

M_{max} = 5127.6 KN-m.

Economical depth $d = 1.1 \sqrt{(M / t_w \sigma_b)} = 1.1. \sqrt{(5127.6 / 10 \text{ x } 138)}$ = 2120 mm D = 1.1. $\sqrt{(5127.6E6 / 12 \text{ x } 138)}$ = 1936 mm

Hence adopt d = 2000 mm and t_w = 12 mm.

Area of web $A_w = t_w$ x d = 12 x 2000 = 24000 mm^2

Area of flange $A_f = (M/\sigma_{bt}$ x d) = (5127.6E6/138 x 2000) = 18578 mm^2

Net are of flange plates $A_p = (2/3)A_f – (1/8)A_w$ = (2/3)x18578 – (1/8)x24000 = 9386 mm^2 Keep gross area of plates = 1.2 x 9386 = 11263 mm^2

Provide two plates 500 x 12 mm each having

A_p = 2x500x12 = 12000 mm^2

Gross area of each angle = 7740 / 2 = 3870 mm^2

Choose 2 ISA 200 x 150 x 12 mm @ 13.8 kg/m, each having A = 4056 mm^2

Step 3. Design of riveted jointing connecting flange angles with theweb: (L.L.+I.L.)per girder,

per mm run =(0.5 x 224.6)/1200(1+1)=0.187 KN/mm Dead load of

track, per girder = 0.5 x 3.62 = 1.81E-3 KN/mm

Total load w on compression flange = 0.187 + 0.00181 = 0.189 KN/mm

(a) For compression flange:

(b) For tension flange: P = 147.2 mm

P = ((59.86 x 2000)/924.4) x ((7080 + 3000)/7080) = 184.4 mm

Max. Permissible pitch = 16 x 12 = 192 mm.

Hence provide uniform pitch of 180 mm throughout the length.

Step 4. Design of riveted joint connecting flange plates to flange angles:

(a) Connection of flange plate to flange angle in compression flange, at ends

V=924.4KN and one plate is available in top flange. A_f = 8112 + 1200/2 = 14112 mm^2

Bearing = 21.5 x 12 x 232E-3 = 59.86 KN, R = 36.3 KN

P = ((36.3 x 2000)/924.4) x ((14112 + 4000)/6000) = 237 mm

(b) Connection of flange plate to flange angles in tension flange at 3.66 m ends:

A_f = 7080 + 10968/2 = 12564 mm^2 (1/8)A_w = 3000 mm^2.

V = 703 KN.

P = ((36.3 x 2000)/703) x ((12564 + 3000)/10968/2)) = 293 mm.

Step 5.Design of end stiffener:

End reaction = 924.4 KN, Permissible bearing stress = 185N/mm^2

Bearing area required = (((3/4) x 924.4E3)/185) = 3748 mm^2 4t_a (150 – 10) = 3748, which gives t_a = 6.7mm

However provide 150 x 75 x 8 mm angles having A = 1742 mm^2 Actual length of stiffener = 2000 – 2 x 12 = 1976 mm Effective length of stiffener = (3/4) x 1976 = 1482 mm

l/r = 1482/69.3 = 21.4

Hence Load carrying capacity = 132.65 x 10568E-3 = 1402 KN.

Design of riveted connection for the end stiffener:

Strength of 20mm dia. rivets in double shear = 2 x (Π/4) x 21.5^2 x 100E-3 = 72.6 KN Strength of rivets

in bearing on web = 21.5 x 12 x 233E-3 = 59.68 KN

No. of rivets required = 924.4/59.86 = 16

Hence provide 8 rivets, in each row, thus making a total of 16 rivets.

Step 6. Design of intermediate stiffeners:

V = 924.4 KN,

Shear stress, f_s = (924.4E3 / 2000 x 12) = 38.5 N/mm^2

Hence for d = 1700 mm, t_w = 12 mm and

f_s = 38.5 N/mm^2 For overall depth 2438 mm and depth 2048 mm,

Try 2 ISA 125 x 75 x 8, Outstand face of web = 125 mm

f_s = (682.6E3 / 2000 x 12) = 28.4 N/mm^2

Hence Max. Spacing = 1900 mm.

Maximum permissible spacing = 1830 mm as per railway bridge code.

<u>Result:</u>

The Design procedure is completed.

<u>Outcome:</u>

At the end of this experiment, student acquires knowledge in designing a deck type plate girder railway bridge for single tract manually

VIVA QUESTIONS

Cantilever Retaining Wall and Counterfort Type Retaining Wall

1. What is a Retaining wall?

   Retaining walls are generally used to retain earth or such materials to maintain unequal levels on its two faces. The soil on the back face is at a higher level and is called back fill. Retaining walls are extensively used in the construction of basements below ground level, wing walls of bridge and to retain slopes in hilly terrain roads.

2. What are the disadvantages of gravity retaining walls?

   Gravity walls of stone masonry were generally used in the earlier days to the height of the earth fill. The advent of reinforced concrete has resulted in thinner retaining walls.

3. What are the types of retaining walls?

   Retaining wall can be classified structurally as
   - Cantilever retaining wall
   - Counterfort retaining wall

4. What is a cantilever retaining wall?

   The most common and widely used retaining wall is of cantilever type. Vertical stem resisting earth pressure one side and the slab bends like a cantilever. The thickness of the vertical slab is large at the bottom and decreases towards the top in proportion to the varying soil pressure.

5. What is a counter fort retaining wall?

   Counter fort retaining walls are used for large heights exceeding 5 mts of earth fill. In counter fort retaining wall the vertical stem is designed as a continuous slab spanning between the counter forts. Counter forts are designed as cantilever beams from the base slab.

6. What are the forces acting on a retaining wall?

   Forces acting on a retaining wall are
   - Lateral earth pressure due to the back fill
   - Vertical forces including weight of soil, stem, heel, toe, and soil fill bbove the toe.
   - The soil pressure developed to resist the earth pressute and other vertical forces acting on the heel and toe.

7. Define Active Earth pressure?

   If the soil exerts a push against the wall by virtue of its tendency to slip laterally and

Seek its natural slope (angle of repose) thus making the wall to move slightly away from the back filled soil mass. This kind of pressure is known as AEP.

8. Define Passive earth pressure?

The pressure or resistance which soil develops in response to movement of the structure towards it is called the Passive Earth Pressure.

9. Give the criteria for the design of gravity retaining wall?

1. Maximum pressure should not exceed the bearing capacity of soi (Base width).
2. No tension should be developed anywhere in the wall

$$e < \frac{b}{6} \qquad \text{i.e. } \bar{x} \leq \frac{2b}{3}$$

3. The wall must be safe against sliding.
4. The wall must be safe against sliding.
5. The wall must be safe against overturning.

10. What are the stability conditions should be checked for the retaining walls?

The stability of retaining walls should be checked against the following conditions the wall should be stable against sliding

(a) The wall should be stable against Overturning

(b) The wall should be stable against Bearing capacity failure.

11. Give the minimum factor of safety for the stability of a retaining wall?

(a) The wall should be stable against sliding = 1.5

(b) The wall should be stable against Overturning For Granular Backfill = 1.5 For cohesive backfill = 2.0

(c) The wall should be stable against Bearing capacity failure. For Granular Backfill = 1.5 For cohesive backfill = 2.0

12. If a retaining wall of 5 m high is restrained from yielding, what will be the total earth pressure at rest per metre length of wall?

Given: the back fill is cohesion less soil having $\varphi = 30°$ and $\gamma = 18$ kN/m^3 . Solution

$$C_o = 1 - \sin\phi = 1 - \sin 30° = 0.5$$

$$P_o = \frac{1}{2} K_o \gamma H^2 = \frac{1}{2} x 0.5 x 18 x 5^2 = 112.5 \text{kN/m length of wall}$$

14. What is meant by backfill?

The material retained or supported by a retaining wall is called backfill.

15. What is meant by surcharge?

The position of the backfill lying above the horizontal plane at the elevation of the top of a wll is called the surcharge.

16. What is a gravity retaining wall?

A gravity retaining wall is the one in which the earth pressure exerted by the backfill is resisted by dead weight of the wall, which is either made of masonry or mass concrete.

17. What is meant by submerged backfill?

The sand fill behind the retaining wall saturated with water is called submerged backfill.

18. What is the function of counterforts in a retaining wall?

The stem of the counterfort retaining wall acts as a continuous slab supported on counterforts. The counterforts take reactions both from the stem as well as the heel slab. Since the active earth pressure on stem acts outwards, and net pressure heel slab acts downwards, the counterforts are subjected to tensile stresses along the outerface of the counterforts.

19. What is meant by back anchoring of retaining wall?

When the height of retaining wall is much more, it becomes uneconomical to provide counterforts. In order to reduce the section of stem etc. in the high retaining walls, the stem may be anchored at its back. The anchor practically takes all the earth pressure and B.M and S.F. in the stem are greatly reduced.

20. When is the design of shear key necessary?

When the wall is unsafe in sliding, shear key will have to be provided.

Solid Slab and RCC Tee Beam Bridges

1. Write the advantages of prestressed concrete bridges

High strength concrete and high tensile steel, besides being economical make for slendersections which are aesthetically superior.In comparison with steel bridges prestressed concrete bridges require very little maintenance.Prestressed concrete bridges can be designed as class 1 type structures without any tensile stresses under service loads, thus resulting in a crack free structure.

2. What are span range for solid slab and T-beam slab decks?

- Span rangesFor solid slab decks –10 to 20 m
- For T-beam slab decks –20 to 40 m

3. Write a short note on post tensioned bridge decks.

Post tensioned bridge decks are generally adopted for longer spans exceeding 20 m. Bridge

Decks with precast post tensioned girders of either T –type or box –type in conjunction with a cast in situ slab are commonly adopted for spans exceeding 30 m.

4. Why the deck slab of prestressed concrete bridges is mostly made of non prestressed concrete?
The pretensioned girders are provided with straight tendons. The use of seven wire strands has been found to be advantageous in comparison with plain or indented wires. The deflected strands are employed in larger girders.

5. What are the advantages of post tensioning system?
   - Post tensioning is suitable for heavy cast in place members.
   - The waiting period in the casting bed is less.
   - The transfer of prestress is independent of transmission length

6. List any four mechanical prestressing systems adopted for bridges in India.
   - Includes weights with or without lever transmission
   - Geared transmission in conjunction with pulley blocks
   - Screw jacks with or without gear drives
   - Wire winding machines

7. What is the span ranges for different types of slabs and beams?
   - Solid slab decks –10 to 20 m
   - T-beam slab decks –20 to 40 m
   - Single or multicell box girder –30 to 70 m
   - Precast box girders –more than 50 m
   - Simply supported, continuous beams –20 to 500 m
   - Composite bridge decks –20 to 30 m
   - Precast prestressed I and T -beam –7 to 36 m
   - Y –beams –15 to 30

8. What are the types of bridges in based on materials?
   1. Timber
   2. Stone masonary
   3. Iron
   4. Steel bridges
   5. R.C.C.Bridges
   6. Prestressed

9. Explain the types if decks slab in R.C.C Bridges?
   1. Tee Beam and slab bridge deck
   2. Continuous bridge deck
   3. Balanced double cantilever bridge

10. What are the factors should be consider while designing of solid slab?
   - Material
   - Environmental
   - Location
   - Labour requirements

11. What are type of loading standards should be considered while designing of Bridge decks?
   1. IRC class AA loading
   2. IRC class 70R loading
   3. IRC class A
   4. IRC class B loading

12. Write the loading are consider in while designing slab deck?
   1. Dead load
   2. Live load (IRC loading standards)
   3. Dynamic load

13. Write the loader provision should consider in solid slab?
1.IRC 21
2.IRC 6

14. Write the types of rational methods in design of bridge?
   1. Courbon's method
   2. Guyon massonent method
   3. Hendry jaeger method

15. Explain the courbon's method condition?
   1. The ratio of span to width of dek is greater than 2 but less than 4
   2. The longitudinal girders are interconnected by at least by at least five symmetrically spaced cross girders

16. What are the maximum & minimum size of R.F?
The maximum size of R.F shall be 40mm dia or a section of equivalent are minimum size of R.F shall be 8 mm dia

17. List out the components of Tee – Beam Bridges?
   - Solid Slab
   - Longitudinal Beam
   - Transverse Beam
   - Column or Pier
   - Footing

18. List out what are the forces should consider in solid slab bridges?
   - Dead load
   - Live Load
   - Seismic Load

19. Write down the design parameters in solid slab bridge?
   - B.M – Positive beam – Negative beam
   - Shear force

20. What are methods of bearing used in Tee – Beam bridges
   - Rubber pad
   - Elastomeric
   - Steel Rockers

RCC Water Tanks

1. Mention the grade of concrete which is used in the construction of water tank.

   Richer concrete mix of grades M20 to M30 are commonly used in the construction of water tanks. High quality concrete, in addition to providing water tightness, also has higher resistance to tensile stresses developed in the tank walls.

2. Mention the three factors that must be considered while designing a RCC tank.
   - Strength
   - Water tightness
   - iii. Overall stability

3. Water are the types of reinforced concrete water tanks?
   - Tanks resting on ground
   - Underground tanks
   - elevated water tanks

4. Mention the reinforcement details that should be provided in a water tanks. Minimum area of steel is 0.3 percent of gross area of section upto 100mm thick, reduced to 0.2 percent in section up to 450mm thick. For sections above 225mm thick, provide two layers of reinforcement. The percentage of reinforcement in base or floor slab resisting directly on ground must be not less than 0.15% of the concrete section. The minimum cover to all reinforcement should be not less than 25mm or the diameter of the bar whichever is greater.

5. Define the following term: Dome:

   A Dome is defined as a thin shell generated by the revolution of a regular curve about one of its axes.

6. Define the following terms:

   i. Latitude:

   The circle of each ring in a dome is called Latitude.

   ii. Meridian circle:

   The circle drawn through two diametrically opposite points on a horizontal diameter and the crown is known as meridian circle.

7. Define the following terms:

   i. Radial:

   The joint between successive horizontal rings is called radial.

   ii. Meridian thrust:

   The reaction between the rings is tangential to the curved surface giving rise to compression along the medians. The compressive stress is called meridional thrust or meridional compression.

8. Mention the thickness and steel requirement of dome.

   A minimum thickness of 7.5cm is provided to protect steel. Minimum steel requirement is 0.15% for mild steel bars and 0.12% for HYSD bars of the sectional area in each direction meridionally as well as along the latitudes.

9. What are the three types of joints in water tank?
   - Movement joints
     (a) Contraction
     (b) Expansion
     (c) Sliding
   - Construction joints
   - Temporary open joints

10. Find out the diameter of a circular tank which is having a flexible base for capacity of 200000 liters. The depth of water is to be 4m, including a free board of 200mm.
    Effective depth of water = 4 -0.2 = 3.8m
    Let D be the inside diameter of the tank.
    Volume, V = $\Pi/4xD^2xdepth$ $(200000x10^3)/10^6$ = $\Pi/4xD^2x3.8$ D= $\sqrt{(200x4)/(\Pi x3.8)}$ D = 8.18m D = 8.2m

11. What is the foundation specification for small capacity tanks?
    For small capacity tanks individual footings for columns can be provided. Infact, the type of footing will depend upon the nature of soil and type of staging. In case of low lying areas of low safe bearing capacity with high ground water table, pile footings are provided. In any case of foundation slab, lean mixes of 1:4:8, 150mm thick may be provided as leveling course.

12. What are the methods available for the analysis of circular tank?
    i. IS code method
    ii. Reissner's method
    iii. Carpenter's method
    iv. Approximate method

13. What are movement joints in water tanks?
    These joints require the incorporation of special materials in order to maintain water-tightness while accommodating relative movement between the side of the joints. All movement joints are essentially flexible joints.

14. What is contraction joint in water tanks?
    A contraction joint is a typical movement joint which accommodates the contraction of the concrete.

15. What is meant by expansion joint in water tanks?
    It is a movement joint with complete discontinuity in both reinforcement and concrete, and is intended to accommodate either expansion or contraction of the structure.

16. What are underground water tanks?
    Underground water tanks are used for storage of water received from water supply mains operating at low pressures, or received from other source.

17. What are conditions under which the walls of underground water tanks designed?
(a) Tank full with water, with no earth fill outside.

(b) Tank empty, with full earth pressure due to saturated earth fill.

18. What are the four components of design of underground water tanks?
(i) Design of long walls
(ii) Design of short walls
(iii) Design of roof slab
(iv) Design of base slab

19. What are two methods of analysis of rectangular tanks?
(i) Approximate analysis
(ii) Exact analysis based on elastic theory

20. Where are domes used?
(i) Roof of circular areas
(ii) Circular tanks
(iii) Hangers
(iv) Exhibition halls, auditoriums and planetariums and
(v) Bottom of tank, bins and bunkers.

Plate Girder Bridges and Truss Girder Bridges

1. What is meant by castellated beam?
A rolled beam with increased depth is to be castellated. To obtain such sections, azigzag line is cut along the beam by an automatic flame-cutting machine. The two halves thusproduced are rearranged so that the teeth match up and the teeth are then welded together.

2. How the beams are failed?
- Bending failure
- Shear failure
- Deflection failure

The designs are based on these three failures which are to be determined

3. What do you mean by bending failure?
Bending failure may be due to crushing of compression flange or fracture of the tensionflange of the beam. Instead of failure due to crushing, the compression flange may fail by acolumn like action with side ways or lateral buckling. Collapse would follow the lateralbuckling.

4. What is the maximum deflection that to be allowed in steel beams?
The deflection of a member, shall not be such as to impair the strength or efficiency of the structure and lead to finishing. The deflection is generally should not exceed 1/325 of the span.

5. What is web crippling?

Web crippling is the localized failure of a beam web due to introduction of anexcessive load over a small length of the beam. It occurs at point of application ofconcentrated load and at point of support of a beam. A load over a short length of beam cancause failure due to crushing and due to compressive stress in the web of the beam below theload or above the reaction. This phenomenon is also known as web crippling or webcrushing.

6. What are laterally supported beams?

The beams which are provided with the lateral supports either by embedding thecompression flange in the concrete slab or by providing effective intermediate (support)restraints at a number of points to restrain the lateral buckling is called laterally supportedbeams.

7. Mention the advantages of using rolled steel wide flange section as beams

- More section modulus
- Lesser area
- Economical

8. Why does buckling of web occur in beams?

- Diagonal compression due to shear
- Longitudinal compression due to bending
- Vertical compression due to concentrated loads

9. Under what situations the plated beams are used?

When a bending moment is large which cannot be resisted by the largest available rolled beam section? The depth of the beam is restricted due to headroom requirements.

10. Why intermediate stiffeners are required for plate girders?

The web of the plate girder relatively being tall and thin it is subjected to buckling. Hence it is stiffened both vertically and horizontally using intermediate stiffeners.

11. What do you mean by curtailment of flanges?

The section of a plate girder is to be designed first at mid span. The bending moment will goes on decreasing towards the supports. Hence the flange plates, provided at the maximum section can be curtailed.

12. What is the purpose of providing the bearing stiffener?

It prevents the web from crushing and buckling sideways, under the action of concentrated loads. It relieves the rivets connecting the flange angles and web, from vertical shear.

14. Name the components of a plate girder.

Web plate Vertical or transverse stiffeners

Flange plate	Bearing stiffeners
Flange angles	Longitudinal or horizontal stiffeners
Flange splice plates	End bearings or end connections
Web splice plates	

15. Mention the basic design assumptions of a plate girder?

- The web plate resists the shear force.
- The shear stress is uniformly distributed over whole cross sectional area of web.
- The flanges resist the bending moment

16. Where the plate girders are used?

The plate girders are used in the buildings where the span is more and heavy loads are expected and in the bridges. Most commonly they are used in the bridges.

17. What are the methods that are adopted to determine the flange design?

- Flange area method
- Moment of inertia method

The former method is an approximate method, which is used for determining the trial section. In this method, it is assumed that the stress distribution in the tension and compression flanges is uniform, whereas in the latter case it is the exact method and is recommended by the IS code. Generally, the section designed by the flange area method is checked by this method.

18. What is the economical depth of a plate girder?

The economical depth of a plate girder is

$d = 1.1\sqrt{\frac{M}{\sigma_{bc} t_w}}$ where σ = permissible bending stress in compression in N/mm2.

tw= thickness of the web plate.

19. The pitch of the rivets connecting cover plates with flanges of rolled steel beam is designed for what force?

These rivets are designed for horizontal shear between the flange plate and flange angles. Since the vertical load is transferred by the flange plates to the flange angles by direct bearing, there will be no vertical shear due to the vertical load. Here the rivets will be in single shear.