STRUCTURAL DESIGN LABORATORY SYLLABUS

1	Introduction to Staad pro
2	Analysis of Slabs.
3	Analysis of Single Story Unsymmetrical Portal Frame.
4	Analysis of Plane Roof Truss(Dl+Ll)
5	Analysis of Plane Roof Truss (Dl+Wl)
6	Design of One Way
7	Design of Two-Way Slab.
8	Analysis and Design of Two Storied RCC Framed Building
9	Design of Isolated Footing
10	Design of Combined Footing

Experiment-1

Introduction to Staad pro

STAAD:- Structural Analysis and Design

Introduction to STAAD Pro:

- It is a structural analysis and design computer program Originally developed by research engineers international at Yorba Linda, CA In 1977. In late 2005, research engineers international was bought by Bently systems.
- An older version called STAAD-III for windows is used by lowa state university for educational purpose for civil & Structural engineers.
- The commercial version, STAAD pro is one of the most widely used structural analysis & design software products world wide. It supports several steel, concrete & timber design codes.
- It can make use of various factors of analysis from one traditional 1^{st} order static analysis and 2^{nd}order p-delta analysis, Geometric non-linear analysis or a buckling analysis. It can also make various forms of dynamic analysis from model extraction to time history & response spectrum analysis.
- Auto is a draft's man software STAAD PRO is Analysis software.

Types of structural designing software’s:

The different types of structural design software’s are

- SAP 2000
- E-TABS
- RAM STRUCTURAL SYSTEM
- MIDAS GEN & CIVIL
- STAAD PRO
- ANSYS

Advantages of STAAD PRO Structural Analysis software:

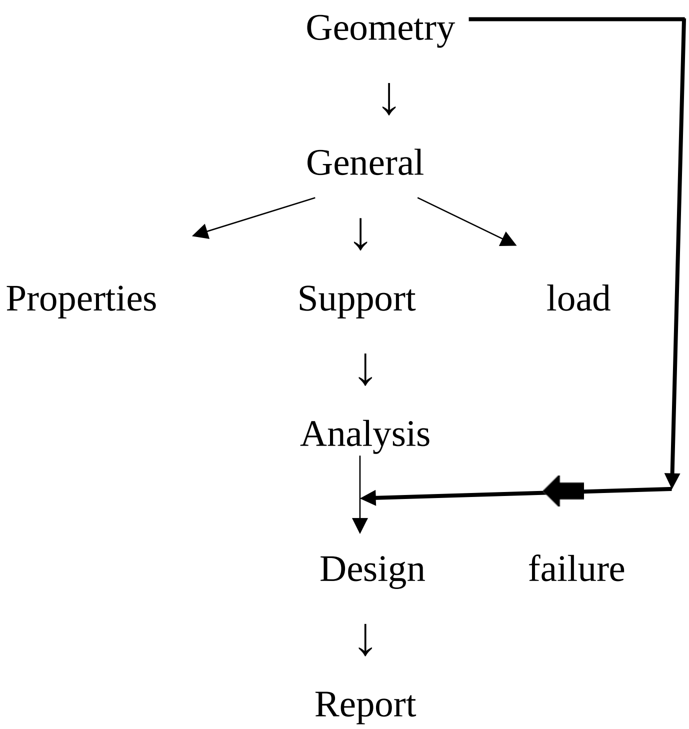
- Extremely Flexible 20/30 modelling environment.
- covers all aspects of structural engineering.
- Broad spectra of design loads.
- Interconnected and open architectures.

Basic codes for Rec structural design:

- IS 456:2000- plain & reinforced concrete for code of practice (IV revision)
- 158001 2007 - Steel design.

Loading standard codes:-

The loads to be considered for structural design are specified in the following standards.



Flow chart of Analysing a model

- In the (part 1 to 8): 1987 code of portion to design loads better then earthquakes to buildings & structures.

Part-1 : Dead load

Part-2 : Imposed (live leads)

Part-3: Wind load

Part-4: Snow load

Part-5: special load & load combinations

- IS 1893: 2002- criteria for earthquake resistant design of Structures (IV revision)
- IS 13920: 1998- Ductile detailing of reinforced concrete structures Subject to seismic forces.

Design Handbook:

The bureau of IS have also published to the 1978 versions of the code Although the handbooks need to be updated to bring them in the with the recently revised (2000 versions) of the code.

- SP16-1980-Design needs (for reinforced concrete) to IS 456:1978
- SP 24-1983-explanatory hard book on IS 450:1978
- SP 34 1987 Handbook on concrete reinforcement & detailing.

Steps involved in analysing a model

- **inputting the job information**

Firstly the information of the project is within after opening the Staad. As the name of the project job clients name & the date when project started.

- **Generating the 30 model Geometry**

There are two methods of creating a structure data in Staad.

(a) Using the command file also called the Staad editor method.

(b) Using the graphical user interface GUI.

- **Assigning the material:**

As after creating the beams & columns will be assign material to them as we require our design is concrete design. Hence, we have assigned the concrete material to the beams & columns.

- **Specifying material properties:**

The properties of the beams & columns are their size (width. depth. of cross section). So, with the help of this command, we have inputted the different properties (as Circle, rectangle, square) and assign these properties of specified members.

- **Specifying material constants:**

As we assigned the concrete material, so default we have the constants of concrete and we don't need to use this command separately or if we need to change the constants we can do say by this command.

- **specifying supports:**

The supports are 1st created (as the created fixed Supports) and then these are assigned to all the lower most of structure where we going to design the foundation.

- **Specifying Loads:**

This is done in following steps.

a) firstly creating all the load cases.

b) Then assigning them to respective members and nodes.

- **Dead loads:**

The load coming on framed structure due to its self-weight of beams, columns, slabs or walls. Thus, load will act as uniformly disturbed loads over the supporting beams.

- **Live loads:**

The live load comes on structure due to extra necessary things in the house. There will be different live loads acting in the structure due to different uses of building.

1. The floor load coming on beams from the trapezoidal toad on the longer beam of floor area.

2. The floor load coming on beams from perpendicular load on the longer beam of floor area

3. There will be some moments.

4. wind load & load combinations.

- **specifying the analysis type**:

We require specifying analysis command which we need in linear static type choosing static check , we will add this command.

- ❖ Post Analysis print command
- ❖ Run analysis
- ❖ Design the structure
- ❖ Report the documentation.

Experiment No-2

Analysis of Single Story Unsymmetrical Portal Frame.

AIM- To analysis and design a 2-D frame

Software used : STAAD PRO

Procedure:-

1. Select Staad Pro v8i

2. Check that the license configuration is according to Indian design Codes.

3. Select New Project and in that Select Plane and right of screen give file name as 2D-Frame analysis and design.

4. check the input units are in meters and Kilonewton's and Click NEXT.

5. Select Add beams and click on Finish

6. select modelling phase which consists of. Five steps

1)Setup

2) Geometry

3) General

4) Analysis and Print

5) Design.

7. Select geometry and enter co-ordinates of our Structure in x, y and z directions as

For node 1: y = 0 z =0

For node 2:- Y = 2 z=0

For node 3: y = 4 z = 0

For node 4 : Y= 0 z = 0

For node 5:- Y = 2 z = 0

For node 6:- y = 4 z = 0

For node7:- Y =0 z = 0

For node 8:- Y = 2 z = 0

For node9:- Y = 4.5 z = 0

8.use ADD BEAM method to connect all the nodes

9. Next Select general option in that Select Properties, define the Properties of beam by giving depth and width of beams and columns and assign the Properties to whole structure

Eg:-

For beams YD=0.45m and ZD=0.30m

For Columns YD=0.30m and ZD=0.30m

10. Next Select SUPPORT in that create support for example fixed and assign the created Supports to the nodes.

11. Next Select LOAD AND DEFINITIONS, click on load Case details and click on Add Select dead & Live Load and give loads

Example:

1.self-weight = -1

2. uniform Force=-2KN/m3

Again, select load case details, click on add, Select Define combinations and give combination factor for dead load and live load as 1.5

12. Now assign the loads on respective member by using assigning option.

13. Next Select material option and check that all the materials are in Concrete are not for concrete design

14 Now Select select all Analysis/Print option in that select all and click on Add

15. Next Select design option in that select Concrete design and select code as Is 456

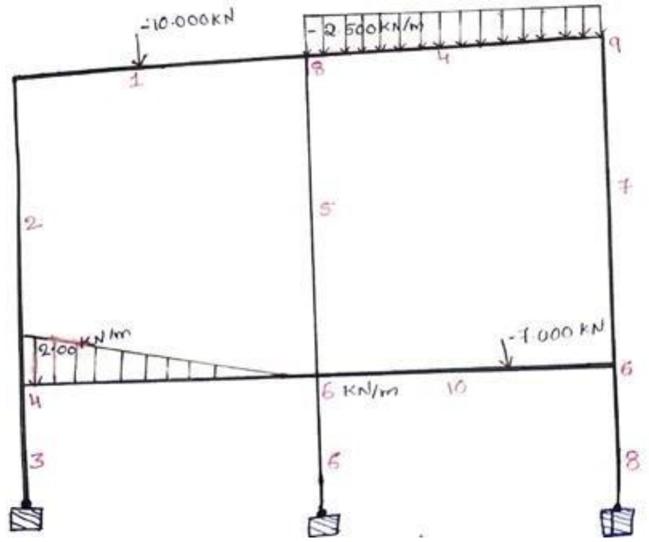
.16. Now select commands option and add design beam , design column & Take off

17. Next Select Parameters like FC, FY main, FYSE, MAX main, max sec etc..

18. Now assign above selected Parameters and commands to our model

19. Now Click on analysis in that Run analysis (Ctrl+F5)

20. Now our design results will be displayed on a separate window. Check that there are no Errors and warnings with at least one error the design will not be carried out.



Result:-

EXPERIMENT-3

ANALYSIS OF SINGLE STOREY UNSYMMETRICAL PORTAL FRAME

Aim:

To analyze the single storeyed unsymmetrical portal frame using STADD pro

Hardware & software required:

Computer with specified configuration with installed STADD Pro Software

PROCUDURE AND ANALYSIS STEPS:

Staad foundation:

Analysis and design carried in Staad and post processing in staad gives the load at various supports. These supports are to be imported into these software to calculate the footing details i.e., regarding the geometry and reinforcement details.

This software can deal different types of foundations :-

SHALLOW (D<B)

1. Isolated (spread) footing
2. Combined (strip) footing
3. Mat (raft) footing

DEEP(D>B)

1. Pile Cap
2. Driller Pier

a). Isolated footing is spread footing which is common type of footing.

b). Combined Footing or Strap footing is generally laid when two columns are very near to each other.

C). Mat foundation is generally laid at places where soil has less soil bearing capacity.

d). pile foundation is laid at places with very loose soils and where deep excavations are required. So depending on the soil at type we has to decide the type of foundation required .Also lot of input data is required regarding safety factors, soil, materials used should begiven in respective units .After input data is give software design the details for each and every footing and gives the details regarding

1. Geometry of footing

2. Reinforcement

3. Column layout

4. Graphs

5. Manual calculations

These details will be given in detail for each and every column.

Another advantage of foundations is even after the design; properties of the members can be updated if required.

The following properties can be updated

1.Column Position

2.Column Shape

3.Column Size

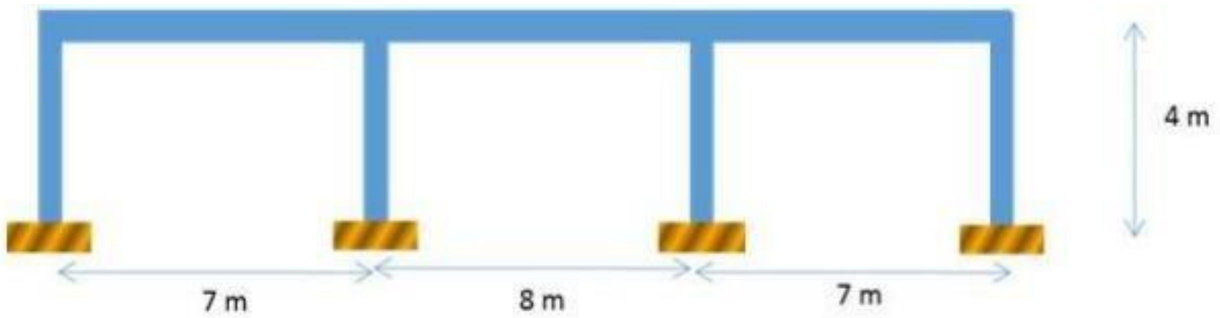
4.Load Cases

5.Support List

The single storey unsymmetrical portal frame represents the center line diagram of the building in staad pro. Each support represents the location of different columns in the structure. This structure is used ingenerating the entire structure using a tool called transitional repeat and link steps. After using the tool the structure that is created can be analyzed in staad pro under various loading cases.

Below figure represents the skeletal structure of the building which is used to carry out the analysis of our building. All the loadings are acted on this skeletal structure to carry out the analysis of our building. This is not the actual structure but just represents the outline of the building in staad pro.

A mesh is automatically created for the analysis of these building.



Load Conditions and Structural System Response :

The concepts presented in this section provide an overview of building loads and their effect on the structural response of typical wood-framed homes. As shown in Table, building loads can be divided into types based on the orientation of the structural action or forces that they induce: vertical and horizontal (i.e., lateral) loads. Classification of loads are described in the following sections.

Building Loads Categorized by Orientation:

Types of loads on an hypothetical building are as follows.

Vertical Loads

Dead (gravity)

Live (gravity)

Snow(gravity)

Wind(uplift on roof)

Seismic and wind (overturning)

Seismic(vertical ground motion)

Horizontal (Lateral) Loads:

Direction of loads is horizontal w.r.t to the building.

Wind Seismic(horizontal ground motion)

Flood(static and dynamic hydraulic forces

Soil(active lateral pressure)

Dead load :

calculation Weight=Volume x Density

Self weight floor finish=0.12*25+1=3kn/m^2

The above example shows a sample calculation of dead load.

Dead load is calculated as per IS 875 part 1

Generally for any structure live load is taken as 25 N/mm for design.

Live loads are calculated as per IS 875 part 2

Wind loads:

In the list of loads we can see wind load is present both in vertical and horizontal loads. This is because wind load causes uplift of the roof by creating a negative(suction) pressure on the top of the roof After designing wind load can be assigned in two ways

1. collecting the standard values of load intensities for a particular heights and assigning of the loads for respective height.

2. calculation of wind load as per IS 875 part 3.

We designed our structure using second method which involves the calculation of wind load using wind speed. In Hyderabad we have a wind speed of 45 kmph for 10 m height and this value is used in Calculation.

Floor load:

Floor load is calculated based on the load on the slabs. Assignment of floor load is done by creating a load case for floor load. After the assignment of floor load our structure looks as shown in the below figure.

The intensity of the floor load taken is: 0.0035 N/mm2

-ve sign indicates that floor load is acting downwards

Load combinations:

All the load cases are tested by taking load factors and analyzing the building in different load combination as per IS456 and analyzed the building for all the load combinations and results are taken and maximum load combination is selected for the design Load factors as per IS456-2000 When the building is designed for both wind and seismic loads maximum of both is taken.

LIVE LOAD	DEAD LOAD	WIND LOAD
1.5	1.5	0
1.2	1.2	1.2
0.9	0.9	0.9

Because wind and seismic do not come at same time as per code. Structure is analyzed by taking all the above combinations.

RESULT :-

EXPERIMENT-4

DESIGN OF ISOLATED FOOTING

AIM:

To analyze and design of isolated footing

SOFTWARE USED :-

STAAD PRO

PROCEDURE:-

1.Select staad pro V8i

2.Check that the license configuration is according to Indian design codes

3.Select new project and in that select space and right of screen give file name as slabanalysis and design

4.check the input units are in meters and kio newtons and

click next5.select add beams and click on finish

6.select modelling phase which consists of five steps.

1.setup

2.geometry

3.general

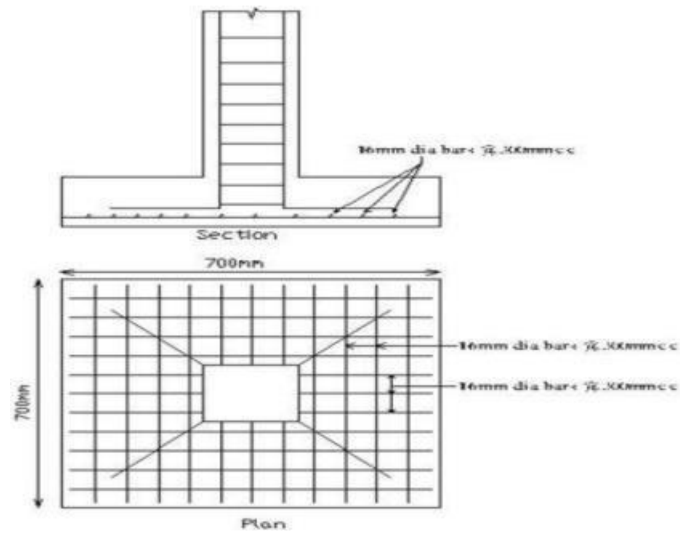
4.analysis and

print5.design

Reinforced Concrete Footings:

Footing comprises of the lower end of a column, pillar or wall which i enlarged withprojecting courses so as to distribute load.

Footings shall be designed to sustain the applied loads, moments and forces and the inducedreactions and to ensure that any settlement which may occur shall be as uniform as possible and the safe bearing capacity of soil is not exceeded. In sloped or stepped footings, the effective cross-section in compression shall be limited by the area above the neutral plane, and the angle of slope or depth and location of steps should be such that the design requirements are satisfied at every section. Design Procedure of Column Footings | Foundation Design Here is a step-by-step guide to Column Footing Design:



Step 1 :-

Area required for

footing Square = B

= (w+w1)/P0

Where, Po = safe bearing

capacity of soil w1 = self weight

of footing

w = self weight of footing

For Rectangle = b/d = B/D A

= b x dNet upward pressure

on the footing q/p = W/A

Step 2 :-

Bending Moment

Critical section for maximum bending moment is taken

at the face of the Column

For a square

footing, Mxx = q x

B/8 (L – a)2 Mxx =

q x L/8 (B – b)2

Myy = q x B/8 (L –

a)2 **Step 3:-**

To fix the depth of the footing shall be greater of the following:

Depth from bending moment

consideration $d = \sqrt{(M/Qb)}$

where, Q = moment of required factor

Depth from shear consideration Check for one way shear

Check for two way shear or punching shear

Critical shear for one way shear is considered at a distance ‘d’ from

face of the column. Shear force, $V = qB\ [\ ½(B – b)\ d]$

Nominal shear stress, $Tv = k\ .\ Tc$ $Tc = 0.16\sqrt{fck}$

Step 4 :-

Check for two way shear

Critical section for two way shear is considered at a distance at a distance d/2 from all the faces of the column.

SF, $V = q\ [\ B2 – (b + d)2\ ]$

SF, $V = q\ [L \times B – (a + d)(b + d)]$

Nominal shear

stress, $Tv =$

$V/2((a+d)(b+d)d)$

for a rectangle $Tv = V/4((b+d)d)$

for a square $Tv = k\ .\ Tc$ $k = 0.5 + \beta > 1$; [Beta β = ratio of sides

of the column $Tc = 0.16\sqrt{fck}$

Area of steel, $Ast = M/((\sigma)stjd)$

CALCULATION FOR BOTTOM STEEL

Mu about X1 X1 = (pe max x length^2/2)= **90.27** KN-m per meter

Mulimit = **450.81** KN-m per meter

The section is singly reinforced

$$A_{st} = \frac{0.5 f_{ck}}{f_y}\left[1-\sqrt{1-\frac{4.6M_u}{f_{ck}bd^2}}\right]bd$$

Hence, Ast = **640.194** mm^2

Min Ast = **484.800** mm^2 (0.12 % for slab, cl 26.5.2.1)

Spacing required = **176.66** mm (considering max of above two calculated values of Ast)

Spacing provided = 150 mm

pt provided = 0.19 %

pt required = 0.16 %

provide a maximum : 0.19 %

Hence provide 12 mm dia bar @ 150 mm c/c parellel to length of footing (|| to Z)

Mu about N1 N1 = (pe max x length^2/2)= **93.30** KN-m per meter

The section is singly reinforced

Calc. Ast = **662.496** mm^2

Min Ast = **484.8** mm^2 (0.12 % for slab, cl 26.5.2.1)

Spacing = **170.71** mm (considering max of above two calculated values of Ast)

Spacing provided = 150 mm

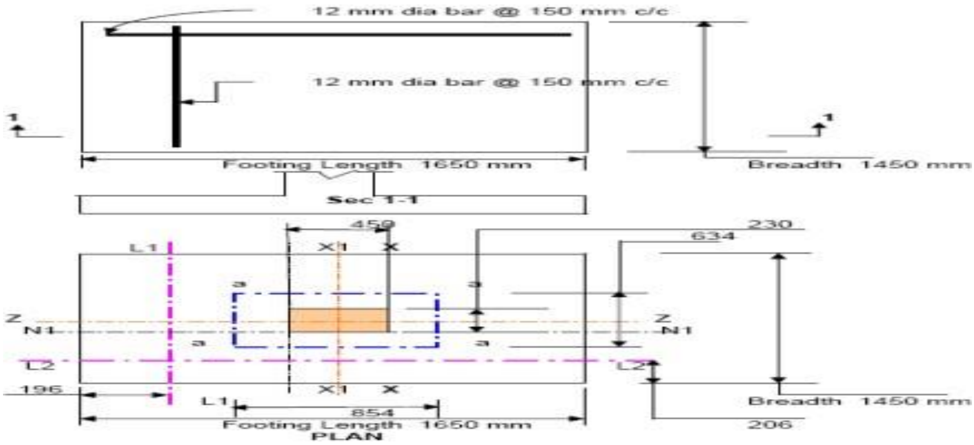
pt provided = 0.19 %

pt required = 0.16 %

provide a maximum : 0.19 %

Hence provide 12 mm dia bar @ 150 mm c/c parellel to breadth of footing (|| to X)

Arrangement of bottom reinforcement as per above design is shown below



CHECK FOR ONE WAY SHEAR :

One way shear at critical section L1- L1

Distance of critical sec. from edge of footing = **0.196** m

Shear force Vs =pe max x 0.196 x 1m width of footing = **98.289** KN

Shear stress τ_v = Vs/bd = **0.243** N/mm^2

τ_c = **0.32** N/mm^2

tv < tc hence O.K.

One way shear at critical section L2- L2

Distance of critical sec. from edge of footing = **0.206** m

Shear force Vs =pe max x 0.206 x 1m width of footing = **103.304** KN

Shear stress τ_v = Vs/bd = **0.256** N/mm^2

τ_c = **0.32** N/mm^2

tv < tc hence O.K.

CHECK FOR TWO WAY SHEAR

Ref. cl 34.2.4 and cl.31.6.3 of IS 456 : 2000

Allowable shear stress $\tau_{v\ allowable}$ = $k_s \tau_c$

k_s = (0.5 + b/c) = **1.011111111** >1

Hence, k_s= **1**

τ_c = 0.25 (fck)$^{0.5}$ = **1.118033989** N/mm^2

$\tau_{v\ allowable}$ = k_s x τ_c = **1.118033989** N/mm^2

Shear force Vs = 501.476 (1.65 x 1.45 - 0.854 x 0.634) = **928.26** KN

Length of critical section = 2 x (854 + 634) = **2976** mm

Area of the critical section (length of critical sec x eff. d) = **1202304** mm^2

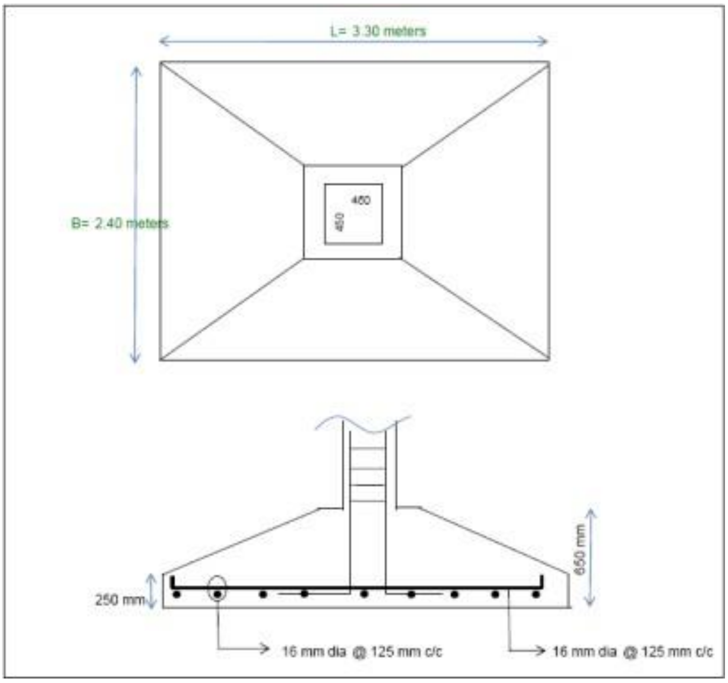
Hence shear stress τ_v = **0.772** N/mm^2

tv < allowable hence O.K.

1 **Footing Size Design**

Load	Pu	1500 KN
Design Load	P	1100 KN
Moment in x dir	Mux	30 KN-m
Moment in y dir	Muy	30 KN-m
Column size	cx	450 mm
	cy	450 mm
SBC	q	150 KN/sqm
Footing Size required	A req	7.33 sqmm
Footing Size Provided	L	3.30 meters
	B	2.40 meters
Area Provided	A prvd	7.92 meters
	Zx	3.17
	Zx	4.36
Net upward pressure	Nup	150 KNm2

Footing Size OK



RESULT:-

EXPERIMENT-5

ANALYSIS DESIGN OF COMBINED FOOTING

To analyze and design the combined Footing using

STADD pro**HARDWARE & SOFTWARE**

REQUIRED:

Computer with specified configuration with installed

STADD Pro Software.**Procedure:**

1. Entering Job information.
2. Truss model geometry.
3. Defining member properties ,sections.
4. Assigning Loads(Load Cases and combination).
5. Defining Pre-Analysis Print out, analysis type and Post-Analysis print out.
6. Defining Design Requirements.
7. The combination of STAAD.pro and STAAD. beams can make your bridge design and analysis easier. STAAD.pro is first used to construct the bridge geometry and STAAD. beams is used to find the AASHTO 2002 load positions that will create the maximum load response.The maximum load response could be any of the following:

8.Maximum plate stresses, moment about the local x axis of a plate (Mx), moment about the local y axis of a plate (My) etc. used to design for concrete deck reinforcement.

9.Maximum support reactions to design isolated, pile cap, and mat foundations. Maximum bending moment or axial force in a member used to design members as per the AASHTO code.

10.Maximum deflection at mid span.

11.These loads that create the maximum load responses can be transferred into STAAD.pro as load cases to load combinations for further analysis and design. Figure shows the bridge design procedure discussed above.

Step 1:-

Area required for footing Square

= B = (w+w1)/P0

Where, Po = safe bearing

capacity of soil w1 = self

weight of footing

w = self weight

For Rectangle = b/d = B/D

A = b x d Net upward pressure on the footing q/p = W/A

Step 2:-

Bending Moment

Critical section for maximum bending moment is taken at the

face of the column For a square footing,

Mxx = q x B/8 (L – a)2

Mxx = q x L/8 (B – b)2

Myy = q x B/8 (L – a)2

Step 3:-

To fix the depth of the footing shall be greater of the following:

Depth from bending moment

consideration d = √(M/Qb)

where,

Q = moment of required factor Depth from shear consideration Check for one way

shear.

Check for two way shear or punching shear

Critical shear for one way shear is considered at a distance 'd' from

face of the column. Shear force, V = qB [½(B – b) d]

Nominal shear stress, Tv = k . Tc Tc = 0.16√fck

Step 4 :-

Check for two way shear

Critical section for two way shear is considered at a distance at a distance d/2 from all the faces of the column.

SF, $V = q [B2 – (b + d)2]$

SF, $V = q [L x B – (a + d)(b + d)]$

Nominal shear stress, $Tv = V/2((a+d)x(b+d)d)$ ——-

{for a rectangle $Tv = V/4((b+d)d)$ ——- {for a square

$Tv = k$.

Tc $k = 0.5 + \beta > 1$; [Beta β = ratio of sides of the column $Tc = 0.16\sqrt{fck}$

Area of
steel, $Ast = M/((\sigma)stjd)$ of footing

Analysis is done as explained in combined footing

RESULT:-