

COMPUTER AIDED CIVIL ENGINEERING DRAFTING ANALYSIS AND DESIGN

LAB MANUAL

**SUBJECT : CACEDA&D LAB
CLASS : BE VI SEM AICTE
CODE : PC 362 CE**

**MUFFAKHAM JAH COLLEGE OF ENGINEERING AND TECHNOLOGY
AFFILIATED TO OSMANIA UNIVERSITY, ACCREDITED BY NAAC
APPROVED BY AICTE
ROAD NO.3, BANJARA HILLS, HYDERABAD**

LIST FO EXPERIMENTS

SL.NO	NAME OF THE EXPERIMENT
INTRODUCTION TO C	
1	DETERMINE THE MAXIMUM B.M AND S.F OF UDL
2	SOLUTION OF NON-LINEAR EQUATIONS USING NEWTON-RAPHSON TECHNIQUE
3	DEFLECTION OF BEAMS
INTRODUCTION TO STAAD	
4	ANALYSIS OF CONTINUOUS BEAM
5	ANALYSIS OF SINGLE STOREY FRAME
6	ANALYSIS OF MULTI STOREY FRAME
7	ANALYSIS OF 3D - FRAME
8	ANALYSIS OF TRUSS
INTRODUCTION TO EXCEL SPREADSHEET	
9	DESIGN OF TWO WAY SLAB
10	DESIGN OF FLAT SLAB

INTRODUCTION

A mainframe computer, informally called a mainframe or big iron, is a computer used primarily by large organizations for critical applications like bulk data processing for tasks such as censuses, industry and consumer statistics, enterprise resource planning, and large-scale transaction processing. A mainframe computer is large but not as large as a supercomputer and has more processing power than some other classes of computers, such as minicomputers, servers, workstations, and personal computers. Most large-scale computer-system architectures were established in the 1960s, but they continue to evolve. Mainframe computers are often used as servers.

1. Micro Computer :

Micro Computer, as name suggests, is a personal computer that is specially designed for personal use and generally consists of single chip that is CPU, data memory, I/O buses, etc. It can be used by one person at a time. Its type includes tablet and smartphone microcomputers, desktop micro computers, workstation and server microcomputers, etc.

Example : Laptop, desktop, IBM-PC, etc.

2. Mini Computer :

Mini Computer, as name suggests, is a type of computer that offers most features and capabilities that large computer generally offers. It generally supports multiple users at a time so one can say that it is a multiprocessing system. It is a smaller computer designed for business applications and services, and also can do time-sharing, batch processing, online processing, etc.

Example : AS/400 computers, Motorola 68040, MV 1500etc

A system configuration (SC) in systems engineering defines the computers, processes, and devices that compose the system and its boundary. More generally, the system configuration is the specific definition of the elements that define and/or prescribe what a system is composed of.

COMPUTER

HARDWARE

Hardware refers to the physical, tangible computer equipment and devices, which provide support for major functions such as input, processing (internal storage, computation and control), output, secondary storage (for data and programs), and communication.

HARDWARE CATEGORIES (Functional Parts)

A computer system is a set of integrated devices that input, output, process, and store data and information. Computer systems are currently built around at least one digital processing device. There are five main hardware components in a computer system: Input, Processing, Storage, Output and Communication devices.

OPERATING SYSTEMS

Is a software that acts as an interface between the user, application software and the computer hardware

EXAPMLE OF OPERATING SYSTEM

1. Microsoft Disk operating system (Ms DOS)
2. Windows (98, 2000, XP, vista)
3. Linux
4. Unix

PARTS OF OPERATING SYSTEM

1. Shell – it is the outer part of an operating system and it is responsible of interacting with the operating system
2. Kernel – Responsible for managing and controlling computer resources such as the processor, main memory, storage devices, input devices, output devices and communication devices

RESOURCE UNER THE OPERATING SYSTEM CONTROL

- The processor
- Main memory
- Input/Output Device
- Secondary storage devices
- Communication devices and ports

Computer software - also known as programs or applications. They are classified into two classes namely - sytem software and application software

A **high-level language (HLL)** is a programming language such as C, FORTRAN, or Pascal that enables a programmer to write programs that are more or less independent of a particular type of computer. Such languages are considered high-level because they are closer to human languages and further from machine languages.

B. Development and Implementation of Programs for the following in C Language

Introduction to C & C++

C PROGRAMMING BASICS TO WRITE A C PROGRAM:

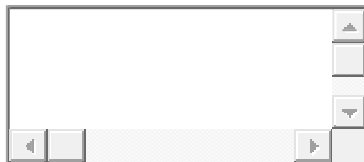
Below are few commands and syntax used in C programming to write a simple C program. Let's see all the sections of a simple C program line by line.

C Basic commands	Explanation
<code>#include <stdio.h></code>	This is a preprocessor command that includes standard input output header file(stdio.h) from the C library before compiling a C program
<code>int main()</code>	This is the main function from where execution of any C program begins.
<code>{</code>	This indicates the beginning of the main function.
<code>/*_some_comments_*/</code>	whatever is given inside the command “/* */” in any C program, won't be considered for compilation and execution.
<code>printf(“Hello_World!“);</code>	printf command prints the output onto the screen.
<code>getch();</code>	This command waits for any character input from keyboard.
<code>return 0;</code>	This command terminates C program (main function) and returns 0.
<code>}</code>	This indicates the end of the main function.

2. A SIMPLE C PROGRAM:

Below C program is a very simple and basic program in C programming language. This C program displays “Hello World!” in the output window. And, all syntax and commands in C programming are case sensitive. Also, each statement should be ended with semicolon (;) which is a statement terminator.

C



```
1 #include <stdio.h>
2 int main()
3 {
4  /* Our first simple C basic program */
5  printf(“Hello World! ”);
6  getch();
7  return 0;
```

```
8 }
```

OUTPUT:

```
Hello World!
```

3. STEPS TO WRITE C PROGRAMS AND GET THE OUTPUT:

Below are the steps to be followed for any C program to create and get the output. This is common to all C program and there is no exception whether its a very small C program or very large C program.

1. Create
2. Compile
3. Execute or Run
4. Get the Output

BASIC STRUCTURE OF A C PROGRAM:

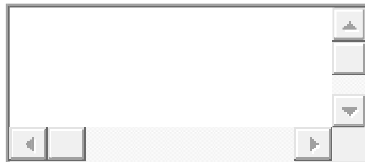
Structure of C program is defined by set of rules called protocol, to be followed by programmer while writing C program. All C programs are having sections/parts which are mentioned below.

1. Documentation section
2. Link Section
3. Definition Section
4. Global declaration section
5. Function prototype declaration section
6. Main function
7. User defined function definition section

EXAMPLE C PROGRAM TO COMPARE ALL THE SECTIONS:

You can compare all the sections of a C program with the below C program.

C



```
1  /*
2  Documentation section
3  C programming basics & structure of C programs
4  Author: fresh2refresh.com
5  Date : 01/01/2012
6  */
7
8  #include <stdio.h> /* Link section */
9  int total = 0; /* Global declaration, definition section */
10 int sum (int, int); /* Function declaration section */
11 int main () /* Main function */
12 {
13  printf ("This is a C basic program \n");
14  total = sum (1, 1);
15  printf ("Sum of two numbers : %d \n", total);
16  return 0;
```

```

17 }
18
19 int sum (int a, int b) /* User defined function */
20 {
21     return a + b;    /* definition section */
22 }

```

OUTPUT:

This is a C basic program Sum of two numbers : 2

DESCRIPTION FOR EACH SECTION OF THE C PROGRAM:

- Let us see about each section of a C basic program in detail below.
- Please note that a C program mayn't have all below mentioned sections except main function and link sections.
- Also, a C program structure mayn't be in below mentioned order.

Sections	Description
Documentation section	We can give comments about the program, creation or modified date, author name etc in this section. The characters or words or anything which are given between “/*” and “*/”, won't be considered by C compiler for compilation process. These will be ignored by C compiler during compilation. Example : /* comment line1 comment line2 comment 3 */
Link Section	Header files that are required to execute a C program are included in this section
Definition Section	In this section, variables are defined and values are set to these variables.
Global declaration section	Global variables are defined in this section. When a variable is to be used throughout the program, can be defined in this section.
Function prototype declaration section	Function prototype gives many information about a function like return type, parameter names used inside the function.
Main function	Every C program is started from main function and this function contains two major sections called declaration section and executable section.
User defined function section	User can define their own functions in this section which perform particular task as per the user requirement.


```

{
printf("INVALID!!!");
}
else
{
if(x>=0 && x<=L1)
{
Vx=-w*x;
Mx=-w*x*x/2;
}
if(x>L1 && x<=(L1+L2))
{
Vx=-w*L1;
Mx=-w*L1*(x-0.5*L1);
}
C=h/2;
I=b*h*h*h/12;
Q=b*h*h/8;
Ss=((Vx*Q)/(I*b)/1000);
Bs=(Mx*C/I)/1000;
}

printf("At a distance %.2f m from left end\nShear force= %.2f N\nBending Moment= %.2f
N-m\nShear Stress= %.2f kN/m2\nBending Stress= %.2f kN/m2\n", x, Vx, Mx, Ss, Bs);

getch();
}

```

EXPERIMENT- 2

SOLUTION OF NON-LINEAR EQUATIONS USING NEWTON-RAPHSON TECHNIQUE

Aim: - To Find real roots of nonlinear equation using Newton Raphson Method

Software: - C language

Program to find the real roots of nonlinear equation using Newton Raphson Method

```
#include<stdio.h>

#include<conio.h>

#include<math.h>

#include<stdlib.h>

/* Defining equation to be solved.

Change this equation to solve another problem. */

#define f(x) 3*x - cos(x) -1

/* Defining derivative of g(x).

As you change f(x), change this function also. */

#define g(x) 3 + sin(x)

Int main()

{

float x0, x1, f0, f1, g0, e;

int step = 1, N;

/* Inputs */

printf("\nEnter initial guess:\n");

scanf("%f", &x0);

printf("Enter tolerable error:\n");

scanf("%f", &e);

printf("Enter maximum iteration:\n");
```

```

scanf("%d", &N);

/* Implementing Newton Raphson Method */

printf("\nStep\t\tx0\t\tf(x0)\t\tx1\t\tf(x1)\n");

do

{

g0 = g(x0);

f0 = f(x0);

if(g0 == 0.0)

{

printf("Mathematical Error.");

exit(0);

}

x1 = x0 - f0/g0;

printf("%d\t\t%f\t\t%f\t\t%f\n",step,x0,f0,x1,f1);

x0 = x1;

step = step+1;

if(step > N)

{

printf ("Not Convergent.");

exit (0);

}

f1 = f(x1);

}while(fabs(f1)>e);

printf("\nRoot is: %f", x1);

getch();

}

```

Output: Newton Raphson Method Using C

Enter initial guess:

1

Enter tolerable error:

0.00001

Enter

10

maximum iteration:

Step x_0 $f(x_0)$ x_1 $f(x_1)$

1 1.000000 1.459698 0.620016 0.000000

2 0.620016 0.046179 0.607121 0.046179

3 0.607121 0.000068 0.607102 0.000068

Root is: 0.607102

EXPERIMENT – 3

DEFLECTION OF BEAMS

Aim: To determine the deflection of beam subjected to different load

Software: C language

```
/*program for determination of deflection of beams*/  
  
#include<stdio.h>  
  
#include<math.h>  
  
int main()  
{  
  
float w, l, e, y, x, t, i, b, d;  
  
/*w=u.d.l ,l-span, e=modulus of elasticity*/  
  
/*i=moment of inertia, y=deflection, b= width, d=depth*/  
  
printf("select the type of beam x= 1 for ssb and 2 for cantilever:.\n");  
scanf("%d",&x);  
  
printf("select the type of load as t= 3 for point load and 4 for udl:.\n");  
scanf("%d",&t)  
  
printf("enter the value of w :\n");  
scanf("%f", &w);  
  
printf("enter the value of l in m:\n");  
scanf("%f", &l);  
  
printf("enter the value of e in N/m*m:\n");  
scanf("%f", &e);  
  
printf("enter the value of b  
in m:\n");  
scanf("%f", &b);
```

```

printf("enter the value of d in m :\n");

scanf("%f", &d);

i=(b*pow(d,3)/12);

/*calculation of deflection of beam*/

if(x==1&&t==3)

{

y=w*pow(l,3)/(48*e*i);

}

else if(x==1&&t==4)

{

y=5/384*w*pow(l,4)/(e*i);

}

else if(x==2&&t==3)

{

y=w*pow(l,3)/(3*e*i);

}

else

{

y=w*pow(l,4)/(8*e*i);

}

printf("\n the deflection of the beam is=%f m",y);

}

```

OUTPUT

Enter the value of W(kN) : 10

Enter the value of l (m): 2

Enter the value of E (N/mm): 2

Enter the value of b(m): 3

Enter the value of $d(m)$: 5

The deflection of the beam = 0.03333

STAAD INTRODUCTION

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

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

Advantages:

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

STAAD Pro. Stands for Structural Analysis And Design Program All structural analysis software generally consists of three parts:

- **Pre Processing:** Generates the model, assembles and organizes all data needed for the analysis.
- **Processing:** Calculates displacements, member forces, reactions, stresses, etc.
- **Post Processing:** Displays the results.

STAAD.Pro Workflow Process: The process of modeling and designing in STAAD.Pro can be summarized into the following general workflow process, which is suggested inherently by the on-screen organization of the tabs within the program:

1. **Basic Geometry:** Define the basic geometry of the structure using beams, columns, plates and/or solid elements.
2. **Section Properties:** Define the sizes of members by width, depth, cross sectional shape, etc.
3. **Materials Constants:** Specify material such as timber, steel, concrete, or aluminum to define Poisson's Ratio, Coefficient of Thermal Expansion, density, etc.
4. **Member Specifications:** Define member orientations, member offsets, member releases where moment transfer is to be limited or eliminated, and conditions that only allow a partial transfer of certain types of forces such as tension-only.
5. **Supports:** Define support locations and boundary conditions including moment fixity, support stiffness, and support angle.
6. **Loads:** Assign loads such as self-weight, dead, live, wind and seismic, and define load combinations.
7. **Analysis Instructions:** Indicate the type of analysis to be performed (regular analysis, P-delta, Buckling, Pushover, etc.) and define associated options.
8. **Post Processing Commands:** Extract analysis results, review deflected shapes, prepare shear and moment diagrams, generate tables to present results, etc.
9. **Design Commands:** Specify (for steel, concrete, timber, etc.)



Fig. STAAD Interface

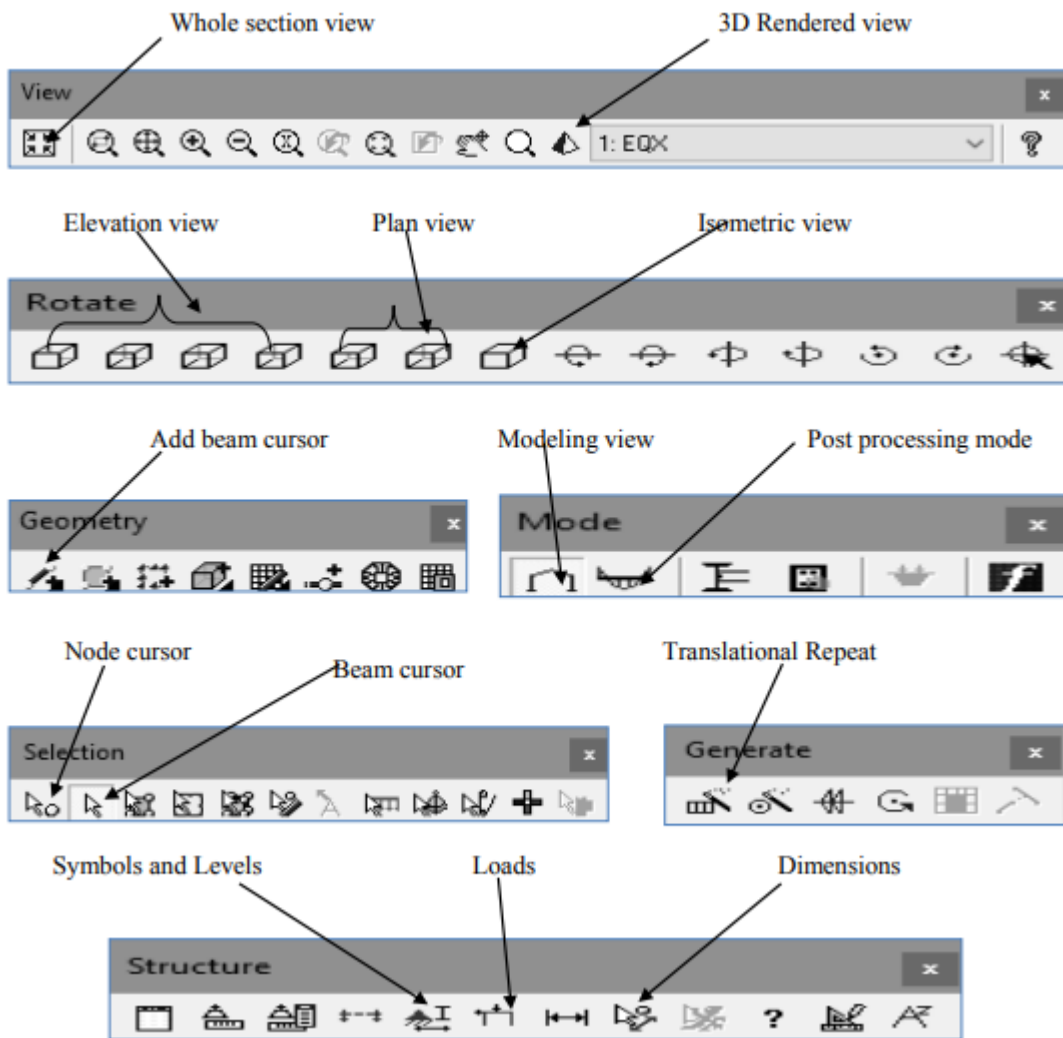


Fig. STAAD Pro. Toolbars and icon views

STAAD Pro. key features:

(A) = Title bar

(C) = Toolbars

(E) = Page Control

(G) = Data area

(B) = Menu bar

(D) = Mode bar

(F) = View window

(H) = Status bar

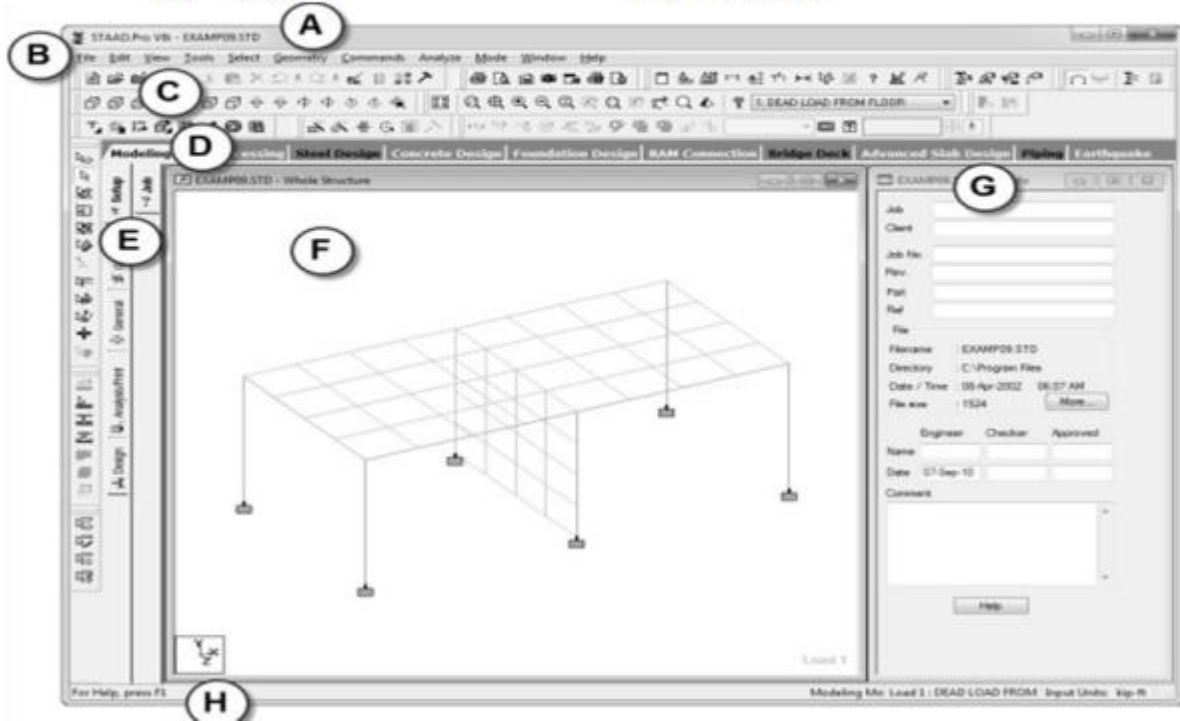
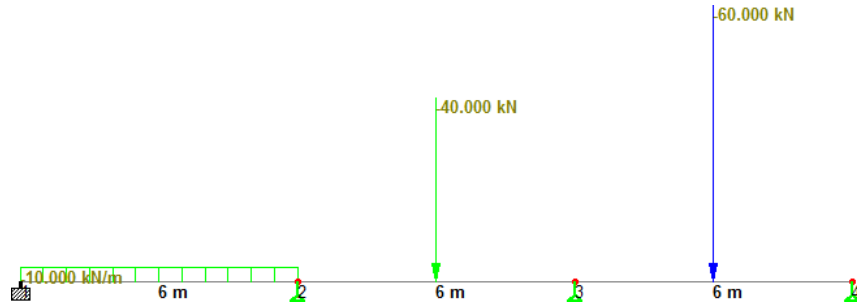


Fig. STAAD Pro. Key features

EXPERIMENT NO. 2

ANALYSIS OF CONTINUOUS BEAM

Aim: To analyze the continuous beam using staad.pro



Software: STAAD Pro.

Procedure: Following are the step wise procedures for Staad Pro Problems (CONTINUOUS BEAM)

1. Double click the STAAD Pro Icon on Desk top
2. Select Plane , Meter, Kilo Newton and write File Name & click Next
3. Add beam , Finish
4. Click on XY plane and cut Screen (Snap node/beam) screen by clicking at right top corner
5. Enter Coordinates of each joint of above beam Close 'Node Screen ' , Close 'Structure screen '
Expand working screen
0 0 0
6 0 0
12 0 0
18 0 0
6. Connect all joints by beam element , join nodes by nodes individually Now the screen should appear like a 3 span beam with nodes clearly seen at the end of each span This step is very important.
7. Now select Commands. Under this select Member Property\Prismatic and click Rectangle.
Enter following details
YD = 0.30
ZD = 0.30

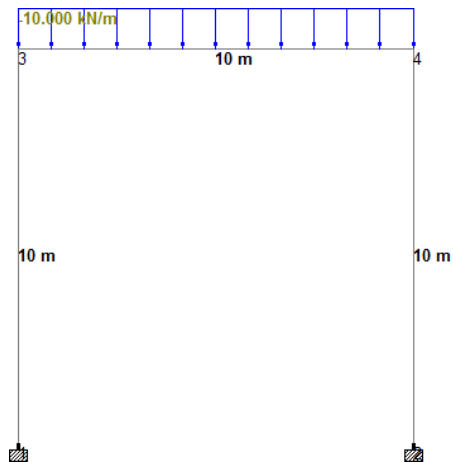
8. Assign supports to the joints which may be fixed, pinned or roller (Fixed But where only restrain should be along FY other to be released as only on reaction is possible for roller support in Y- direction)
9. Select Commands\Loading\Primary Load Number 1 , Loading type 'DEAD' , Title 'Load Case 1' , Add , Close
10. Now click on ' Load case 1 ' , New , Load Items , Member load , Uniform Load , W1 , - 10 knm Add Then Concentrated force P -10 d1 3 Add , then P -60 d1 2.
11. Add close Now click on UNI GY -10 kN/m 'use cursor to Assign, Click on first Span.
12. Similarly assign 40kN and 60kN on remaining two spans and then click close To verify whether all loads are assigned correctly or not Click on Load details menu on menu bar And click load view, now load diagram with values should be seen on main diagram.
13. Now Commands\Analysis\Perform Analysis, select all, ok.
14. Now Click Analysis\Run analysis Click Staad Analysis After a time, display will show number of errors & warning messages if anythen
15. Now Staad pro has executed analysis of continuous beam and the results of the analysis are to be seen on Post processing mode. Click 'Go to Post Processing mode ' Similarly 'Done ' 'Apply ' 'OK' The post processing mode window will appear as shown in the fig. Below.
16. To see the bending moment diagram of the analysed beam click on the beamtab at left side of the working window. Click on Mzbutton on the menu bar at top of the working window. You shouldsee the BMD as shown in the figure below(Holding down the ctrl button scroll the mouse wheel to adjust the scale to your convenience). Go to reactions menu>>view value >>beam results and check the boxes ends, maximum and midpoint under bending to see the value of BMD at respective locations.
17. To see the shear force diagram click on the beamtab at left side of the working window. Click on Fybutton on the menu bar at top of the working window. You should see the SFD as shown in the figure below (Holding down the ctrl button scroll the mouse wheel to adjust the scale to your convenience). Go to reactions menu>>view value >>beam results and check the boxes ends, maximum and midpoint under shear to see the value of SFD at respective l ocations.
18. To see the displacement of the beam click on the nodes>displacement tab in the menu on left side of the working area. You should see the diagram as shown in the figure below(Holding down the ctrl button scroll the mouse wheel to adjust the scale to your convenience). To view the displacement value you can go to reactions menu>>view value >>beam results and check the box Max resultant under displacement.
19. To see the reactions at the support of the beam go to nodes>>reactions tab on left side of the working area.

Result: The analysis of continuous beam is done by using STAAD Pro.

EXPERIMENT NO. 2

ANALYSIS OF SINGLE STOREY FRAME

Aim: To analyze the single storey frame using staad.pro



Software: STAAD Pro.

Procedure: Following are the step wise procedures for Staad Pro Problems (SINGLE STOREY FRAME)

1. Double click the STAAD Pro Icon on Desk top
2. Select Plane , Meter, Kilo Newton and write File Name & click Next
3. Add beam , Finish
4. Click on XY plane and cut Screen (Snap node/beam) screen by clicking at right top corner
5. Enter Coordinates of each joint of above beam Close 'Node Screen ' , Close 'Structure screen '
Expand working screen
0 0 0
10 0 0
0 10 0
10 10 0
6. Connect all joints by beam element , join nodes by nodes individually Now the screen should appear like a single storey frame with nodes clearly seen at the end of each span This step is very important.
7. Now select Commands. Under this select Member Property\Prismatic and click Rectangle.
Enter following details

YD = 0.30

ZD= 0.30

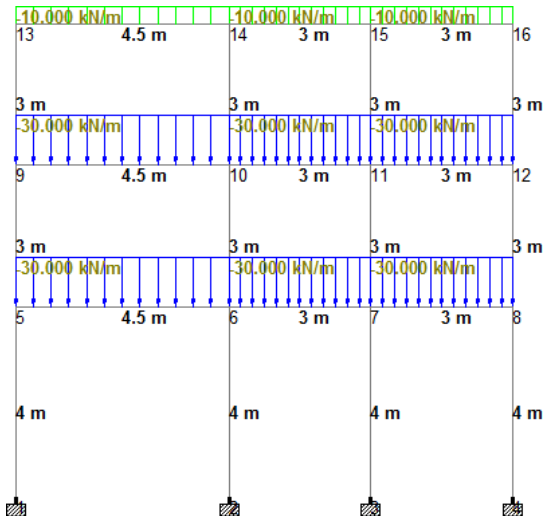
8. Assign supports to the joints which may be fixed, pinned or roller (Fixed But where only restrain should be along FY other to be released as only on reaction is possible for roller support in Y- direction)
9. Select Commands\Loading\Primary Load Number 1 , Loading type 'DEAD' , Title 'Load Case 1' , Add , Close
10. Now click on ' Load case 1 ' , New , Load Items , Member load , Uniform Load , W1 , - 10 kN/m Add close Now click on UNI GY -10 kN/m 'use cursor to Assign, Click on first Span.
11. Now Commands\Analysis\Perform Analysis, select all, ok.
12. Now Click Analysis\Run analysis Click STAAD Analysis After a time, display will show number of errors & warning messages if anythen
13. Now Staad pro has executed analysis of continuous beam and the results of the analysis are to be seen on Post processing mode. Click 'Go to Post Processing mode ' Similarly 'Done ' 'Apply ' 'OK' The post processing mode window will appear as shown in the fig. Below.
14. To see the bending moment diagram of the analysed beam click on the beam tab at left side of the working window. Click on Mz button on the menu bar at top of the working window. You should see the BMD as shown in the figure below (Holding down the ctrl button scroll the mouse wheel to adjust the scale to your convenience). Go to reactions menu>>view value >>beam results and check the boxes ends, maximum and midpoint under bending to see the value of BMD at respective locations.
15. To see the shear force diagram click on the beam tab at left side of the working window. Click on Fy button on the menu bar at top of the working window. You should see the SFD as shown in the figure below (Holding down the ctrl button scroll the mouse wheel to adjust the scale to your convenience). Go to reactions menu>>view value >>beam results and check the boxes ends, maximum and midpoint under shear to see the value of SFD at respective locations.
16. To see the displacement of the beam click on the nodes>displacement tab in the menu on left side of the working area. You should see the diagram as shown in the figure below(Holding down the ctrl button scroll the mouse wheel to adjust the scale to your convenience). To view the displacement value you can go to reactions menu>>view value >>beam results and check the box Max resultant under displacement.
17. To see the reactions at the support of the beam go to nodes>>reactions tab on left side of the working area.

Result: The analysis of single storey frame is done by using STAAD Pro.

EXPERIMENT NO. 3

ANALYSIS OF MULTI STOREY FRAME

Aim: To analyze the multi storey frame using staad.pro



Software: STAAD Pro.

Procedure: Following are the step wise procedures for Staad Pro Problems (MULTI STOREY FRAME)

1. Double click the STAAD Pro Icon on Desk top
2. Select Plane , Meter, Kilo Newton and write File Name & click Next
3. Add beam , Finish
4. Click on XY plane and cut Screen (Snap node/beam) screen by clicking at right top corner
5. Enter Coordinates of each joint of above beam Close 'Node Screen ' , Close 'Structure screen ' ,
6. Connect all joints by beam element , join nodes by nodes individually Now the screen should appear like a single storey frame with nodes clearly seen at the end of each span This step is very important.
7. Now select Commands. Under this select Member Property\Prismatic and click
8. Assign supports to the joints which may be fixed, pinned or roller (Fixed But where only restrain should be along FY other to be released as only on reaction is possible for roller support in Y- direction)
9. Select Commands\Loading\Primary Load Number 1 , Loading type 'DEAD' , Title 'Load Case 1' , Add , Close. More loads may be added if required.
10. Now click on ' Load case 1 ' , New , Load Items , Member load 'use cursor to Assign,

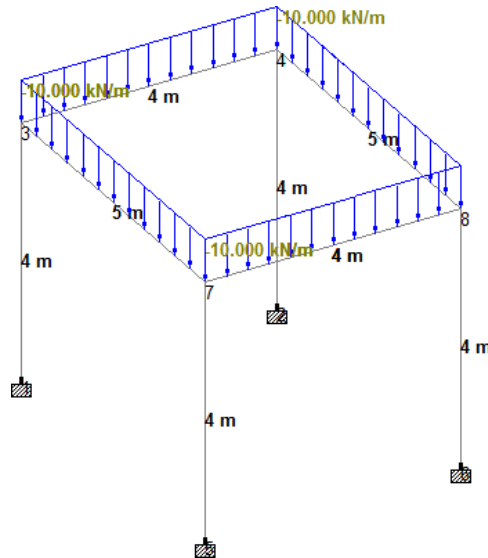
11. Now Commands\Analysis\Perform Analysis, select all, ok.
12. Now Click Analysis\Run analysis Click STAAD Analysis After a time, display will show number of errors & warning messages if anythen
13. Now Staad pro has executed analysis of continuous beam and the results of the analysis are to be seen on Post processing mode. Click 'Go to Post Processing mode ' Similarly 'Done ' 'Apply ' 'OK' The post processing mode window will appear as shown in the fig. Below.
14. To see the bending moment diagram of the analysed beam click on the beam tab at left side of the working window. Click on Mz button on the menu bar at top of the working window. You should see the BMD as shown in the figure below (Holding down the ctrl button scroll the mouse wheel to adjust the scale to your convenience). Go to reactions menu>>view value >>beam results and check the boxes ends, maximum and midpoint under bending to see the value of BMD at respective locations.
15. To see the shear force diagram click on the beam tab at left side of the working window. Click on Fy button on the menu bar at top of the working window. You should see the SFD as shown in the figure below (Holding down the ctrl button scroll the mouse wheel to adjust the scale to your convenience). Go to reactions menu>>view value >>beam results and check the boxes ends, maximum and midpoint under shear to see the value of SFD at respective locations.
16. To see the displacement of the beam click on the nodes>displacement tab in the menu on left side of the working area. You should see the diagram as shown in the figure below(Holding down the ctrl button scroll the mouse wheel to adjust the scale to your convenience). To view the displacement value you can go to reactions menu>>view value >>beam results and check the box Max resultant under displacement.
17. To see the reactions at the support of the beam go to nodes>>reactions tab on left side of the working area.

Result: The analysis of multi storey frame is done by using STAAD Pro.

EXPERIMENT NO. 4

ANALYSIS OF 3D - FRAME

Aim: To analyze the 3D Frame using staad.pro



Software: STAAD Pro.

Procedure: Following are the step wise procedures for Staad Pro Problems (MULTI STOREY FRAME)

1. Double click the STAAD Pro Icon on Desk top
2. Select Plane , Meter, Kilo Newton and write File Name & click Next
3. Add beam , Finish
4. Click on XY plane and cut Screen (Snap node/beam) screen by clicking at right top corner
5. Enter Coordinates of each joint of above beam Close 'Node Screen ' , Close 'Structure screen ' ,
6. Connect all joints by beam element , join nodes by nodes individually Now the screen should appear like a single storey frame with nodes clearly seen at the end of each span This step is very important.
7. Now select Commands. Under this select Member Property\Prismatic and click
8. Assign supports to the joints which may be fixed, pinned or roller (Fixed But where only restrain should be along FY other to be released as only on reaction is possible for roller support in Y- direction)
9. Select Commands\Loading\Primary Load Number 1 , Loading type 'DEAD' , Title 'Load Case 1' , Add , Close. More loads may be added if required.

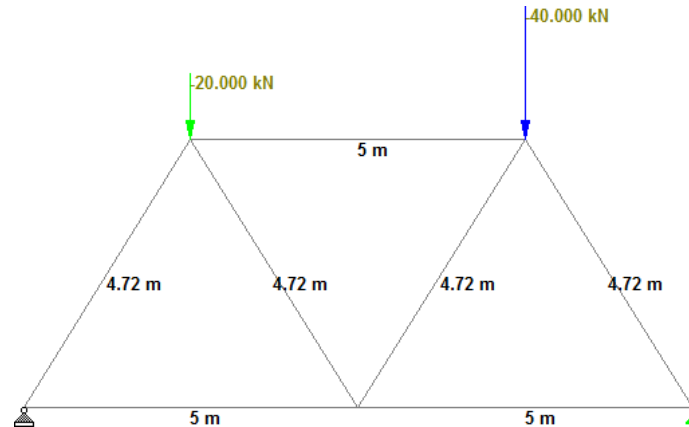
10. Now click on “Load case 1”, Assign Self weight as “-1” to the entire structure also assign Dead Load of Wall to the beams
Under Load Case 2 Create Live Load and specify the range of Floor Load
Under Load Case 3 Define Load Combination of 1.5(DL+LL)
11. Now Commands\Analysis\Perform Analysis, select all, ok.
12. Now Click Analysis\Run analysis Click STAAD Analysis After a time, display will show number of errors & warning messages if any then
13. Now Staad pro has executed analysis of continuous beam and the results of the analysis are to be seen on Post processing mode. Click ‘Go to Post Processing mode ‘ Similarly ‘Done ‘ ‘Apply ‘ ‘OK’ The post processing mode window will appear as shown in the fig. Below.
14. To see the bending moment diagram of the analysed beam click on the beam tab at left side of the working window. Click on Mz button on the menu bar at top of the working window. You should see the BMD as shown in the figure below (Holding down the ctrl button scroll the mouse wheel to adjust the scale to your convenience). Go to reactions menu>>view value >>beam results and check the boxes ends, maximum and midpoint under bending to see the value of BMD at respective locations.
15. To see the shear force diagram click on the beam tab at left side of the working window. Click on Fy button on the menu bar at top of the working window. You should see the SFD as shown in the figure below (Holding down the ctrl button scroll the mouse wheel to adjust the scale to your convenience). Go to reactions menu>>view value >>beam results and check the boxes ends, maximum and midpoint under shear to see the value of SFD at respective locations.
16. To see the displacement of the beam click on the nodes>displacement tab in the menu on left side of the working area. You should see the diagram as shown in the figure below(Holding down the ctrl button scroll the mouse wheel to adjust the scale to your convenience). To view the displacement value you can go to reactions menu>>view value >>beam results and check the box Max resultant under displacement.
17. To see the reactions at the support of the beam go to nodes>>reactions tab on left side of the working area.

Result: The analysis of 3d frame is done by using STAAD Pro.

EXPERIMENT NO. 5

ANALYSIS OF TRUSS

Aim: To analyze the pratt and warren truss using staad.pro



Software: STAAD Pro.

Procedure: Following are the step wise procedures for Staad Pro Problems (MULTI STOREY FRAME)

1. Double click the STAAD Pro Icon on Desk top
2. Select Plane , Meter, Kilo Newton and write File Name & click Next
3. Add beam , Finish
4. Click on XY plane and cut Screen (Snap node/beam) screen by clicking at right top corner
5. Enter Coordinates of each joint of above beam Close 'Node Screen ' , Close 'Structure screen ' ,
6. Connect all joints by beam element , join nodes by nodes individually Now the screen should appear like a single storey frame with nodes clearly seen at the end of each span This step is very important.
7. Now select Commands. Under this select Member Property\Prismatic and click
8. Assign supports to the joints which may be fixed, pinned or roller (Fixed But where only restrain should be along FY other to be released as only on reaction is possible for roller support in Y- direction)
9. Select Commands\Loading\Primary Load Number 1 , Loading type 'DEAD' , Title 'Load Case 1' , Add , Close. More loads may be added if required. Assign the nodal load the points.
10. Now Commands\Analysis\Perform Analysis, select all, ok.

11. Now Click Analysis\Run analysis Click STAAD Analysis After a time, display will show number of errors & warning messages if any then
12. Now Staad pro has executed analysis of continuous beam and the results of the analysis are to be seen on Post processing mode. Click 'Go to Post Processing mode ' Similarly 'Done ' 'Apply ' 'OK' The post processing mode window will appear as shown in the fig. Below.
13. To see the axial force diagram click on the beam tab at left side of the working window. Click on Fx button on the menu bar at top of the working window. You should see the SFD as shown in the figure below (Holding down the ctrl button scroll the mouse wheel to adjust the scale to your convenience). Go to reactions menu>>view value >>beam results and check the box maximum .
14. To see the displacement of the beam click on the nodes>displacement tab in the menu on left side of the working area. You should see the diagram as shown in the figure below(Holding down the ctrl button scroll the mouse wheel to adjust the scale to your convenience). To view the displacement value you can go to reactions menu>>view value >>beam results and check the box Max resultant under displacement.
15. To see the reactions at the support of the beam go to nodes>>reactions tab on left side of the working area.

Result: The analysis of truss is done by using STAAD Pro.

INTRODUCTION TO SPREADSHEET (EXCEL)

A spreadsheet is a computer application that is designed to add, display, analyze, organize, and manipulate data arranged in rows and columns. It is the most popular application for accounting, analytics, data presentation, etc. Or in other words, spreadsheets are scalable grid-based files that are used to organize data and perform calculations. It can also use the tool's features and formulas to help you make sense of your data. For example, track data in a spreadsheet and see sums, differences, multiplication, division, and fill dates automatically, among other things. Microsoft Excel, Google sheets, Apache open office, LibreOffice, etc are some spreadsheet software. Among all these software, Microsoft Excel is the most commonly used spreadsheet tool and it is available for Windows, macOS, Android, etc.

A collection of spreadsheets is known as a workbook. Every Excel file is called a workbook. Every time when you start a new project in Excel, you'll need to create a new workbook. There are several methods for getting started with an Excel workbook. To create a new worksheet or access an existing one, you can either start from scratch or utilize a pre-designed template.

A single Excel worksheet is a tabular spreadsheet that consists of a matrix of rectangular cells grouped in rows and columns. It has a total of 1,048,576 rows and 16,384 columns, resulting in 17,179,869,184 cells on a single page of a Microsoft Excel spreadsheet where you may write, modify, and manage your data.

In the same way as a file or a book is made up of one or more worksheets that contain various types of related data, an Excel workbook is made up of one or more worksheets. You can also create and save an endless number of worksheets. The major purpose is to collect all relevant data in one place, but in many categories (worksheet).

FEATURE OF SPREADSHEET

As we know that there are so many spreadsheet applications available in the market. So these applications provide the following basic features:

1. Rows and columns:

Rows and columns are two distinct features in a spreadsheet that come together to make a cell, a range, or a table. In general, columns are the vertical portion of an excel worksheet, and there can be 256 of them in a worksheet, whereas rows are the horizontal portion, and there can be 1048576 of them.

2. Formulas:

In spreadsheets, formulas process data automatically. It takes data from the specified area of the spreadsheet as input then processes that data, and then displays the output into the new area of the spreadsheet according to where the formula is written. In Excel, we can use formulas simply by typing “=Formula Name(Arguments)” to use predefined Excel formulas. When you write the first few characters of any formula, Excel displays a drop-down menu of formulas that match that character sequence. Some of the commonly used formulas are:

- =SUM(Arg1: Arg2): It is used to find the sum of all the numeric data specified in the given range of numbers.
- =COUNT(Arg1: Arg2): It is used to count all the number of cells(it will count only number) specified in the given range of numbers.
- =MAX(Arg1: Arg2): It is used to find the maximum number from the given range of numbers.
- =MIN(Arg1: Arg2): It is used to find the minimum number from the given range of numbers.
- =TODAY(): It is used to find today’s date.
- =SQRT(Arg1): It is used to find the square root of the specified cell.

3. Function: In spreadsheets, the function uses a specified formula on the input and generates output. Or in other words, functions are created to perform complicated math problems in spreadsheets without using actual formulas. For example, you want to find the total of the numeric data present in the column then use the SUM function instead of adding all the values present in the column.

4. Text Manipulation: The spreadsheet provides various types of commands to manipulate the data present in it.

5. Pivot Tables: It is the most commonly used feature of the spreadsheet. Using this table users can organize, group, total, or sort data using the toolbar. Or in other words, pivot tables are used to summarize lots of data. It converts tons of data into a few rows and columns.

USE OF SPREADSHEETS

The use of Spreadsheets is endless. It is generally used with anything that contains numbers. Some of the common use of spreadsheets are:

- **Finance:** Spreadsheets are used for financial data like it is used for checking account information, taxes, transaction, billing, budgets, etc.
- **Forms:** Spreadsheet is used to create form templates to manage performance review, timesheets, surveys, etc.
- **School and colleges:** Spreadsheets are most commonly used in schools and colleges to manage student’s data like their attendance, grades, etc.
- **Lists:** Spreadsheets are also used to create lists like grocery lists, to-do lists, contact detail, etc.
- **Hotels:** Spreadsheets are also used in hotels to manage the data of their customers like their personal information, room numbers, check-in date, check-out date, etc.

COMPONENTS OF SPREADSHEETS

The basic components of spreadsheets are:

1. TitleBar:

The title bar displays the name of the spreadsheet and application.

2. Toolbar:

It displays all the options or commands available in Excel for use.

3. NameBox:

It displays the address of the current or active cell.

4. Formula Bar:

It is used to display the data entered by us in the active cell. Also, this bar is used to apply formulas to the data of the spreadsheet.

5. Column Headings:

Every excel spreadsheet contains 256 columns and each column present in the spreadsheet is named by letters or a combination of letters.

6. Row Headings:

Every excel spreadsheet contains 65,536 rows and each row present in the spreadsheet is named by a number.

7. Cell:

In a spreadsheet, everything like a numeric value, functions, expressions, etc., is recorded in the cell. Or we can say that an intersection of rows and columns is known as a cell. Every cell has its own name or address according to its column and rows and when the cursor is present on the first cell then that cell is known as an active cell.

8. Cell referring:

A cell reference, also known as a cell address, is a way for describing a cell on a worksheet that combines a column letter and a row number. We can refer to any cell on the worksheet using cell references (in excel formulae). As shown in the above image the cell in column A and row 1 is referred to as A1. Such notations can be used in any formula or to duplicate the value of one cell to another (by using = A1).

9. Navigation buttons:

A spreadsheet contains first, previous, next, and last navigation buttons. These buttons are used to move from one worksheet to another workbook.

10. Sheet tabs:

As we know that a workbook is a collection of worksheets. So this tab contains all the worksheets present in the workbook, by default it contains three worksheets but you can add more according to your requirement.

EXPERIMENT 6

DESIGN OF TWO WAY SLAB

Aim: To design a two slab using MS Excel

Software Used : Microsoft EXCEL

Theory :

When a reinforced concrete slab is supported by beams on all the four sides and the loads are carried by the supports along with both directions, it is known as two way slab. In two way slab, the ratio of longer span (l) to shorter span (b) is less than 2. i.e Longer span (l)/Shorter span (b) < 2.

These types of slabs are mostly used on the floor of multi-storey buildings. In two way slab, as the loads are carried in both directions (longer and shorter direction) main reinforcement bars are provided in both directions. The loads are carried by two sides are equal. Slab with reinforcement in both directions is more economical than one way slabs.

Procedure:

1. Open MS EXCEL and Prepare a tabular format
2. Thickness of the slab is to be calculated and load calculations are to be done for finding out moments
Dead Load = thickness of slab * 25
Total factored Load = 1.5 * (Dead Load + Live Load + Floor Finish)
3. Calculation of moment is based on bending moment coefficients (α_x -ve, α_y +ve α_x -ve α_y +ve)
 $M_x = w_u * \alpha_x * L_x^2$
 $M_y = w_u * \alpha_y * L_y^2$
4. Check for depth and area of reinforcement along x and y direction
5. Check for shear

Few Formulae in Excel

$$M_x = w_u * \alpha_x * L_x^2$$

$$M_y = w_u * \alpha_y * L_y^2$$

$$d = \text{ROUNDUP}(\text{SQRT}((M_u * 10^6) / (0.138 * f_{ck} * b)), 1)$$

$$A_{st} = (0.5 * (f_{ck} / f_y) * b * d) * (1 - (\text{SQRT}(1 - ((4.6 * M_u * 10^6) / (f_{ck} * b * d^2))))))$$
$$A_{stmin} = ((0.12 * b * D) / 100)$$

$$\text{IF}(A_{st} < A_{stmin}, A_{stmin}, A_{st})$$

$$\text{Spacing} = ((\text{PI}() * 0.25 * \phi^2 * 1000) / A_{st})$$

Result: Print out of Excel Sheet to be attached

EXPERIMENT 7

DESIGN OF FLAT SLAB

Aim: To design a flat slab using MS Excel

Software Used : Microsoft EXCEL

Theory :

A flat slab is a two-way reinforced concrete slab that usually does not have beams and girders, and the loads are transferred directly to the supporting concrete columns. The flat plate is a two-way reinforced concrete framing system utilizing a slab of uniform thickness, the simplest of structural shapes. The flat slab is a two-way reinforced structural system that includes either drop panels or column capitals at columns to resist heavier loads and thus permit longer spans.

1. Advantages of Flat Slab

1. Flexibility in room layout
 - Partition walls can be placed anywhere.
 - Offers a variety of room layouts to the owner.
 - False ceilings can be omitted.
2. Reinforcement placement is easier
 - As reinforcement detailing of the flat slab is simple, it is easier to place
3. Ease of Framework installation.
 - Big table framework can be used in flat slab
4. Building height can be reduced.
 - Floor height can be reduced and consequently the building height will be reduced.
 - Approximately 10% of the vertical member could be saved.
 - Foundation load will also reduce.
5. Less construction time
 - Use of a big table framework helps to reduce construction time

Uses of column heads (capital):

- Shear strength of flat slab is increased by using column heads.
- Column heads reduce the clear or effective span, and therefore, reduce the moment in the flat slab floor.

Uses of drop panels:

- Drop panels increase the shear strength of flat slab floor.
- Drop panels increase flat slab's negative moment capacity.
- Drop panels reduce deflection by stiffening the flat slabs.

Procedure:

1. Open MS EXCEL and Prepare a tabular format
2. Thickness of the slab is to be calculated and load calculations are to be done for finding out moments
Dead Load = thickness of slab * 25
Total factored Load = 1.5 * (Dead Load + Live Load + Floor Finish)
3. Calculation of moment as per IS codal provisions (Direct Design method)
4. Check for depth.
5. Design Moment, Negative Design Moment, Positive Design Moment, Column Strip and Middle Strip moments to be calculated
6. Calculation of Area of steel along column strip, middle strip (Longer and shorter span)
7. Check for punching shear

Formulae in Excel

Design Moment

$$M_o = (W * L_n) / 8 * L_1 =$$

$$\text{MAX}(L_x : L_y)$$

$$L_n = 0.65 * L_1$$

$$L_2 = \text{MIN}(L_x : L_y)$$

d from deflection (mm) =

$$\text{ROUNDUP}(((L_x * 1000) / 32) - 30, 0) \text{ d prov} =$$

$$\text{IF}(d \text{ from deflection} < 95, 95, d \text{ from deflection})$$

$$\text{Dead Load (kN/m}^2) = ((d \text{ prov} + 30) / 1000) * 25$$

$$\text{Width of Column Strip} = 0.5 * L_1 * 1000$$

$$\text{Width of Middle Strip} = (L_2 * 1000) - (\text{Width of Column Strip})$$

$$\text{Design Moment} = (\text{Factored Load} * L_2 * L_n$$

$$* L_n) / 8$$

Negative Design Moment = 0.65 * Design Moment

$$\text{Positive Design Moment} = 0.35 * \text{Design Moment}$$

$$\text{Negative Col Strip Moment} = 0.75 * \text{Negative Design Moment}$$

$$\text{Negative Middle Strip Moment} = 0.25 * \text{Negative Design Moment}$$

$$\text{Moment Positive Col Strip Moment} = 0.60 * \text{Positive Design Moment}$$

$$\text{Moment Positive Middle Strip Moment} = 0.40 * \text{Positive Design Moment}$$

$$d = \text{ROUNDUP}(\text{SQRT}((M_u * 10^6) / (0.138 * f_{ck} * b)), 1)$$

$$A_{st} = (0.5 * (f_{ck} / f_y) * b * d) * (1 - (\text{SQRT}(1 - ((4.6 * M_u * 10^6) / (f_{ck} * b * d^2))))))$$

$$A_{st \text{ min}} =$$

$$((0.12 * b * D) / 100)$$

$$\text{IF}(A_{st} < A_{st \text{ min}},$$

$$A_{st \text{ min}}, A_{st})$$

$$\text{Spacing} = ((\text{PI}) * 0.25 * \phi^2 * 1000) / A_{st}$$

Result: Print out of Excel Sheet to be attached