

ADVANCED CAD LABORATORY

LAB MANUAL

Year : 2016 - 2017
Course Code : BST102
Regulations : IARE - R16
Semester : III
Branch : CIVIL ENGINEERING,
(M.Tech Structures)

Prepared by

Mr.SRINIVAS ANGADI, ASSISTNT PROFESSOR
Mr.S. SELVAPRAKASH, ASSISTNT PROFESSOR



INSTITUTE OF AERONAUTICAL ENGINEERING
(Autonomous)
Dundigal, Hyderabad - 500 043



INSTITUTE OF AERONAUTICAL ENGINEERING
(Autonomous)
Dundigal, Hyderabad - 500 043

1. PROGRAM OUTCOMES:

B.TECH - PROGRAM OUTCOMES (POS)	
PO-1	Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems (Engineering knowledge).
PO-2	Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences (Problem analysis).
PO-3	Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations (Design/development of solutions).
PO-4	Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions (Conduct investigations of complex problems).
PO-5	Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations (Modern tool usage).
PO-6	Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice (The engineer and society).
PO-7	Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development (Environment and sustainability).
PO-8	Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice (Ethics).
PO-9	Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings (Individual and team work).
PO-10	Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions (Communication).
PO-11	Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments (Project management and finance).
PO-12	Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change (Life-long learning).

2. PROGRAM SPECIFIC OUTCOMES

PROGRAM SPECIFIC OUTCOMES (PSO's)	
PSO-1	Professional Skills: The ability to understand, analyze and develop computer programs in the areas related to algorithms, system software, multimedia, web design, big data analytics, and networking for efficient design of computer-based systems of varying complexity.
PSO-2	Problem-Solving Skills: The ability to apply standard practices and strategies in software project development using open-ended programming environments to deliver a quality product for business success.
PSO-3	Successful Career and Entrepreneurship: The ability to employ modern computer languages, environments, and platforms in creating innovative career paths to be an entrepreneur, and a zest for higher studies.

3. ATTAINMENT OF PROGRAM OUTCOMES AND PROGRAM SPECIFIC OUTCOMES:

S.No	Experiment	Program Outcomes Attained	Program Specific Outcomes Attained
WEEK-1	Program using arrays and function for matrix multiplication	PO-2,PO-3	PSO-1
WEEK-2	Programs to draw bending moment and shear force diagrams graphics in C	PO-2,PO-3	PSO-1
WEEK-3	Program for design of slab using excel	PO-2	
WEEK-4	Program for design of beam using excel	PO-3	PSO-1
WEEK-5	Program for design of column using excel	PO-3	PSO-1
WEEK-6	Program for design of footing using excel	PO-3	PSO-1
WEEK-7	Analysis of Truss using STADD pro	PO-2,PO-3	PSO-1
WEEK-8	Analysis of Multistoried space frame using STADD pro	PO-2,PO-3	PSO-1
WEEK-9	Analysis of Bridge Deck Slab using STADD pro	PO-2,PO-3	PSO-1

4. MAPPING COURSE OBJECTIVES LEADING TO THE ACHIEVEMENT OF PROGRAM OUTCOMES:

Course Objectives	Program Outcomes												Program Specific Outcomes		
	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO 10	PO 11	PO 12	PSO1	PSO2	PSO3
I		√	√									√	√		
II		√	√									√	√		

5. SYLLABUS:

ADVANCED CAD LAB								
II Semester: M.Tech Structures								
Course Code	Category	Hours / Week			Credits	Maximum Marks		
AIT101	Core	L	T	P	C	CIA	SEE	Total
		-	-	3	2	30	70	100
Contact Classes: Nil	Tutorial Classes: Nil	Practical Classes: 36			Total Classes: 36			
OBJECTIVES: The course should enable the students to: Learn how to analyze a problem and design the solution for the problem. I. Analyze the different structural members and give efficient design for the structure II. Strengthen the ability to identify and apply the suitable algorithm/program/design method for the given real world problem.								
LIST OF EXPERIMENTS								
WEEK-1	MATRIX MULTIPLICATION							
Program using arrays and function for matrix multiplication								
WEEK-2	BENDING MOMENT AND SHEAR FORCE							
Programs to draw bending moment and shear force diagrams graphics in C								
WEEK-3	DESIGN OF SLAB							
Program for design of slab using excel								
WEEK-4	DESIGN OF BEAM							
Program for design of beam using excel								
WEEK-5	DESIGN OF COLUMN							
Program for design of column using excel								
WEEK-6	DESIGN OF FOOTING							
Program for design of footing using excel								
WEEK-7	ANALYSIS OF TRUSS							
Analysis of Truss using STADD pro								
WEEK-8	ANALYSIS OF MULTISTORIED SPACE FRAME							
Analysis of Multistoried space frame using STADD pro								

WEEK-9	ANALYSIS OF BRIDGE DECK SLAB
Analysis of Bridge Deck Slab using STADD pro	
Reference Books:	
<ol style="list-style-type: none"> 1. Design & Construction of Multistoreyed Buildings- PWD Association – TamilNadu, Chempauk. 2. Structural Design of Multi-storeyed Buildings –Second edition U.H.Varyani-South Asian Publishers, New Delhi 3. Learn Staad Pro- Rajendran, Design Tech, Civil Cadd centre,Coimbatore-641012 4. Hand book of Reinforced Concrete Design –V.L.Shah & S.R.Karve- Structures Publication, Pune. 5. Microsoft excel 2013 Bible by John Walkenbanch 	
Web References:	
<ol style="list-style-type: none"> 1. http://www.academia.edu/5782510/Bridge_Design_using_the_STAAD.Pro_Beava_AASHTO_Code 2. https://www.academia.edu/5108944/Seismic_analysis_of_plan_irregular_multi-storied_building_using_STAAD_pro 3. http://onlinecivilforum.com/site/index.php/2016/10/23/rcc-design-excel-sheet/ 	
SOFTWARE AND HARDWARE REQUIREMENTS FOR A BATCH OF 24 STUDENTS:	
HARDWARE: Desktop Computer Systems: 24 nos	
SOFTWARE: Application Software: STADD PRO	

6. INDEX:

S.No	Experiment	Page No
1	MATRIX MULTIPLICATION	9
2	BENDING MOMENT AND SHEAR FORCE	11
3	DESIGN OF SLAB	17
4	DESIGN OF BEAM	25
5	DESIGN OF COLUMN	31
6	DESIGN OF FOOTING	36
7	ANALYSIS OF TRUSS	42
8	ANALYSIS OF MULTISTORIED SPACE FRAME	58
9	ANALYSIS OF BRIDGE DECK SLAB	62

WEEK-1

MATRIX MULTIPLICATION

AIM:

To write a program using arrays and function for matrix multiplication

Hardware & Software required:

Computer with specified configuration with installed C programming language software

ALGORITHM:

1. Start
2. Read the operator to be p
- 3 Using the switch statement for operator.
 - a) Case `*`: call mull()
4. Stop

Procudure for Multiplication of Matrices

1. Start
2. Read the value of two matrices using nested for loops
3. Display the two matrices using nested for loop
4. Then multiply two matrices using the following statement using nested for loop $c[i][j] = c[i][j] + a[i][p] * b[p][j]$
5. Display the resultant matrix using nested for loops.
6. Stop

C Program for Multiplication of Matrices

```
/*to find product of two matrices*/
void findproduct(int a[][10],int b[][10],int r,int d,int c)
{
    int i,j,k;
    int m[10][10];
    for(i=0;i<r;i++)
    for(j=0;j<c;j++)
    {
        m[i][j]=0;
        for(k=0;k<d;k++)
        m[i][j]=m[i][j]+a[i][k]*b[k][j];
    }
    printf("\t*****\n");
    printf("\tThe product of two matrixes is:\n");
    printf("\t*****\n");
    printmatrix(m,r,c);
}
```

INPUT: Enter the choice + or *
*

OUTPUT:

A= 1 1 1 B= 1 0 0

1 1 1 0 1 0

1 1 1 0 0 1

Resultant Matrix is:

1 1 1

1 1 1

1 1 1

WEEK-2

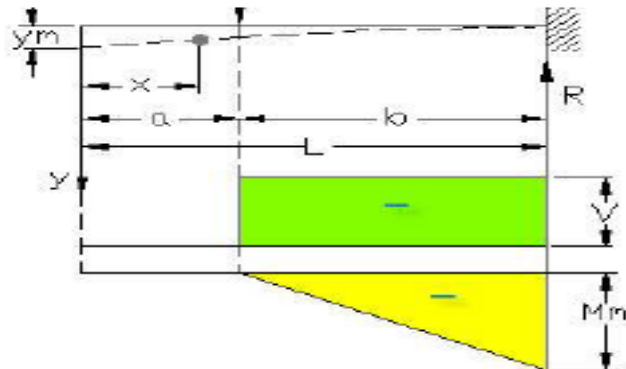
BENDING MOMENT AND SHEAR FORCE

AIM:

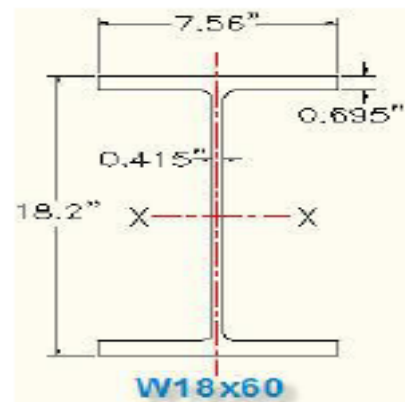
Programs to draw bending moment and shear force diagrams.

Hardware & Software required:

Computer with specified configuration with installed Microsoft Office



Cantilever with Load at any point



W18x60

Effect of load P only

	INPUT	
Load, P =	15000	lbs
Load location in inches, a =	80	in
Beam span in feet, Lf =	18	ft
Location in inches from free end, x =	120	in
AISC Beam Section =	W18x60	
Beam weight per foot, w =	60	lbs/ft
Beam section modulus about x-x, Sx =	108	in ³
Section 2nd moment of area about x-x, I =	984	in ⁴
Modulus of elasticity, E =	29000000	lbs/in ²
Allowable tension/compression stress, Fa =	22000	lbs/in ²

feet	18
inches =	feet * 12
=	216

CALCULATE

Beam span in inches, L =	Lf * 12	
=	216	in
Right support force, R =	P	
=	15000	lbs
Max beam shear force, V =	- R	
=	-15000	lbs
b =	(L - a)	
=	136	in
Moment at x if x > a, Mx1 =	- P*(x - a)	
=	-800000	in-lbs
Max moment, Mm =	- P * b	
=	-2040000	in-lbs

Beam Deflection due to P only

CALCULATE continued

Max beam deflection at x = 0, y _{MP} =	(P*b ² /(6*E*I))*(3*L - b)	
=	0.830	in
Beam deflection at x < a, y =	(P*b ² /(6*E*I))*(3*L - 3*x - b)	
=	0.246	in

x > a Not OK

$$\begin{aligned}
 \text{Beam deflection at } x = a, \quad y_a &= P \cdot b^3 / (3 \cdot E \cdot I) \\
 &= 0.441 \quad \text{in} \\
 \text{Beam deflection at } x > a, \quad y &= (P \cdot (L - x)^2 / (6 \cdot E \cdot I)) \cdot (3 \cdot b - L + x) \\
 &= 0.252 \quad \text{in} \quad \boxed{x > a, \text{ OK}} \\
 \text{Max allowable deflection} &= L / 360 \\
 &= 0.600 \quad \text{in}
 \end{aligned}$$

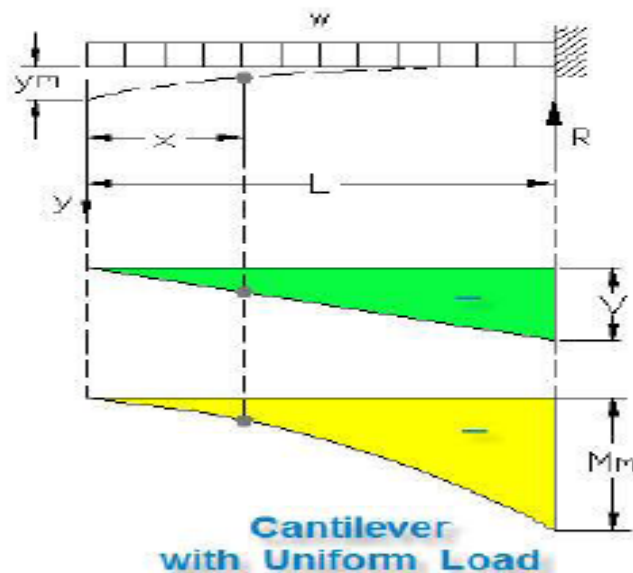
End Slopes due to P only

CALCULATE continued

$$\begin{aligned}
 \text{Beam left end slope, } \theta_L &= (P \cdot L^2) / (2 \cdot E \cdot I) \\
 &= 0.0123 \quad \text{radians} \quad 0.703 \quad \text{deg} \\
 \text{Beam right end slope, } \theta_R &= 0
 \end{aligned}$$

CASE-B CANTILEVER WEIGHT LOAD w LB/FT

Beam Shear & Moment Diagrams



Effect of beam weight only

CALCULATE continued

$$\begin{aligned}
 \text{Total beam weight, } W &= w \cdot L_f \\
 &= 1080 \quad \text{lbs} \\
 \text{Right support force, } R_B &= W \\
 &= 1080 \quad \text{lbs} \\
 \text{Max beam shear force, } V &= R_B \\
 &= 1080 \quad \text{lbs} \\
 \text{Beam shear force at } x, \quad V &= - (w/2) \cdot x \\
 &= -600 \quad \text{lbs}
 \end{aligned}$$

$$\begin{aligned}\text{Max beam moment, } M_m &= -(w/12) * L^2 / 2 \\ &= -118640 \quad \text{in-lbs} \\ \text{Moment at x, } M_{x2} &= -(w/12) * x^2 / 2 \\ &= -36000 \quad \text{in-lbs}\end{aligned}$$

Effect of beam weight only

CALCULATE continued

$$\begin{aligned}\text{Beam deflection due to load w at x, } y_x &= ((w/12)/(24 * E * I)) * ((3 * L^4 - 4 * L^3 * x + x^4)) \\ &= 0.014 \quad \text{in} \\ \text{Max beam deflection due to load w, } y_{MW} &= ((w/12) * L^4) / (8 * E * I) \\ &= 0.048 \quad \text{in} \\ \text{Max allowable deflection} &= L / 360 \\ &= 0.600 \quad \text{in}\end{aligned}$$

Effect of beam weight only

End Slopes

CALCULATE continued

$$\begin{aligned}\text{Beam left end slope, } \theta_L &= ((w/12) * L^3) / (8 * E * I) \\ &= 0.0002943 \quad \text{radians} \quad 0.017 \quad \text{deg} \\ \text{Beam right end slope, } \theta_R &= 0\end{aligned}$$

CASE-C CANTILEVER WEIGHT AND CONCENTRATED LOADS

SET x = L ABOVE FOR MAX MOMENT

$$\text{Total moment at x if } x > a, \quad M_{xT} = M_{x1} + M_{x2}$$

$$= -636000 \quad \text{in-lbs}$$

x > a, OK

$$\text{Moment stress at x, } F_x = M_{xT} / S_x$$

$$= 5889 \quad \text{lbs/in}^2$$

$$\text{Safety Factor at x, } SF_x = F_a / F_x$$

$$= 3.74 \quad \text{SF}_x > 2.00, \text{ OK}$$

Total Cantilever Deflection due to beam weight and load P.

Principal of Superposition

Within the elastic limit the total deflection at a point in a beam is equal to the sum of the deflections due to all of the applied loading systems.

$$\text{Total beam deflection at x if } x < a, \quad y_x = y_A + y_a$$

$$= 0.260 \quad \text{in}$$

x > a, OK

$$\text{Max beam deflection at free end, } y_{MAX} = y_{MW} + y_{MB}$$

$$= 0.877 \quad \text{in}$$

$$\text{Max allowable deflection} = L / 360$$

$$= 0.600 \quad \text{in}$$

Total Beam Slope at x

$$\text{Total beam end slope (at } x = 0), \quad \theta_t = \theta_A + \theta_B$$

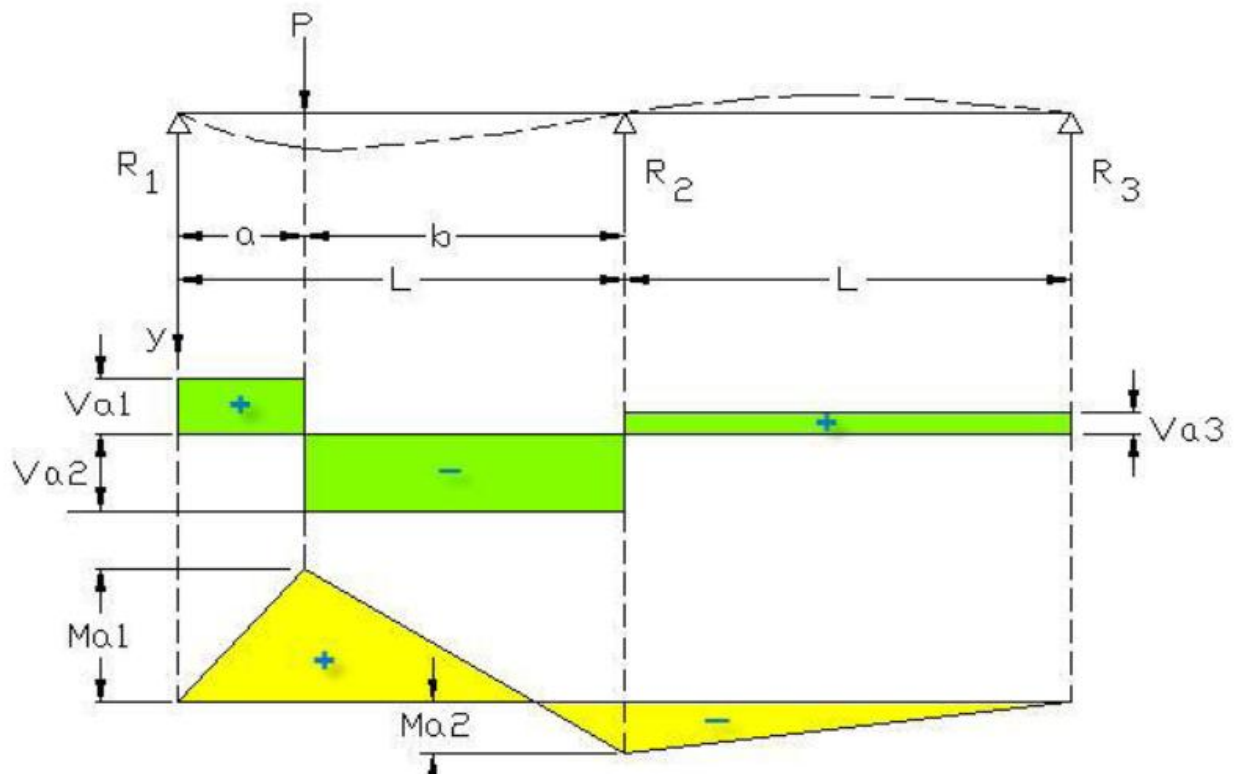
$$0.01258 \quad \text{radians}$$

$$0.72 \quad \text{deg}$$

CASE-A CONTINUOUS BEAM PINNED WITH CONCENTRATED LOAD

Beam Shear & Moment Diagrams

Two Equal Spans



Continuous Beam Pinned Concentrated Load

Effect of load P only

INPUT

Load, P = 5000 lbs
 Load location in inches, a = 100 in
 Beam span in feet, Lf = 18 ft
 Load location from left support, x = 120 in (x < or = b)
 Beam Section = W12x72
 Beam weight per foot, w = 72 lbs/ft
 Beam section modulus, Sx = 97.40 in^3
 Section moment area about x-x, I = 597 in^4
 Modulus of elasticity, E = 29000000 lbs/in^2
 Allowable stress, Fa = 22000 lbs/in^2

feet	10
inches =	feet * 12
=	120

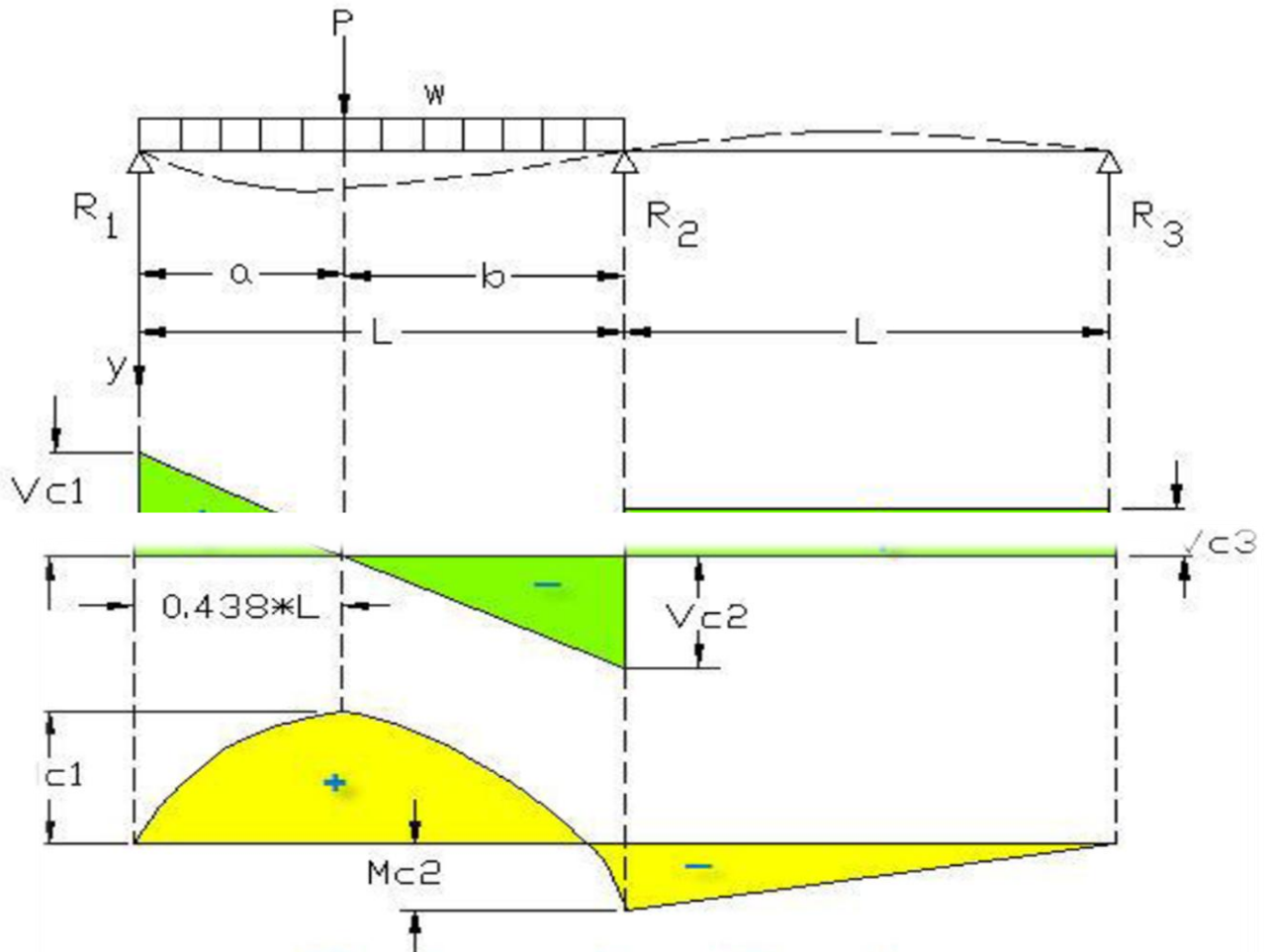
CALCULATE

Beam single span in inches, L = Lf * 12
 = 216 in
 b = L - a
 = 116 in
 Left support force, R₁ = V_{a1} = (P*b / (4*L^3))*(4*L^2 - a*(L + a))
 = 2231 lbs
 Support force at x = L, R₂ = - (P*a / (2*L^3))*(2*L^2 - b*(L + a))
 = -1405 lbs
 Right support force, R₃ = V_{a3} = (P*b / (4*L^3))*(L + a)
 = 5 lbs
 Max moment at x = a, Ma1 = (P*a*b / (4*L^3))*(L + a)
 = 223052 in-lbs
 Moment at x = L, Ma2 = - (P*a*b / (4*L^2))*(L + a)
 = -98208 in-lbs
 Shear, Va2 = (P*a / (4*L^3))*(4*L^2 + b*(L + a))
 = 2769 lbs

CASE-B CONTINUOUS BEAM PINNED WITH DISTRIBUTED LOAD

Beam Shear & Moment Diagrams

Two Equal Spans



Continuous Beam Pinned Concentrated & Distributed Loads

CALCULATE continued

Case C = Case A + Case B

Left support force, $R_1 = V_{c1} = V_{a1} + V_{b1}$

= 2798 lbs

Support force at $x = L$, $R_2 = V_{c2} = V_{a2} + V_{b2}$

= -2215 lbs

Right support force, $R_3 = V_{c3} = V_{a3} + V_{b3}$

= 86 lbs

Case C = Case A + Case B

Max moment at $x = a$, $M_a = M_{a1} + M_{b1}$

= 249843 in-lbs

Moment at $x = L$, $M_{a2} = M_{a2} + M_{b2}$

= -115704 in-lbs

feet	10
inches =	feet * 12
=	120

WEEK-3

DESIGN OF SLAB

AIM:

To write a program for design of slab using excel.

HARDWARE & SOFTWARE REQUIRED:

Computer with specified configuration with installed Microsoft Office

PROCEDURE:

Manual Procedure:

Design of Slabs

- Reinforced concrete slabs are classified into

1. One way slabs
2. Two way slabs
3. Flat slabs
4. Flat plates

One way Slabs

- When slabs are supported on two opposite edges continuously or when the load is transported in one direction only on supporting the slab on four edges.
- In one way slabs the direction in which loads are transmitted is called the span.
- One way slabs are usually made to span in the shorter direction since the corresponding bending moments and shear forces are the least.
- Steel reinforcement is also provided in span direction.
- Reinforcement is also provided in the transverse direction to distribute any unevenness that may occur in loading and for temperature and shrinkage effects in that direction. Such a steel is called distribution reinforcement or secondary reinforcement.
- The main reinforcement is calculated based on the bending moment and should not be less than the minimum specified by the code.

Loading on Slabs

- Loading on the slabs consist of

1. Dead weight of the slab
2. Dead weight of floor finishes
3. Weight of fixtures
4. Weight of partitions
5. Live load or imposed load

- Live load varies depending upon the use and types of building.

- For design purpose, live loads are considered as either acting on the full span or assumed to be absent altogether in the span.

- In continuous slabs, these are so placed as to get the maximum bending moment and shear effect in the structure.

In the design of slabs for other structures like bridges, the effect of partial loading of the slab may have to be considered.

- According to IS:875, the loading on slabs for buildings are calculated as follows:

- Self weight at 25kN/m^3 for R.C.

- Finishes and portions at 1.5kN/m^2

- Characteristic imposed loads, depending upon the type of building.

- For roofs $= 1.5\text{kN/m}^2$ with access
 $= 0.75\text{kN/m}^2$ without access

- For floors $= 2.0\text{ kN/m}^2$ for residential floors
 $= 3.0\text{ kN/m}^2$ for office floors

Arrangement of Loading

- In buildings, the live load is arranged to get critical values of bending moments and shear forces.

- To get the maximum sagging moment, the alternative should be loaded with live loads.

- To get the maximum hogging moment, alternative adjacent spans should be loaded with live load.

•If the live load is less than three fourth of dead load, then all spans are loaded with live loads, in addition to dead loads. [i.e. $DL > 1.33LL$]

Analysis of Slabs spanning in one way under udl

- One way slabs are analyzed as beams of unit width.
- IS: 456 allows use of simple coefficient for calculation of moments and forces for continuous beams and slabs of uniform loading with more or less equal spans and continuous, if the difference in span are not more than 15 percent of the larger of the spans.
- For obtaining the bending moment, the coefficient shall be multiplied by the total load and effective span.
- For shear force, the coefficient shall be multiplied by the design load.

Design for shear in slabs

- While designing slabs, thickness is so chosen that the shear can be resisted by concrete itself, and slab does not need extra shear reinforcements.
 - In extreme cases, where thickness tends to be very large that shear reinforcements are allowed to be used for slabs.
 - When the slab thickness is less than 300mm, the slabs show increased shear strength.
 - Enhanced shear is equal to $k\tau_c$, where “k” is called the shear strength enhancement factor.
- Table 14: Shear Enhancement Factors

•K varies from 1.30 for 150mm or less to 1.0 for slabs of thickness 300mm or more.

$$K = \left[\frac{400}{d} \right]^{\frac{1}{4}}$$

$$\left[\frac{400}{d} \right] \leq 1.0$$

Design of Slabs

According to IS: 456

- 1.No shear reinforcement should be provided if $D \leq 200\text{mm}$
- 2.Design slabs without extra shear reinforcement.
3. If $D \geq 200\text{mm}$, spacing of shear reinforcement can be increased to “d” instead of “0.75d” as in beams.
4. Maximum shear should not be more than that prescribed in IS:456.

5. Spacing for bars in slab for main reinforcement

1.3d or 2.300mm whichever is smaller

6. Spacing of bars in slab for secondary steel

1.5d or 2.450mm whichever is smaller

1. Minimum cover to the steel in slabs is 20mm.

2. Minimum distribution steel is 0.15% of bD for MS and 0.12% of bD for HYSD.

3. Diameter of reinforcing bars shall not exceed one eighth the total thickness.

4. Basic effective span-to-depth ratios is applicable to slabs also.

5. $M_u = f_{st} A_{st} \times (\text{lever arm})$ is useful for calculating the required area of steel reinforcement.

Design of One Way Slabs

1. Assume depth based on deflection and percent steel.

2. Determine maximum BM assuming one meter width and effective span.

3. Determine effective depth of slab from $M_u = k f_c k b d^2$ and $b = 1000\text{mm}$, find $D = d + \text{Cover}$.

4. Check slab depth for shear

5. Determine depth for satisfying all requirements and determine steel required for the moment.

6. Check for minimum distribution steel.

7. Select a suitable diameter of main reinforcement and determine its spacing.

Moments in Two way Restrained Slabs with Corners Held Down Analysis by Coefficients

•Restrained slabs are defined in codes as those slabs which are cast integral with RC frames and which are not free to lift up at the corners.

•These slabs may be continuous or discontinuous at the edges. Those which are discontinuous at edges are also referred to be simply supported.

•Coefficients specified in IS:456-2000, Table 26 in Annexure D can be used for analysis of such slabs.

Moments in Two way Restrained Slabs with Corners Held Down

Analysis by Coefficients

•Conditions to be satisfied for use of these coefficients are:

- 1.Loading on adjacent spans should be the same
- 2.Span in each direction should be approximately equal.

•The span moment per unit width and the negative moments at continuous edges for the slabs are calculated from the equations in terms of “lx”,

$$M_x = \alpha_x w l_x^2 \quad \text{for short span}$$

$$M_y = \alpha_y w l_y^2 \quad \text{for longer span}$$

Design of Two-way Slabs

- Restrained two way slab is divided into middle strip and edge strips. Middle strip is forming three-fourth of slab width in width directions.
- Torsion steel must be provided at discontinuous edges as specified in code.
- Coefficients are given in Table 26 of IS: 456-2000. These coefficients are derived from the yield line theory of slabs.
- Table given for coefficients is applicable for slabs carrying uniformly distributed loads not for concentrated loads, for which analysis should be done, separately.
- Span moments/edge moments per unit width are calculated by determining ratio of “ly” and “lx” and for different edge conditions.

Coefficients for Moments

•Slabs restrained against corners lifted up

$$\alpha_y^+ = \frac{24 + 2n_d + 1.5n_d^2}{1000}$$

•nd = Number of discontinuous edges = 0,1,2,3 and 4.

$$\alpha_x^+ = \frac{2}{9} \left[\frac{3 - \sqrt{18 \alpha_y^+ (C_{s1} + C_{s2})}}{r} \frac{1}{(C_{l1} + C_{l2})^2} \right]$$

• C = 1.0 for a discontinuous edge

= $\sqrt{7/3}$ for continuous edge

•Subscripts “s” and “l” denote “short edge” and “long edge”

- Subscripts “1” and “2” represent two edges in short and longer direction

Important Design Issues from Table 26

- 1.Edge moments of continuous supports are 1.33 times the span moments.
- 2.Long span moment coefficient “ α_y ” is a constant for given end conditions of slab, irrespective of the span ratios.
- 3.Short span coefficient varies sharply with variation of the ratio of spans
- 4.While using Table 26 for a series of slabs, moments calculated at an interior support will sometimes be different on two sides of that support because of the differences in continuity condition of slabs on opposite sides of support.

Arrangement of Reinforcements

- While using design of two-way slabs with the help of coefficients, restrained slabs are considered to be divided into middle and edge strips.
- Moments given in Table 26 apply only to middle strips, and no further redistribution is allowed for these moments.
- Edge strips have to be reinforced only with nominal minimum steel for crack control.

Arrangement of Reinforcements

- Middle strip should have steel (+ve and –ve) calculated for various sections. In edge strips , steel is placed as positive steel at the bottom of slab.
- Negative moments may be experienced at discontinuous edges since, in practice, they are not supported on rollers but partially restrained at their ends. The magnitude of this moment depends on the degree of fixity at the edge of the slab and is intermediate.

Arrangement of Reinforcement

- Usual practice is to provide at these edges top reinforcement for negative moment equal to

As per IS:456.2000, 50 percent of the steel provided at mid span should be extended along these edges, and the negative steel has to extend into the span 0.1 times the span length from the face of the beam.

Design of Two-way Slabs

- The slab thickness should be calculated based on the greater value of the negative B.M on the short span.

$$M_u = K f_{ck} b d^2$$

Hence

- Total thickness = d (short) + 0.5ϕ + cover
- Total thickness = d (long) + 0.5ϕ + ϕ + cover

The slab should satisfy span/effective depth ratio to control deflection.

–Simply supported = 28

–Continuous = 32

• Depth of slab selected from deflection criterion will be generally greater than the minimum required from strength criterion.

• Short span steel will be placed in the lower layer.

Restrained moments are obtained for the middle strips only. The reinforcement is distributed uniformly in the middle strips.

• Each direction is to be provided only with the minimum reinforcement placed at the bottom of the slabs. In addition, corner steel reinforcement should be provided at the discontinuous edges. Corner reinforcement consists of two mats, are placed on the top and the other at the bottom of the slab, with each mat having steel in both x and y directions.

• Where the slab is discontinuous on both sides of a corner, full torsion steel has to be provided.

The area of the full torsion reinforcement per unit width in each of the four layer should be as follows.

• (Area of full corner steel) = $\frac{3}{4}$ [Area required for the maximum span per unit width in each of four layers]

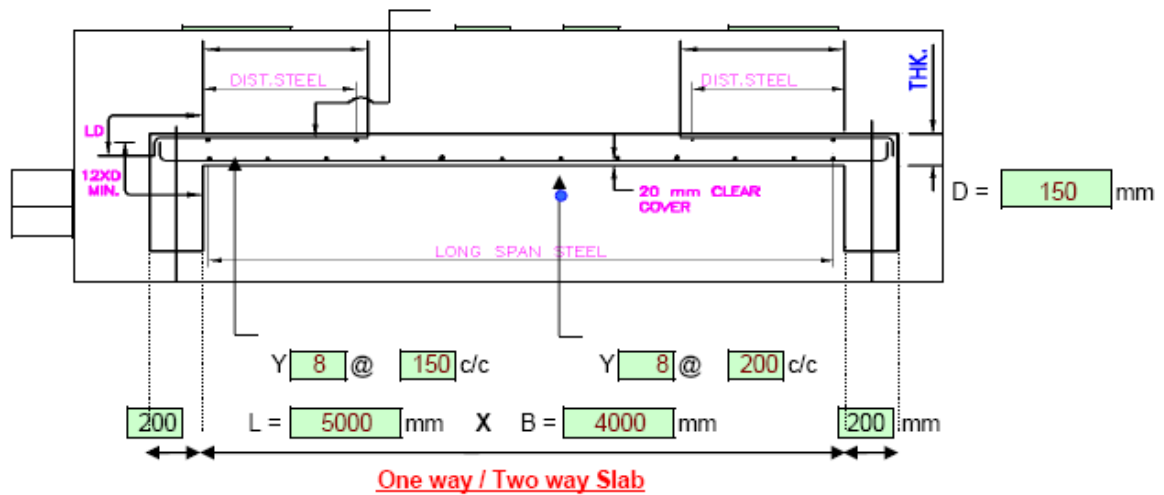
• These steels are to be provided for a distance of one-fifth the short span .

Based on the Manual Procudere, excel sheet should be designed

Design of Slab in EXCEL

Slab No.

Total No. of Slab



Project :-

Name of work :-

Sr. No.	Discription	Dia of bar in	Nos. of Bar	Length of bar in mm	Wt. Of bar Kg/Mt	Quantity In Kg
1	Main steel	8	28	5470	0.395	60.50
2	Distribution Steel	8	26	4470	0.395	45.91
3	End Support Main steel (Top)	10	24	3120	0.617	46.20
		10	30	3120	0.617	57.75

Total Quantity of steel	Dia of Bar	Kg.	Quantity of all beams (Kg)
	8	106.405	106.4051
	10	103.952	103.95216
	12	0	0
	16	0	0
	20	0	0
	25	0	0
	32	0	0
	Total	210.357	210.35726

WEEK-4

DESIGN OF BEAM

AIM:

To write a program for design of beam using excel.

HARDWARE & SOFTWARE REQUIRED:

Computer with specified configuration with installed Microsoft Office

PROCEDURE:

Manual Procedure:

DESIGN FOR BEAM

- ☐ RCC beam is subjected to bending moments and shear.
- ☐ Due to the vertical external load, bending compresses the top fibers of the beam and elongates the bottom fibers.
- ☐ The strength of the RCC beam depends upon the composite action of concrete and steel.

WORKING STRESS METHOD

- The Stresses in an element is obtained from the working loads and compared with permissible stresses.
- The method follows linear stress-strain behavior of both the materials.
- Modular ratio can be used to determine allowable stresses.
- Material capabilities are under estimated to large extent. Factor of safety are used in working stress method.
- The member is considered as working stress.
- Ultimate load carrying capacity cannot be predicted accurately.
- The main drawback of this method is that it results in an uneconomical section

LIMIT STATE METHOD

- The stresses are obtained from design loads and compared with design strength.

- In this method, it follows linear strain relationship but not linear stress relationship (one of the major difference between the two methods of design).
- The ultimate stresses of materials itself are used as allowable stresses.
- The material capabilities are not under estimated as much as they are in working stress method. Partial safety factors are used in limit state method.

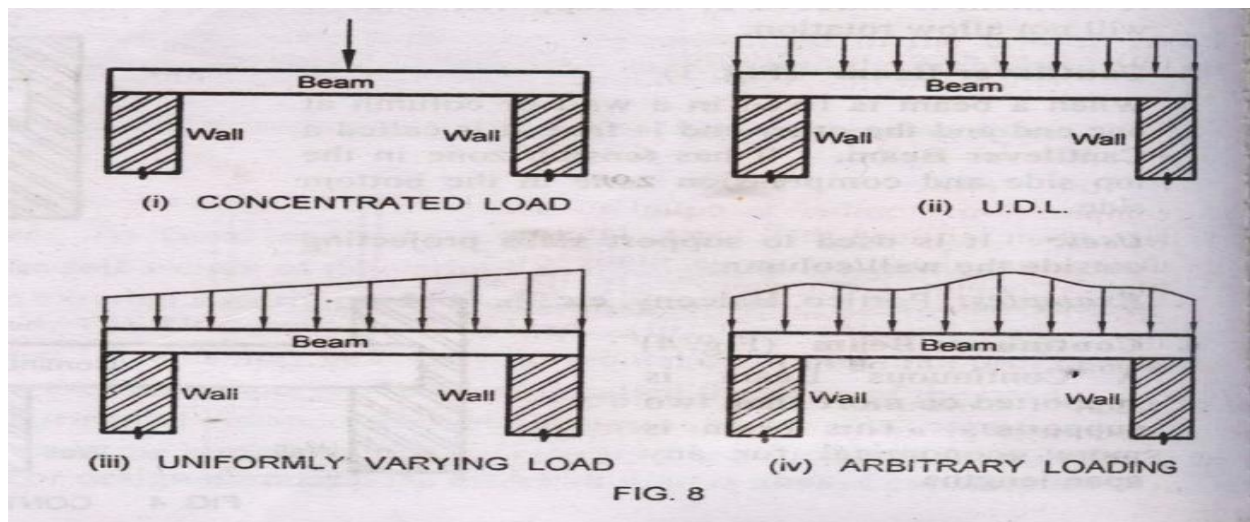
LOAD ACTING ON A STRUCTURE

DEAD LOAD:

- ❖ DEAD Load is self-weight of the various components in the building.

LIVE LOAD:

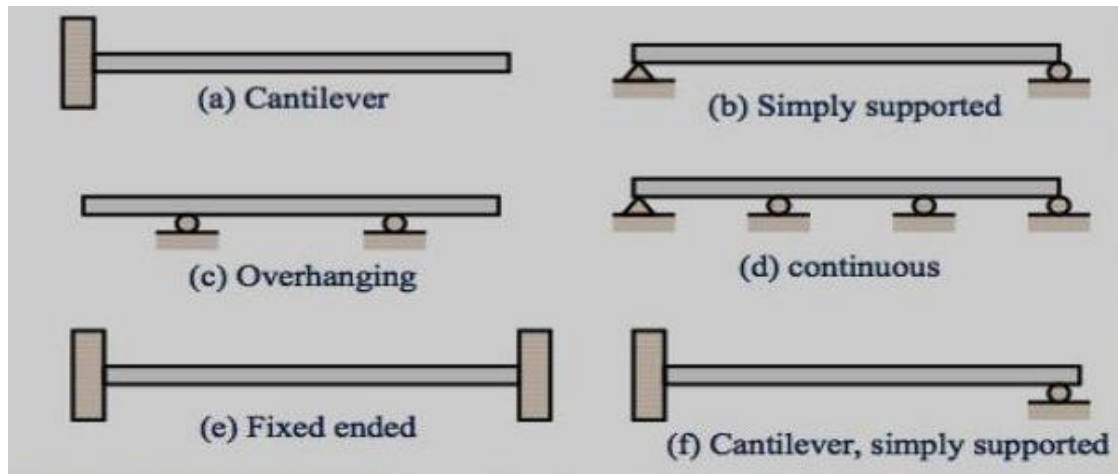
- ❖ LIVE Load is the external superimposed load on a structure.
- Uniformly distributed load.
- Uniformly varying load.
- Concentrated load.
- Arbitrary load.



TYPES OF BEAMS

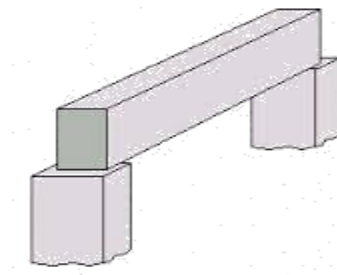
- ☐ Simply Supported Beam.
- ☐ Fixed Beam.
- ☐ Cantilever Beam.
- ☐ Continuous Beam.
- ☐ Overhanging Beam.

- ❑ Cantilever, Simply supported beam



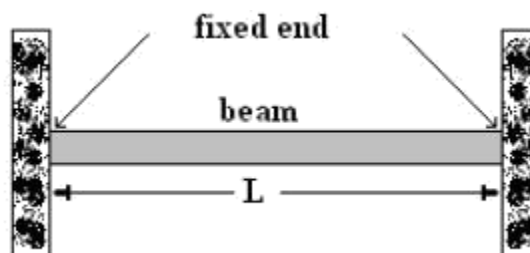
SIMPLY SUPPORTED BEAM

- ❑ It is a beam that is freely supported at two ends on walls or columns.
- ❑ In actual practice no beam rests freely on the supports (columns or walls)



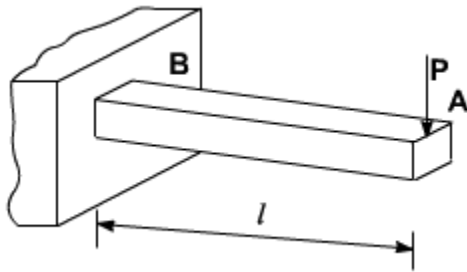
FIXED BEAM

- ❑ In this beam both ends of the beam are rigidly fixed into the supports.
- ❑ Main reinforcement bars and stirrups are also provided



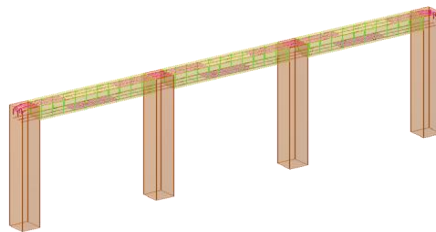
CANTILEVER BEAM:

- ❑ One end of the beam is fixed to wall or column and the other end is free.
- ❑ It has tension on top and compression on bottom



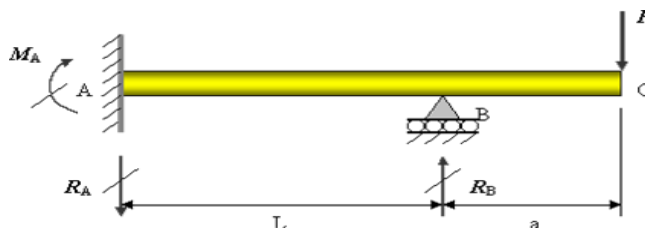
CONTINUOUS BEAM

- ❑ A continuous beam is a statically indeterminate multi span beam on hinged support.
- ❑ The end spans may be cantilever, may be freely supported or fixed supported. At least one of the supports of a continuous beam must be able to develop a reaction along the beam axis.



OVERHANGING BEAM:

- ❑ In overhanging beam its end extends beyond column or wall support.
- ❑ Overhanging of the beam is the unsupported portion of the beam, it may be on side or both the sides.



TYPES OF RCC BEAMS:

❑ SINGLY REINFORCED BEAM:

- ❑ Singly reinforcement beam have steel provided only one side tension and another side compression. tension takes steel load or tensile load and compression takes concrete or compressive load.

DOUBLY REINFORCED BEAM

- Doubly reinforced sections contain reinforcement both at the tension and at the compression face, usually at the support section only

Beam Design

Beam Data

width	200 mm
depth	600 mm
clear cover to main	15 mm

d'	36 mm	$= cc + s_{dia} + m_{dia}/2$
eff depth	565 mm	$= d - d'$

Material Grades

Concrete	20 MPa
Steel	415 MPa

Moment	123 KN-m
--------	----------

Mu/bd ²	1.93	
x _u max	270	$= (700/(1100 * (0.87 * f_y))) * d$
M _u lim	176	$= 0.36 * f_{ck} * b * x_{u\max} * (d - (0.42 * x_{u\max}))$
M _u lim/bd ²	2.76	

Beam is designed as Singly Reinforced Beam

Area of Steel	Tension (Ast)	Compr (Asc)
Percentage	0.613 %	-----
Area of Steel	692 sqmm	

Refer Table 2 SP 16 pg 48

Tension Reinforcement

Type	Bar dia	Nos	Area of Steel
Layer 1	25 mm	2	982 sqmm
Layer 2	16 mm	2	402 sqmm
Layer 3	-	2	

Total Steel Provided 1384 sqmm
Provided Steel OK 1.226 %

Compression Reinforcement

Type	Bar dia	Nos	Area of Steel
Layer 1	12 mm	2	
Layer 2	-		
Layer 3			

Total Steel Provided #VALUE!

Shear Force (Vu)	200 KN
------------------	--------

ζ _v	1.771	$= V_u / (b * d)$	
ζ _c	0.562	Refer Table 61 SP 16 pg 179	or $= (0.85 * \sqrt{0.8 * f_{ck}} * \sqrt{1 + 5 \beta}) / (6 \beta)$
ζ _c max	2.8	Refer Table J SP 16 pg 175	

Type	Bar Dia	Nos	Area of Steel
Layer 1	16 mm	2	402 sqmm
Layer 2	12 mm	4	452 sqmm
Layer 3	-		

Total Steel Provided 855 sqmm 0.757 %

Sectional Dimensions OK
Shear Reinforcements required

Type of stirrup	2 legged
Stirrup diameter	8 mm
Spacing	150 c/c

Steel Calculation

Grade Check
7.1

SRB		
a	0.75	$=(0.87435/100) * (fy/fck)^2$
b	-3.611	$=(0.87/100) * (fy)$
c	1.930	$=Mu/bd^2$
-p	0.613	$=(b \pm \sqrt{b^2 - 4ac})/2a$
Ast	692	$=(p * b * d)/100$

DRB		
a	0.75	$=(0.87435/100) * (fy/fck)^2$
b	-3.611	$=(0.87/100) * (fy)$
c	2.762	$=Mulim/bd^2$
-p	0.955	$=(b \pm \sqrt{b^2 - 4ac})/2a$
Astlim	1079	$=(p * b * d)/100$

Mu2	-53	$=Mu - Mulim$
Ast2	-278	$=Mu2/((0.87*fy)*(d-d'))$
Ast	801	$=Astlim + Ast2$

0.0629
0.1

d'/d	0.10	
fsc	353	Refer Table F SP 16 pg 13
fcc	8.92	$=0.466*fck$
Asc	-291	$=Mu2/((fsc-fcc)*(d-d'))$

Min steel %	0.205	$=0.85\% / fy$
Ast	692	
Asc	-291	
Min Steel	231	$=(0.85*b*d) / fy$
Max Steel	4516	$=0.04*b*d$
Ast	692	
Asc		

Shear Calculations

Pt provided	0.757	$=(Ast*100)/(b*d)$
Pc provided		$=(Asc*100)/(b*d)$
β	3.068	$=(0.8*fck)/(6.89*Pt)$

Shear Capacity of Concrete (Vs)	63	$=\zeta c * b * d$
Shear Stg to be carried by Stirrup (Vus)	137	$=Vu - Vs$

Spacing		
actual req	150	$=(Asv*0.87fy*d)/Vus$
min	454	$=(Asv*0.87fy)/(b*0.4)$
max	423	$=0.75d$
max	300	$=300mm$

provide the
least of the
4

WEEK-5

DESIGN OF COLUMN

AIM:

To write a program for design of Column using excel.

HARDWARE & SOFTWARE REQUIRED:

Computer with specified configuration with installed Microsoft Office

PROCEDURE:

Manual Procedure:

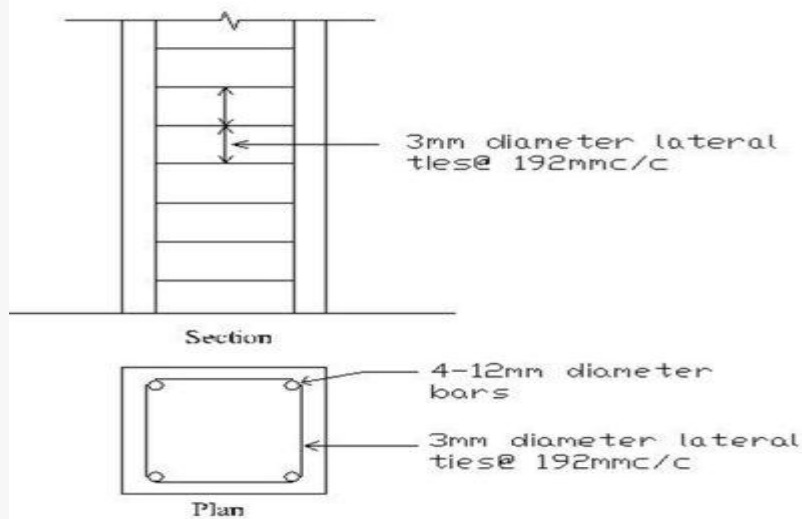
Design of column

Reinforced Concrete Columns

A column is a very important component in a structure. It is like the legs on which a structure stands. It is designed to resist axial and lateral forces and transfer them safely to the footings in the ground.

You can manually calculate the superimposed loads on a column in a structure using a simple process outlined in this linked article. You might also like this RCC Column design app which can then be used to calculate longitudinal steel reinforcement in a column for a given axial load.

Columns support floors in a structure. Slabs and beams transfer the stresses to the columns. So, it is important to design strong columns.



Reinforced Cement Concrete Column Plan and Section

A column is defined as a compression member, the effective length of which exceeds three times the least lateral dimension. Compression members whose lengths do not exceed three times the least lateral dimension may be made of plain concrete.

The axial load carrying capacity of a column is deduced from the formula

$$P_u = 0.4 f_{ck} A_c + 0.67 f_y A_{sc} \text{ as per IS 456-2000.}$$

Please see the link for formulas to calculate axial loads in columns. I would recommend using advanced structural design software like ETabs or Staad Pro for design of structures. Column design does not depend only on axial loads, but also on many other factors. There are bending moments and torsional forces induced due to beam spans, wind loads, seismic loads, point loads and many other factors.

In this article, we are going to discuss in detail the basis of classification of columns and different types of reinforcement required for a certain type of column.

A column may be classified based on different criteria such as:

1. Based on shape

- Rectangle
- Square
- Circular
- Polygon

2. Based on slenderness ratio

The ratio of the effective length of a column to the least radius of gyration of its cross section is called the slenderness ratio.

- Short RCC column, ≤ 10
- Long RCC column, > 10
- Short Steel column, ≤ 50
- Intermediate Steel column > 50 & < 200
- Long Steel column > 200

3. Based on type of loading

- Axially loaded column
- A column subjected to axial load and uniaxial bending
- A column subjected to axial load and biaxial bending

4. Based on pattern of lateral reinforcement

- Tied RCC columns
- Spiral RCC columns

Minimum eccentricity

$$e_{\min} > l/500 + D/30 > 20$$

Where, l = unsupported length of column in 'mm'

D = lateral dimensions of column

Types of Reinforcements for columns and their requirements

Longitudinal Reinforcement

- Minimum area of cross-section of longitudinal bars must be atleast 0.8% of gross section area of the column.
- Maximum area of cross-section of longitudinal bars must not exceed 6% of the gross cross-section area of the column.
- The bars should not be less than 12mm in diameter.
- Minimum number of longitudinal bars must be four in rectangular column and 6 in circular column.
- Spacing of longitudinal bars measures along the periphery of a column should not exceed 300mm.

Transverse reinforcement

- It may be in the form of lateral ties or spirals.
- The diameter of the lateral ties should not be less than $1/4^{\text{th}}$ of the diameter of the largest longitudinal bar and in no case less than 6mm.

The pitch of lateral ties should not exceed

- Least lateral dimension

- 16 x diameter of longitudinal bars (small)
- 300mm

Helical Reinforcement

The diameter of helical bars should not be less than $1/4^{\text{th}}$ the diameter of largest longitudinal and not less than 6mm.

The pitch should not exceed (if helical reinforcement is allowed);

- 75mm
- $1/6^{\text{th}}$ of the core diameter of the column

Pitch should not be less than,

- 25mm
- 3 x diameter of helical bar

Pitch should not exceed (if helical reinforcement is not allowed)

Least lateral dimension

- 16 x diameter of longitudinal bar (smaller)
- 300mm

Company's Name:	IARE Consultants Department Civil Engineering	Job No.:	ABC/1001
Client:		Design by:	UVW
Project:		Date:	17-Mar-17
		Page No.	C/101

SPREADSHEET OF DESIGN OF RECTANGULAR COLUMN SECTION BY LIMIT STATE METHOD
for AXIAL COMPRESSIVE LOAD & BIAxIAL BENDING MOMENTS, INCLUDING
SLENDerness EFFECT, AS PER IS:456-2000

Calculates range of safe loads for a Column Section with given Concrete grade and Reinforcement and checks adequacy of the section for the given loads.

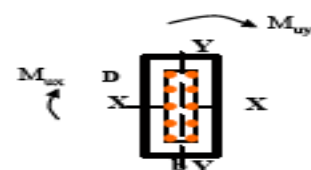
Column Dimensions:

Breadth, 'b' =	230 mm
Depth, 'D' =	450 mm
Concrete Grade =	M 25
Yield Strength of Steel, f_y =	415 N/mm ²
Concrete Cover to main bars =	60 mm

Details of Reinforcement:

Diameter of bars =	20 mm
No. of bars on 230 mm face =	2
No. of bars on 450 mm face =	5
Total number of bars =	10

Total A_{st} = 3142 mm²
Percentage of Reinforcement = 3.035 < 4% O.K.

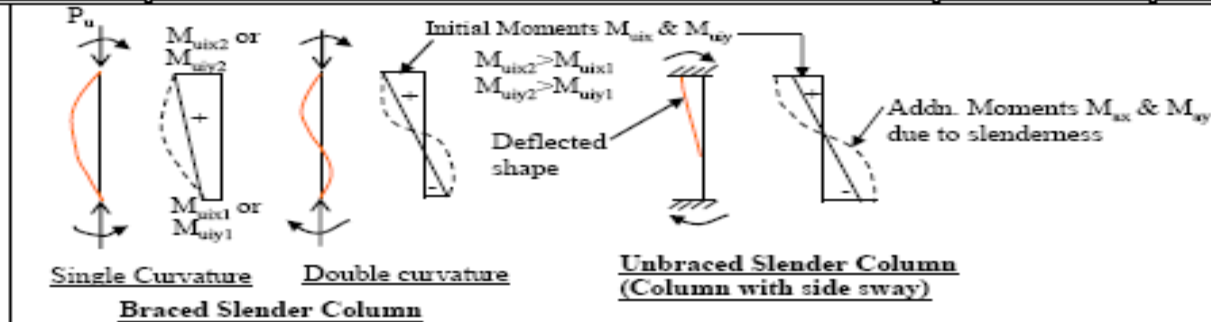


COLUMN SECTION
(Max. nos. of bar that can be shown in the section at each face = 6 only)

Col.	Applied Ultimate Loads <i>(See figures on next page)</i>					Effective Length		Unsupported Length l (m)	Column Type Braced or Unbraced
	Axial Load Pu (kN)	Initial Moment M_{ux} (kNm)	Initial Moment M_{uy} (kNm)	M_{ux1}^* (+ or -)	M_{ux2}^* (+ or -)	M_{uy1}^* (+ or -)	M_{uy2}^* (+ or -)		
C1	350	300	1.317	0.164	0.968	3.000	3.000	3.000	Braced
C2	0	0	0	0	0	0.000	0.000	0.000	Braced
C3	0	0	0	0	0	0.000	0.000	0.000	Braced
C4	0	0	0	0	0	0.000	0.000	0.000	Braced
C5	0	0	0	0	0	0.000	0.000	0.000	Braced
C6	0	0	0	0	0	0.000	0.000	0.000	Braced
C7	0	0	0	0	0	0.000	0.000	0.000	Unbraced
C8	0	0	0	0	0	0.000	0.000	0.000	Braced
C9	0	0	0	0	0	0.000	0.000	0.000	Unbraced
C10	0	0	0	0	0	0.000	0.000	0.000	Braced

Note: * at M_{ux1} and M_{uy1} indicates moment is +ve for single curvature bending, and -ve for double curvature bending.

Company's Name:	IARE Consultants	Job No.:	ABC/1001
Client:	Department Civil Engineering	Design by:	UVW
Project:	0	Date:	17-Mar-17
	0	Page No.	C/102



For calculations of Final Design Moments, see worksheet on 'Slenderness eff.'.

Summary of Results:

Col. Mk.	Axial Load (kN)	Final Design Moments		Permissible Moments		Max. Tensile Stress in Steel (N/mm^2)		$\left[\frac{M_{ux}}{M_{ux1}}\right]^{\alpha_x} + \left[\frac{M_{uy}}{M_{uy1}}\right]^{\alpha_y}$ to be less than 1.0	Status
		$M_{ux}(kNm)$	$M_{uy}(kNm)$	$M_{ux1}(kNm)$	$M_{uy1}(kNm)$	f_{ux}	f_{uy}		
C1	350	300.00	13.40	150.74	69.52	351.03	270.75	2.183	Revise Section
C2	0	0.00	0.00	141.09	61.79	360.9	345.43	0.000	Section O.K.
C3	0	0.00	0.00	141.09	61.79	360.9	345.43	0.000	Section O.K.
C4	0	0.00	0.00	141.09	61.79	360.9	345.43	0.000	Section O.K.
C5	0	0.00	0.00	141.09	61.79	360.9	345.43	0.000	Section O.K.
C6	0	0.00	0.00	141.09	61.79	360.9	345.43	0.000	Section O.K.
C7	0	0.00	0.00	141.09	61.79	360.9	345.43	0.000	Section O.K.
C8	0	0.00	0.00	141.09	61.79	360.9	345.43	0.000	Section O.K.
C9	0	0.00	0.00	141.09	61.79	360.9	345.43	0.000	Section O.K.
C10	0	0.00	0.00	141.09	61.79	360.9	345.43	0.000	Section O.K.

See Charts for range of permissible values of P_u with M_{ux1} and M_{uy1} .

WEEK-6

DESIGN OF FOOTING

AIM:

To write a program for design of Footing using excel.

HARDWARE & SOFTWARE REQUIRED:

Computer with specified configuration with installed Microsoft Office

Reinforced Concrete Footings

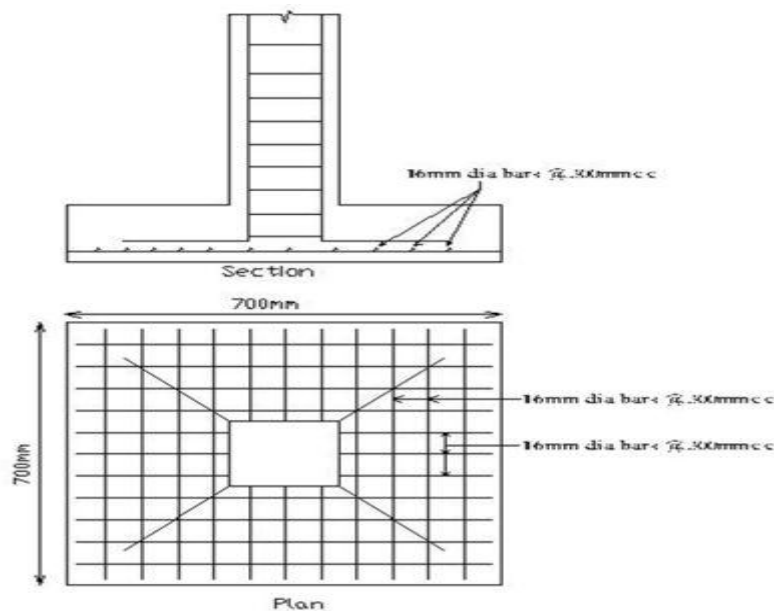
Footing comprises of the lower end of a column, pillar or wall which is enlarged with projecting courses so as to distribute load.

Footings shall be designed to sustain the applied loads, moments and forces and the induced reactions 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

$$\text{Square} = B = (w + w_1) / P_0$$

Where, P_0 = safe bearing capacity of soil

w_1 = self weight of footing

w = self weight of footing

For Rectangle = $b/d = B/D$

$$A = b \times d$$

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,

$$M_{xx} = q \times B/8 (L - a)^2$$

$$M_{xx} = q \times L/8 (B - b)^2$$

$$M_{yy} = q \times B/8 (L - a)^2$$

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.

$$\text{Shear force, } V = qB \left[\frac{1}{2}(B - b) d \right]$$

$$\text{Nominal shear stress, } T_v = k \cdot T_c$$

$$T_c = 0.16\sqrt{f_{ck}}$$

Step 4

Check for two way shear

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

$$\text{SF, } V = q \left[B^2 - (b + d)^2 \right]$$

$$\text{SF, } V = q \left[L \times B - (a + d)(b + d) \right]$$

$$\text{Nominal shear stress, } T_v = \frac{V}{2((a+d)(b+d)d)} \text{ ———— } \{ \text{for a rectangle} \}$$

$$T_v = \frac{V}{4((b+d)d)} \text{ ———— } \{ \text{for a square} \}$$

$$T_v = k \cdot T_c$$

$$k = 0.5 + \beta > 1 \quad ; \text{ [Beta } \beta = \text{ratio of sides of the column} \text{]}$$

$$T_c = 0.16\sqrt{f_{ck}}$$

$$\text{Area of steel, } A_{st} = \frac{M}{(\sigma)stjd}$$

ISOLATED FOOTING WITH BIAXIAL MOMENT

CALCULATION FOR BOTTOM STEEL

M_u about X1 X1 = (pe max x length²/2)=

90.27 KN-m per meter

Mulimit = 450.81 KN-m per meter

The section is singly reinforced

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

Hence, Ast = 640.194 mm²

Min Ast = 484.800 mm² (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 parallel to length of footing (|| to Z)

M_u about N1 N1 = (pe max x length²/2)=

93.30 KN-m per meter

The section is singly reinforced

Calc. Ast = 662.496 mm²

Min Ast = 484.8 mm² (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

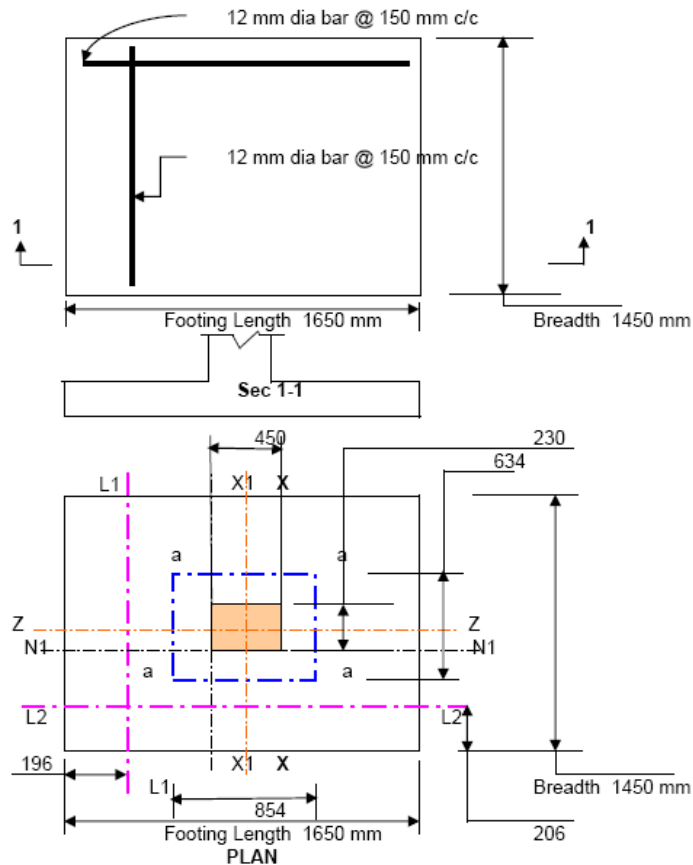
pt provided = 0.19 %

pt required = 0.16 %

provide a maximum : 0.19 %

Hence provide 12 mm dia bar @ 150 mm c/c parallel 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 $V_s = p_e \max \times 0.196 \times 1\text{m width of footing} =$

98.289 KN

Shear stress $\tau_v = V_s/bd =$ 0.243 N/mm² $\tau_c =$ 0.32 N/mm²

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 $V_s = p_e \max \times 0.206 \times 1\text{m width of footing} =$

103.304 KN

Shear stress $\tau_v = V_s/bd =$ 0.256 N/mm² $\tau_c =$ 0.32 N/mm²

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 $\tau_{v \text{ allowable}} =$ $k_s \tau_c$ $k_s = (0.5 + b/c) =$ 1.011111111 >1Hence, $k_s =$ 1 $\tau_c = 0.25 (f_{ck})^{0.5} =$ 1.118033989 N/mm² $\tau_{v \text{ allowable}} = k_s \times \tau_c =$ 1.118033989 N/mm²Shear force $V_s = 501.476 (1.65 \times 1.45 - 0.854 \times 0.634) =$

928.26 KN

Length of critical section = $2 \times (854 + 634) =$

2976 mm

Area of the critical section (length of critical sec x eff. d) =

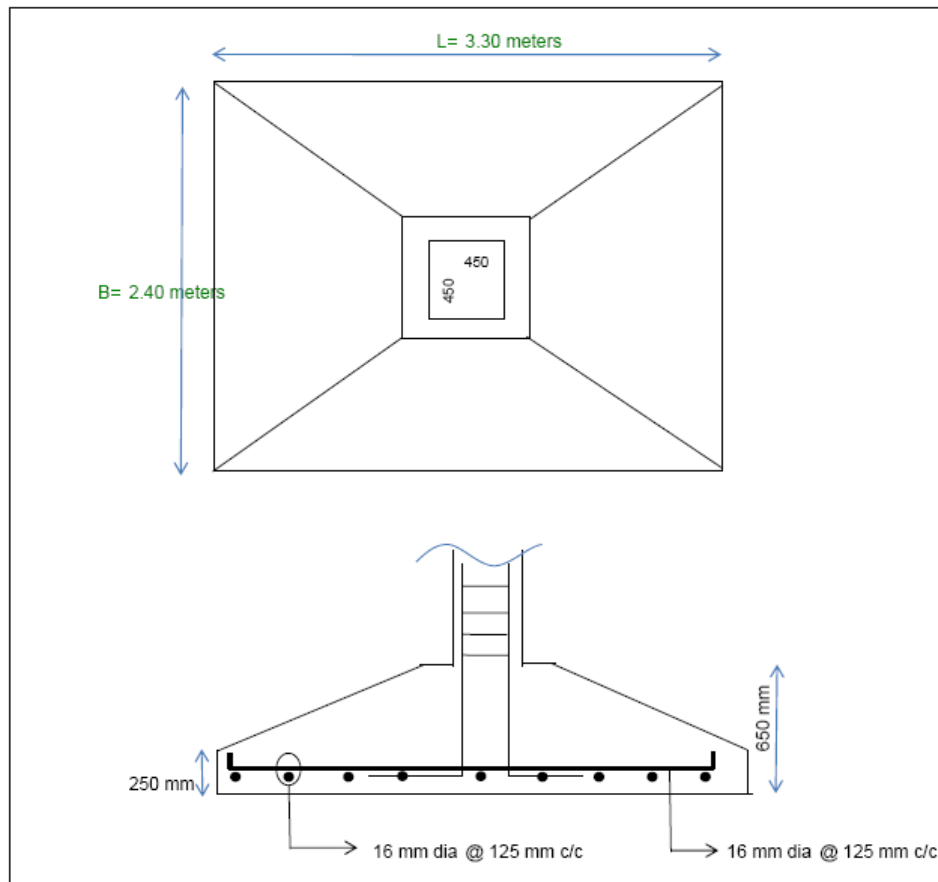
1202304 mm²Hence shear stress $\tau_v =$ 0.772 N/mm²

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



WEEK-7

ANALYSIS OF TRUSS

AIM:

To determine the Analysis of Truss using STADD pro

APPARATUS:

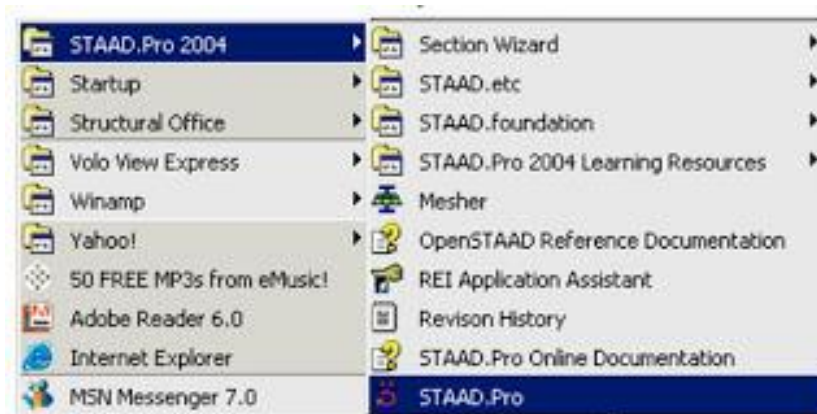
STADD pro software, compute,

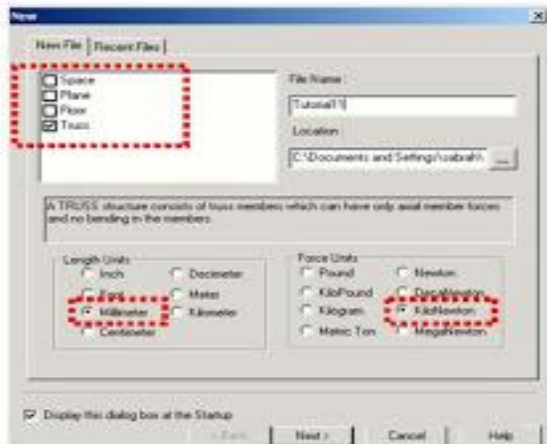
PROCEDURE:

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

How to Start Staad Pro

The Pic below shows how to start Staad Pro





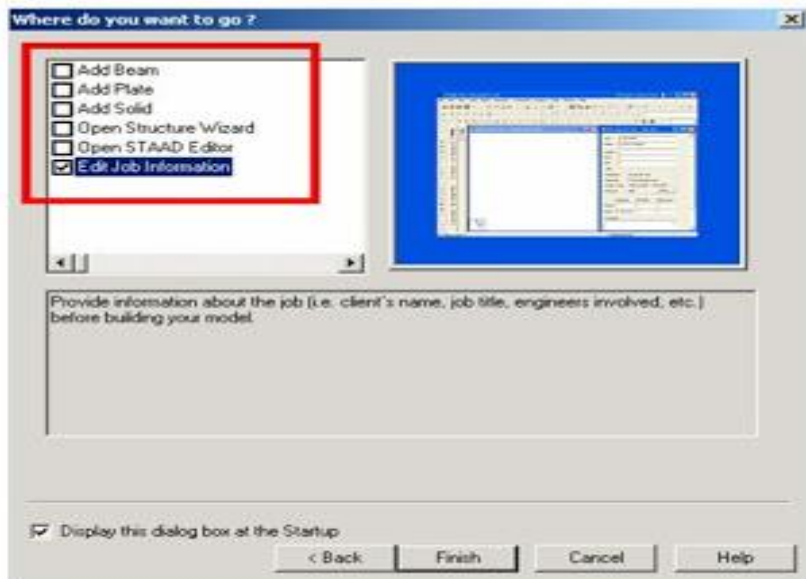
For a new file, select the type of structure you want to solve. Here select truss type.

Select from here the units, that we will deal with.

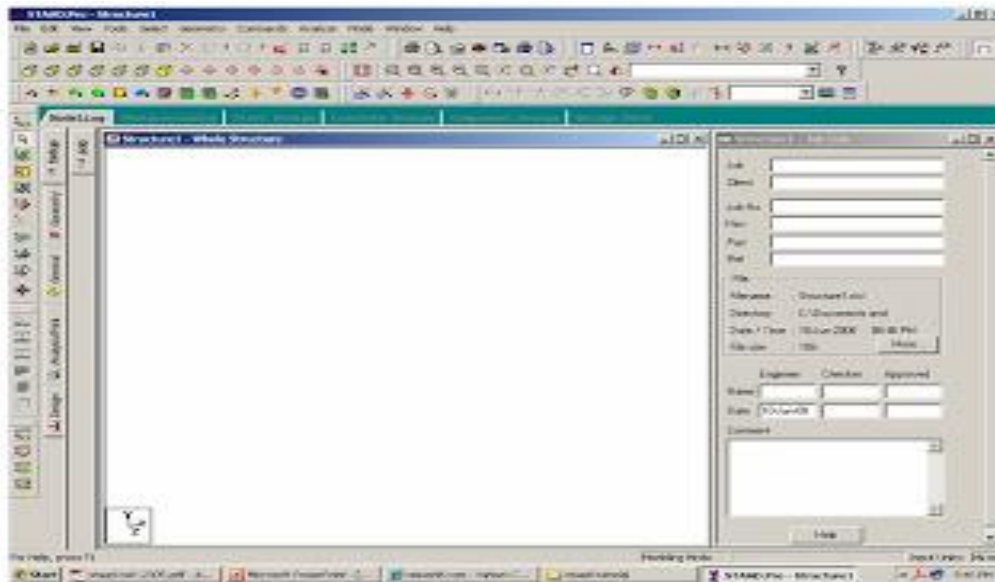
Assume that we have mm, kN.

In New File select

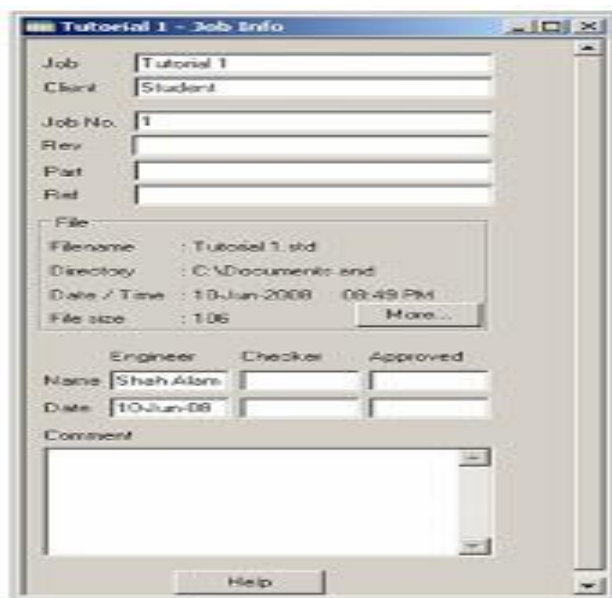
1. Truss
2. Units (here KN and mm in this example)



The staad Graphical interface will appear as shown in below picture



1. Entering Job Information

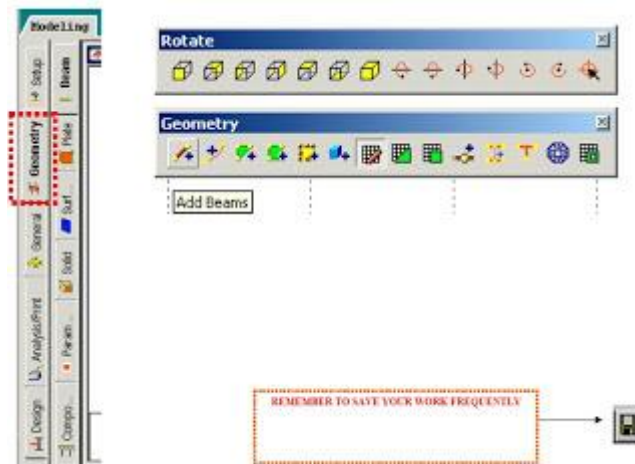
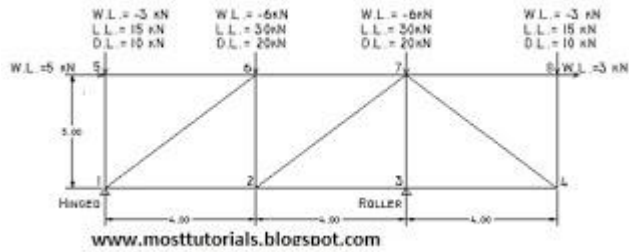
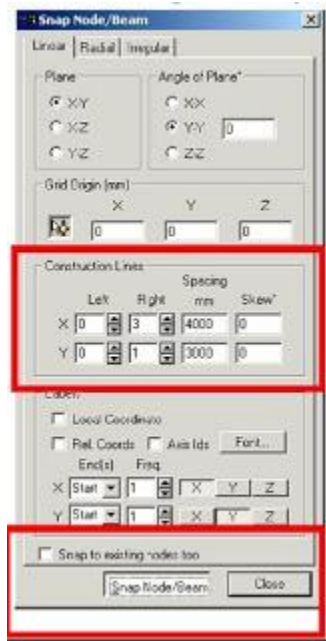


Building Model (structure) Geometry

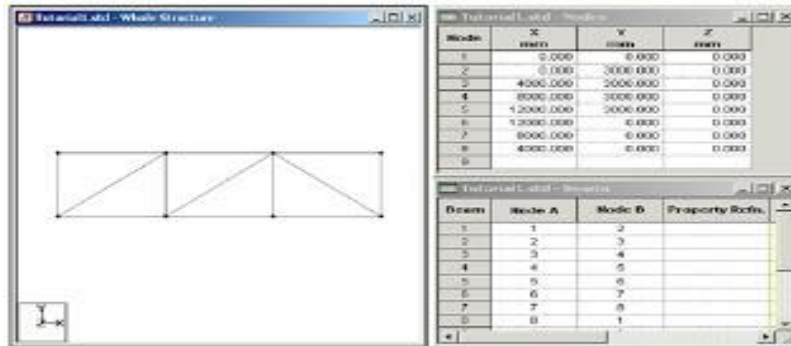
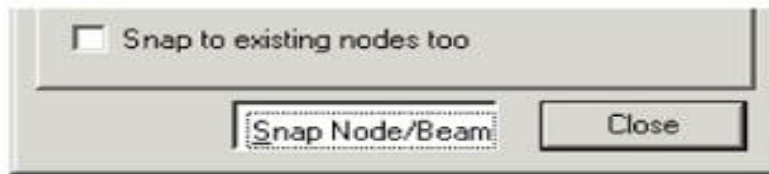
- Defining Truss Geometry

One of the methods that you can construction lines and then draw on these lines the truss members

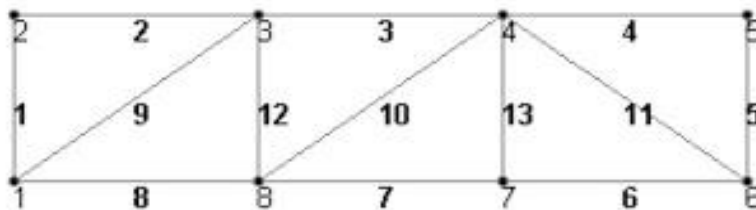
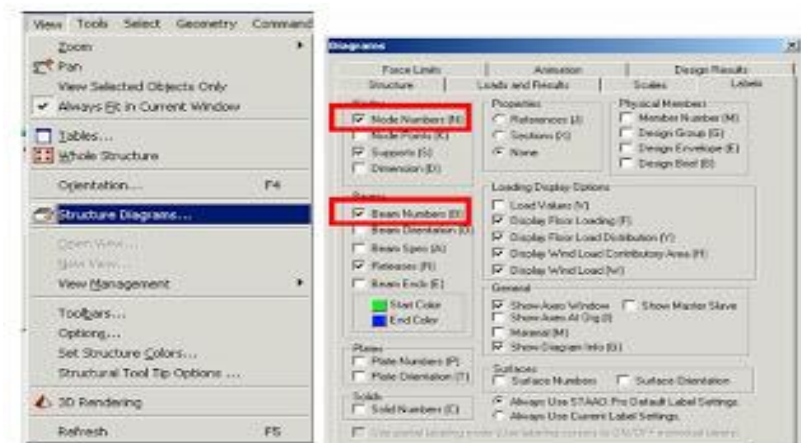
Noting that the no. of construction lines is excluding 1st line..



After clicking Snap Node/Beam ,use the mouse and connect between nodes created at the intersections of construction lines



How to see Diagram Labels (Nodes Numbers, Beam Numbers etc)



Node and beam labels are a way of identifying the entities we have drawn on the screen, and very useful when dealing with the output results

3. Defining member properties, sections



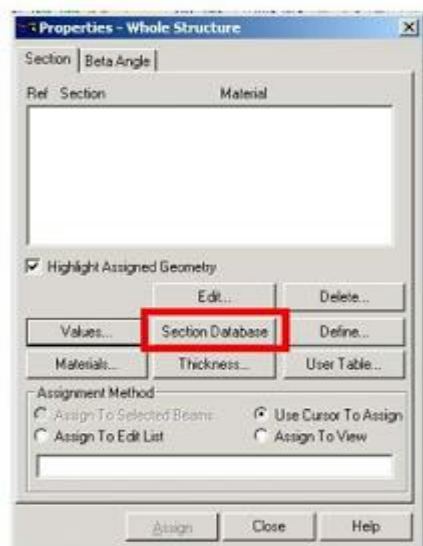
Property

In which we can define or choose sections properties of the members of the truss.

- Spec.: In which we can define or choose members' specifications.
- Support: In which we can define the supports properties (restraints).
- Load: In which we can define the applied loads, load cases, load combinations.
- Material In which we can define the material properties (ex. E, density, etc.)

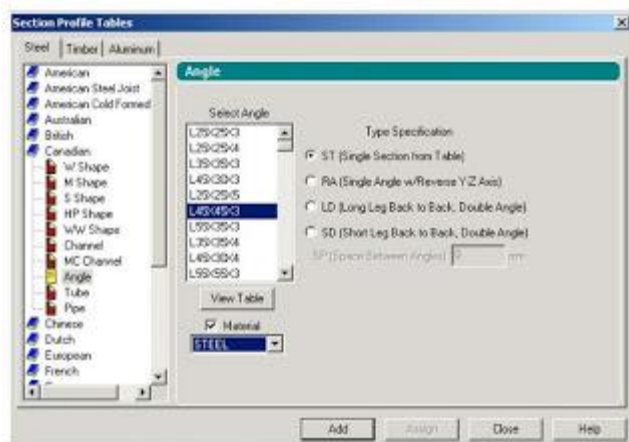
Assume Preliminary

- Sections:

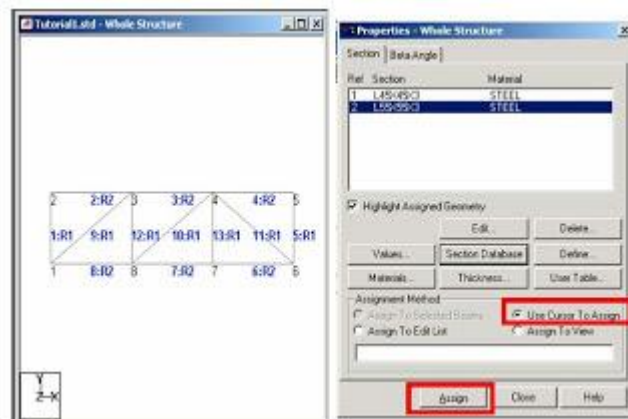


All Top and Bot chords are one size **L 55*55*3**

All Diagonals and verticals are one size **L 45 * 45* 3**

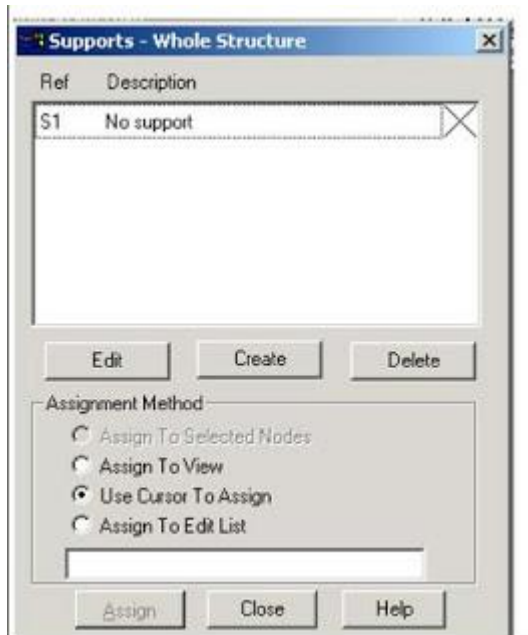


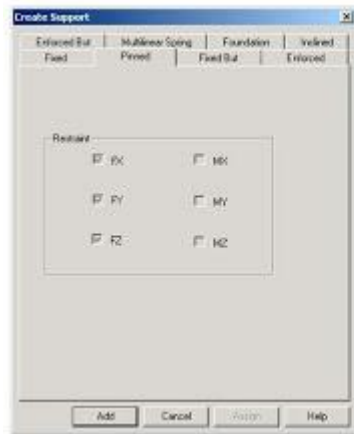
Assigning the sections created to the model



Supports

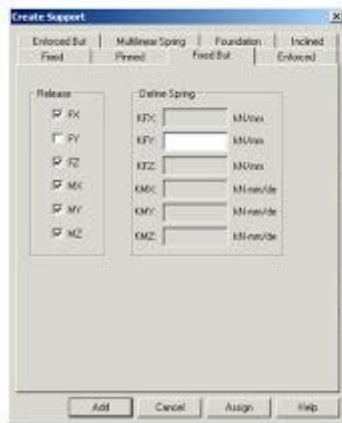
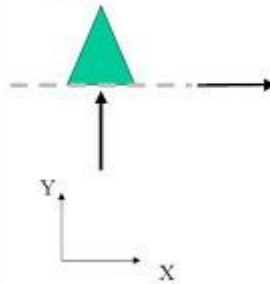
By clicking on the support icon the shown window will appear. We have to create new kinds of supports





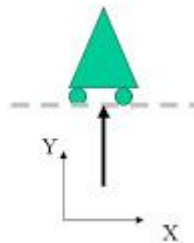
Hinged or Pinned support
Restrains (Fx, Fy, Fz)

<defaults>

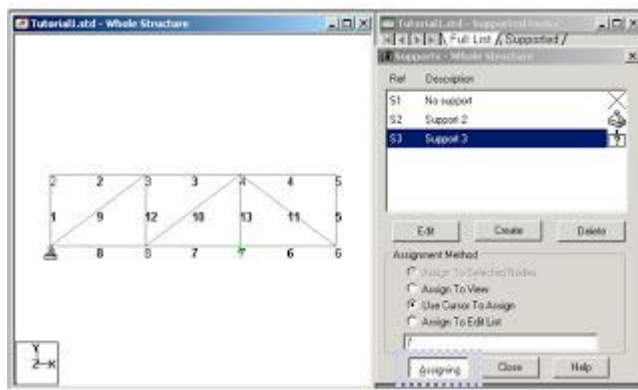


(Fixed but) Support

All directions are restrained
or fixed and we choose what
are the directions that we
want to release.



Use the mouse and click on the nodes according to its support type as shown below

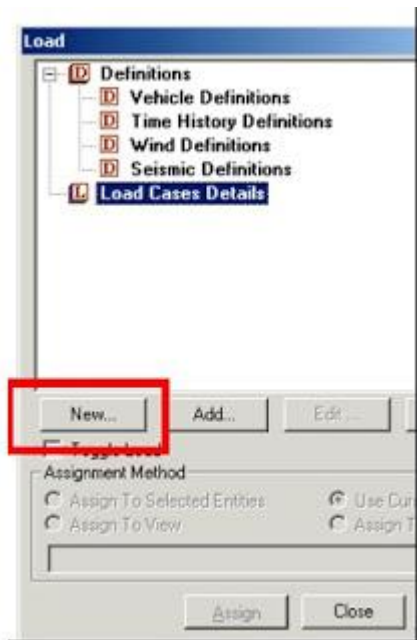


- **4. Defining Loads**

The creation and assignment of load cases involves the following two steps:

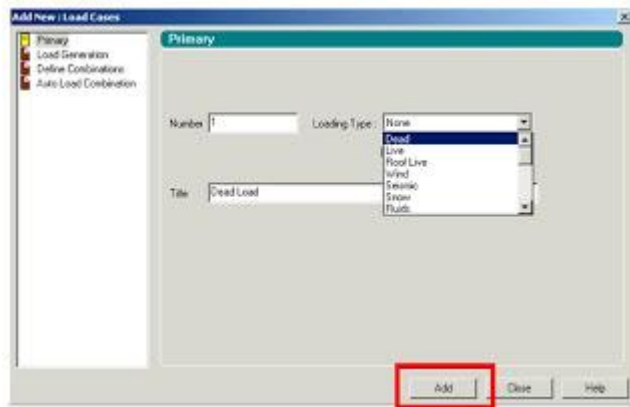


1. First, we will be creating all 3-load cases.
2. Then, we will be assigning them to the respective members/nodes

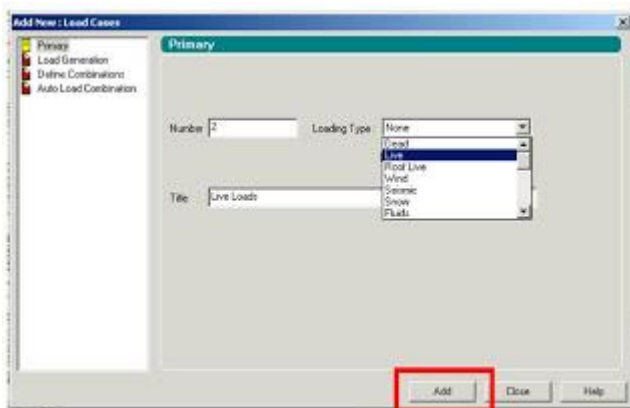


For example

Dead Load Case can be Load Case No.1



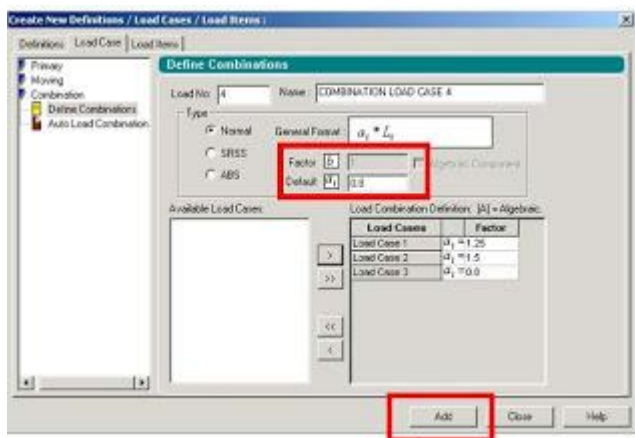
Live Load Case can be Load Case No. 2 and Wind Load is the Case No.3



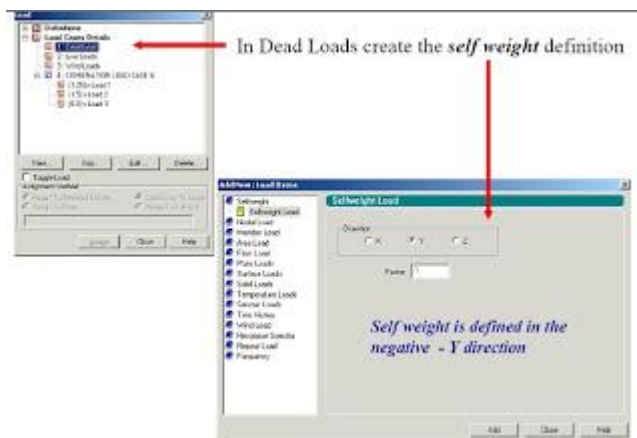
Also, we can define **load combinations** according to required.

For example, we can create a load combination

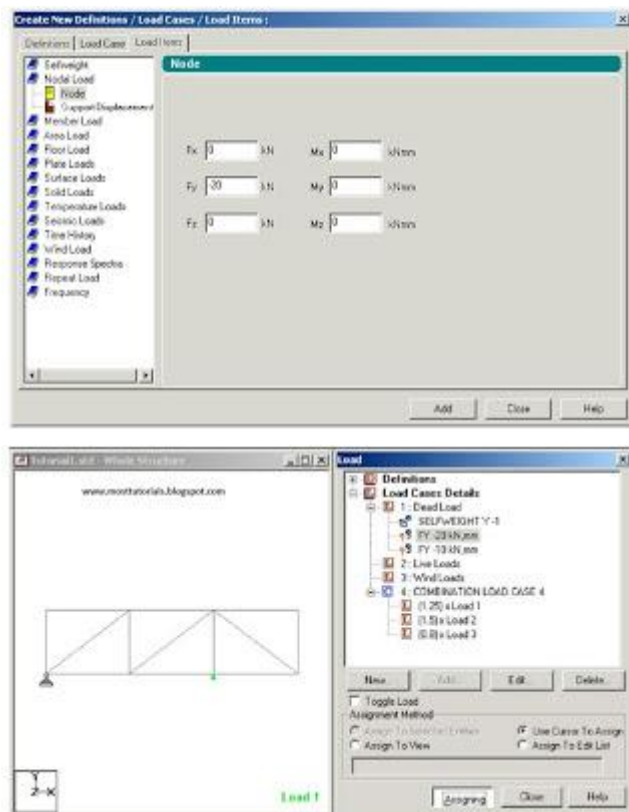
1.25 D.L. + 1.5 L.L + 0.8 W.L.

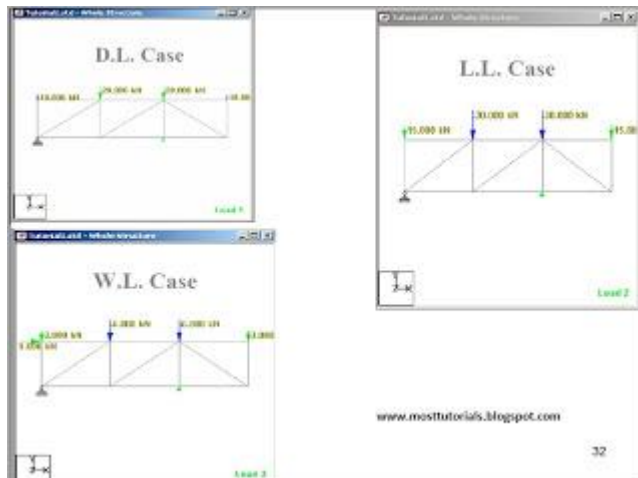


And In Dead Load Create the Self weight as

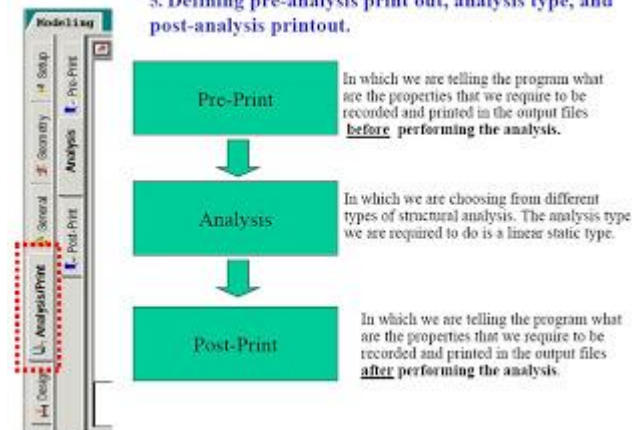


In each Load Case Create the Nodal Loads on truss as specified before. Make sure of the direction of forces according to global coordinates





5. Defining pre-analysis print out, analysis type, and post-analysis printout.



Analysis

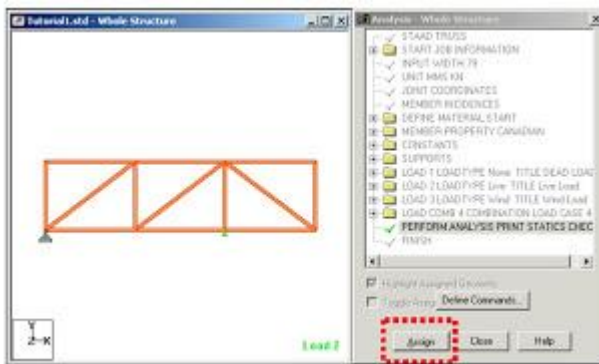
To specify the Analysis command, first go to **Analysis/Print** Page from the left side of the screen.

Then, click on the Analysis sub-page from the second row of pages as shown below.

Choose Statics Check and Click **Add** then Close.

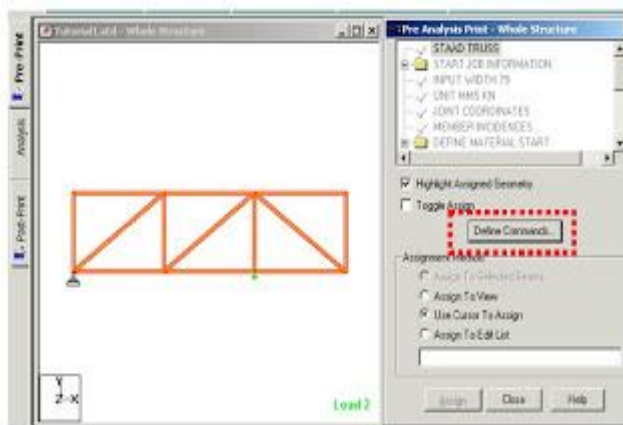
After Creating the command of (Perform Analysis-Check); We have to assign the members that want to be this type of analysis for it.

Click **Assign** and then use the cursor and choose all the members, thus all the members should be highlighted as shown below.

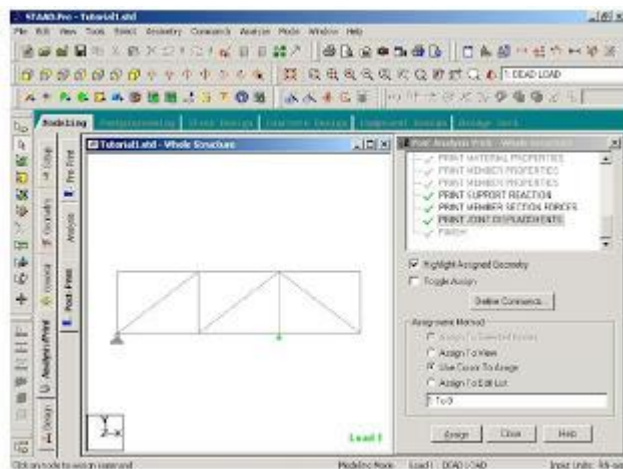


Pre-Print

To add the Pre-Print Command click " Define Commands"



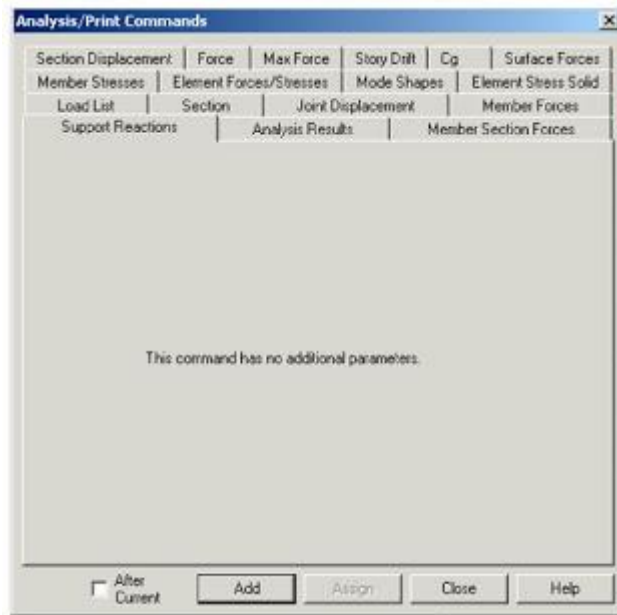
For example here we have chosen to print the support reactions,member forces,joint displacements



Post Print Commands

- Support reactions
- Analysis results
- Member Forces
- Max Forces

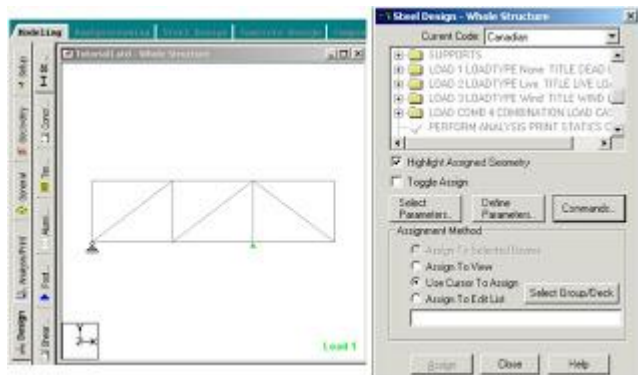
Add which property you need and then use cursor to ASSIGN to which member in the truss



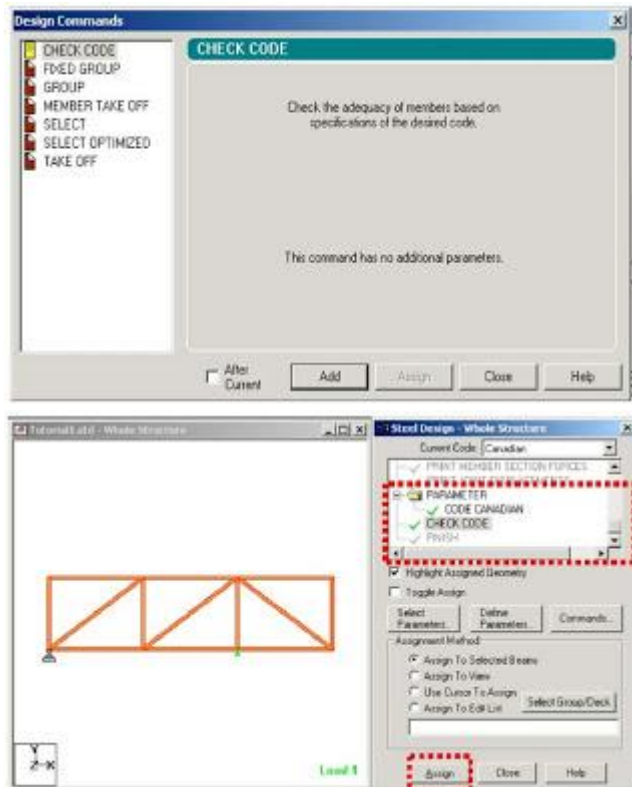
6. Defining Design Requirements

Steps

1. To Specify steel design parameters, go to Design/Steel page from the left side of the screen. Make sure that under the Current Code selections on the top right hand side ,Canadian is selected.

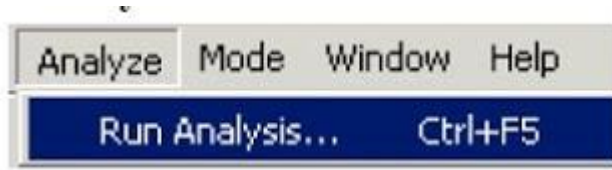


There are many Design Commands in the STAAD Design subroutine. Here, we will use only to **Check Code**, regarding adequacy of members.

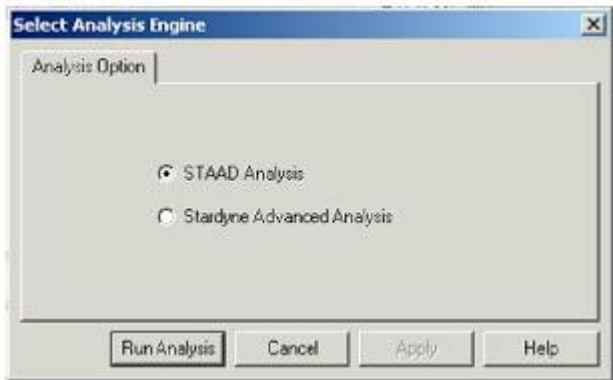


7-Analysis & Viewing Results

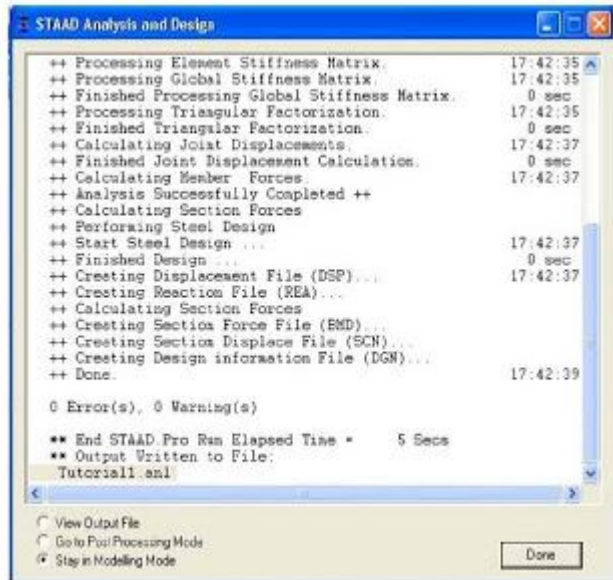
STAAD Performs Analysis and Design simultaneously. In order to Perform Analysis and Design, select the Run Analysis option from the **Analyze** menu



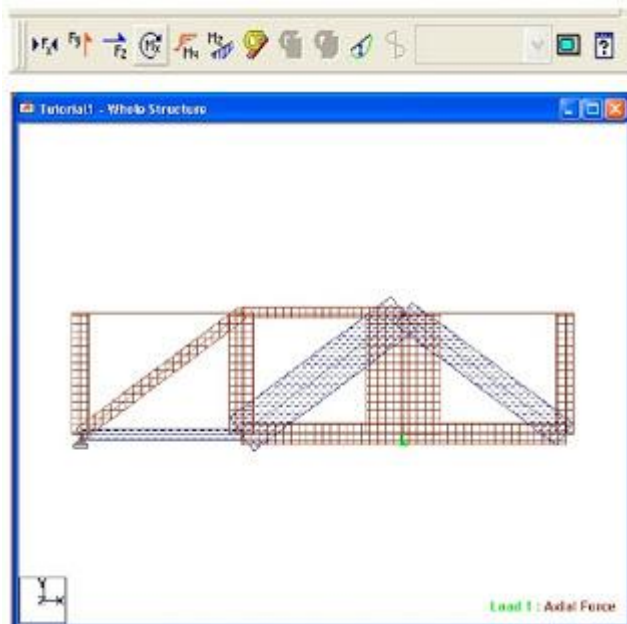
When you select the Run Analysis option from the Analyze menu, the following dialog box appears. We are presented with the choice of 2 engines. The **STAAD engine** and the **STARDYNE Advanced Analysis engine**. The **STARDYNE** Analysis engine is suitable for advanced problems such as **Buckling Analysis, Modal Extraction** using various method ,etc
STAAD engine is suitable for this tutorial. Click on th **Run Analysis** button.



The solving process is shown in pop up screen.



Visualization of Some Results



WEEK-8

ANALYSIS OF MULTISTORIED SPACE FRAME

AIM:

Analysis of Multistoried space frame using STADD pro

HARDWARE & SOFTWARE REQUIRED:

Computer with specified configuration with installed STADD Pro Software

PROCEDURE AND ANALYSIS STEPS

STAAD

Staad is powerful design software licensed by Bentley .Staad stands for structural analysis and Design Any object which is stable under a given loading can be considered as structure. So first find the outline of the structure, where as analysis is the estimation of what are the type of loads that acts on the beam and calculation of shear force and bending moment comes under analysis stage. Design phase is designing the type of materials and its dimensions to resist the load. this we do after the analysis.

To calculate s.f.d and b.m.d of a complex loading beam it takes about an hour. So when it comes into the building with several members it will take a week. Staad pro is a very powerful tool which does this job in just an hour's staad is a best alternative for high rise buildings.

Now a days most of the high rise buildings are designed by staad which makes a compulsion for a civil engineer to know about this software.

These software can be used to carry rcc ,steel, bridge , truss etc according to various country codes.

Staad foundation:

Staad foundation is a powerful tool used to calculate different types of foundations. It is also licensed by Bentley software's. All Bentley software's cost about 10 lakhs and so all engineers can't use it due to heavy cost.

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) Foundation

DEEP ($D > B$)

1. Pile Cap
2. Driller Pier

1. Isolated footing is spread footing which is common type of footing.

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

other.

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

4. 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 be given 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.

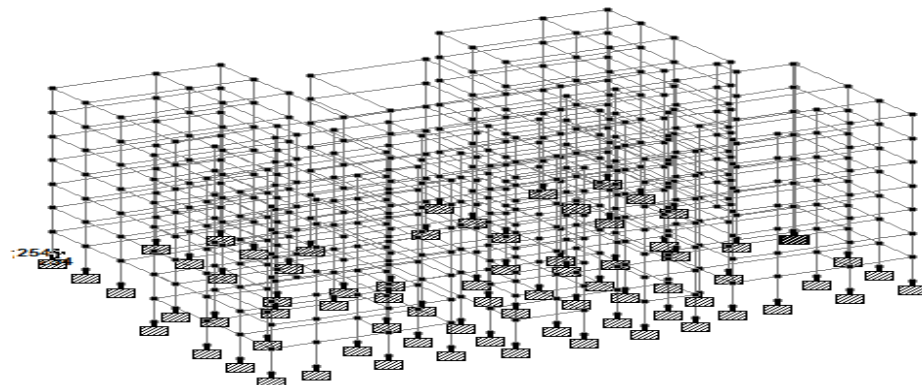
Another advantage of foundations is even after the design; properties of the members can be updated if required.

The following properties can be updated

Column Position
Column Shape
Column Size
Load Cases
Support List

The multistoreyed 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 in generating 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= $0.12 \times 25 + 1 = 3 \text{ kn/m}^2$

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**

Live load	dead load	wind load
1.5	1.5	0
1.2	1.2	1.2
0.9	0.9	0.9

When the building is designed for both wind and seismic loads maximum of both is taken.

Because wind and seismic do not come at same time as per code.

Structure is analyzed by taking all the above combinations.

WEEK-9

ANALYSIS OF BRIDGE DECK SLAB

AIM:

Analysis of Bridge Deck Slab using STADD pro

HARDWARE & SOFTWARE REQUIRED:

Computer with specified configuration with installed STADD Pro Software

PROCEDURE AND ANALYSIS STEPS

The combination of STAAD.pro and STAAD.beava can make your bridge design and analysis easier. STAAD.pro is first used to construct the bridge geometry and STAAD.beava 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:

1. Maximum plate stresses, moment about the local x axis of a plate (M_x), moment about the local y axis of a plate (M_y) etc. used to design for concrete deck reinforcement.
2. Maximum support reactions to design isolated, pile cap, and mat foundations.
3. Maximum bending moment or axial force in a member used to design members as per the AASHTO code.
4. Maximum deflection at mid span.

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 1 shows the bridge design procedure discussed above.



Figure 1: Bridge Design process in STAAD.pro

Creating the Bridge Geometry/Structural Analysis

Figure 2 shows a bridge with the dimensions.

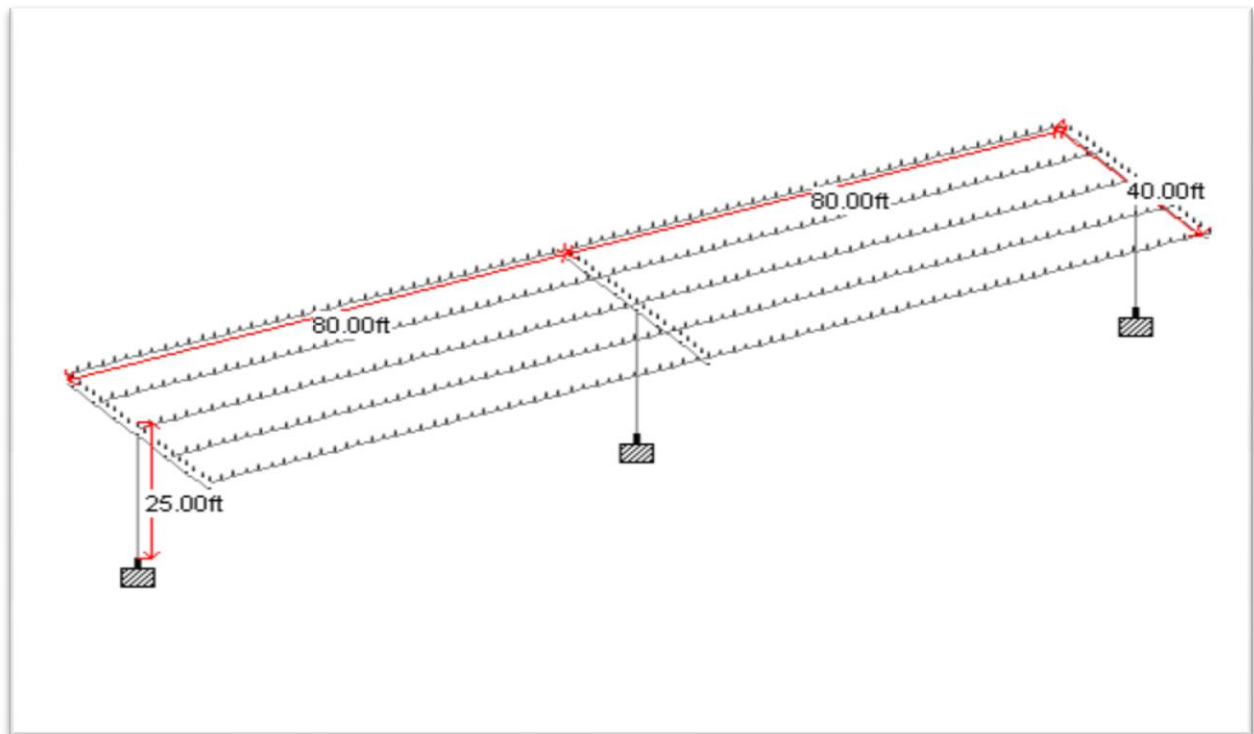


Figure 2: Bridge Dimensions

Step By Step Procedure

1. Open STAAD.pro with the default units of **Kip-Ft** and use the **Space** option.
2. Click the **Next** button and select the **Add Beam** mode. Click **Finish**.
3. The goal of the next few steps is to draw the stick model of the Bridge Structure (i.e. the beams and the girders). Select the **X-Z** grid option. Create two grid lines in the **x** direction at 80ft spacing. Create four grid lines in the **z** direction at 10ft spacing.
4. Click on the **Snap Node/Beam** button and draw the beams and the girders as shown below. First draw five 160ft girders. Then draw the 40ft beams in the **z** direction.
5. Click on **Geometry->Intersect Selected Members->Highlight**. STAAD.pro will highlight all the beams that intersect each other no common nodes. To break these beams at the intersection point, click on **Geometry->Intersect Selected Members->Intersect**
6. The beams have been created. The columns will now be created using the translational repeat command. Select nodes 6, 14, and 7 using the nodes cursor as shown below. The node numbers may vary depending upon how the model was constructed. Select **Geometry->Translational Repeat** from the menu. Select the **y** direction for the translational repeat and select enter a **Default Step Spacing** of **-25ft**

7. The deck of this bridge structure will be created in the following steps using the **Generate Surface Meshing Tool**. In reality, spacing of beams etc. may not be regular and hence it may become difficult to create the deck of the bridge using the **Generate Surface Meshing Tool**. The **Parametric Meshing Mode** could become very useful in these circumstances.

8. Select the **Geometry->Generate Surface Meshing** tool from the menu. Select four nodes that outline the 160ft x 40ft deck. Simply click on node 1, 3, 11, and 10 and select node 1 again to complete the command. Select the **Quadrilateral Meshing** option.

9. 2ft x 2ft element size is adequate for this type of model. Hence input the parameters as shown below.

10. The mesh will be created. To view the mesh properly, you will need to click on the **Setup Control Tab** on your left.

11. You should note that the girders and beams are automatically broken down into smaller elements. In reality, the girders are physically attached to the deck hence it is ok to mesh them. The concrete beams parallel to the z-axis are not attached to the deck. The load from the deck is transferred to the 40ft steel girders. The load from the girders is then transferred to the concrete beams. Hence, we should merge the beams in the z direction. Select the **Select->Beam Parallel To->z** from the menu.

12. Select **Geometry->Merge Selected Members** to merge the split concrete beams. Select each entry and press the **Merge** button.

13. By merging the beams together, the concrete column to beam connectivity is lost. Hence, click on **Geometry->Intersect Selected Members->Highlight**. STAAD.pro will highlight all the beams that intersect each other no common nodes. To break these beams at the intersection point, click on **Geometry->Intersect Selected Members->Intersect**.

14. The geometry has been created. The properties and specifications have to be assigned.

Concrete Deck	Plate elements - 12in thick Material - Concrete
Steel Girders – Parallel to x axis	40ft Beam elements – W24X103 Material – Steel Offset = 0.5 x Depth of Beam + 0.5 thickness of slab = 0.5 x 2.044167 ft + 0.5 x 1ft = 1.5220835 ft at both ends.
Concrete Beam – Parallel to the z axis	2ft x 2ft Sections Material – Concrete Offset = 0.5 x Depth of Concrete Beam + Depth of Steel Beam + 0.5 thickness of slab = 0.5 x 2ft + 2.044167 ft + 0.5 x 1ft = 3.544167 ft at both ends.
Concrete Columns	2ft Circular Sections Material – Concrete Offset = Depth of Concrete Beam + Depth of Steel Beam + 0.5 thickness of slab = 2ft + 2.044167 ft + 0.5 x 1ft = 4.544167ft at the end connected to the concrete beams.

15. Click on the **General->Property** control tab on your left and click on the **Section Database** button.
16. Select the **W24x103** section from the American **W shape** database and click on the **Add** button.
17. Select the **W24x103** section that has been created. Click on the Select the **Select- >Beam Parallel To->X** from the menu and click on the **Assign** button on the right.
18. Select the **Define** button on the right and select the **Circle** section profile. Input a 2ft diameter and press the **Add** button.
19. Select the **Rectangle** section profile and input 2ft in the **YD** and **ZD** input boxes. Press the **Add** button. Press the **Close** button.
20. Select the Cir 24 section that has been created. Click on the Select the **Select->Beam Parallel To->Y** from the menu and click on the **Assign** button on the right.
21. Select the **Rect 24.00x24.00** section that has been created. Click on the Select the **Select->Beam Parallel To->Z** from the menu and click on the **Assign** button on the right.
22. Press the **Thickness** button the right and input 1ft in the **Node 1** input box and press the **Add** button.
23. Select the newly create **Plate Thickness** entry in the **Properties** dialog box. Select all the plates using the **Plates Cursor** and press the **Assign** button.
24. Select the **Spec** sub-control tab on the left.
25. Press the **Beam** button. Select the **Offset** tab. Select the start option from the **location** selection box. Select the local option from the **Direction** selection box. Input an offset of -1.5220835 in the **Y** input box. Press the **Add** button.
26. Press the **Beam** button. Select the **Offset** tab. Select the **End** option from the **Location** selection box. Select the **Local** option from the **Direction** selection box. Input an offset of -1.5220835 in the **Y** input box. Press the **Add** button.
27. Assign these specifications to the steel girders parallel to the x-axis. Select the **START LOCAL 0 – 1.52208 0** specification that has been created. Click on the **Select->Beam Parallel To->x** from the menu and click on the **Assign** button on the right.
28. Select the **END LOCAL 0 -1.52208 0** specification that has been created. Click on the Select the **Select->Beam Parallel To->x** from the menu and click on the **Assign** button on the right.
29. Repeat Steps# 25 to 28 for the concrete beams but use a y-offset of -3.544167 ft at both ends. For Step# 28, you will need to select beams parallel to the z-axis.

30. For the columns assign a local x-offset of 4.544167ft at the end connected to the concrete beams.
31. Click on the **View->3D Rendering** in the menu.
32. Select the **General->Support** tab on the left and click on the **Create** button in the right hand side **Data Area**. Click on the **Add** button (i.e create a fixed support entry).
33. Select the newly created **S2 Support 2** entry and using the nodes cursor select nodes 16, 17, and 18. Click on the **Assign** button.
34. Select the **General->Load** tab on the left and click on **Load Case Details** on the right hand side in the **Data Area**.
35. Click on the **Add** button on the right. Input the **“Dead Load”** in the **Title** input box. Select **“Dead”** in the **Loading type** selection box. Press the **Add** button. Click the **Close** button.
36. Select the newly created **1: Dead Load** entry in the data area. Press the **Add** button. Select the **Selfweight** item and press the **Add** button.
37. Click on the **Analysis/Print** control tab item on the left and press the **Add** button.
38. Click on the **Analyze->Run Analysis** menu. Use the **STAAD Analysis** option and click on the **Run Analysis** button.
39. If the analysis completed successfully, you should look at the exaggerated deflected shape of the bridge under the action of selfweight. Try to find out any connectivity problems etc. You can go to the **Post-Processing** mode by clicking on **Mode- >Postprocessing** command in the menu.
40. You can look at the bending moment diagram for the bridge by clicking on the **Beam->Forces** control tab on your left in the **Post-Processing** mode.
41. You can look at the stress distribution diagram for the bridge by clicking on the **Plate** control tab on your left. Select the **Max Absolute** stress type from the **Stress Type** selection box and click on the **Ok** button.