aadspro
Structural Engineering Software

Analysis, autoscan design, detailing, scheduling and sequencing program or aadspro is a new generation design package which can be used for a number of applications in Civil Engineering. It is integrated with commonly used programs like STAAD.Pro and AutoCAD and ETABS.

The program consists of different modules which can perform the design of different parts of a structure. With this version we are introducing the reinforced concrete design of Doubly reinforced Beam, Slab, Column, Footing, T beam. All of the designs are as per limit state.

aadspro also has the option for designing all beams and slabs in a floor. By this the user gets automatic detailing of reinforcement.
This program is user friendly both for data and graphic input. Results can be viewed not only in Tabular form but also in a drawing format in AutoCAD. aadspro also gives option for saving to or importing from Excel.

Another important point in aadspro is that it thoroughly checks the code provisions like, minimum concrete cover, etc. and give message alerts if the values are exceeding the permissible limits as per Indian British and American codes.

CONTENTS

1. Creating Data base

aadspro using the output of STAAD Pro and ETABS as its input. In aadspro module such as Beam detailing ,Column detailing and Footing detailing, the output database of STAAD Pro and ETABS are used and in column detailing the STAAD Pro output file (anl file) is also used as input.

There are different procedure in creation of the database in STAAD Pro and ETABS.

Database Creation in STAAD Pro

User can create the Database file by selecting the menu Tools>SQL Query>Advanced Query in STAAD Pro. A database file (.mdb) is created with same name of the model is created in the same path of the model file. (See Fig: 1.1)

Creating STAAD Pro output file (anl) file

The anl file is automatically created in the same path of the model file after the analysis is performed. See STAAD Pro Manual for more information.

Database Creation in ETABS

Use File menu > Export > Save Input/output As Access Database File command to select the input/output data to be exported to a Microsoft Access database format.

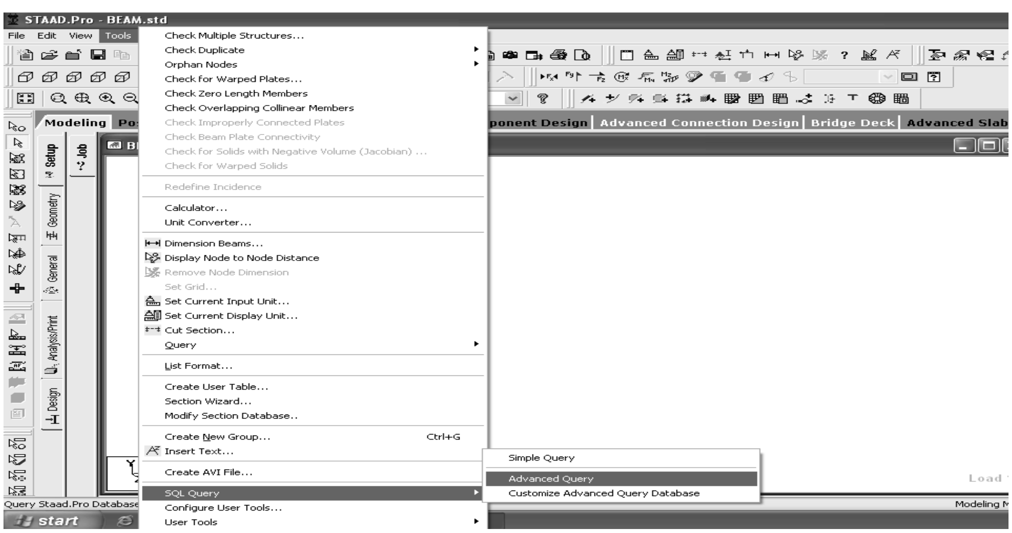


Fig 1.1

2. Settings

Click on the 'Tools' from main menu and select 'settings' from sub menu. (Fig 2.1)

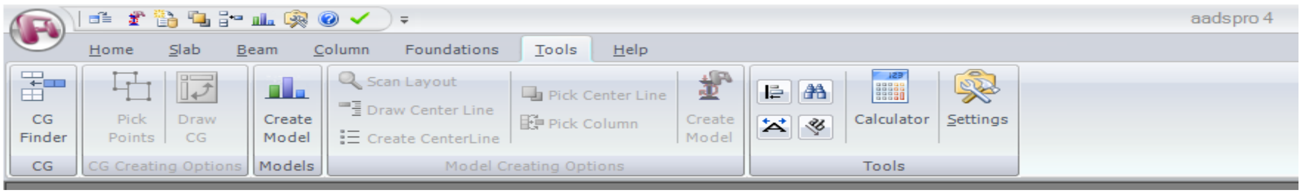
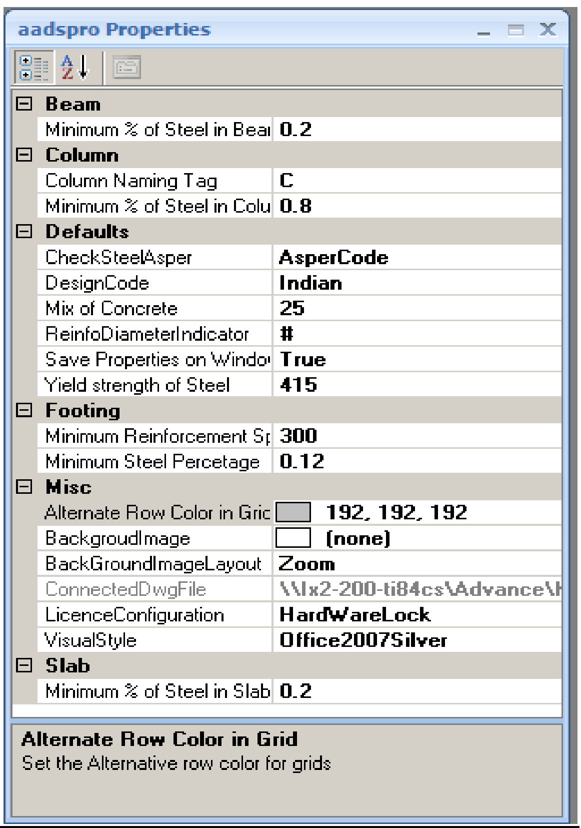


Fig 2.1

User can set the properties



- Background Image
  By this property user can change the background image aadspro.

- Background Image Layout
  User can change the layout style of the current Background Image.

- Column Naming Tag
  Column Naming Tag to be used in detailing Process.

- Check Steel As per
  This property help the user to set the default steel percentage used in design process. In this user can use either as per Code option or User Defined option .The As per Code option will check the steel percentage as per the code provision given in standard code selected by the user. By User Defined option user can set the percentage manually. User can change the manual steel percentage setting for Beam, Slab, Column, Footing in their respective Minimum % of steel Properties.

- Design Code
  This property set the standard design code used in different design process. Aadspro gives provision to use Indian, British and American codes.

- Fck Value
  Set Current the mix of concrete used in design module of structural elements.

- Reinfo Dia Indicator
  By using this property user can customize the Diameter indication tag used in Detailing process.

- Connected Dwg File
  User can see which AutoCAD file is currently connected to aadspro.

- Alternate grid row Color
  Set the color of the alternate row of the grid used in aadspro.

- Visual Style
  User can change the Visual Appearance of aadspro by changing the three option given (Office2007Black, Office2007Blue, Office2007Silver)

- License Configuration
  Sets how the software license is configured, aadspro uses three option in license configuration

1. Hard Ware Lock - License configured through a hardware lock supplied with aadspro software.
2. License File -License configured through a License file.
3. Web License - License through web. User can login to aadspro with a aadspro account. User account can be created from www.aadspro.com

3. MODEL CREATOR

Model Creator is powerful tool for creating Staad Pro or ETABS model directly from AutoCAD. It minimizes the time and design procedure in model creation.It automatically detects the beam and column from AutoCAD and makes the model within minutes.

3.1 Center line scanning

This chapter provides a step-by-step procedure for the Centerline scanning of slabs.
A layout of the structure is provided in AutoCAD as shown in the fig 3.1

- 'Fillet' the beam layout.
- All the beam lines & columns in the layout should be in a particular layer.
- All columns should be in polyline.
- All the beam lines should break inside at the column.
- Layout should be in cm.

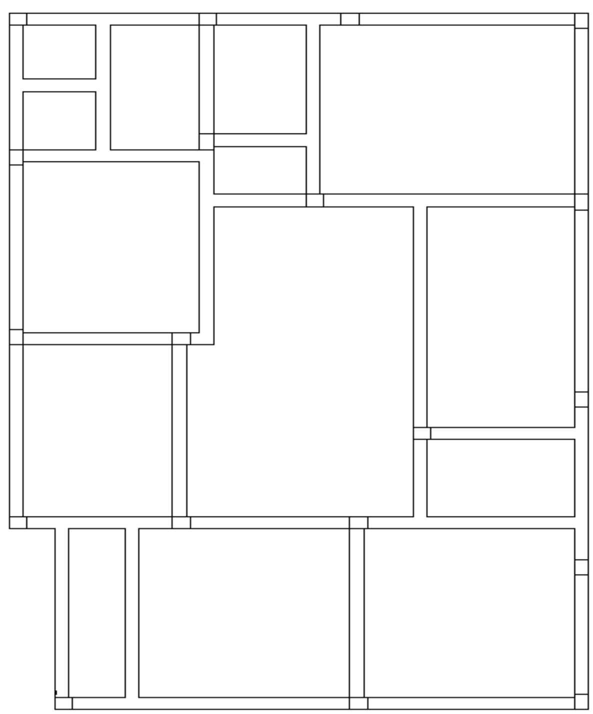


Fig.3.1

- Select 'Tools' from the Main menu and 'create model' from sub menu.(Fig 3.2)

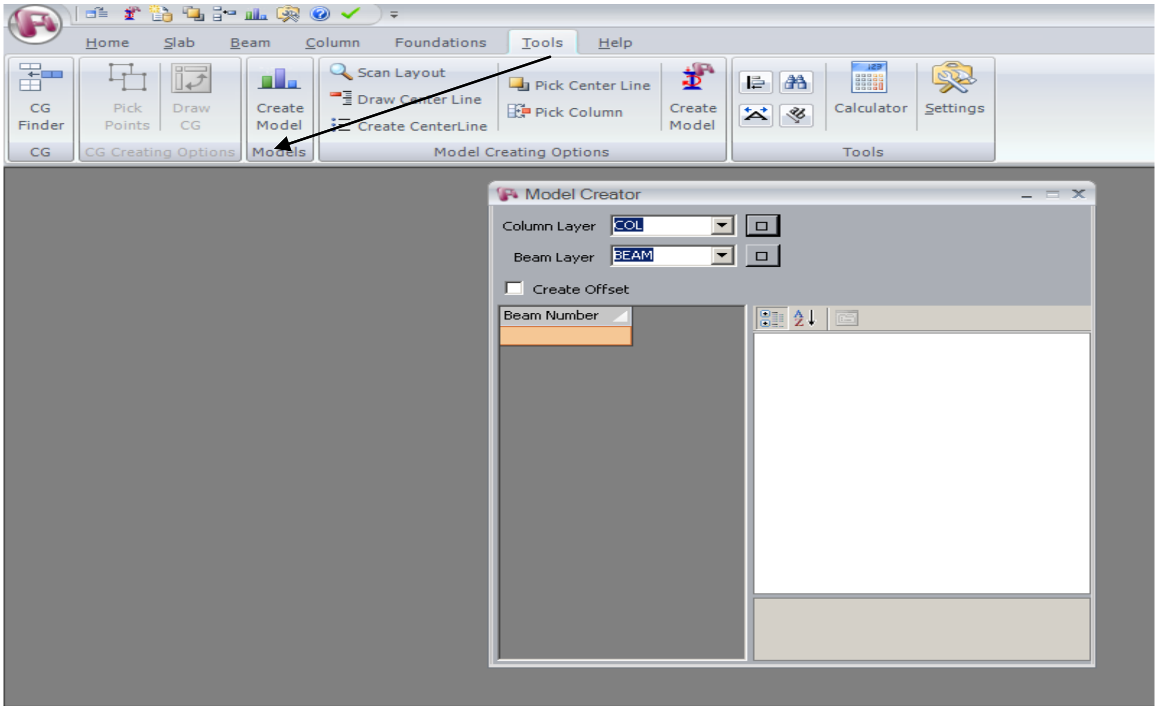
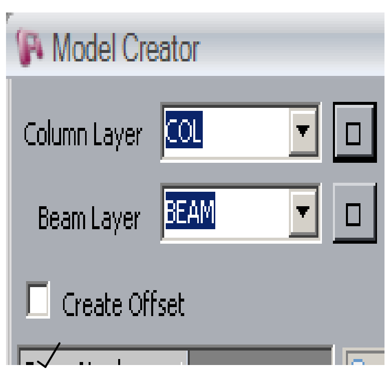


Fig.3.2

- Before starting the Scanning process user must select Column and Beam layer from the layout drawing. User can either select the layer from the drop down list or select the layer directly from the AutoCAD drawing by clicking the button in right side of dropdown list. Create Offset option will make the offset correction in the created model.



- Click on the **Scan Layout** from tool bar (**Fig 3.3)**

The Scan Layout option will help the user to scan AutoCAD layout drawing directly. The Columns drawn in the drawing must be in poly line and beams in line format and in a specified layer. By setting the Column and Beam Layer help the program to find out the beam and columns from drawing. Note that the Beam lines must break in every beam junction for better results. By Clicking this button aadspro prompt the user to select the desired layout area in AutoCAD to create the model. After the scanning process is over aadspro create the center of the layout in the drawing in specified layer.

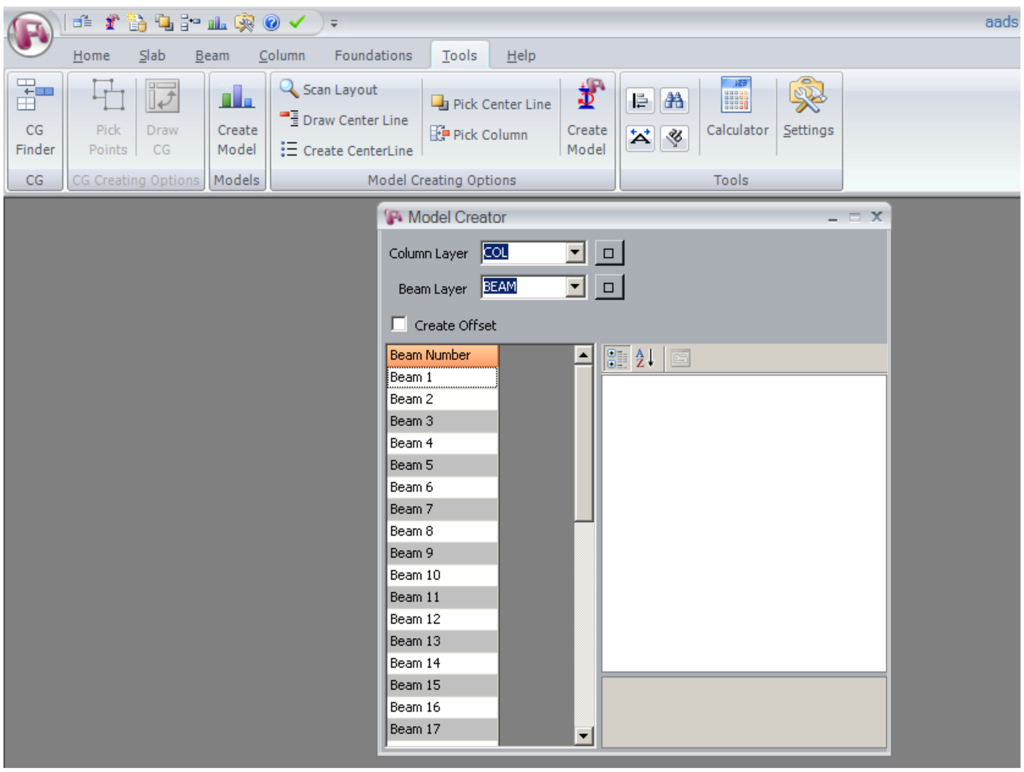


Fig 3.3

- Click on the **Draw center line** from tool bar

  This option will help the user to create the lines in the AutoCAD. When using this option user must pick two point to draw a line. Aadspro will take the line drawn after clicking kept as a beam line in model. AutoCAD will prompt to pick two point until the user press the escape button or give any invalid input.

- Click on the **Create center line** from tool bar

This option will help the user to create the center line of two lines selected by the user in AutoCAD. When using this option user must select two lines to draw a center line. Aadspro will make a center line of selected line and take it as a beam line in model . AutoCAD will prompt to pick two lines until the user press the escape button or give any invalid input

- **Result**

Drawing obtained in AutoCAD (Fig 3.4)

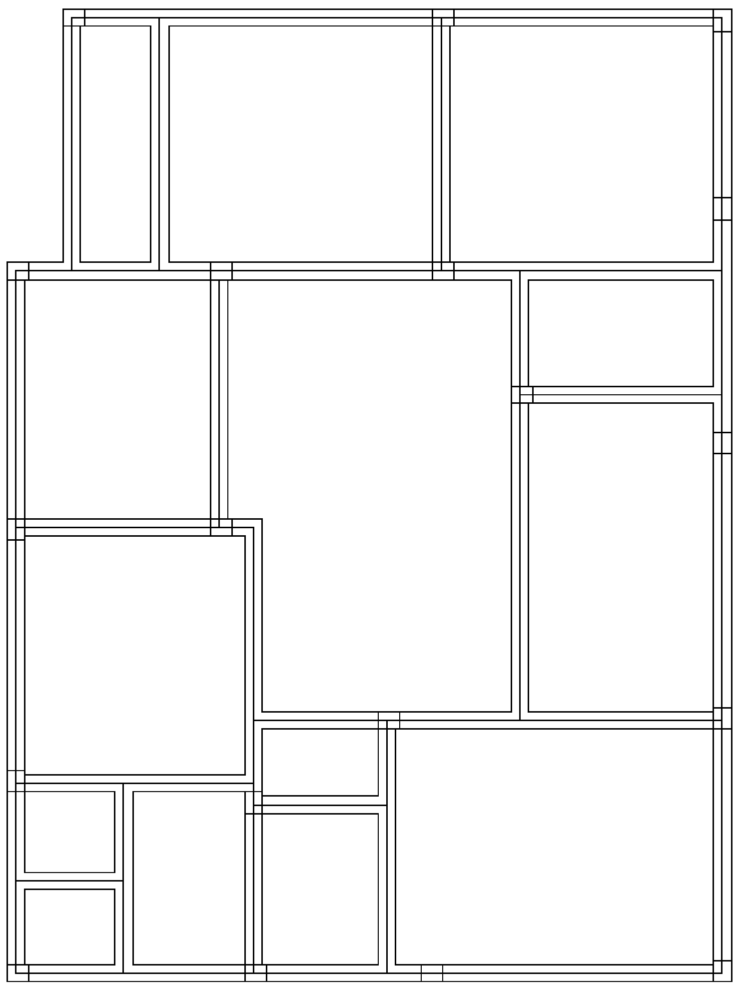


Fig 3.4

3.2 Creating model in STAAD

- Click on the **Pick Center Line** from tool bar (Fig 3.5)

This option will help the user to select a line as beam line that is already created in AutoCAD. If the user gives a layer (Beam Layer) in the time of selection, aadspro only take the line only in that layer otherwise aadspro takes all the lines in that selection area

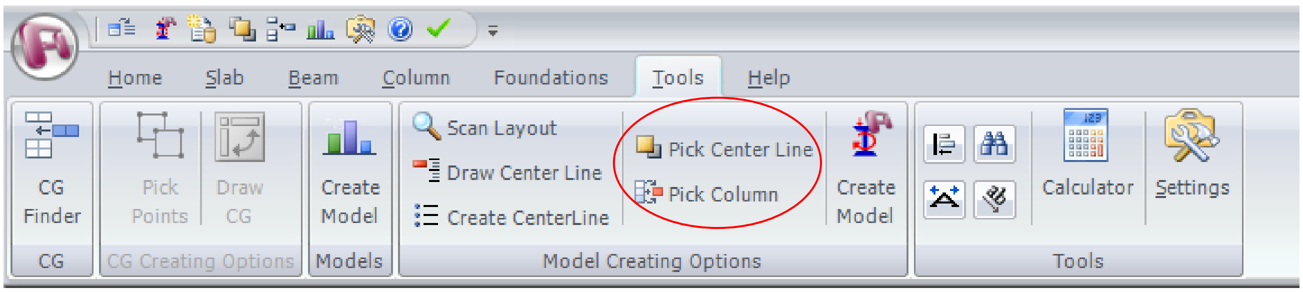


Fig 3.5

- Click on the **Pick Column** from tool bar

This option will help the user to select the column in the layout drawing. User must give the Column Layer to detect the column and it must be in a poly line format.

- Click on the **Create Model** from tool bar

This option will produce the model after verifying all the center line created or selected by the user. When user click on this button aadspro prompt the user to save the file to any location that the user need.

At the time of saving, user can select the model type (STAAD Pro and ETABS).When user select a STAAD Pro model aadspro make the STAAD Pro file (.std) in the selected location and Open in STAAD Pro Application.

If the user select a ETAB Model aadspro create an e2k (*.e2k) file in the selected location. User can import this model file in ETABS application.

- **Result**

System automatically provides Staad file with support. (Fig 3.6)

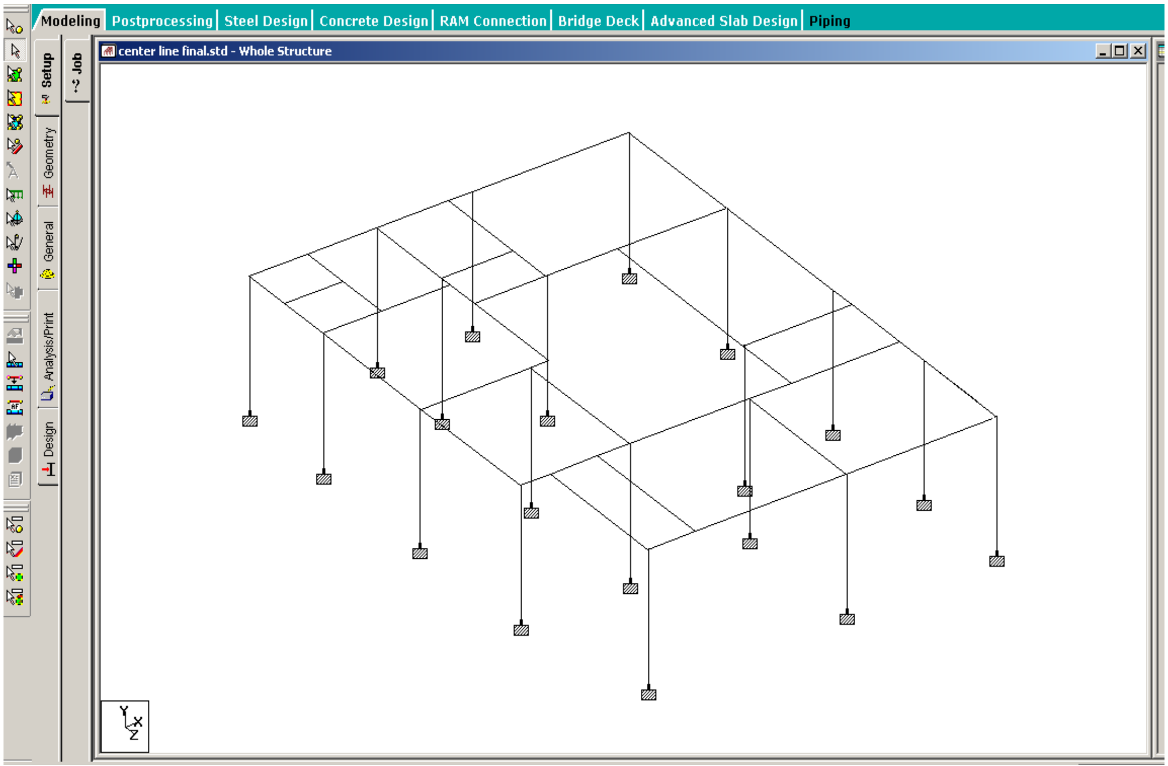


Fig 3.6

4. FOOTING

4.1 Footing Detailing

This module helps the users to create the footing schedule and footing layout from STAAD Pro and ETABS database. User can import STAAD Pro and ETABS database file to create schedule and layout. aadspro will analysis and check all the foundation columns and extract the design parameters from the imported database. aadspro makes the design values by these parameters and the other inputs given through it, and group it with their width and design values .

This chapter provides a step-by-step tutorial for the layout of footings of a multi-storied building.

- Description of structure
- Steps involved in Staad
- Steps involved in aadspro
- Results

4.1.1 Description of structure

The structure is a double storied building. Plan of the building is shown in fig (4.1.1).

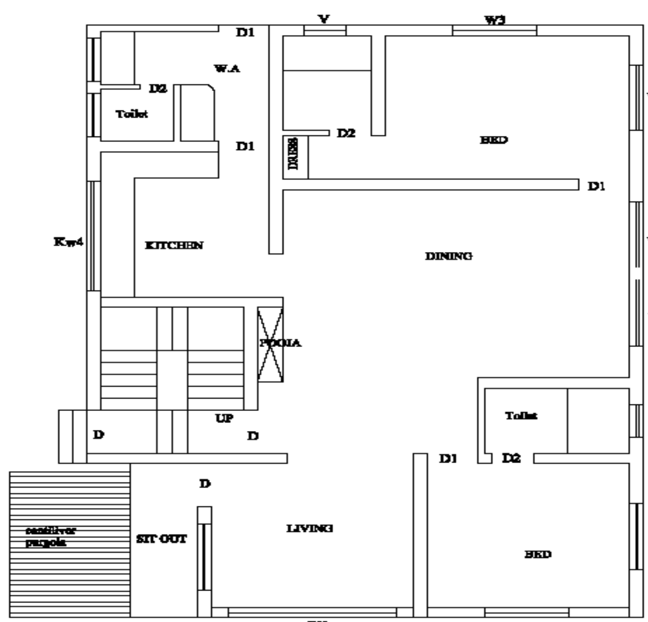


Fig 4.1.1

Layout in AutoCAD

A layout is provided in AutoCAD as shown in the fig 4.1.2.

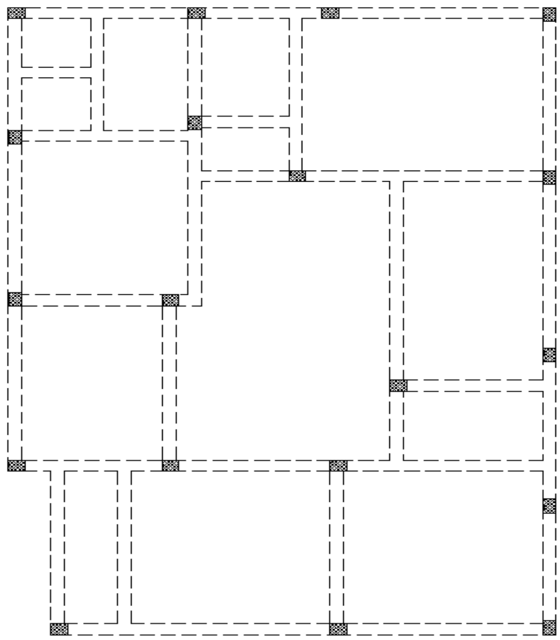


Fig .4.1.2

4.1.2 Steps involved in Staad

Staad mdb file is required for the design of footing in aadspro, Creating Staad mdb file.

4.1.3 Steps involved in aadspro

Select 'Foundation' from the Main menu and 'Detailing' from the sub menu. (fig 4.1.3).

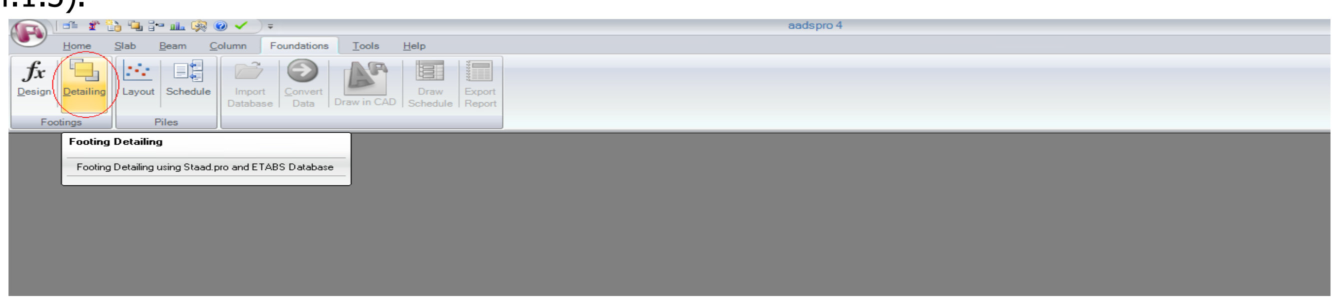


Fig .4.1.3

The detailing window helps the user to customize the detailing and design process. By setting these properties user can customize the detailing with variety of option.

1. Click on the Import Database from tool bar (fig4.1.4)

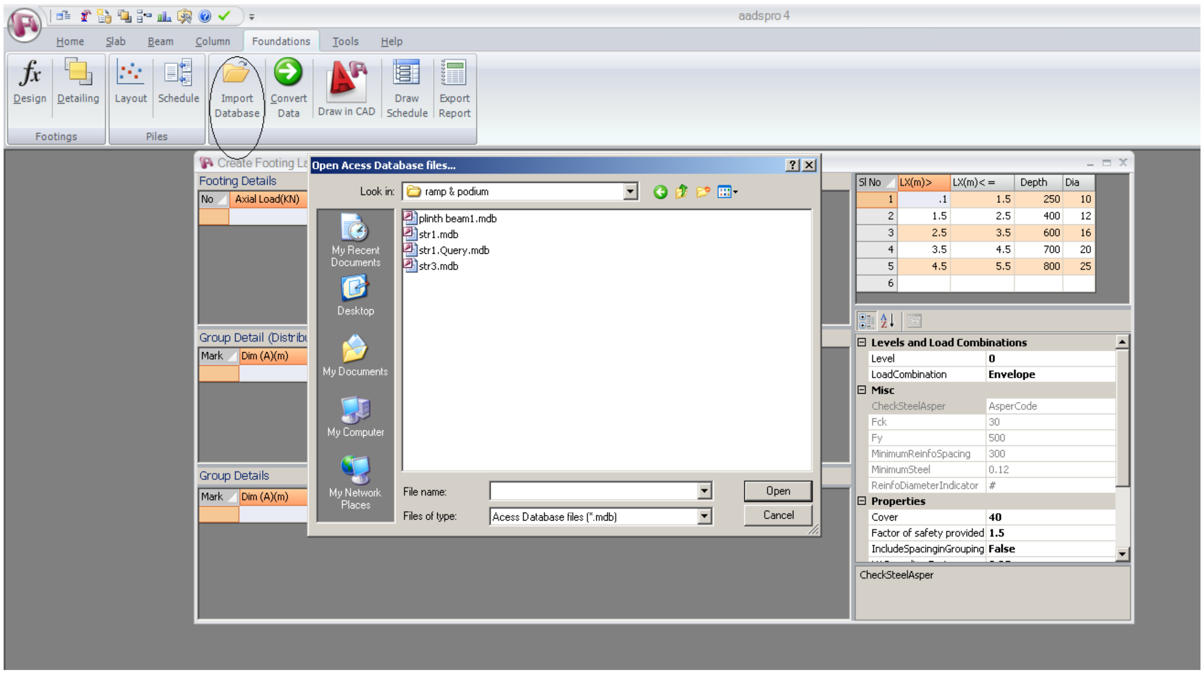
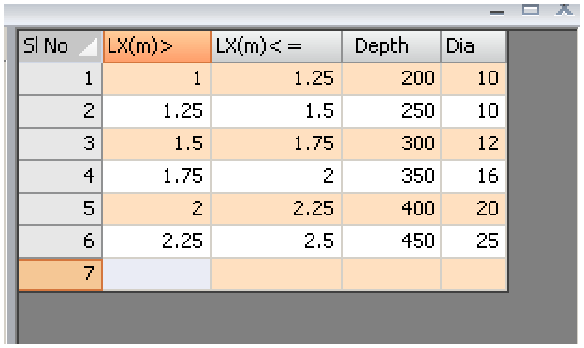


Fig 4.1.4

Import Database button allow the user to import the database file to the detailing system in aadspro. After selecting the database aadspro displays the details of the footing in the imported model (Axial Load, Moment in X direction, Moment in Y direction etc..) in the Footing details grid.

2. User can set data (fig4.1.5 & fig4.1.6)



Sl No	LX(m)>	LX(m)< =	Depth	Dia
1	1	1.25	200	10
2	1.25	1.5	250	10
3	1.5	1.75	300	12
4	1.75	2	350	16
5	2	2.25	400	20
6	2.25	2.5	450	25
7				

Fig .4.1.

Depth & dia grid.

This grid helps the user to give the input for the footing depth and diameter to be used while the design process is going on through aadspro design module. aadspro find out the corresponding depth and diameter from this grid with comparing the lx value and it will be used in design process. User can edit these values if the user wants any changes in depth and diameter.

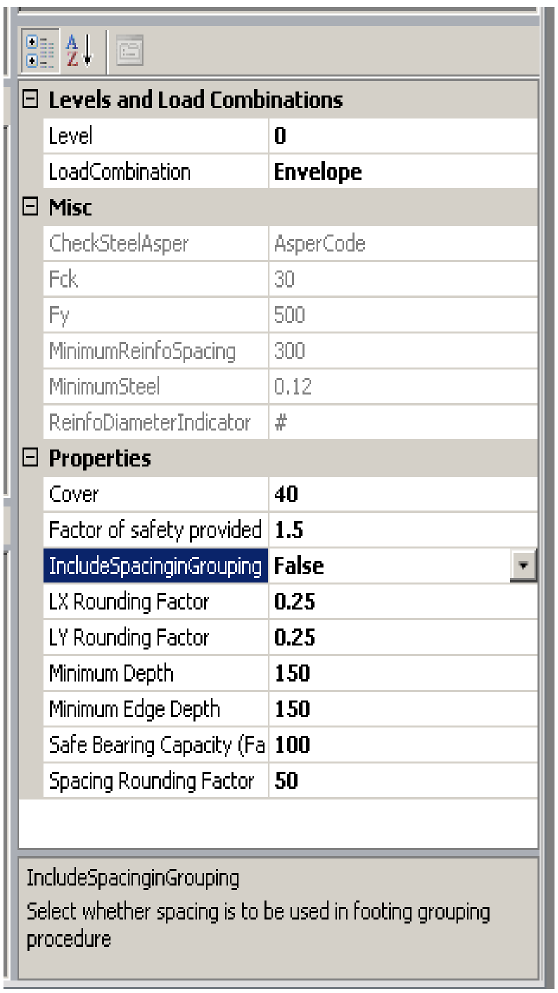


Fig .4.1.6

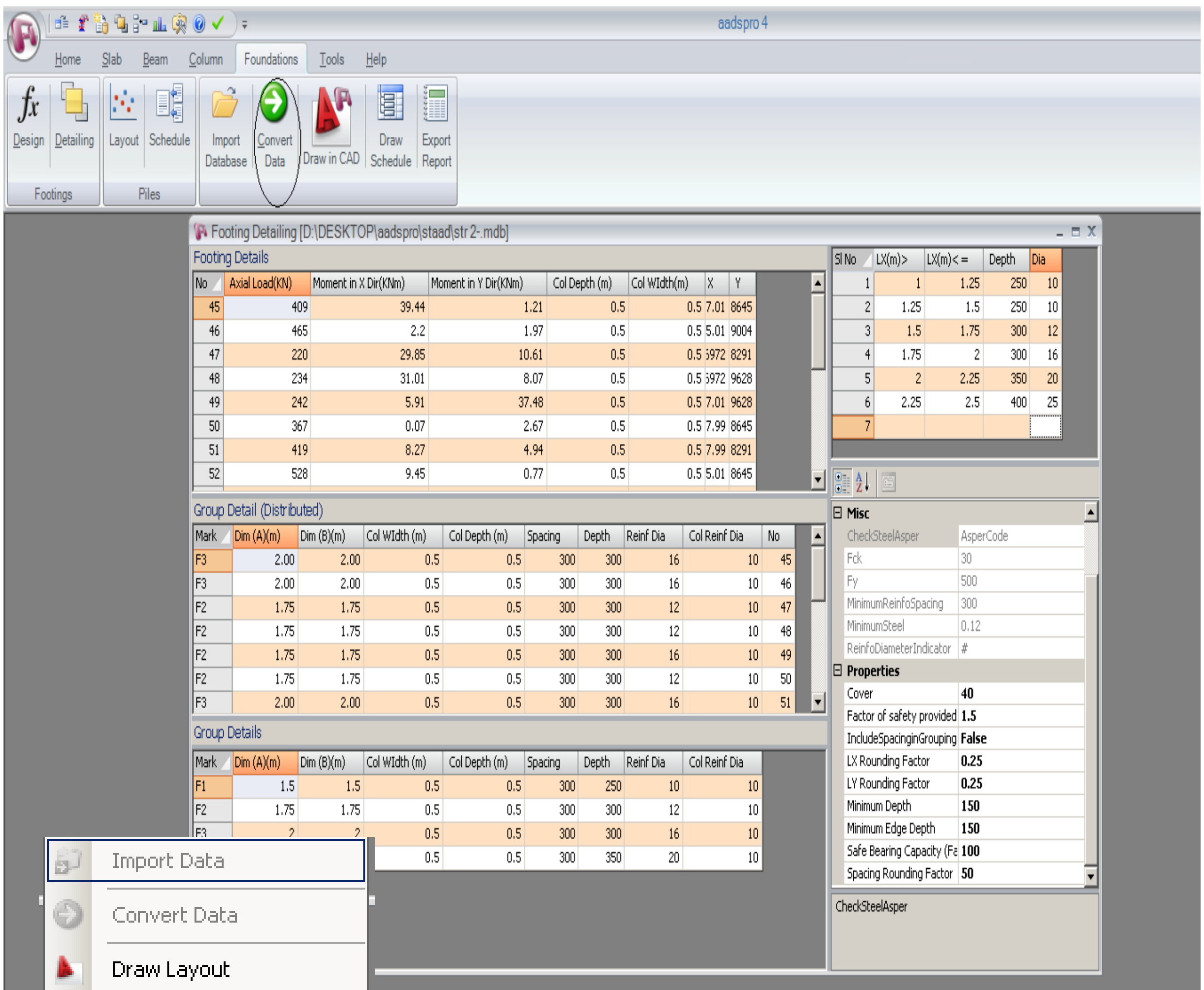
Properties

- Level
  Select the beam level to draw in cad.
- Load combination
  The load combination property list out all the load combination in model. aadspro displays the Design Load Envelope at the time of importing the database. User can change these values by selecting any load combination from the list.
- Cover
  The nominal cover use in design process.
- Factor of safety provided
  Set the factor of safety provided in the current design process
- Include spacing grouping
  Select whether spacing is to be used in footing grouping procedure.
- LX & LY Rounding Factor
  Group the footing by rounding the LX and LY value to reduce the number of footing groups.
- Minimum Depth
  Set the minimum depth of the footing
- Minimum Edge Depth
  Set the minimum edge depth of the footing
- Safe bearing capacity
  Safe bearing capacity of soil.
- Spacing Rounding factor
  Set the value of the rounding factor for spacing. It will helps in footing grouping to reduce the number of footing groups

3. Click on the Convert Data from tool bar (fig 4.1.7)

Convert data button design the imported foundation with the aadspro footing design module. Before going to the conversion user must set some parameters in depth setting grid and also in the property window.

The depth setting grid allow aadspro to take the depth of the footing in the design time with the footing size. In Property window user can set some parameter for the design process.



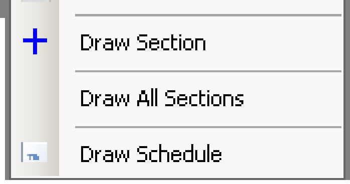


Fig 4.1.7

By right click on the 'Footing Detailing' user can draw Layout, Footing Section and schedule in Auto Cad.

4.1.3 Results

1.Click on the Draw in Cad from tool bar (fig 4.1.8)

Draw in CAD button draws the footing layout in AutoCAD with designed footing size and give notation to each footing with their group name. User can select the level of beam to be draw in Auto CAD by selecting level in property window.

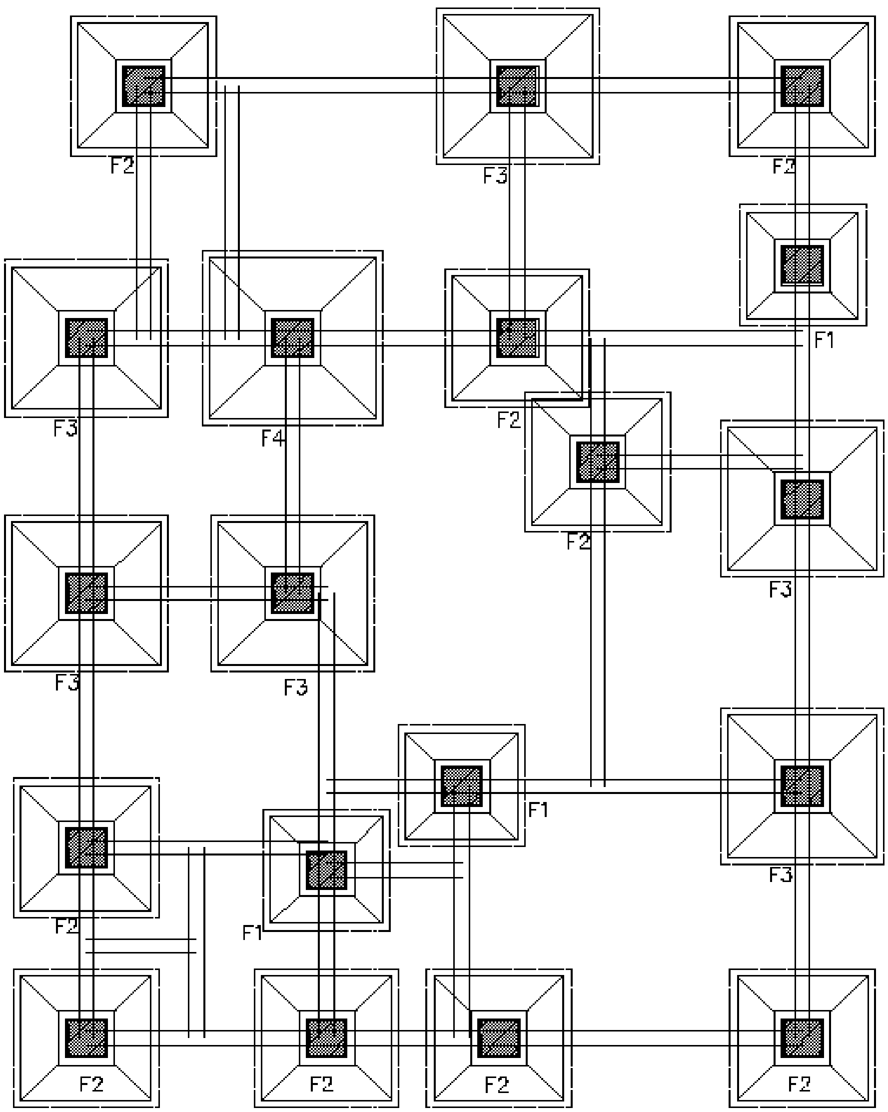


Fig 4.1.8

2. Click on the Draw Schedule from tool bar (fig 4.1.9)

Draw schedule draws a schedule in AutoCAD with footing name ,Dimension A, Dimension B, Dimension C and the Reinforcement details in a tabular format.

FOOTING SCHEDULE					
Name	A	B	C	Reinf A	Reinf B
F1	1.5	1.5	250	#10@300c/c	#10@300c/c
F2	1.75	1.75	300	#12@300c/c	#12@300c/c
F3	2	2	300	#16@300c/c	#16@300c/c
F4	2.25	2.25	350	#20@300c/c	#20@300c/c

Fig 4.1.9

3. Click on the Export report from tool bar

Export Report button export the details to excel format.

4.2 Footing Design

This module contains both Analysis and Design of a footing section as per limit state method.

4.2.1 Steps involved in aadspro

Open aadspro. Select 'Foundation' from the Main menu and 'Design' from drop down menu. (fig.4.2.1)

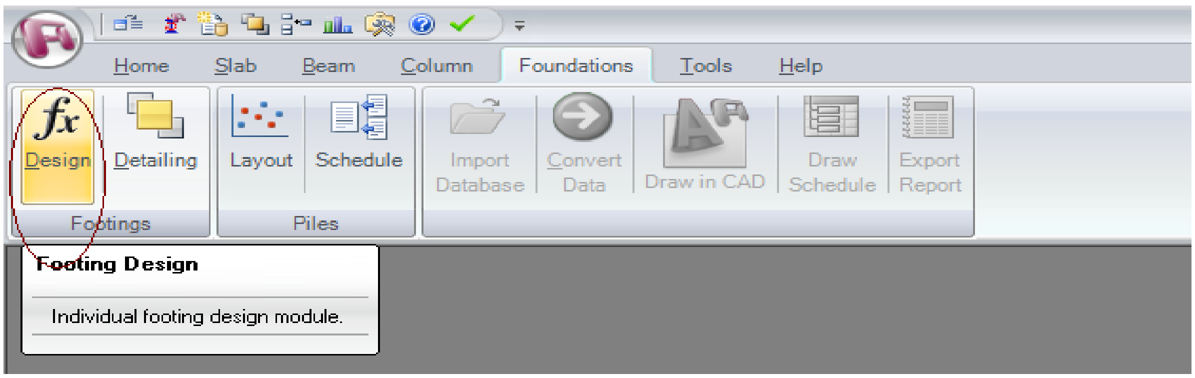


Fig.4.2.1

1. User can set data in footing design grid (fig 4.2.2)

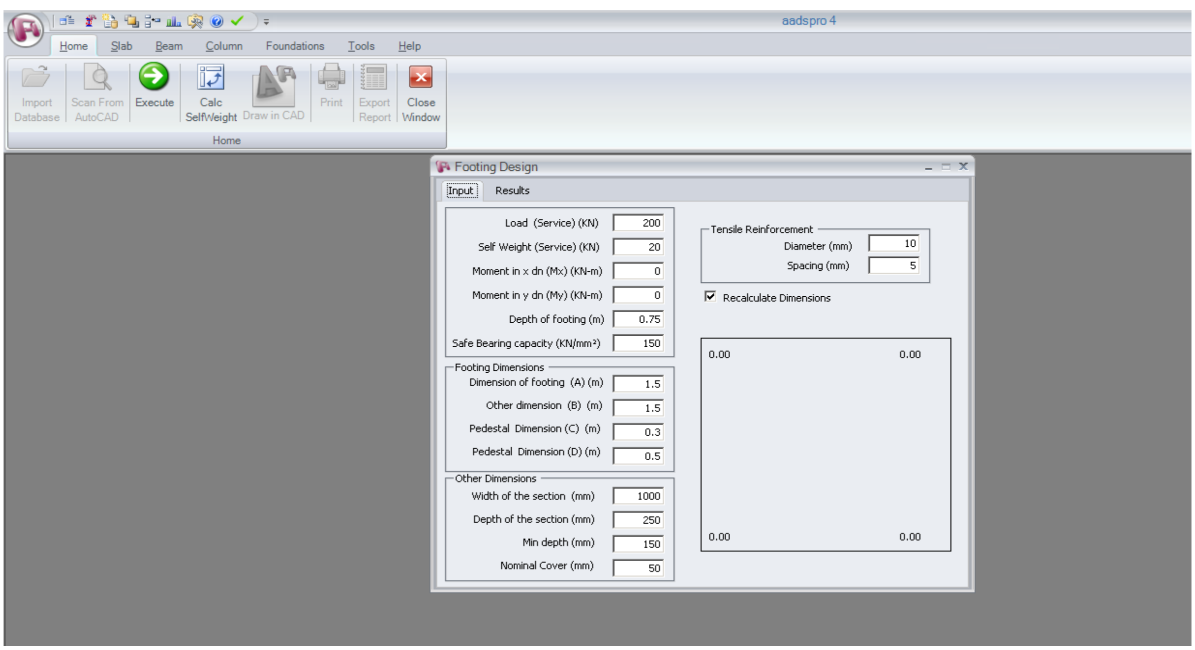


Fig.4.2.2

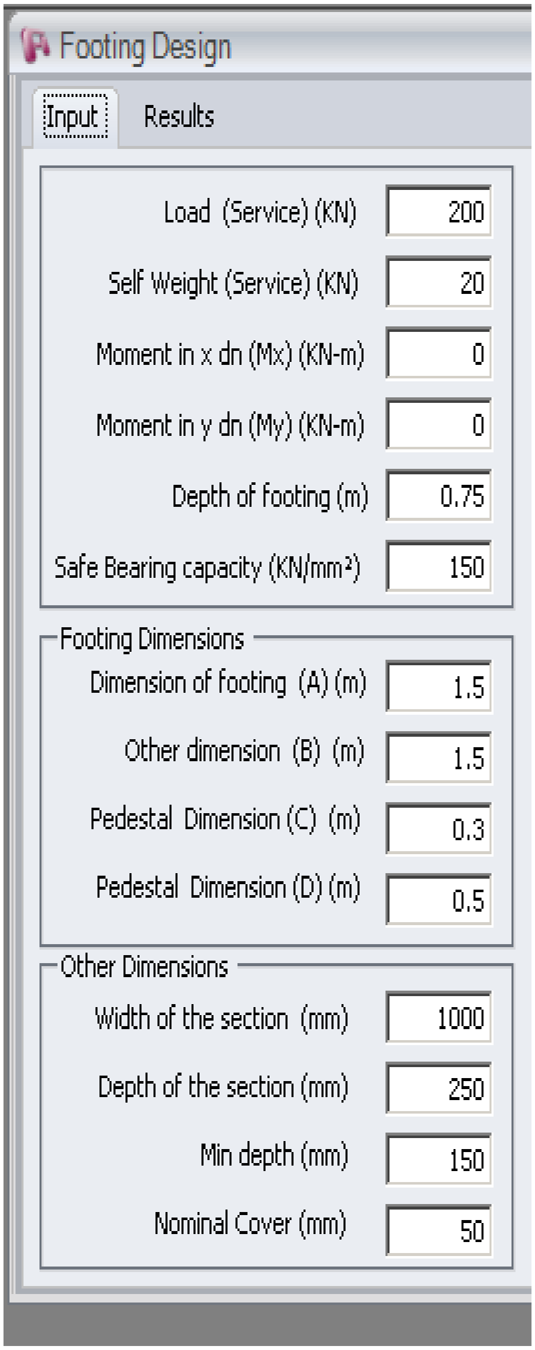


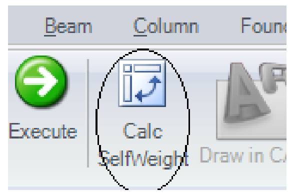
Fig. 4.2.3

- Load (service)

Load acting on it.

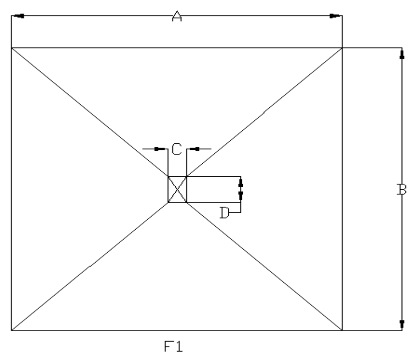
- Self weight

Self weight can calculate by <u>calc self weight</u> button from drop down menu by applying footing dimensions in footing design grid.



- Moment in X & Y direction
- Depth of footing
- Safe bearing capacity of soil
- Dimension of footing A&B, Pedestal

dimensions C&D



- Width of section

Program automatically calculates the width of section.

- Depth of section
- Minimum depth

Minimum depth is 150 mm

- Nominal cover

4.2.2. Example for footing design (fig 4.2.4)

Consider the example given below and its solution by manual calculation. Explanations are also attached after solutions to see how the program arrives at results.

Example

A square column of size 400x400 and 2.75 metre effective length carries an axial load of 1500 kN including its weight. Design the square footing for the column. The service bearing capacity of the soil is 150 kN/m^2. Use M25 concrete and Fe 415 steel.

Solution

Load on the column = 1500 kN

Weight of footing = 150 kN

Total load = 1650 kN

Area of footing = $\frac{1650}{150}$ = 11 m^2

$\therefore$ Side of the footing = $\sqrt{11}$ = 3.3 m

$\therefore$ Provide 3.4m x 3.4m footing

Check for Maximum & Minimum soil pressure:

Moment about x direction, Mx = 35 KNm

Moment about y direction, My = 25 KNm

Z = L^3/6

= 3.4^3/6 = 6.55 m^3

P/A = $\frac{1500}{3.4x3.4}$

= 129.76 KN/m^2

Mx/Z = 35/6.55 = 5.34KN/m^2

My/Z = 25/6.55 = 3.8 KN/m^2

Maximum soil pressure = $\frac{P}{A}+\frac{Mx}{Z}+\frac{My}{Z}$

= $129.76+5.34+3.8$

= 138.9 KN/m^2 < 150 KN/m^2

Minimum soil pressure = $\frac{P}{A}-\frac{Mx}{Z}-\frac{My}{Z}$

= $129.76-5.34-3.8$

= 120.6 KN/m^2 > 0

Hence safe for Maximum and Minimum soil pressure.

Net upward pressure intensity = $\frac{1500}{3.4\times 3.4}$

= 129.76 KN/m^2

P_u = 1.5 x 129.76

= 194.64 KN/m^2

Depth of the footing

The critical section for bending moment is taken corresponding to the column face.

Projection beyond the column face = $(3.4-0.4)/2$ = 1.5m

B.M. at the critical section, M_u = $1.5\times 129.76\times 1.500\times \frac{1.500}{2}$

= 218.97kNm

B.M. at the critical section, M_u = 218.97x3.4 = 744.50KN/m

The cross sectional shape of footing, resisting this BM is trapezoidal.

The trapezoidal section may be considered as the equivalent rectangular section of width,

b = $b_w+(B-b_w)/8$

$$= 0.4 + (3.4 - 0.4)/8$$

= <u>0.775m</u>

R = 0.138 fck = 0.138 x 25

$$d = \sqrt{\frac{M_u}{Rb}} = \sqrt{\frac{744.50 \times 10^6}{0.138x25 \times 775}} = 527.68\text{mm}$$

= 530mm

Assume a clear cover of 50 mm and dia.of bars 10mm,

D = 530 + 50 + 5 = 585 mm

Provide overall depth, D = 900mm (for shear consideration)

d = 900-50-5 = 845 mm

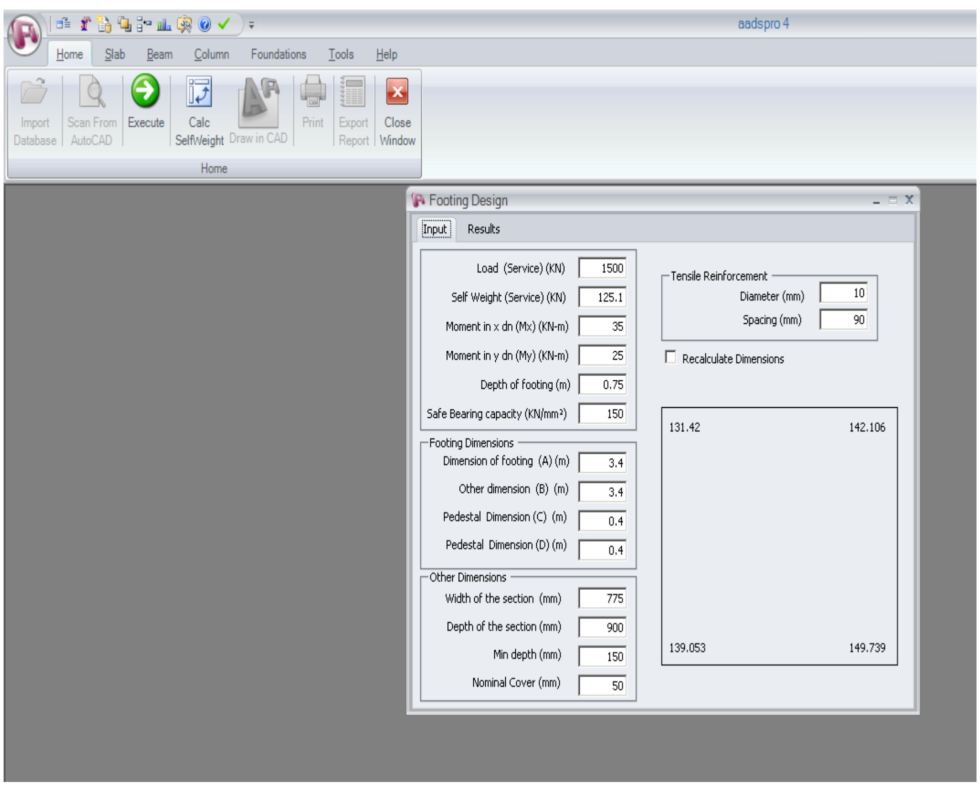


Fig 4.2.4

Calculation for Reinforcement;

$$0.87 f_y A_{st} d\left(1-\frac{A_{st} f_y}{bdf_{ck}}\right) = M_u$$

$$0.87x415xA_{st}x845\left(1-\frac{A_{st}415}{775x845x25}\right) = 218.97x10^6$$

$$305087.25A_{st} - 7.733A_{st}^{\ 2} - 218.97x10^6 = 0$$

A_{st} = $731.28mm^2$

Percentage of reinforcement, Pt = $100\left(\frac{A_{st}}{bd}\right)$

= $100\left(\frac{731.28}{775x845}\right)$

= 0.11%

Hence provide A_{st} 731.28 mm^2

Check for shear stresses

(i) Critical section for one-way shear

The critical section for one-way shear is at a distance equal to the effective depth from the column face.

Min depth of footing at edge = 0.2D = 0.2x900 = 180mm

Provide 180mm>150mm minimum specified in IS 456-2000, Cl: 34.1.2

Overall depth at critical section, D' = $900-\left(\left(\frac{900-180}{1500}\right)x845\right)$

= 495mm

Effective depth at critical section, d' = 495-50-5 = 440mm

Width of footing at critical section, b' = b+2d

= 775+ (2 x 845) = 2465mm

Shear force at the critical section, V_u = 1.5 x 129.76x (1.500 -0.845)

= 127.48 kN

$$\tau_v = \frac{V_u}{b' \times d'} = \frac{127.48 \times 10^3}{2465 \times 440}$$

= 0.12 N/mm^2

τ_c for pt = 0.11% = 0.29 N/mm^2

Here $\tau_v < \tau_c$, hence the section is safe for one way shear.

(ii) Critical section for punching shear

Critical section for punching shear is all-round the column at a distance $\frac{d}{2} = \frac{845}{2} = 423$ mm from the column face.

Critical Perimeter, b' = 4(400 + 845)= 4 x 1245

Overall depth at critical section, D' = $900 - \left(\left(\frac{900 - 180}{1500} \right) x423 \right)$

= 696.96mm

Effective depth at critical section, d' = 697-55 = 642mm

Shear force at the critical section, V_u = 1.5 x 129.76x (3.4^2 -1.245^2)

= 1948.34 kN

Punching shear stress, τ_v = $\frac{F_u}{4b_0 d}$

$$= \frac{1948.34 \times 10^3}{4 \times 1245 \times 642}$$

= 0.61 N/mm^2

Allowable stress, τ_c = $0.25\sqrt{f_{ck}}$

= $0.25\sqrt{25}$

= 1.25 N/mm^2

Here $\tau_v < \tau_c$, section is safe for two way shear.

Provided, A_{st} = 731.28mm^2

Assuming 10 dia bars, required spacing = $1000 \div \left(\frac{731.28}{78.5}\right)$

= 100mm

$\therefore$ Provide 10mm-dia bars @100 mm c/c

4.2.2. Results

1. Click on the Execute from tool bar (fig 4.2.5)

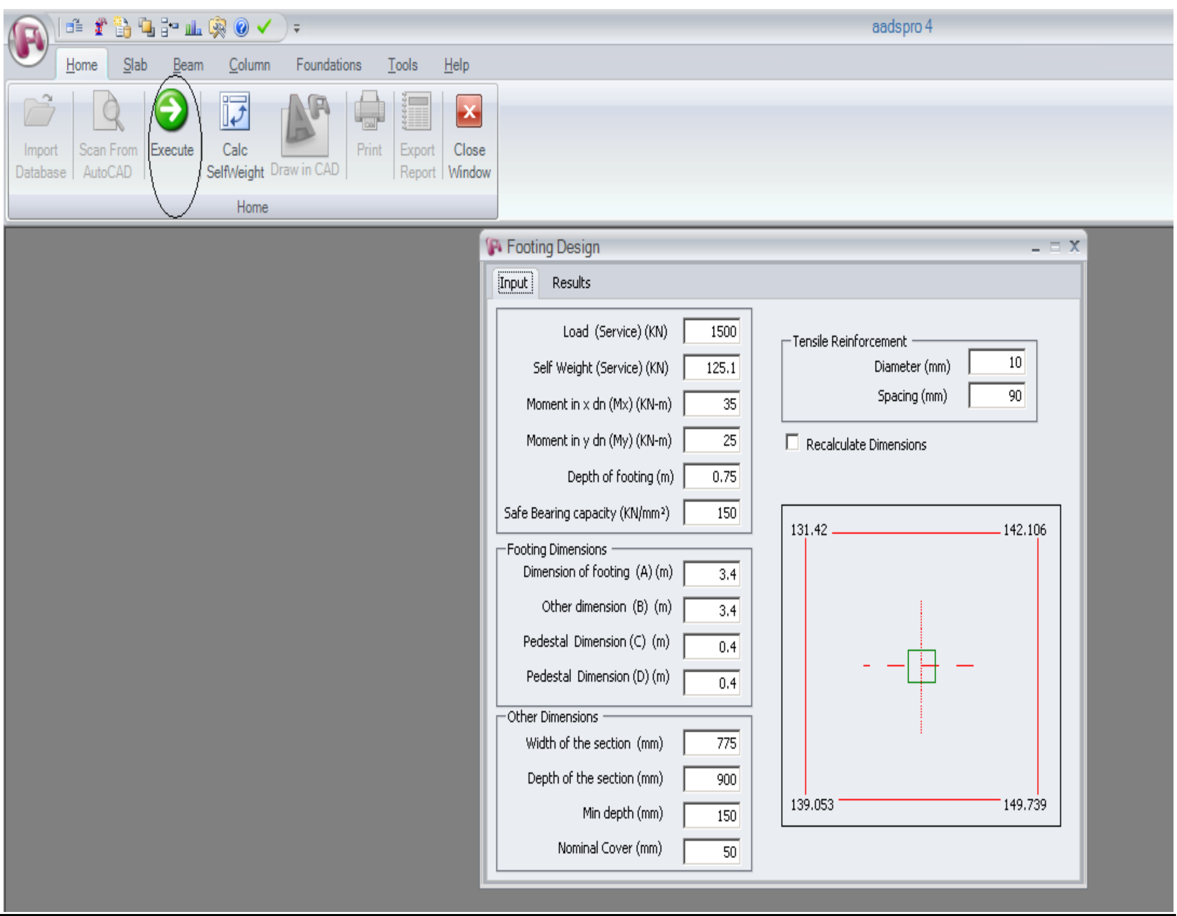


Fig.4.2.5

Make it use for a required one by changing the dimensions either by typing or by clicking on the buttons. Corresponding changes are seen in the attached figure and each time it calculates and shows the corner stresses in the figure.

2. Click on the Results from footing design grid (fig 4.2.6)

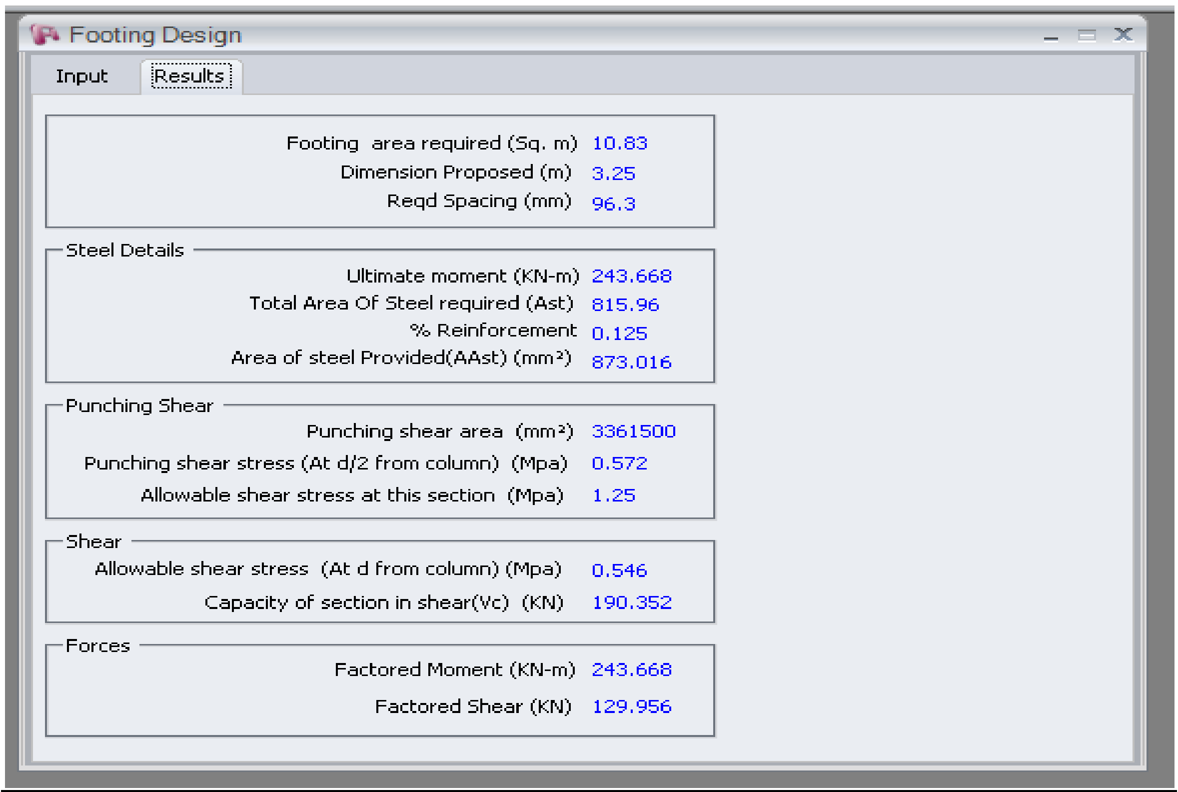


Fig.4.2.6

Comparison of results

1. Factored values of moment and shear

	Values by program	Values by manual calculation
Moment	243.668 KNm	218.97KNm
Shear	129.956KN	127.48KN

Here almost all values are exactly same as that of manual calculation.

2. Program also gives details like,

   A_{st} required = 815.96 mm^2

   A_{st} provided = 873.016 mm^2

   Spacing of 10 dia bars = 90 mm c/c

5. PILE

5.1 Pile Detailing

5.1.1 Description of the building

The structure is a triple storied building; in which pile is to be designed (fig 5.1.1). plan of the building is shown in figure .

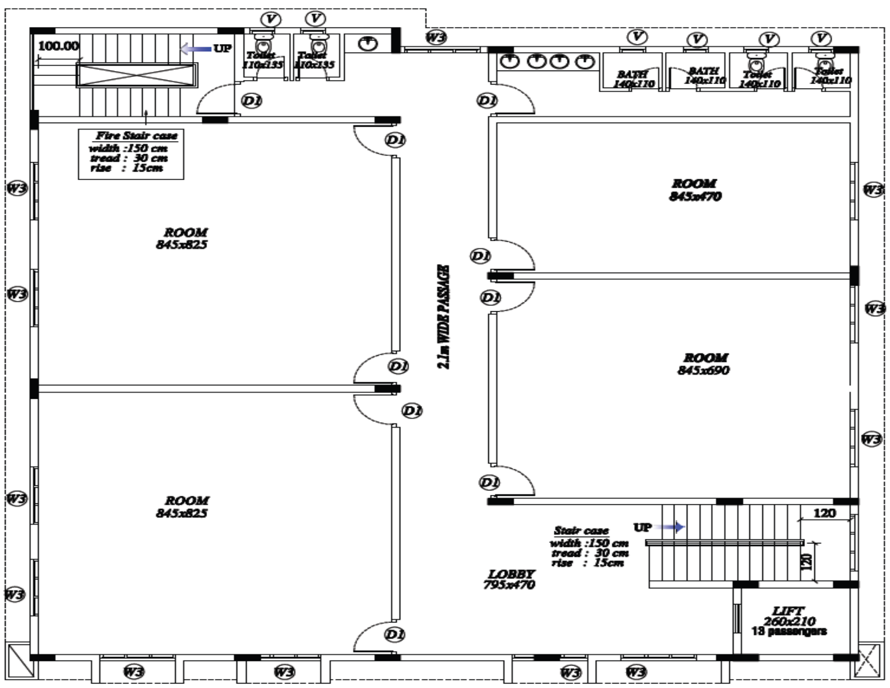


Fig.5.1.1

5.1.2 Steps involved in Staad

Staad mdb file is required for the design of pile in aadspro,

Creating Staad mdb file

- Open the Staad file and analyze it.
- Select Tools >SQL Query>Advanced Query for STAAD Pro version 2006 & higher (fig.5.1.2).

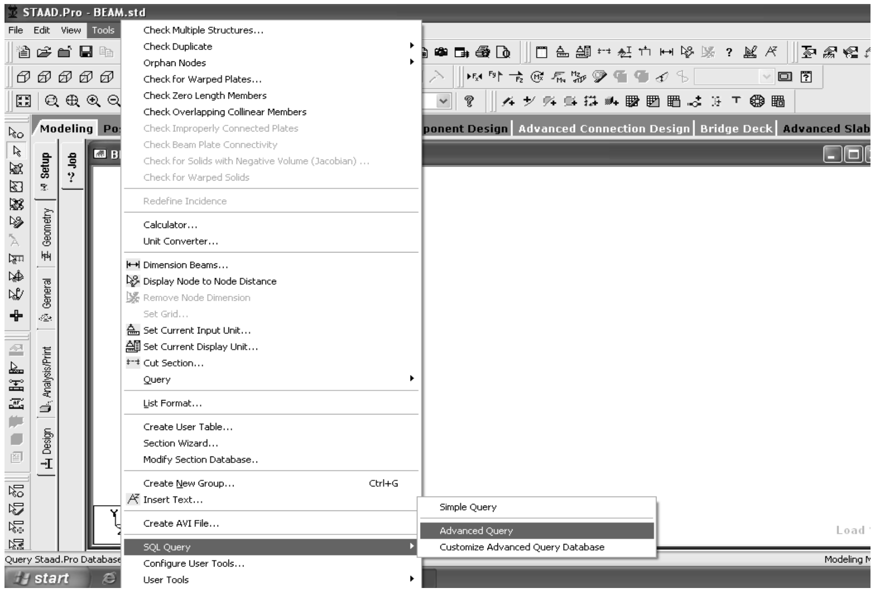


Fig .5.1.2

- Steps involved in aadspro

Open aadspro**.** (AutoCAD will be open automatically)

Select Foundation from main menu.

- Click on the Layout from tool bar (fig5.1.3)

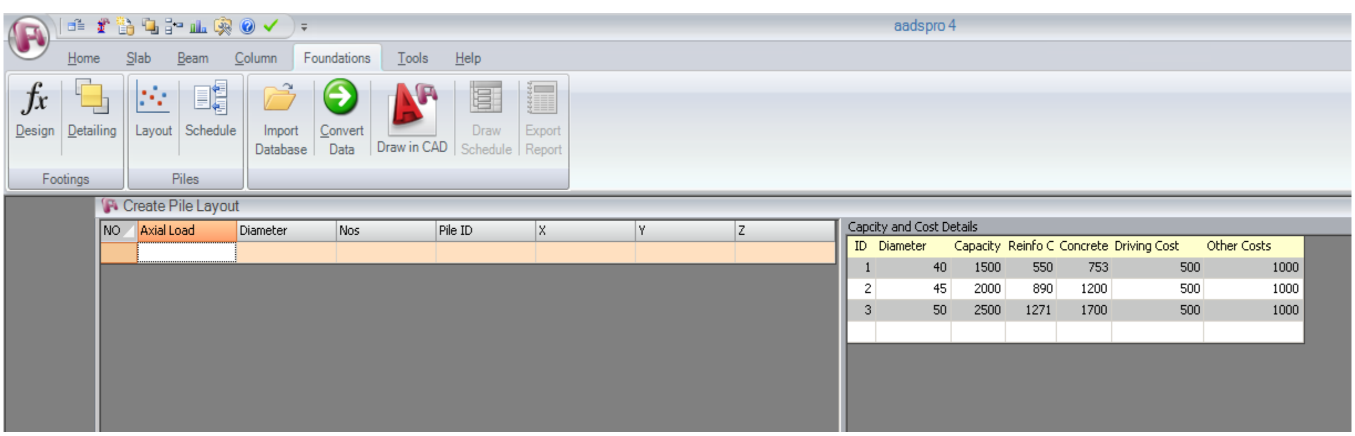


Fig 5.1.3

The layout window helps the user to customize the detailing and design process. By setting these properties user can customize the detailing with variety of option.

- Click on the Import Database from tool bar (fig 5.1.4)

Import Database button allow the user to import the database file to the detailing system in aadspro .After selecting the database aadspro displays the details of the Pile in the imported model (Axial Load, Moment in X direction, Moment in Y direction etc..) in the Pile details grid.

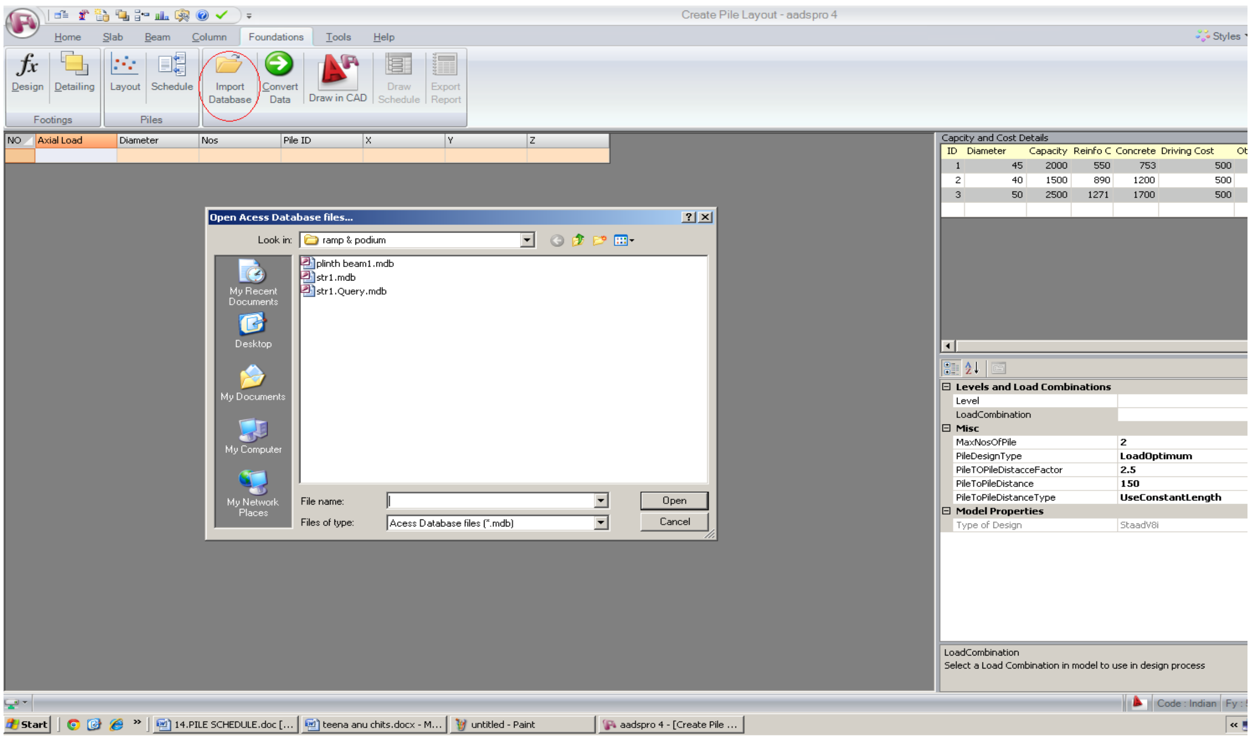


Fig 5.1.4

User can set

- Diameters of piles
- Corresponding capacities of pile from soil report
- Level (i.e. the plinth beam level in staad pro.)
- Load combination (Maximum ie envelope)
- Maximum number of pile
- Pile design type
- Pile to pile distance factor
- Pile to pile distance
- Click on the Convert Data from tool bar (fig 5.1.5)

Convert data button design the imported Pile with the aadspro pile design module. Before going to the conversion user must set some parameters in depth setting grid and also in the property window. The depth setting grid allows aadspro to take the depth of the Pile in the design time with the pile size.

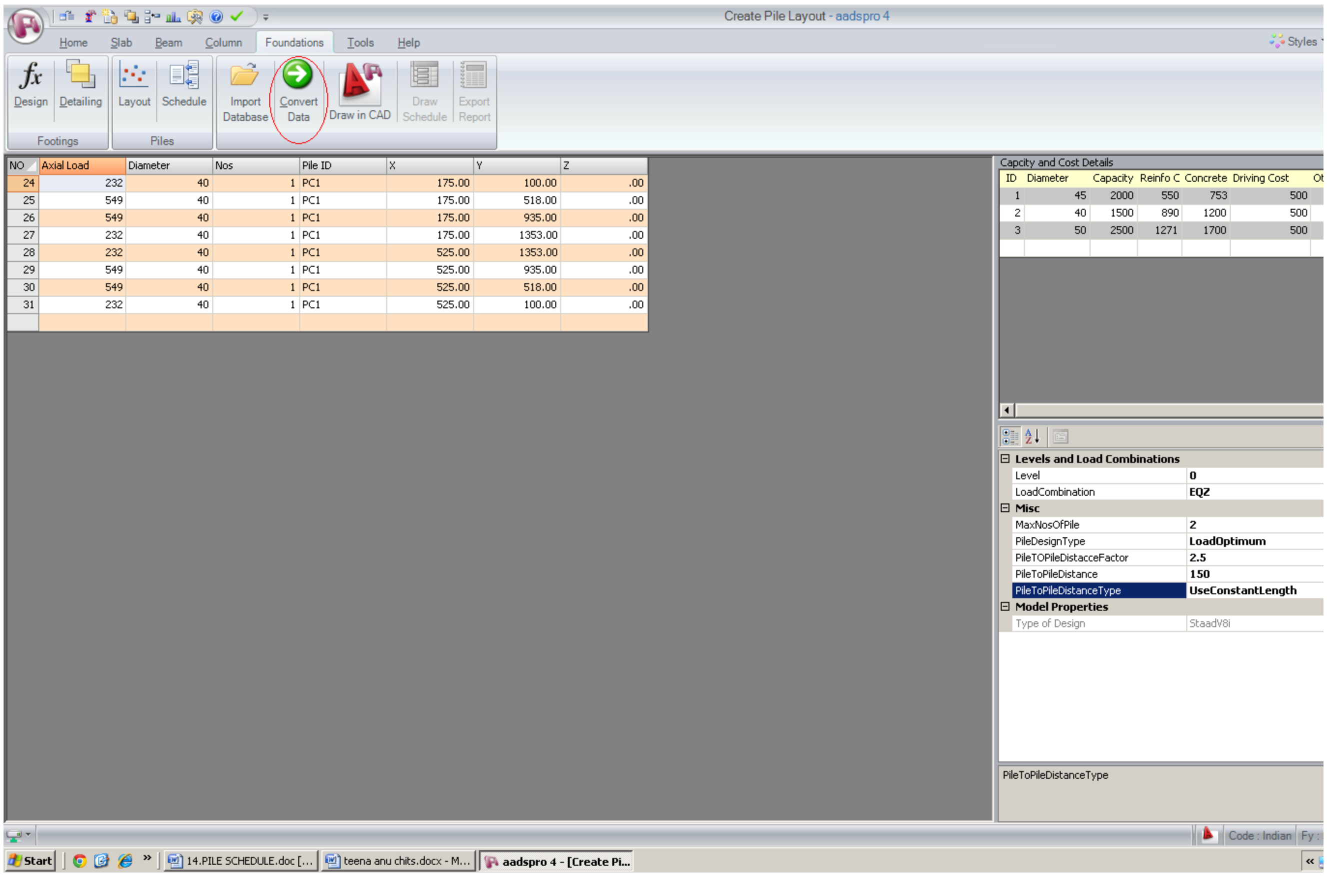


Fig 5.1.5

5.1.3 Results

- Click on the Draw in Cad from tool bar (Fig 5.1.6)

Draw in CAD button draws the Pile layout in AutoCAD with designed Pile size and give notation to each Pile with their group name.

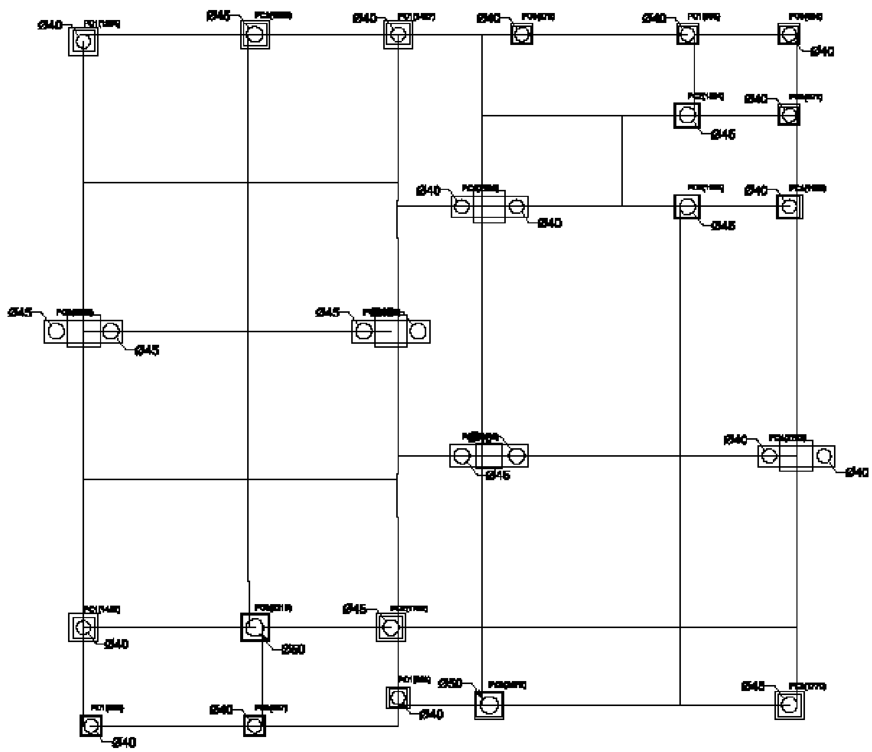


Fig 5.1.6

- User can add piles directly to staad file by Right click- Add pile to staad pro model (Fig 5.1.7)

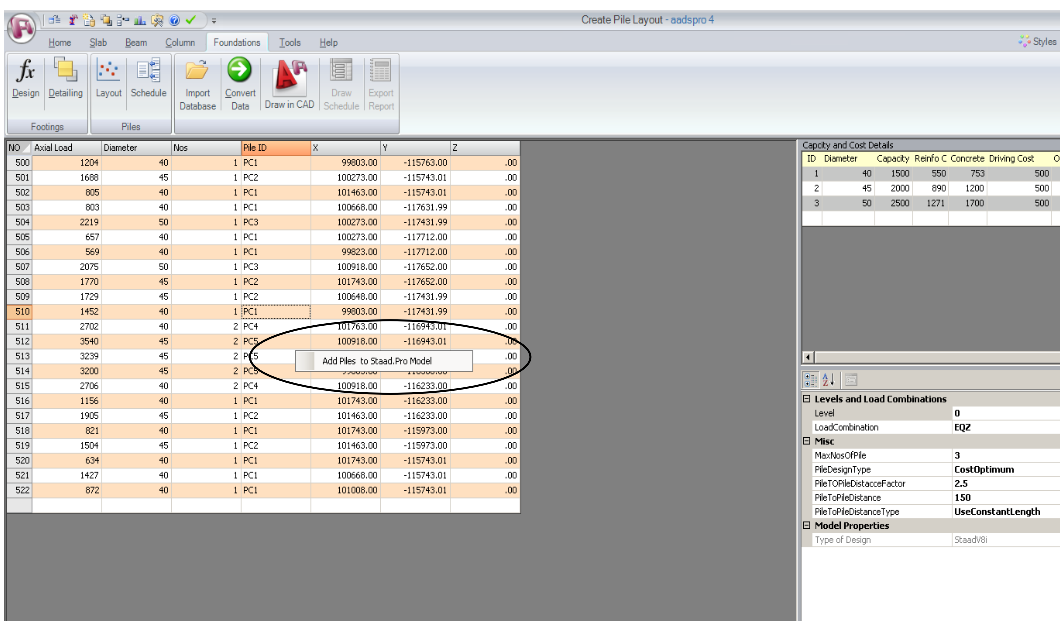


Fig 5.1.7

5.2 Pile Schedule

1. Click on the **Schedule** from tool bar (Fig 5.2.1)

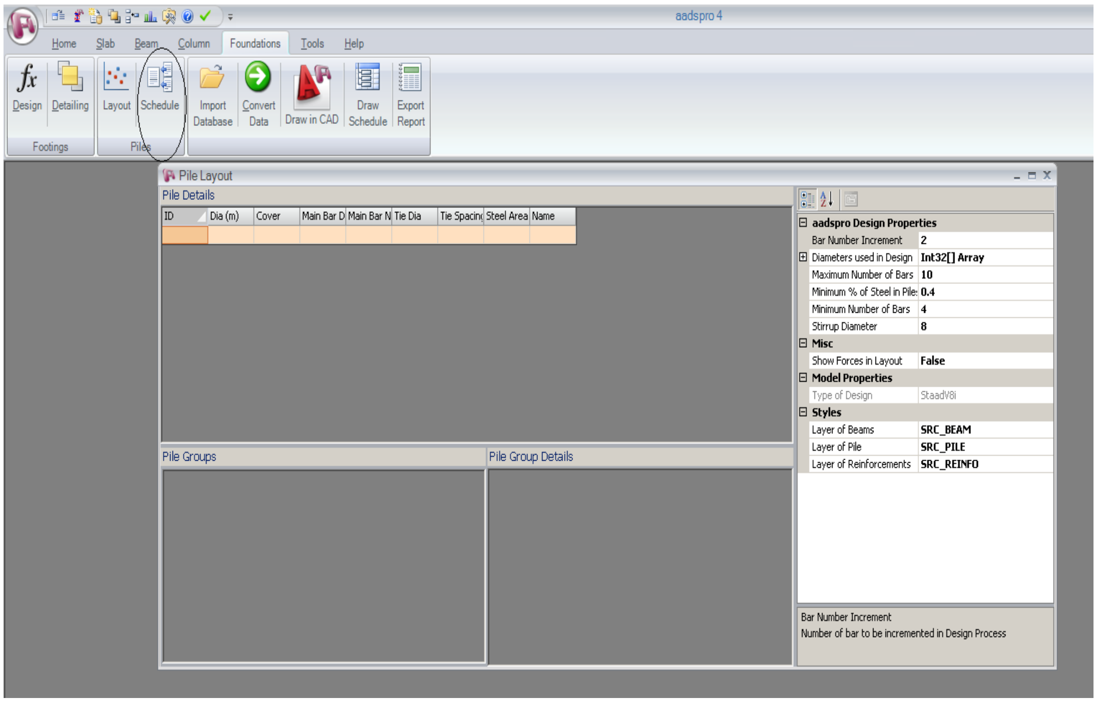


Fig 5.2.1

2. Click on the **Import Database** from tool bar

Import Database button allow the user to import the database file to the detailing system in aadspro.

User can set

- Bar number increment
- Diameter used in design
- Maximum number of bar
- Minimum percentage steel in pile
- Minimum number of bar
- Stirrup diameter

3. By Right click **Design all using aadspro** from the bar (Fig 5.2.2)

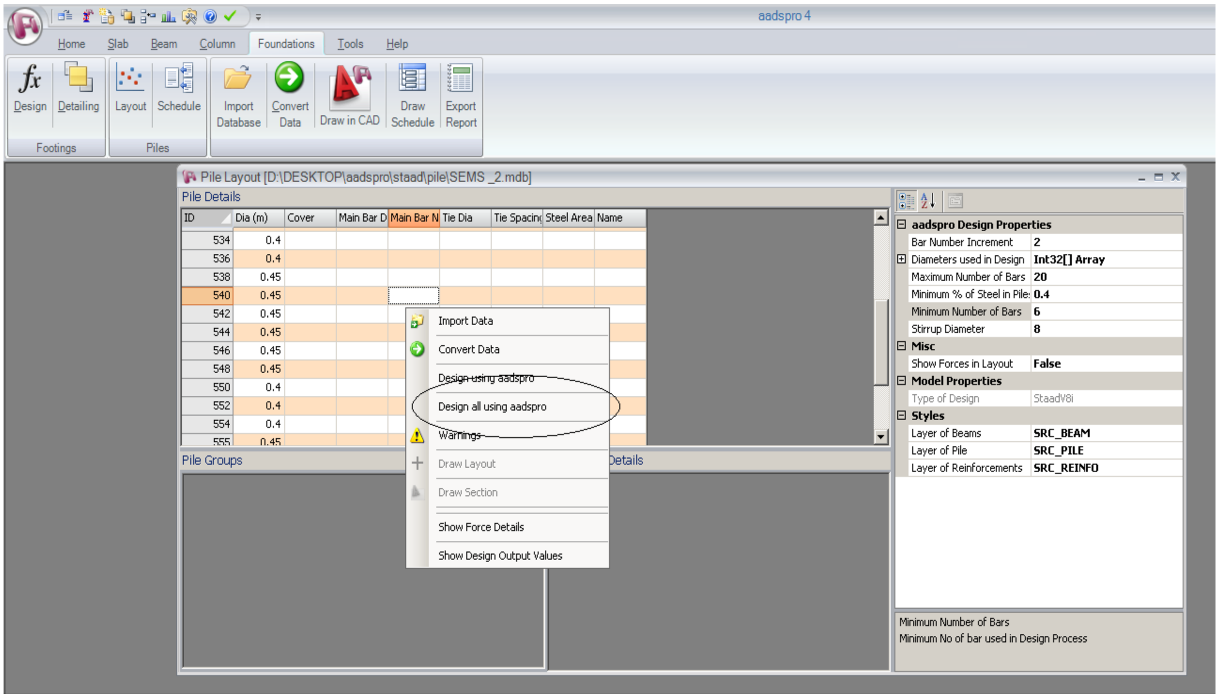


Fig 5.2.2

It will produce pile details, pile groups and pile group details. (Fig 5.2.3)

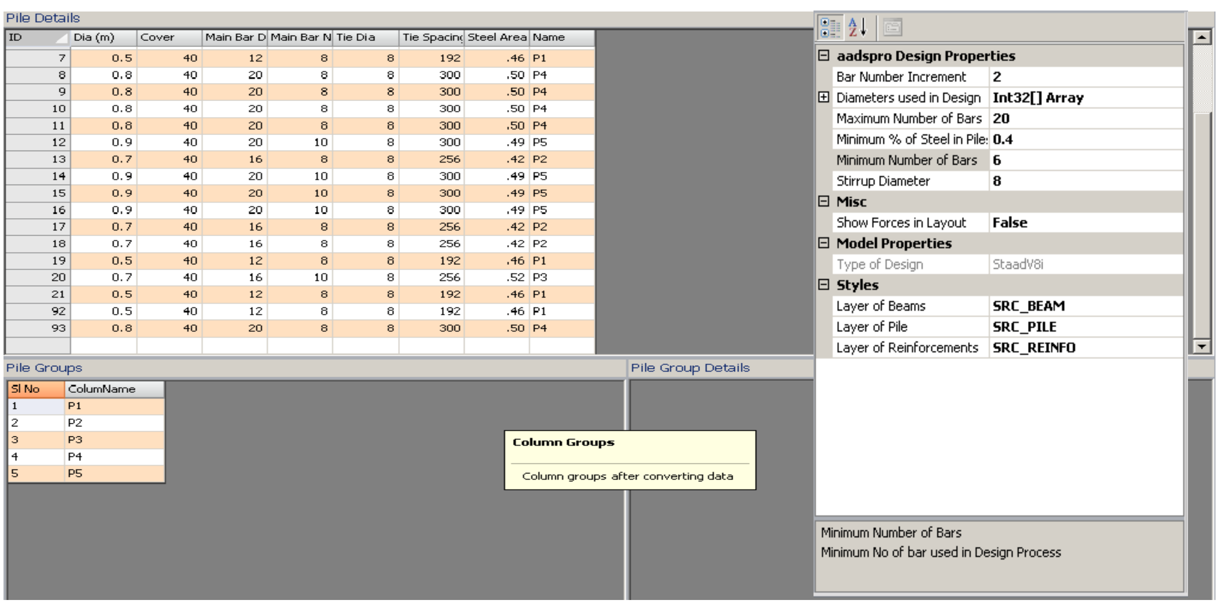
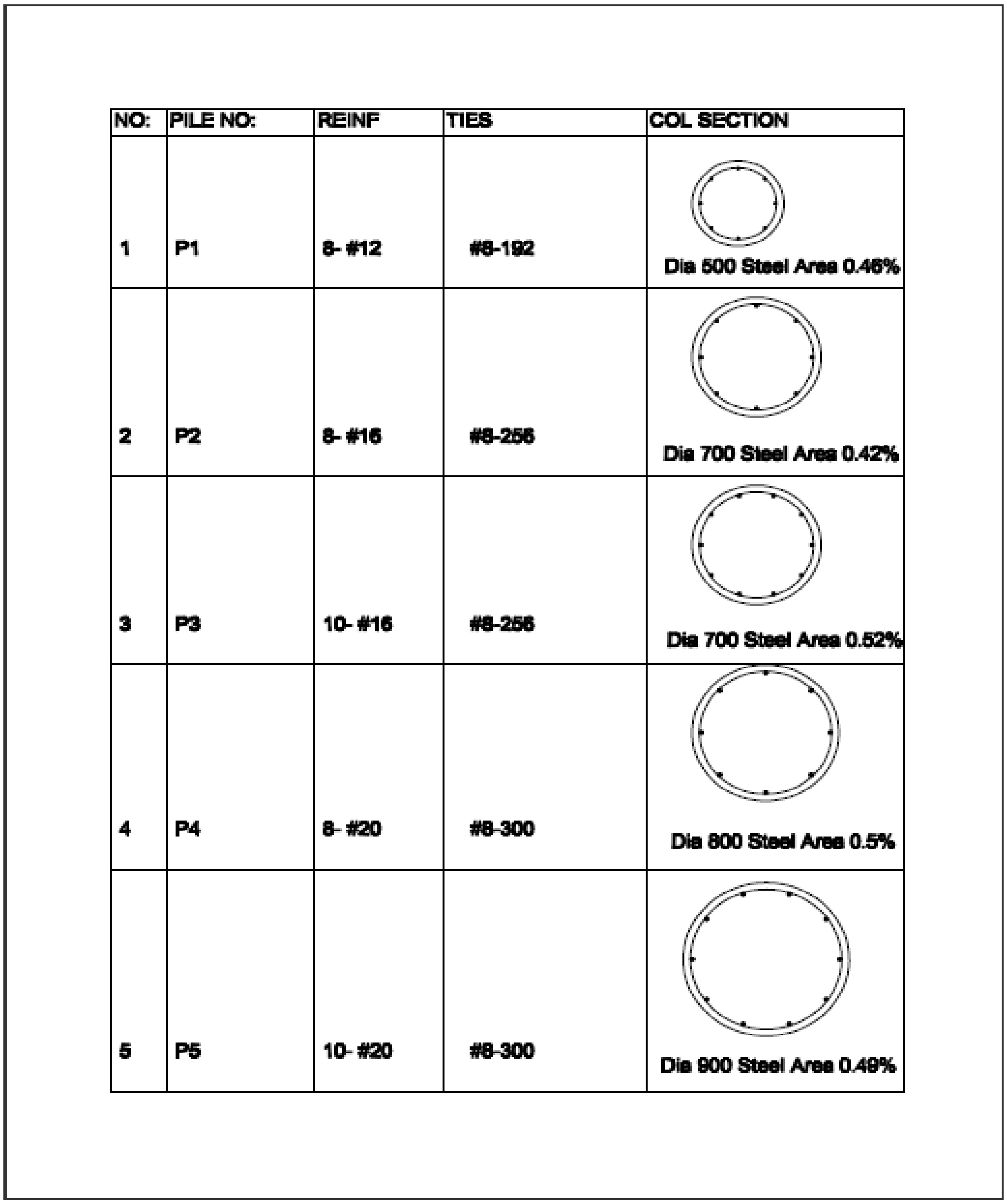


Fig 5.2.3

4. Results

Click on the **Draw Schedule** from tool bar (Fig 5.2.4)

Draw schedule button draws the pile schedule in AutoCAD with designed pile number, reinforcement, tie, and pile section with diameter and percentage steel.



NO:	PILE NO:	REINF	TIES	COL SECTION
1	P1	8- #12	#8-192	Dia 500 Steel Area 0.46%
2	P2	8- #16	#8-256	Dia 700 Steel Area 0.42%
3	P3	10- #16	#8-256	Dia 700 Steel Area 0.52%
4	P4	8- #20	#8-300	Dia 800 Steel Area 0.5%
5	P5	10- #20	#8-300	Dia 900 Steel Area 0.49%

Fig 5.2.4

6. BEAM

6.1 Beam Detailing

aadspro beam detailing system is a simple and powerful tool for creating beam detailing drawing with minimum user effort. It reduces the time for creating detailing drawing in AutoCAD. A simple user interface helps these activity more simple and accurate. All the standard code checking is done through this system.

This chapter provides a step-by-step tutorial for the detailing of a beam of a multi-storied building.

6.1.1 Description of the structure

The structure is a double stored building; in which the beam is to be designed. Plan of the building is shown in fig (6.1.1).

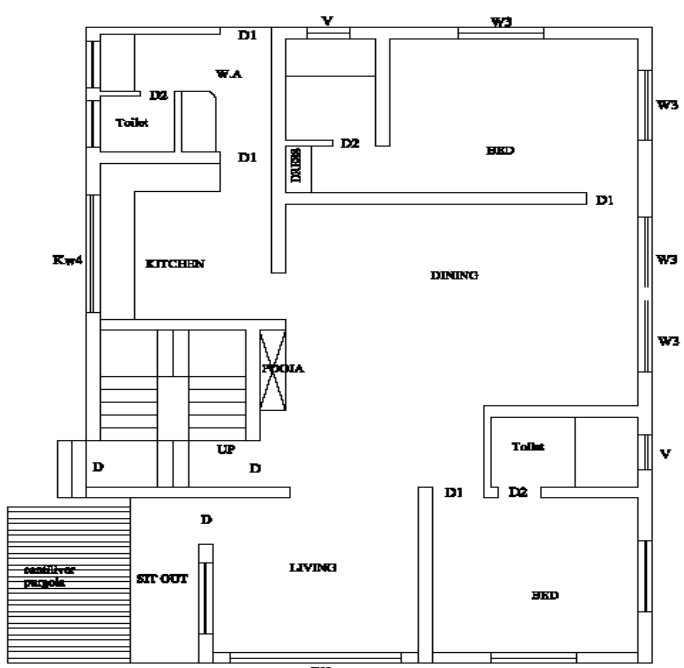


Fig 6.1.1

6.1.2 Steps involved in Staad

Staad mdb file is required for the design of beam in aadspro, Create a Staad mdb file

6.1.3 Steps involved in aadspro

Open aadspro. (AutoCAD will be open automatically)
Select 'Beam' from main menu.

- Click on the **Using STAAD. Pro'** from tool bar (fig1.3.1)

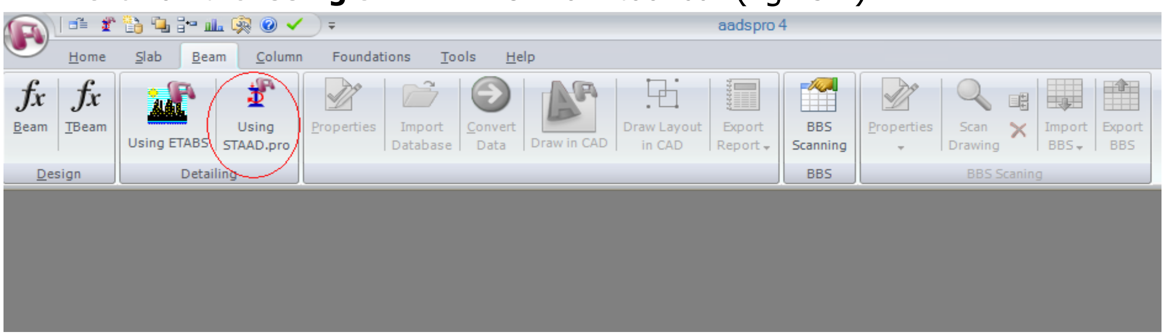


Fig 6.1.2

- Click on the **Import Database** from tool bar (fig 6.1.3)

Import Database button allow the user to import the database file to the detailing system in aadspro.
Import the staad mdb. File, .After selecting the database file aadspro check the input file and display the Level/Stories in Storey list box

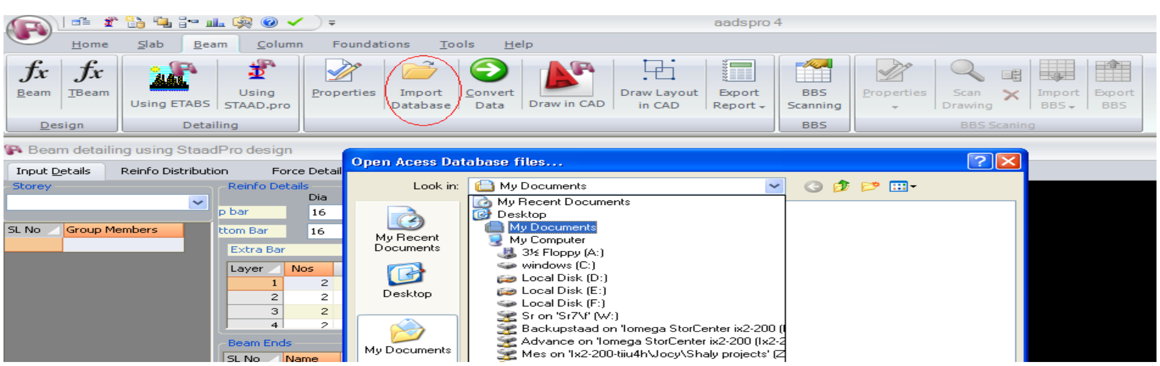


Fig 6.1.3

- User can set (Fig 6.1.4)

This window helps the user to customize the detailing and design process. By setting these properties user can customize the detailing with variety of option.

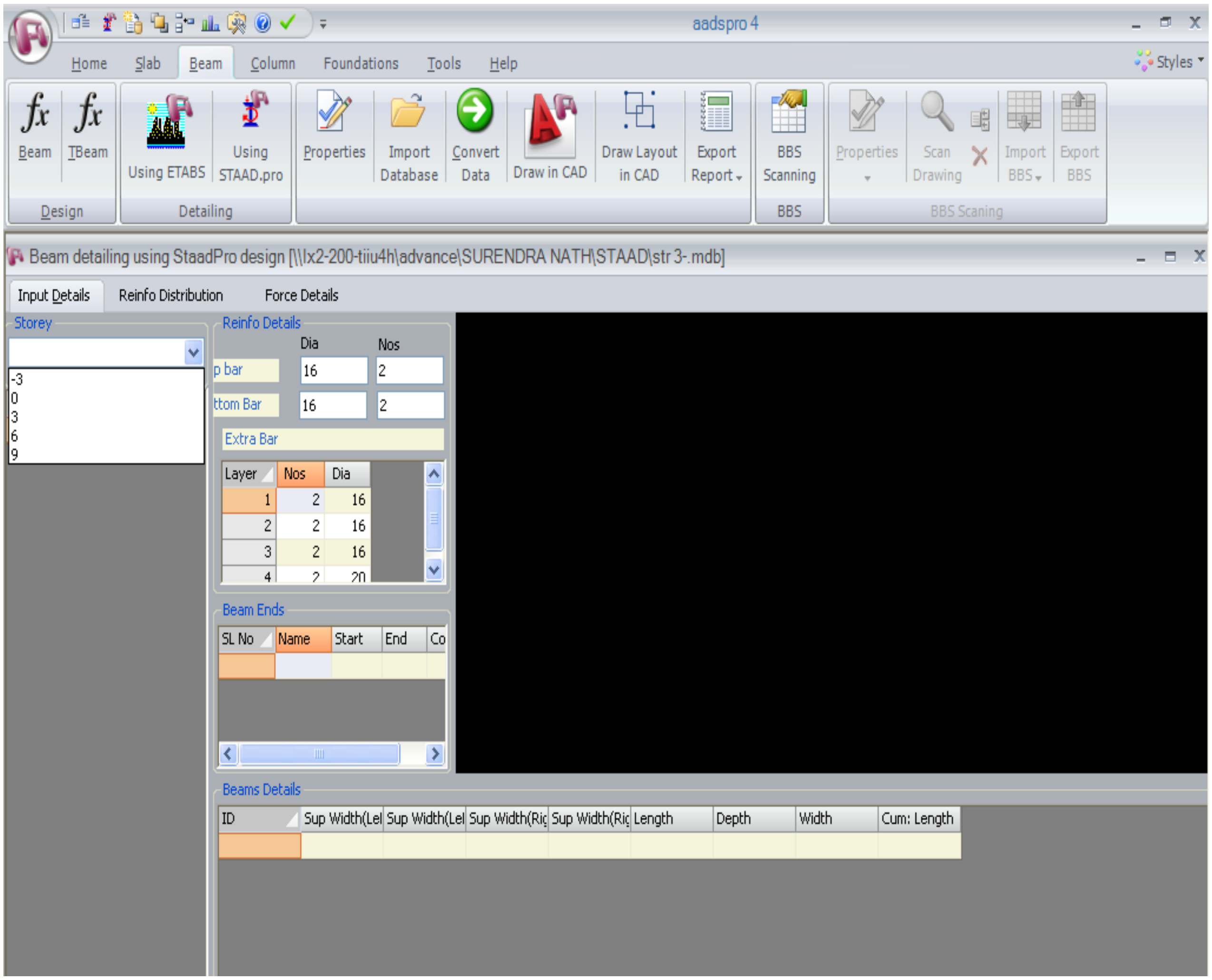


Fig 6.1.4

- Reinforcement Details
  Number and dia of bar for - Top bar
  - Bottom bar
  - Extra bars
- Story - select the suitable story at the beam level.

- **Set Properties** (Fig 6.1.5, 6.1.6)

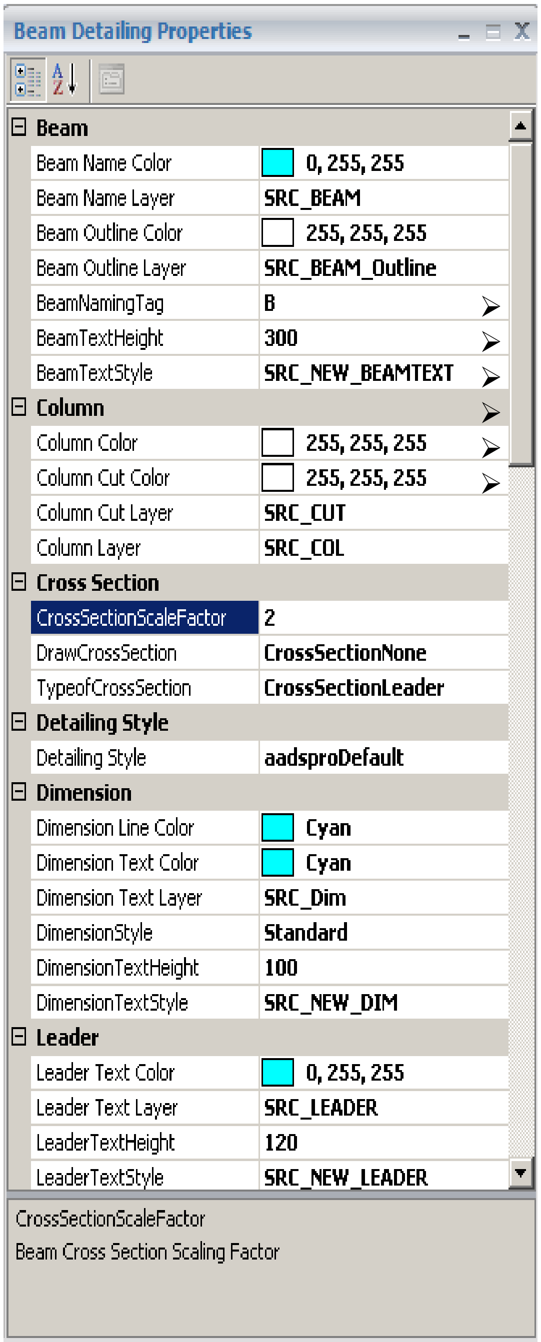


Fig 6.1.5

Beam
- Beam Name Color
- Beam Name Layer
- Beam outline color
- Beam outline Layer
- Beam Naming Tag
- Beam Text height
- Beam Text Style

} ` As per project layout

Column
- Column Color
- Column Cut Color`
- Column Cut Layer
- Column Layer

Cross section
- Cross section scale factor
- Beam cross section scaling factor
- Draw cross section
- Cross section none - No cross section.
- Cross section support - Cross section is created in two support point of a beam.
- Cross section mid - Cross section is created in the midpoint of a beam.

Type of cross section

Cross Section Leader: Cross section is created with multi leader.

Cross Section Simple: Cross section is created with simple number notations.

DETAILING STYLE

Aadspro Defaults

Default value can change from main menu-tools-settings – addspro properties.

Style1
Style2
Style3
Custom

} Can select suitable style.

User can change style.

LEADER
- Leader text color
- Leader text layer
- Leader text height

} as per project layout

DIMENSION

- Dimension line color
- Dimension text color
- Dimension text layer
- Dimension style
- Dimension text height
- Dimension text style

} as per project layout

Load combination

Set a load combination in model to use in design process.

MISC

Aggregate Spacing - Set the spacing need to accommodate the aggregate used.

Cover in design – Cover to be used in design process of beam.

Cover in drawing - Cover to shown in the detailing drawing for better viewing.

Development length factor - Development Length factor given to reinforcement.

Extra bar length factor - Length calculating factor for extra bars based on the beam span length i.e., an extra length is calculated and added to the current reinforcement length with the given factor.

Gap Between beam groups -Distance between two beam section.

Gap Between reinforcement - Gap provided in between reinforcement bars.

Show ID –

sets Which type of Beam number is shown in drawing

Show Beam in Model: Displays the id of the beam in STAAD Pro or ETABS model

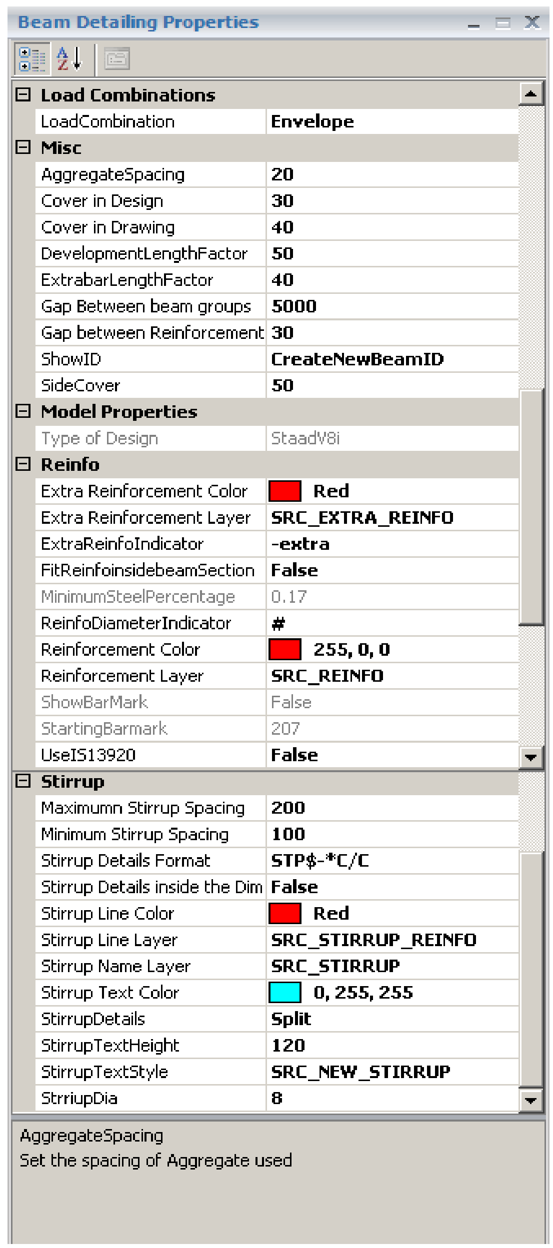


Fig 6.1.6

Create New Beam ID: Create new id starting from 1.
Side cover– Side cover of beam.

MODEL PROPERTIES – Type of data base loaded in beam detailing.
E.g.: Staad V8i
Staad 2004

REINFO
Extra reinforcement color, Layer and indicator can change as per project
Fit reinfo inside beam section -
True – reinforcement cut inside with the beam section.
False – reinforcement up to the development length.
Reinfo Diameter indicator - Set the reinforcement diameter indication tag.
Reinforcement color As per project layout
Reinforcement layer
Use IS13920 - Use IS-13920 Standard in beam detailing.

STIRRUP
Maximum & Minimum stirrup spacing
Maximum & minimum stirrup spacing to be given in detailing.
Stirrup Detail format
In the given format '$' legend replaces the diameter of the stirrup bar and '*' legend replaces the spacing of stirrup.
Stirrup details inside dimension
True – Stirrup details come inside the dimension.
False – stirrup detail come outside the dimension.
Stirrup line color, line layer, text color, text height, text style can change as per project.
Stirrup details – How stirrup details shown in detailing
Grouped – Stirrup details shown as maximum group
Split - Stirrup details shown as splited form.

- Click on the **convert data** from tool bar (Fig 6.1.7)

Convert data button design the beam with the aadspro beam design module. Before going to the conversion user must set input details and also in the property window.

Convert data button converts all the data in selected level and imported to the detailing system. In this process aadspro check all the beam and columns in that level and make the continuity details of the beam and make them into groups.

In detailing using STAAD Pro database aadspro make the reinforcement details with the forces and moments values from the database and used in detailing. aadspro calculate steel details for each section in beam. But in the case of ETABS database aadspro take the steel details directly from the database itself.

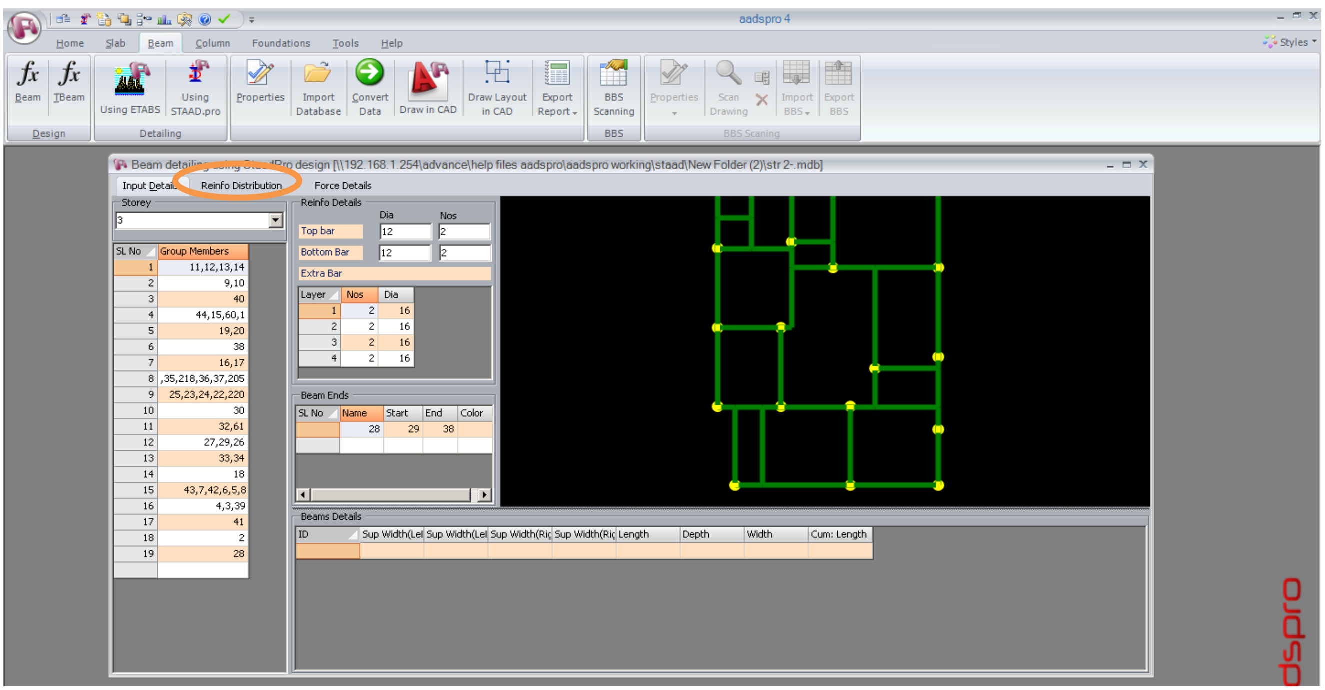
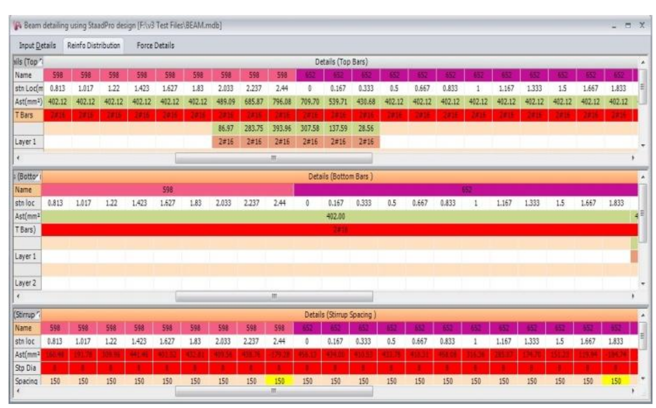


Fig 6.1.7

Reinfo Distribution:

The Reinfo Distribution window helps user to customize the steel distribution of the selected beam group. By selecting any beam group aadspro displays the details of the steel area in each station location. In Reinfo Distribution aadspro displays the steel details of the top, bottom and stirrup details of the selected beam



Force Details:
Force details list all the positive and negative moments, Torsion and shear details to grid in tabular format. This is used for creating design reports.

6.1.4 Results

- Click on the **Draw in Cad** from tool bar (Fig 6.1.8)

Draw in CAD button draws all beam groups listed in the grid just right side of the beam detailing window to AutoCAD. Before drawing to the AutoCAD user can edit the Reinforcement detail for any changes needed. Make sure that Beam Properties must be set before going to draw. User can draw a single beam group by right click the grid and selecting the 'Draw section menu'.

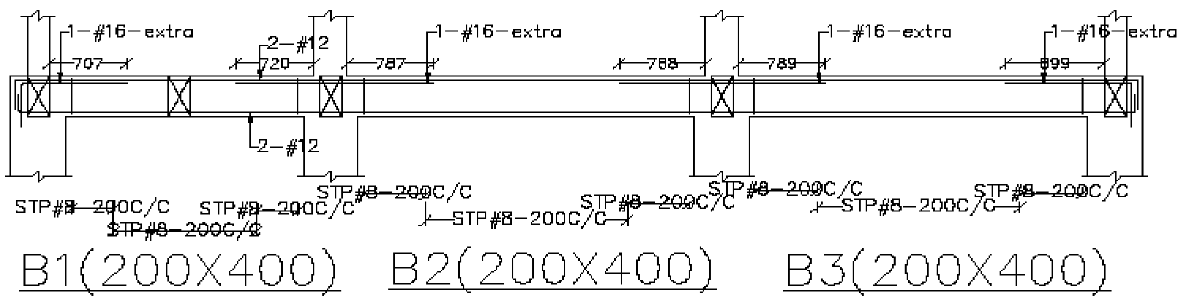


Fig 6.1.8

- Click on the **'Draw layout in cad'** from tool bar (Fig 6.1.9)

This button creates a layout of beam with the beam name in AutoCAD

- Click on the **'Export Report'** from tool bar.
- Export Report button export the details to excel format.

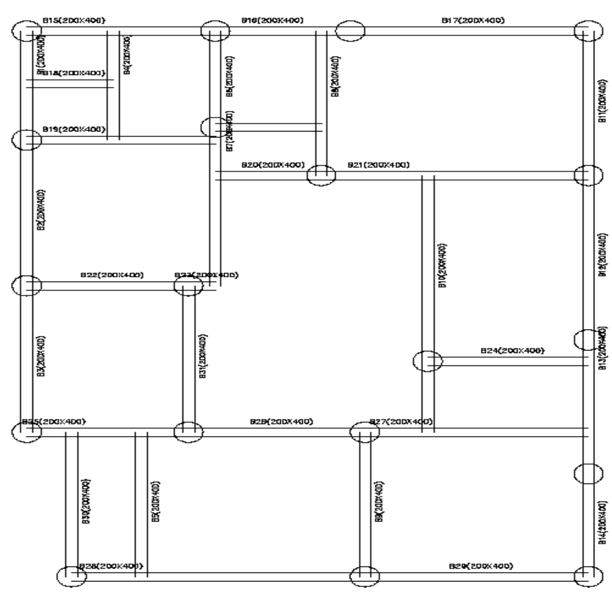


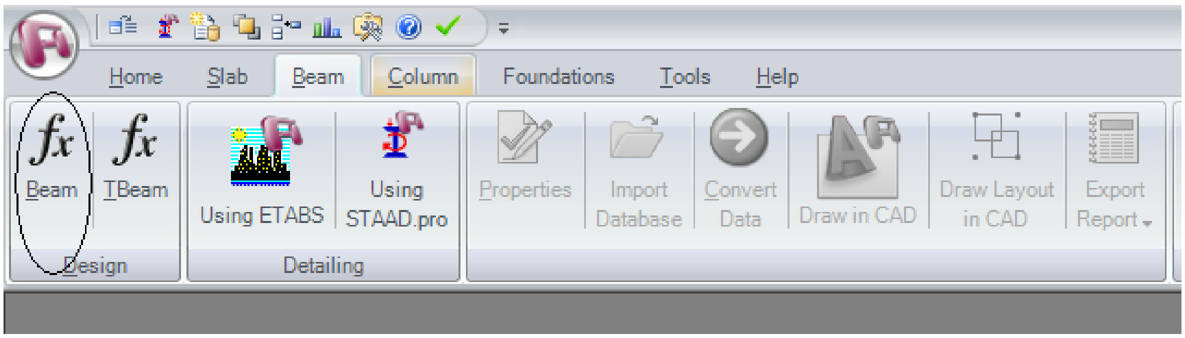
Fig 6.1.9

6.2 Beam Design

This module contains Design of a beam section as per limit state method.

1.1 Steps involved in aadspro

- Open aadspro. Select 'beam' from the Main menu and 'Design' from drop down menu. (fig.6.2.1).



(fig.6.2.1)

- User can set data in beam design grid (fig 6.2.2)

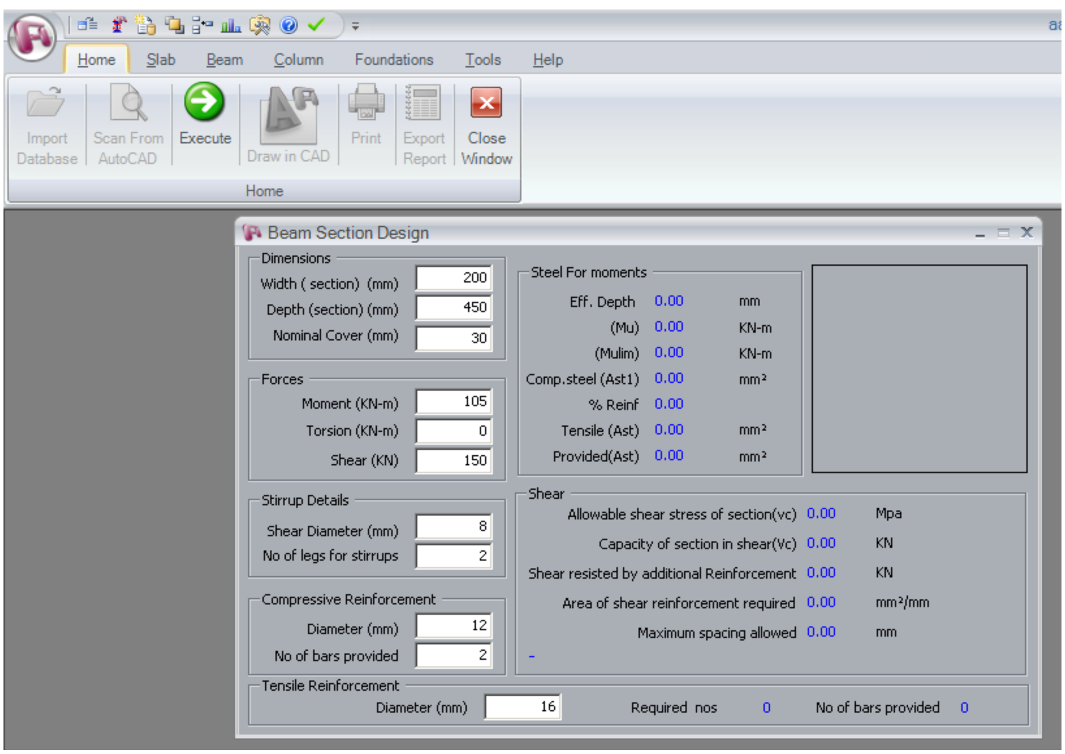
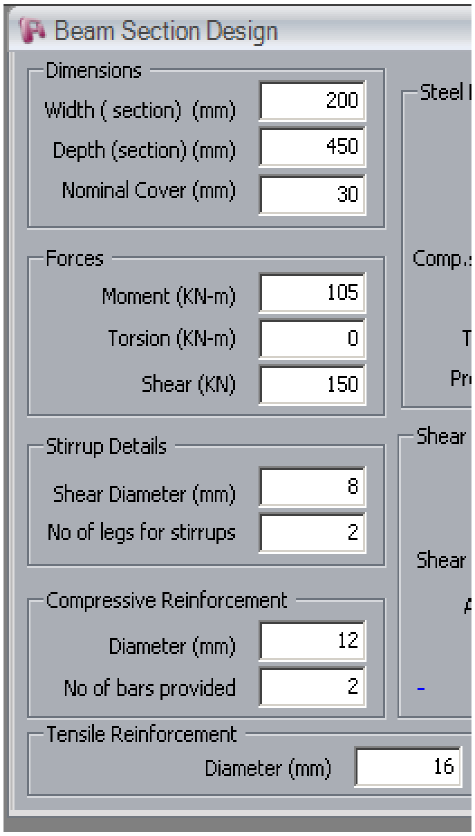


Fig 6.2.2



- In Beam Section Design plate, input the dimensions and clear cover by typing new values in the boxes.
- Similarly input the factored values of moment, shear and torsion.
- Input the reinforcement and clear cover by typing new values in the boxes.
- Program calculates the actual shear stress of the section and compares this with the allowable capacity. Minimum shear reinforcement or additional shear reinforcement, if required any, will also be provided.

Consider the examples given below and its solutions by manual calculation. Explanations are also attached after solutions to see how the programme arrives at results.

<u>Example 1</u>

Determine the reinforcement required for a rectangular beam section with the following data.

Width: 200mm

Depth: 500mm

Factored B.M: 310kNm

Factored torsional moment: 10 kNm

Factored shear force : 150 kN

Use M25 grade concrete and Fe 415 steel.

Solution

Overall depth = 500 mm

Assuming 25 mm dia. bars of two layers at a clear cover of 30 mm and 10mm dais stirrups,

Vertical distance between longitudinal bar is 20mm.

Effective cover, d' = 30 +5+25+20+(25/2)

= 92.5mm

Effective depth, d = 500 – 92.5

= 407.5 mm

X_{umax}/d = 0.48 (As per Cl: 38.1 of IS 456-2000)

X_{umax} = 0.48 x 407.5

= 195.6 mm

Longitudinal reinforcement

Equivalent bending moment, M_{e1} = $M_u + M_t$

Where, M_t = $T_u\left[\dfrac{1+\dfrac{D}{b}}{1.7}\right]$

Mt = $10\times10^6\left[\dfrac{1+\dfrac{500}{200}}{1.7}\right]$

= 20.59 kNm

Since $M_u > M_t$, calculate M_{e1} as per IS 456-2000 Cl: 41.4.2., otherwise calculate M_{e2} as per IS 456-2000 Cl : 41.4.2.1.

∴ M_{e1} = $310+20.59$

= 330.59 kNm

Limiting moment of resistance, M_{ulim} = $0.36(x_{umax}/d)(1-0.42x_{umax}/d)bd^2f_{ck}$

= $0.138\ f_{ck}bd^2$ (for fy415)

= $0.138 \times 25 \times 200 \times 407.5^2$

= 114.58 KNm

Percentage of steel at limiting

Condition, p_{tlim} = $41.4(f_{ck}/f_y)(x_{umax}/d)$

= $41.4 \times (25/415)\ 0.48$

= 1.197%

Area of steel at limiting condition, A_{stlim} = $(p_{tlim}bd)/100$

= $1.197 \times 200 \times 407.5/100$

= $975.64mm^2$

Calculation for tensile steel

Additional moment, M_{u2} = $M_{e1}-M_{ulim}$

= 330.59 – 114.58

= 216.01 KNm

Area of additional steel, Ast_2 = $M_{u2}/(0.87fy(d-d'))$

= 216.01 x 10^6 / (0.87 x 415(407.5-92.5))

= $1899.305\ mm^2$

Total tensile steel, A_{st} = $A_{stlim}+A_{st2}$

= 975.64 +1899.305

= $2874.95\ mm^2$

No of bar required = Ast / (Area of bar used)

= 2874.95 / (490.625)

= 5.85 nos

Hence provide 6 bars of 25 mm dia in tension side (A_{st} = 2943 mm^2)

Calculation for compression steel

Stress in concrete, f_{cc} = $0.446f_{ck}$

= 0.446 x 25

= 11.15N/mm^2

Stress in compression steel, f_{sc} = $700(1-(d'/x_{umax}))$

= 700(1-(92.5/ 195.6))

= 368.967 N/ mm^2

Here, d'/d = 92.5 / 407.5

= 0.22

Maximum value of f_{sc} (for fy 415) = 353 N/ mm^2 (from SP 16, Table F)

Note: For fy 250 steel the maximum value of f_{sc} is 0.87fy

Area of compression steel, A_{sc} = $M_{u2}/ ((f_{sc}-f_{cc})(d-d'))$

= (216.01 x 10^6)/ ((353-11.15) x (407.5-92.5))

= 2003.63 mm^2

No. of bar required in comp. side = A_{sc} / (Area of bar used)

= 2003.63 / (490.625)

= 4.08no.s

Hence provide 4 bars of 25mm dia.(A_{sc} = 2003.63 mm^2)

Check for maximum and minimum area of steel:

Minimum area of steel required, A_{smin} = $\frac{0.85bd}{fy}$

= $\frac{0.85x200x407.5}{415}$

= 166.93 mm^2 < A_{st} provided

Maximum area of steel required, A_{max} = 0.04bD

= $0.04x200x500$

= 4000.00 mm^2 > A_{st} provided

Transverse reinforcement

Equivalent shear, V_e = $V_u + 1.6\frac{T_u}{b}$

= $150\times10^3 + 1.6x\frac{10\times10^6}{200}$

= 230 kN

Equivalent shear stress, τ_{ve} = $\frac{V_e}{bd}$

= $\frac{230\times10^3}{200\times407.5}$

= 2.8 N/mm^2 < $\tau_{c,\max}$

(for M25 $\tau_{c,\max}$ = 3.1 N/mm^2)

Shear is safe. Hence revision not required.

% Steel provided, p_t = $100 \times \frac{A_{st}}{bd}$

= $100 \times \frac{2943}{200 \times 407.5}$

= 3.61%

τ_c , From table 19 of IS 456: 2000 for pt = 3.61%,

= 0.92 N/ mm^2

Since $\tau_c < \tau_{ve}$, transverse shear reinforcement is essential.

As per IS 456-2000, Cl: 41.4.3., The area of two legged stirrups should satisfy the equation,

$$A_{sv} = \frac{T_u.S_v}{b_1.d_1\langle 0.87 f_y \rangle} + \frac{V_u.S_v}{2.5 d_1 \langle 0.87 f_y \rangle}$$

$$A_{sv} = \frac{10 \times 10^6.S_v}{115 \times 415 \times \langle 0.87 * 415 \rangle} + \frac{150 \times 10^3.S_v}{2.5 \times 415 \times \langle 0.87 * 415 \rangle}$$

Assuming 10 dia stirrups,

2 x 78.54 = 0.58 S_v + 0.4 S_v

S_v = 160 mm

Also, area of transverse reinforcement should satisfy the following condition,

$$A_{sv} = \frac{\langle \tau_{ve} - \tau_c \rangle b.S_v}{0.87 f_y}$$

$$2 \times 78.54 = \frac{\langle 2.8 - 0.92 \rangle 200.S_v}{0.87 \times 415}$$

S_v = 150.83 mm

<u>Choosing the minimum, provide 10 dia stirrups @ 150mm c/c.</u>

As per IS 456-2000, Cl: 26.5.1.7, this spacing should not exceed the least of the following,

i) x_1 = 200-(2x30)+10 = 150mm

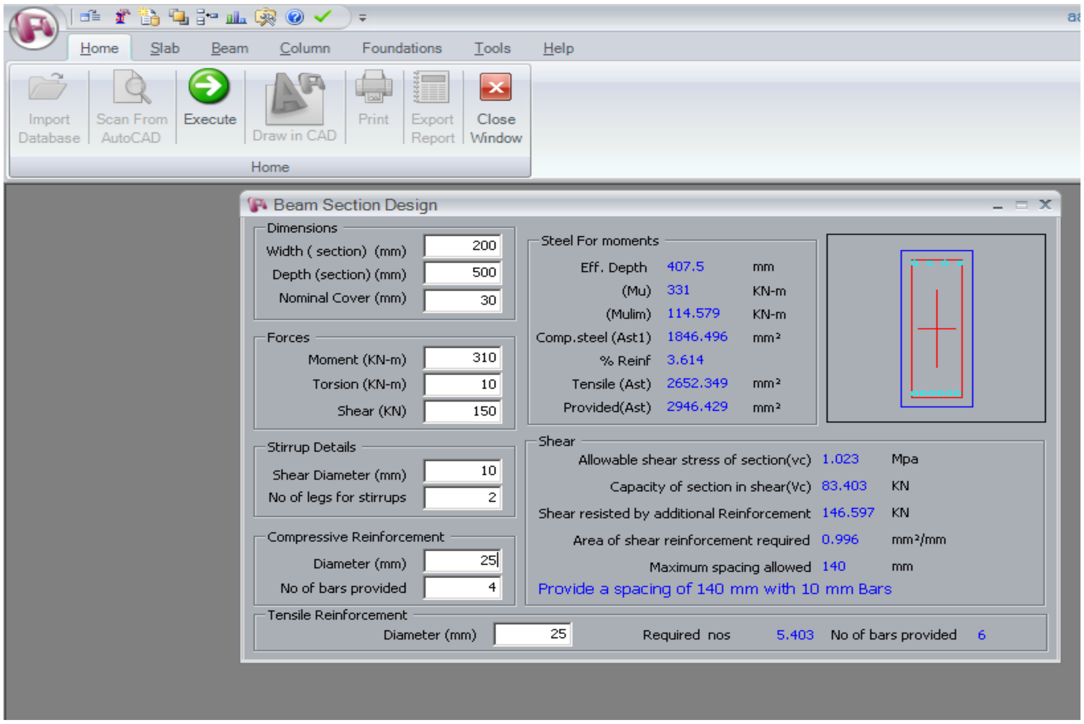
ii) $\frac{x_1 + y_1}{4}$ = $\frac{150+450}{4}$ = 150mm

iii) y_1 = 500-(2x30)+10 = 450mm

Hence provide 10mm dia 2 legged stirrups @ 150 mm c/c.

Now let us see how the program arrives at results.

1. First enter the Characteristic values of concrete and steel. Here this values are f_{ck} = 25 and f_y = 415.
2. Enter the dimensions of the beam section. Given width = 200 and overall depth = 500. Assuming25mm dia bars and 10mm dia stirrups at a clear cover of 30mm, effective depth = 407.5mm. The programme automatically calculates the effective depth with respect to the area of tensile reinforcement (provided) and this is displayed just above the message boxes.
3. Here, select 25mm dia bars as compressive reinforcement, 25 mm dia bars as tensile reinforcement and 10mm dia stirrups as shear reinforcement.
4. Enter the values of moment = 310 kNm, Torsion = 10 kNm and Shear force = 150KN.
5. Cross sectional details of the beam is neatly shown in the diagram attached to the design plate.



Now let us see how the program arrives at results.

1. Program automatically calculates the moment and shear. A comparison with manual calculation is shown below.

	Values from Program	Values by manual Calculation
Effective depth	407.5mm	407.5mm
Mulim	114.58kNm	114.58kNm
Area of tensile reinforcement	2946.429mm^2	2874.95mm^2
Spacing of stirrups	140 mm	150 mm

Here almost all values are exactly same as that of manual calculation.

6.3 BBS (Bar Bending Schedule) of –BEAM

This chapter provides a method for getting the bar bending schedule of existing beams. This can be used to change the existing bar bending schedule of beams.

6.3.1 In AutoCAD

Beam detail is provided in AutoCAD as shown in the fig 6.3.1

- Stirrup details should be in #8-220 formats, the entire leader should touch the line. Beam naming tag should be there.

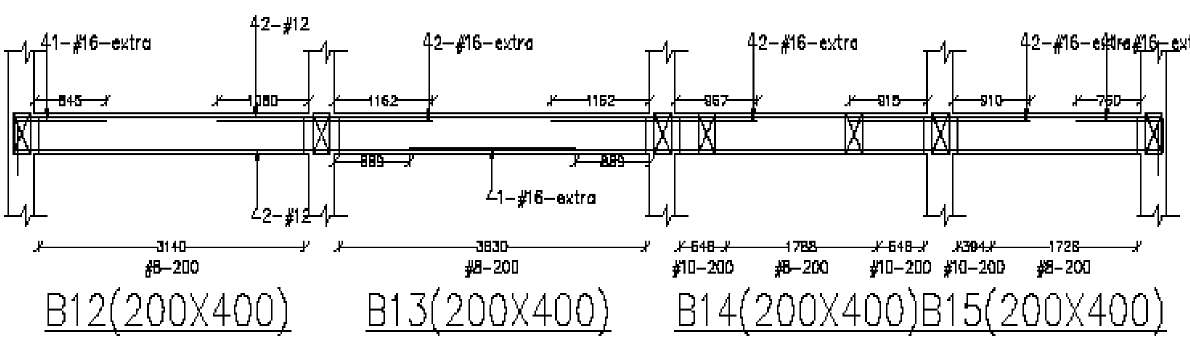


Fig 6.3.1

6.3.2 In aadspro

Select 'Beam' from the Main menu and 'Beam BBS Scan' from drop down menu. Beam BBS Scan window will be displayed as below (Fig 6.3.2)

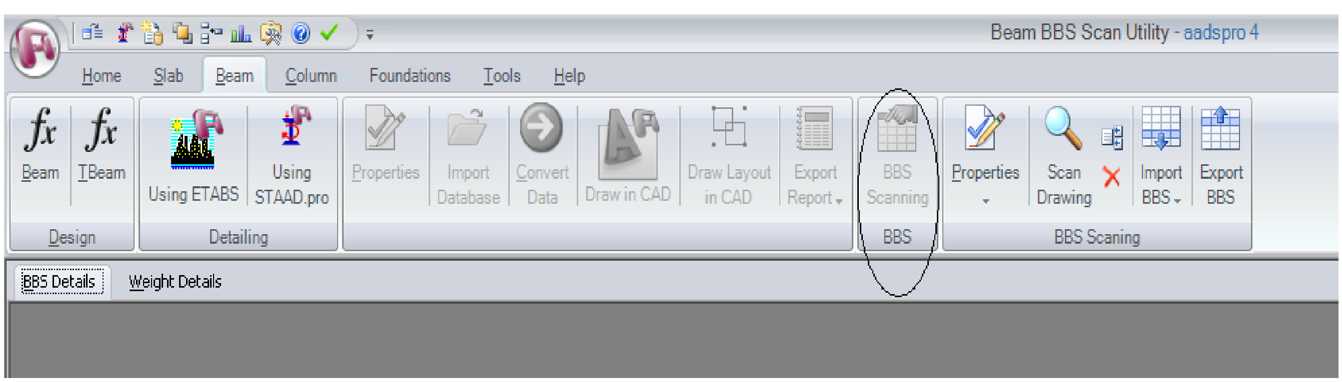


Fig 6.3.2

6.3.3 Click on the **properties** from tool bar (fig 6.3.3)

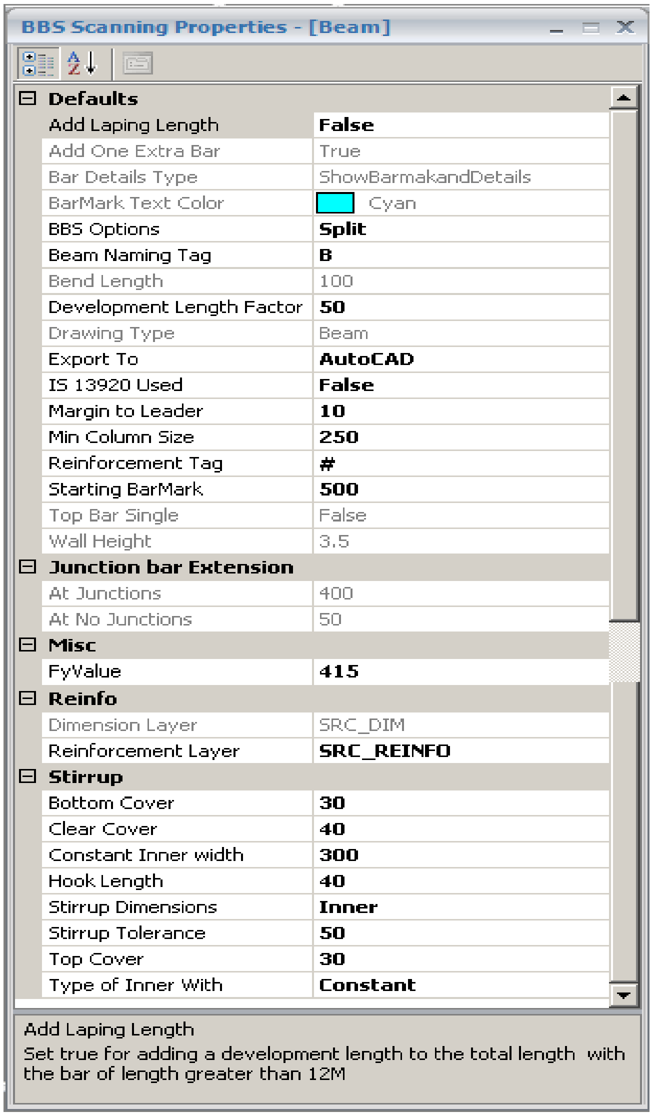


Fig 6.3.3

- ➢ Default
- Add lapping length

Set true for adding a development length to the total length with the bar of length greater than 12m.

- BBS Option

Bar bending schedule can 'Grouped' or 'split' type

- Beam Naming Tag

Beam naming tag used in drawing

- Development length factor

Development length factor used

- Export to

Bar bending exporting option, 3 options are there

1. Auto Cad
2. Simple Excel
3. Excel with picture.

- IS 13920 Used

Set true for if IS 13920 is used.

- Margin to Leader

Setting the tolerance of the leader position with the reinforcement line in drawing

- Min column size

Minimum size of the column:

- Reinforcement Tag

Reinforcement tag used in drawing

- Starting Bar mark:

Starting bar mark to be set for bar bending schedule

Misc

- Fy value

Set fy value

- ➢ Reinfo
- Reinforcement layer

Layer of reinforcement drawn in Auto Cad

- ➢ Stirrup
- Bottom cover given in beams
- Clear cover
- Constant inner width

- Constant width for inner stirrups in 4 Legged stirrups
- Hook length
- Hook length multiplier

- Stirrup dimensions
- Calculate the stirrup dimension with inner or outer
- Stirrup tolerance
- Minimum stirrup tolerance in mm
- Top cover
- Top cover given in beams
- Type of inner width
- Sets the width type of inner stirrups in 4 Legged stirrups
- Constant- will set the value in inner stirrup constant width property.
- As per width- will give the 1/3 value of the width after reducing the clear cover of the beam.

- Click on the **Scan drawing** from tool bar (fig 6.3.4)

The scan Beam Drawing button help the users to select the beam area from layout drawing in AutoCAD. Before scanning the layout the drawing must formatted to suit with aadspro scanning process.

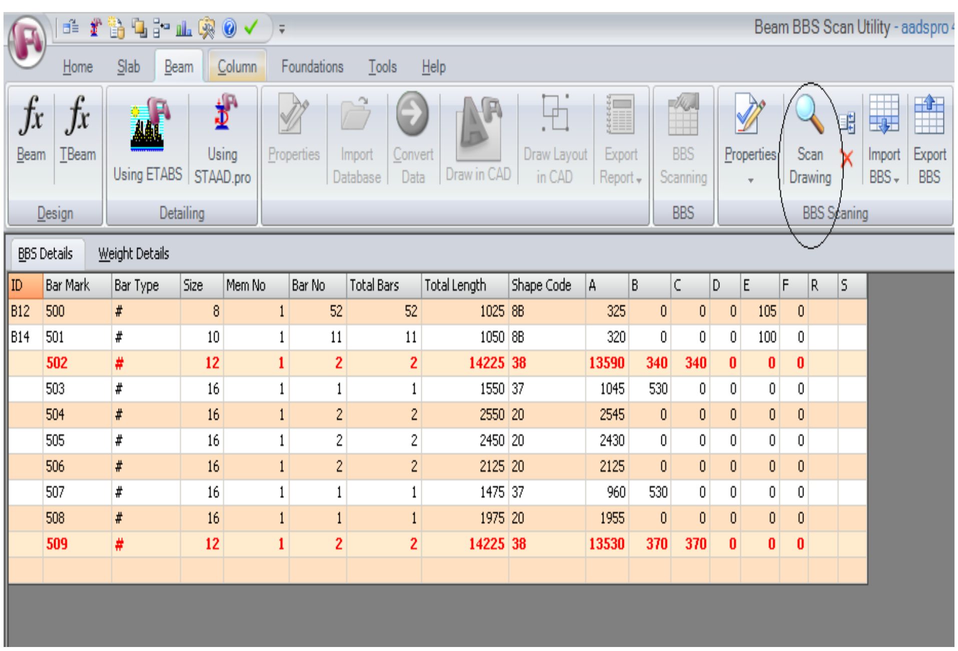


Fig. 6.3.4

Here,

- Bar mark = Bark mark given in Result (500,501 etc as shown in Fig 6.3.4)
- # = Type of bar.
- Size = Bar dia size
- Member No = No of repetitions of the member.
- Bar No = No of bars of same length and dia.
- Tot Bars (Total Bars) = Member No x Bar No
- Len(Bar)mm = Length of bar in mm
- Shape code = Shape code depends on the code selected, British, Indian & American.
- A, B, C etc = Representations in shape code.

SHAPE CODES

As per British code

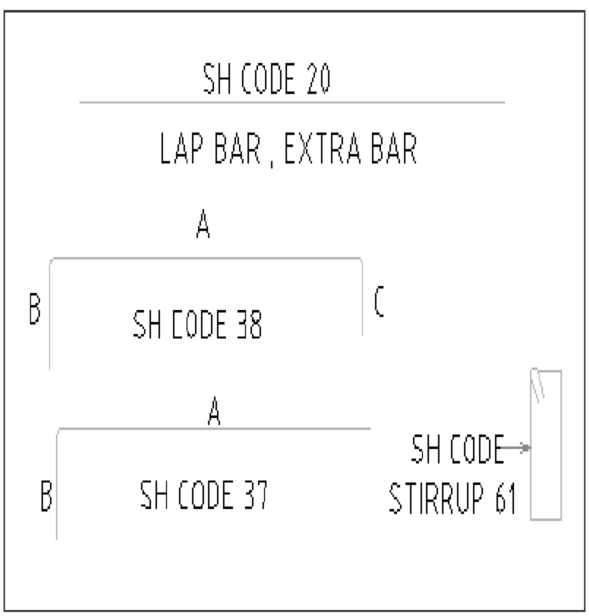


Fig 6.3.5

As per Indian code

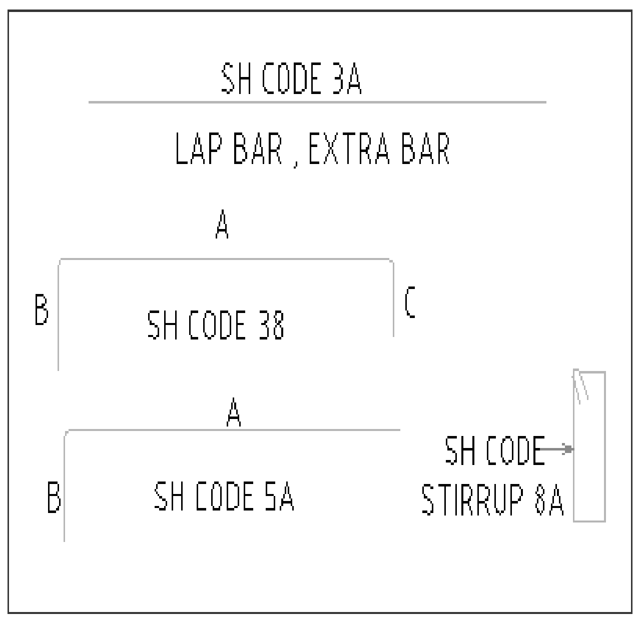


fig 6.3.6

1.3 Results

- Bar Bending Schedule (Fig 6.3.7 & Fig 6.3.8) is obtained for the drawing in AutoCAD
- Here 1-#16-503 means 1 number 16 mm dia bar and 503 represents bar no for BBS.
- 2L-17-#8-500-200 means 2 legged 17numbers of 8 mm dia stirrups @ 200 mm spacing and 500 represents bar number for BBS.

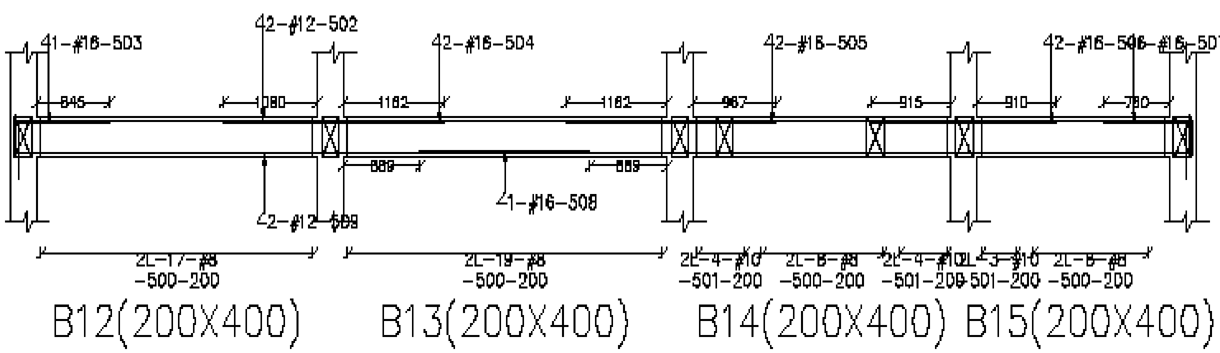


Fig 6.3.7

Bar Bending Schedule

ID	Bar Mark	Bar Type	Size	Mem No	Bar No	Total Bar	Total Length	Shape Code	A	B	C	D	E	F	R	S
B12	500	#	8	1	52	52	1025	88	325	0	0	0	105	0		325, 105
B14	501	#	10	1	11	11	1050	88	320	0	0	0	100	0		320, 100
	502	#	12	1	2	2	14225	38	13590	340	340	0	0	0		340, 13590, 340
	503	#	16	1	1	1	1550	37	1045	530	0	0	0	0		1045, 530
	504	#	16	1	2	2	2550	20	2545	0	0	0	0	0		2545
	505	#	16	1	2	2	2450	20	2430	0	0	0	0	0		2430
	506	#	16	1	2	2	2125	20	2125	0	0	0	0	0		2125
	507	#	16	1	1	1	1475	37	960	530	0	0	0	0		960, 530
	508	#	16	1	1	1	1975	20	1955	0	0	0	0	0		1955
	509	#	12	1	2	2	14225	38	13530	370	370	0	0	0		370, 13530, 370

Fig 6.3.8

7. SLAB

7.1 Slab Detailing

This chapter provides a step-by-step tutorial for the detailing of a slab of a multi-storied building.

7.1.1 Description of structure

7.1.2 Steps involved in auto cad

7.1.3 Steps involved in aadspro

7.1.4 Results

7.1.1 Description of the structure

The structure is a double storied building; in which the first floor slab is to be designed. Plan of the building is shown in fig. (7.1.1).

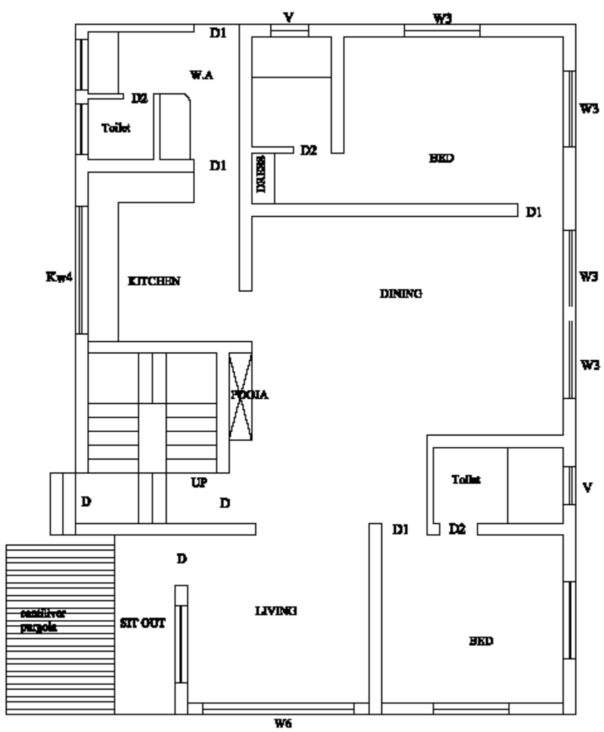


Fig. 7.1.1

7.1.2 Steps involved in auto cad

A layout of the slab is provided in AutoCAD as shown in the fig 7.1.2.

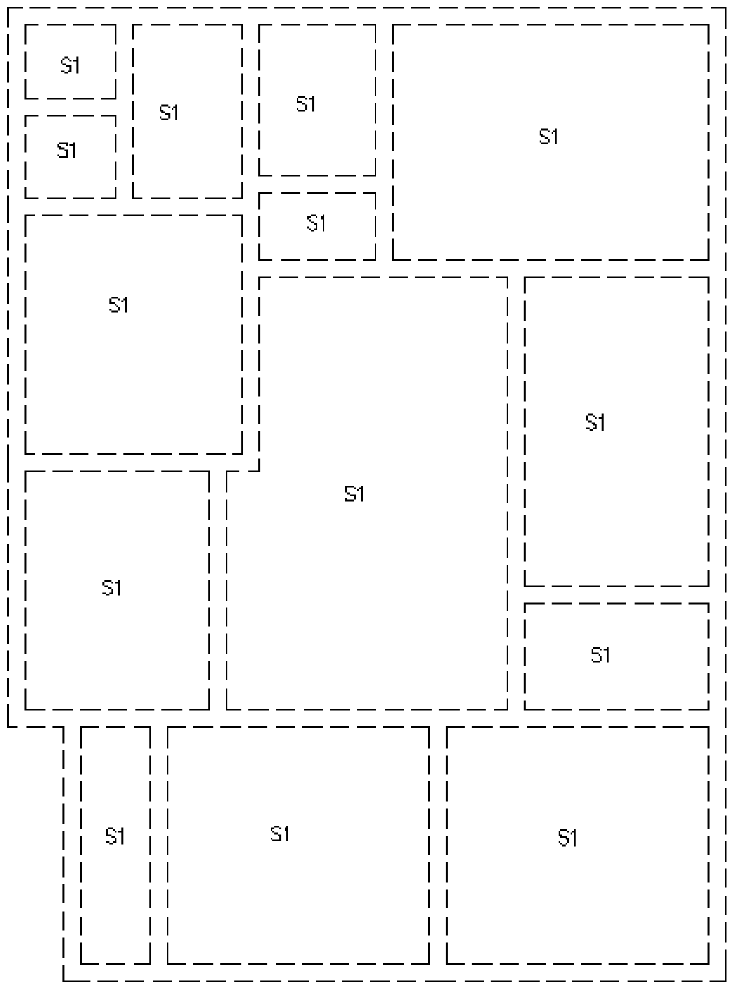
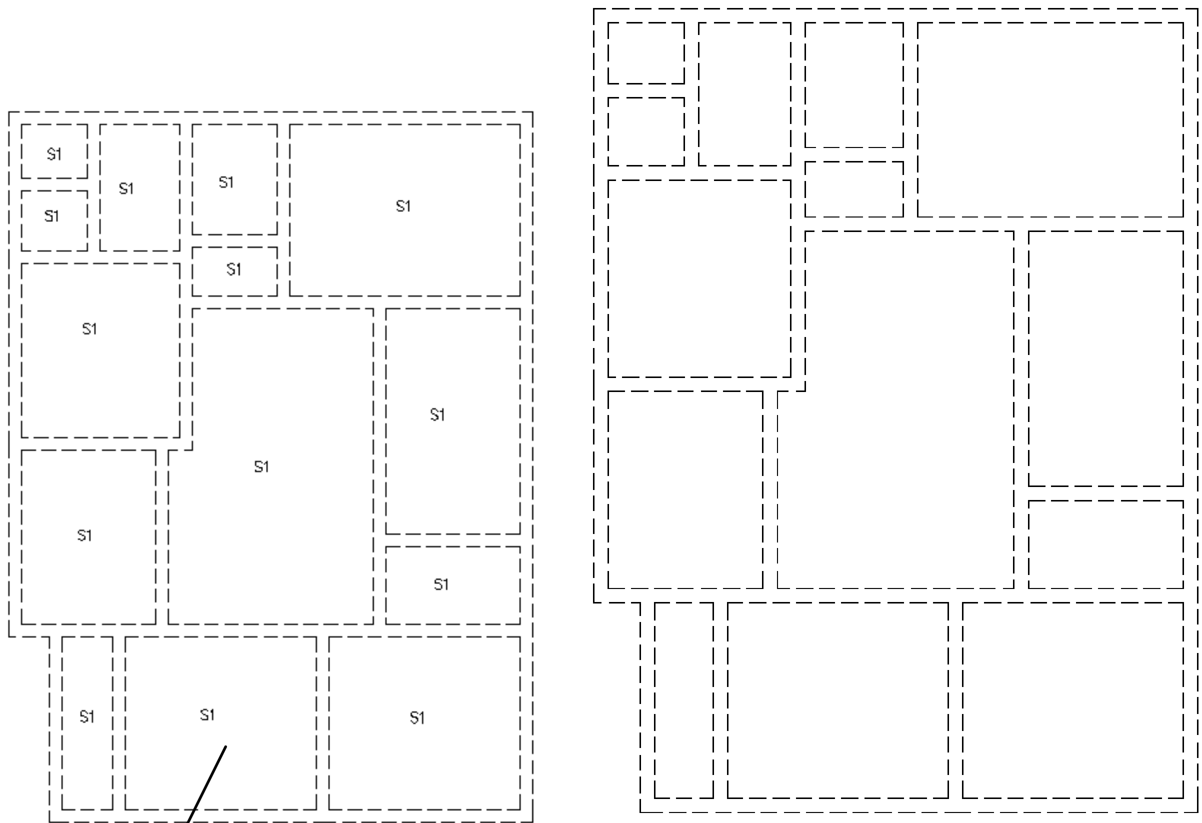


Fig.7.1.2

Make two layouts as shown in fig. 7.1.3



Tag and Depth

Notation	Depth(mm)
S1	100
S1	110
S1	150
S1	150
S1	180

Fig 7.1.3

Layout should be filleted, the Z coordinate must be 0, give any slab name like 'S1' in each slab of one layout, one layout for bottom bar reinforcement and other for top bar reinforcement.

7.1.3 Steps involved in aadspro

Click on the **slab detailing** from tool bar (Fig 7.1.4)

Slab detailing module is powerful tool for generating slab detailing drawing to AutoCAD. In this module aadspro automatically detects the slab area and edge conditions by selecting the layout area in AutoCAD.

This tool helps to minimize the drafting and design time .The simple user interfaces help the users to make the activity more easy and understandable. User can give their own parameters, detailing strategies and detailing style in this process.

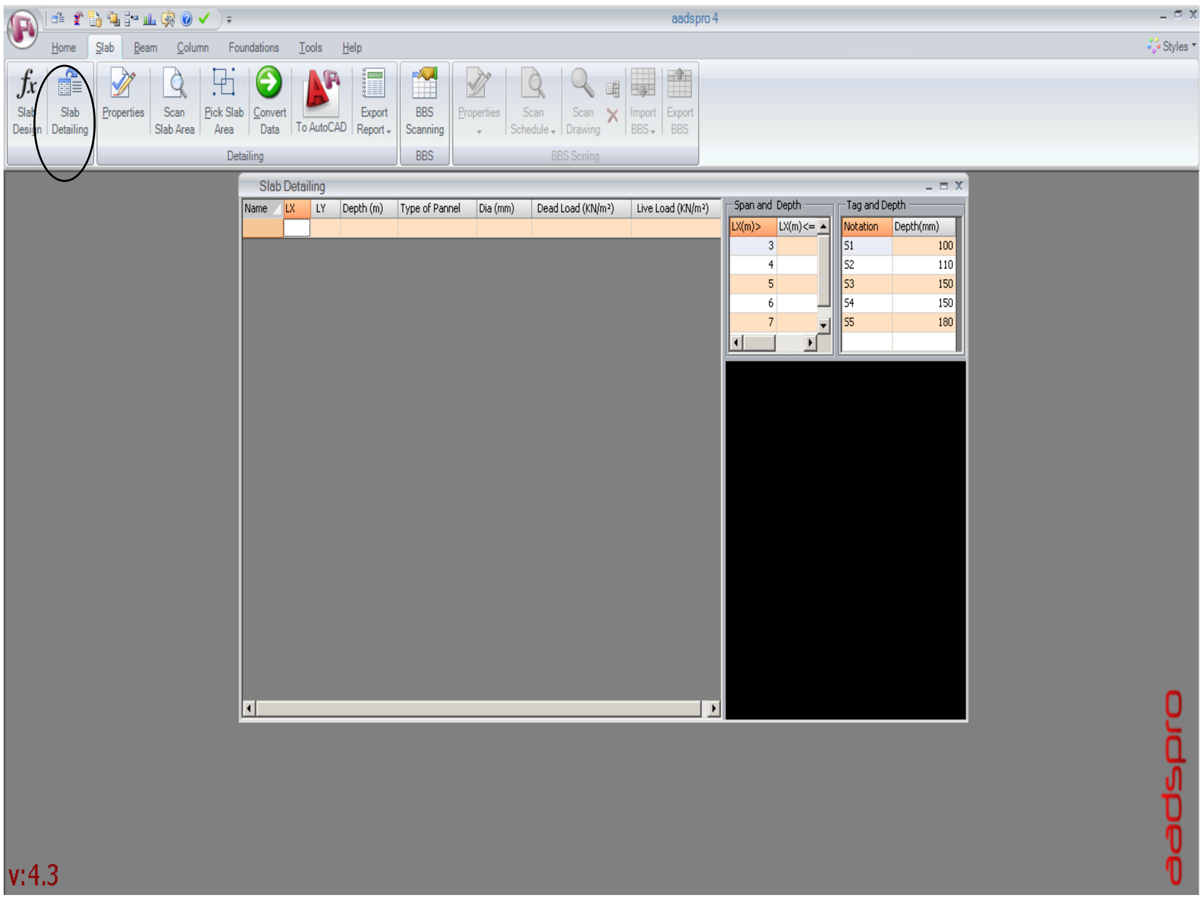


Fig 7.1.4

The layout window helps the user to customize the detailing and design process. By setting these properties user can customize the detailing with variety of option.

- Click on the **properties** from tool bar (Fig 7.1.5)

The property window help the user to customize the detailing and design process. By setting these properties user can customize the detailing with variety of option .User can change the visual appearance of drawing and also the design parameters

➢ **Bottom bar**

Color & Layer of Bottom bars lines drawn in auto cad in detailing process.

- Bottom continuous or discontinuous bar edge length multiplication factor

This property help aadspro to draw the line in the Bottom bar continuous or discontinuous edge. This length is calculated by multiplying the span length with the given factor

eg: if the span length is 3.5 meter and the factor is 0.17 the continuous bar edge length will be 3.5x0.17 = 0.6 m

For discontinuous bar edge length will be 3.5x0.12=0.42m

➢ **Bottom bar Dimension**

Layer & color of the Bottom bars dimension lines drawn in auto cad in detailing process.

➢ **Defaults**

Default value can change from main menu-tools-settings- aadspro properties.

➢ **Miscellaneous**

- Beam width
- Default diameter
- Dimension style

} As per project layout

- Drawing unit
- Live load (factored Kn/m^2)
- Minimum spacing
- Detailing type
- Select suitable detailing type from drop menu

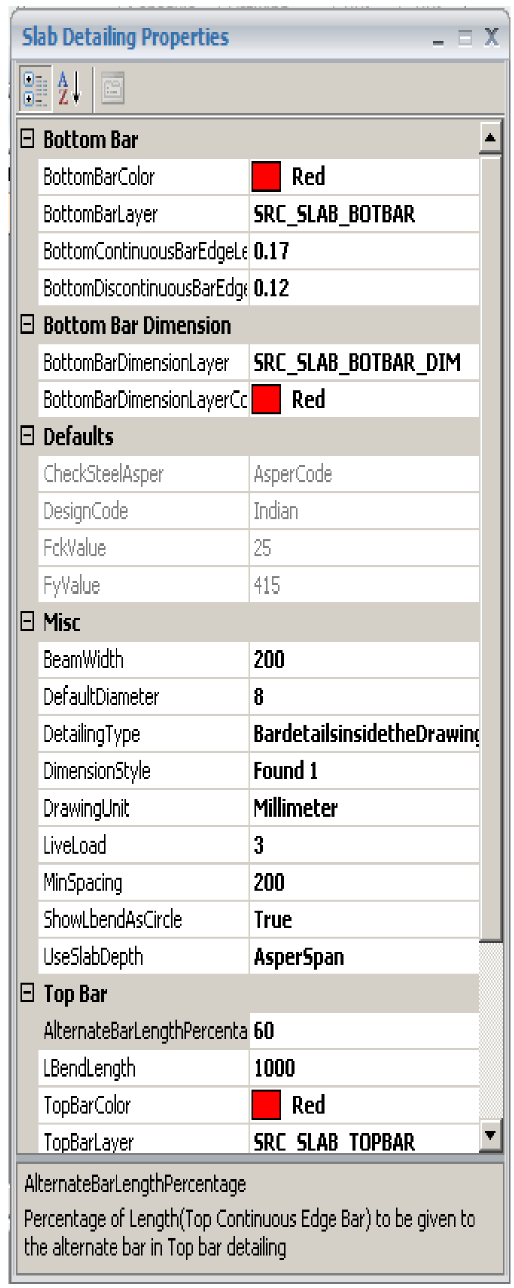


Fig 7.1.5

- Show bend As circle
  Set 'True' to show L bend as a circle
  Set 'False' to show L bend as a L shape
- Use slab depth 1. AsperUserDefinedLX
  This option helps the user to select the depth of slab with the span LX. The show how user can set the depth of slab. In this user can enter the depth value that they need to various LX value.
  For e.g. When a slab has a LX of 3.5 the system will check these entered values and find the desired depths from the given list. Here aadspro take 120 as depth

2.As per User Defined Tag
This option help the user to select the depth of slab with the given notation in AutoCAD At the scanning time. The fig. (b) Show how user can set the depth of slab . In this user can Enter the depth value that they need in various notations. In the scanning process time aadspro
assign these depths to the scanned slab area

3.As per span
In this option aadspro assigns a depth to each slab by checking each slab spans and assign a depth with standard code provisions.

➢ **Top Bar**

- Alternate Bar length percentage
  Percentage of length (top continuous edge bar) to be given to the alternate bar in top bar detailing.
- L Bend Length
  Set the length of the bending bar in the discontinuous edge of the slab
- Top bar color & Layer
  Color & Layer of Top bars lines drawn in auto cad in detailing process.
- Top continuous or discontinuous bar edge length multiplication factor
  This property help aadspro to draw the line in the Top bar continuous or discontinuous edge. This length is calculated by multiplying the span length with the given factor
  eg: if the span length is 3.5 meter and the factor is 0.3 the continuous bar edge length will be 3.5x0.3 = 1.05 m
  For discontinuous bar edge length will be 3.5x0.2=0.7m

➢ **Top Bar Dimension**

Layer & color of the Top bars dimension lines drawn in auto cad in detailing process.

- Click on the **Scan slab area** from tool bar (fig 7.1.6)

The scan Slab button help the users to select the slab area from layout drawing in AutoCAD. Before scanning the layout the drawing must formatted to suit with aadspro scanning process. Users must note the below listed conditions before scanning.

1. The Appropriate notation (eg :S1,S2,S3 etc..) to be placed inside the layout to find out the slab area
2. The enclosing beam lines must be filleted properly to find out the slab area.

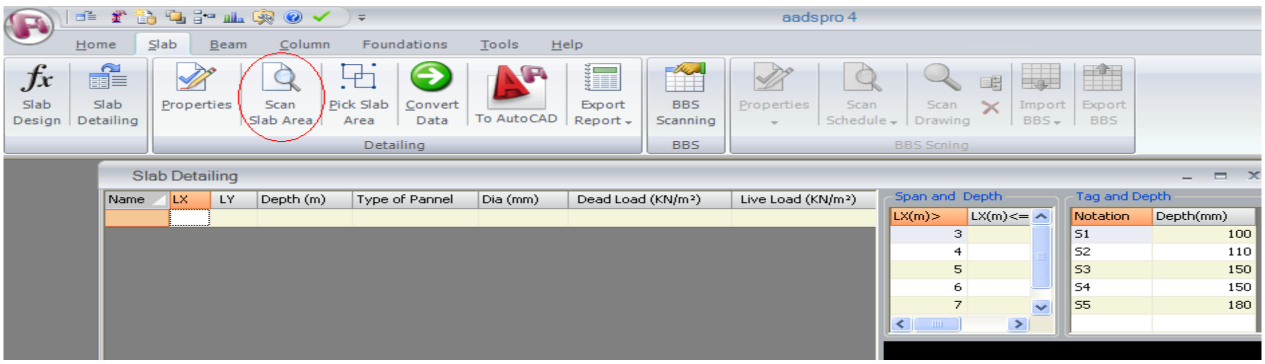


Fig 7.1.6

1.3.3 Click on the **Convert data** from tool bar (fig 7.1.7)

The convert data button in aadspro analyzes all the created slab areas and makes their continuity details and assign edge conditions of each slab area.

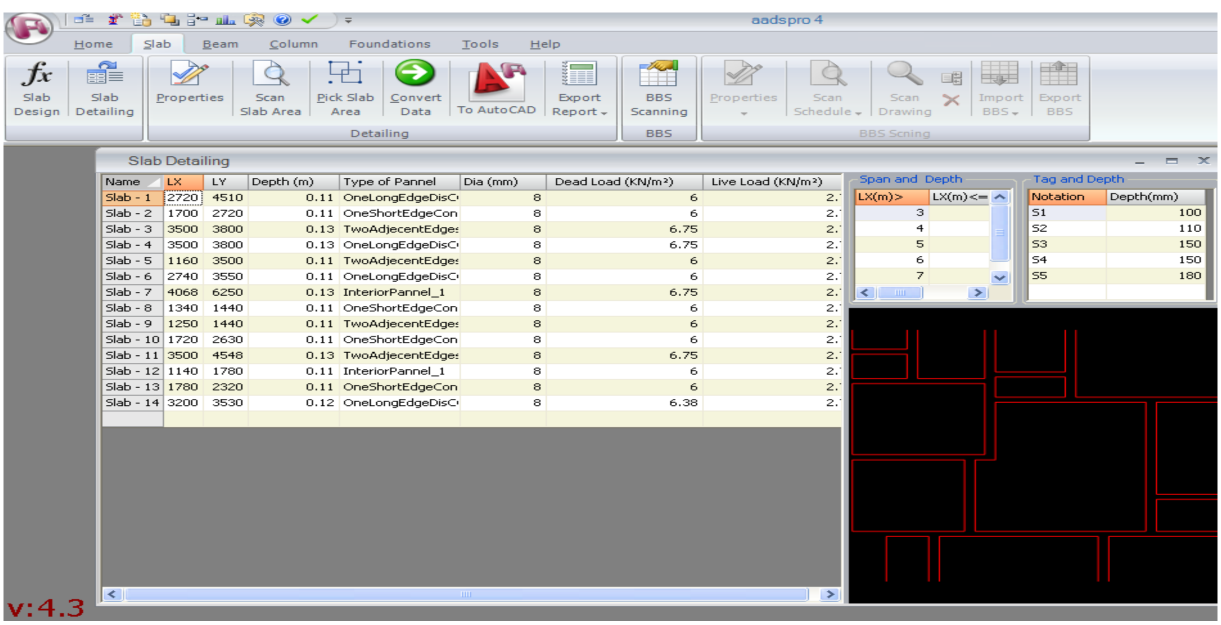


Fig 7.1.7

7.1.4 Results

- Click on the '**To Auto Cad'** from tool bar

The Draw in AutoCAD button will prompt the user to pick two points in AutoCAD to draw the bottom and top detailing of slabs with the current converted data.

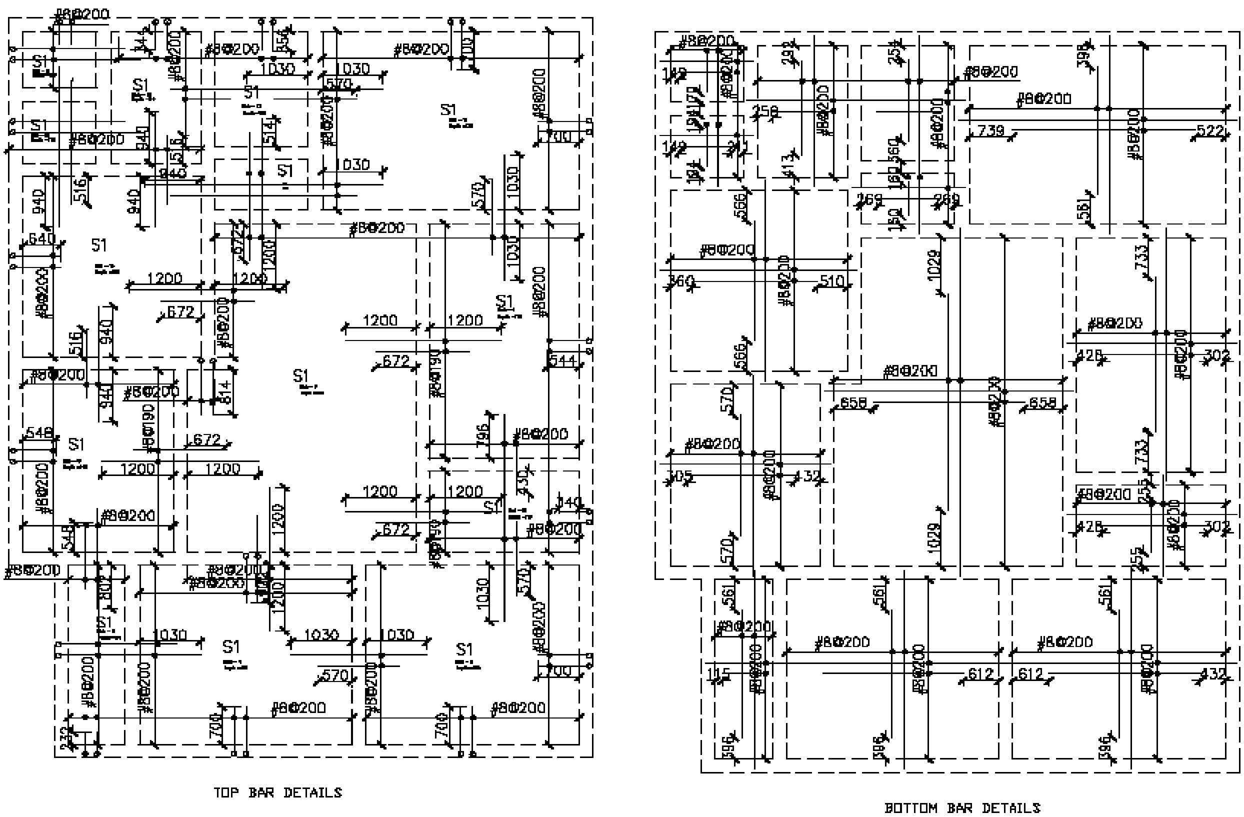


Fig 7.1.8

- Click on the **Schedule** from tool bar

Draw schedule button draws the slab schedule in AutoCAD with designed pile number.

1	#8 AT 200 C/C	11	#8 AT 200 C/C
2	#8 AT 200 C/C	12	#8 AT 200 C/C
3	#8 AT 200 C/C	13	#8 AT 200 C/C
4	#8 AT 200 C/C	14	#8 AT 200 C/C
5	#8 AT 200 C/C	15	#8 AT 190 C/C
6	#8 AT 200 C/C	16	#8 AT 190 C/C
7	#8 AT 200 C/C	17	#8 AT 200 C/C
8	#8 AT 200 C/C	18	#8 AT 200 C/C
9	#8 AT 200 C/C	19	#8 AT 200 C/C
10	#8 AT 200 C/C	20	#8 AT 200 C/C

7.2 Slab Design

7.2.1 Two way slab design

This module contains both Analysis and Design of a slab section as per limit state method.

7.2.1.1. Steps involved in aadspro

Open aadspro. Select 'slab' from the Main menu and 'Design' from drop down menu. (Fig 7.2.1)

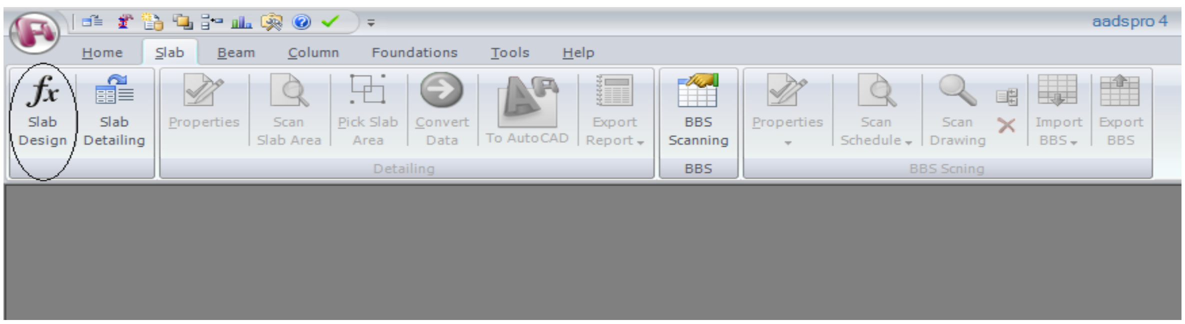


Fig 7.2.1

- User can set data in slab design grid (fig7.2.2)

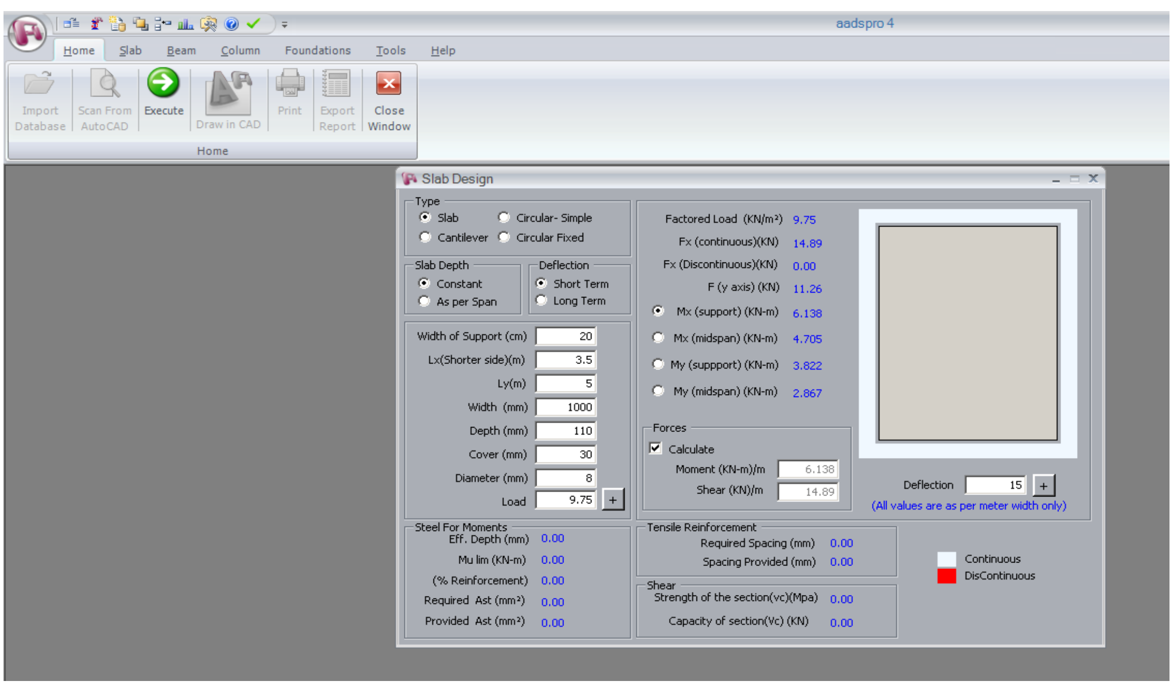


Fig 7.2.2

- Type
  Select type of slab

- Slab depth
  Constant:-
  This option help the user to fix the depth.
  As per span:-
  In this option aadspro assigns a depth to each slab by checking each slab spans and assign a depth with standard codel provisions.

- Deflection
  Select deflection whichever required.
  The corresponding deflection will be shown in deflection box.

- Width of support(cm)
-
- Lx(short side)(m) (effective span shorter direction)

- Ly(m) (effective span longer direction

- Width(mm)

- Depth of slab

- Cover

- Diameter

- Load
  Load detail can set as per project on 'Loads' grid

- Material properties, f_{ck} & f_y is a default value, can be change by clicking Tools-Settings-addspro properties.

Program thoroughly checks the code provisions like max. Diameter of tensile reinforcement, maximum spacing of reinforcement in longitudinal and transverse directions, minimum concrete cover, etc. and gives message alerts if the values are exceeding the limits as per IS: 456 – 2000.

Following illustrations give more idea about the performance of the program.

Consider the example below and its solution by manual calculation.

Example 1

Design a slab over a room 6 m x 4.5m. The slab has a bearing of 150 mm all round at the edges on supporting masonry walls. The corners of the slab are held down. The live load on the slab is 3000 N/ m^2. Use M 15 concrete and Fe 415 steel.

Solution

Distance between the c/c of bearings

in the shorter direction = $4.5 + .15$

= 4.65 m

C/c distance in the longer direction = $6 + .015$

= 6.15 m

Assume over all depth as 1/30 of span = $\left(\frac{1}{30}\right) x 4.65$

= 155mm

Provide depth = 160 mm

<u>Provide a slab of depth160mm with 10 mm dia. bars and clear cover of 25mm</u>

Effective depth (shorter direction) = $160 - 25 - 5$ mm

= 130 mm

Effective depth (longer direction) = $160 - 25 - 5 - 10$

= 120mm

Clear span + eff. depth in shorter direction = 4.5 + 0.130 = 4.63

Clear span + eff. depth in the longer direction = 6.0 + 0.120 = 6.120

$\therefore$ Effective span (Shorter of the two)

In the shorter direction, l_x = <u>4.63m</u>

In the longer direction, l_y = <u>6.12m</u>

Span ratio, l_y / l_x = $6.12 / 4.63$ = 1.32 Say 1.30

Referring to Table 26 of I.S. 456-2000,

Four edges discontinuous condition and span ratio 1.3, we get

$\alpha_x = 0.079$ and $\alpha_y = 0.056$

Load Calculation

Self weight of the slab (160mm)	=	0.16 × 25	=	4.00 KN/m^2
Wt of finishes (20 mm)	=	0.02 × 25	=	0.50 KN/m^2
Live Load (given)			=	3.00 KN/m^2
Total Load			=	7.5KN/m^2

With a partial safety factor of 1.5,

Total factored load = 7.5 × 1.5 = 11.25 N/m^2

Consider a 1 meter wide strip of slab,

Factored short span moment, M_{ux} = $\alpha_x w l_x^2$

= $0.079 x 11.25 x 4.63^2$

= 19.05 KNm

Factored long span moment, M_{uy} = $\alpha_y w l_x^2$

= $0.056 x 11.25 x 4.63^2$

= 13.51KNm

$M_{u, lim}$ = $0.138 f_{ck} bd^2$

= $0.138 x 15 x 1000 x 130^2$

= 34.98KNm

Here $M_{ux} < M_{u,lim}$, Therefore the section is under-reinforced.

Steel for short span

$$\frac{M_{ux}}{bd^2} = \frac{19.05x10^6}{1000x130^2}$$

= 1.13

Percentage of steel = $50\left[\frac{\sqrt{1-\frac{4.6}{f_{ck}}x\frac{M_{ux}}{bd^2}}}{\frac{f_y}{f_{ck}}}\right]$

$$= 50\left[\frac{1-\sqrt{1-\frac{4.6}{15}x1.13}}{\frac{415}{15}}\right]$$

= 0.35%

Minimum percentage of steel = 0.12% < 0.35%

Area of steel, A_{st} = $\frac{0.35}{100}\times1000\times130$

= 455.00 mm^2

Spacing of 10 mm diameter bars = $\frac{78.5\times1000}{455.00}$

= 172.53mm

Therefore provide 10mm ϕ bars @ 170 mm c/c.

Steel for long span

$$\frac{M_{uy}}{bd^2} = \frac{13.51x10^6}{1000x120^2}$$

= 0.94

Percentage of steel = $50\left[\frac{1-\sqrt{1-\frac{4.6}{f_{ck}}\times\frac{M_{uy}}{bd^2}}}{\frac{f_y}{f_{ck}}}\right]$

= $50\left[\frac{1-\sqrt{1-\frac{4.6}{15}\times 0.94}}{\frac{415}{15}}\right]$

= 0.283%

A_{st} = $\frac{0.283}{100}\times 1000\times 120$

= 339.6 mm^2

Spacing of 10 mm diameter bars = $\frac{78.54\times 1000}{339.6}$

= 231.27mm

Therefore provide 10 mm dia. bars @ 230mm c/c

Now let us see how the program arrives at results (Fig 7.2.3)

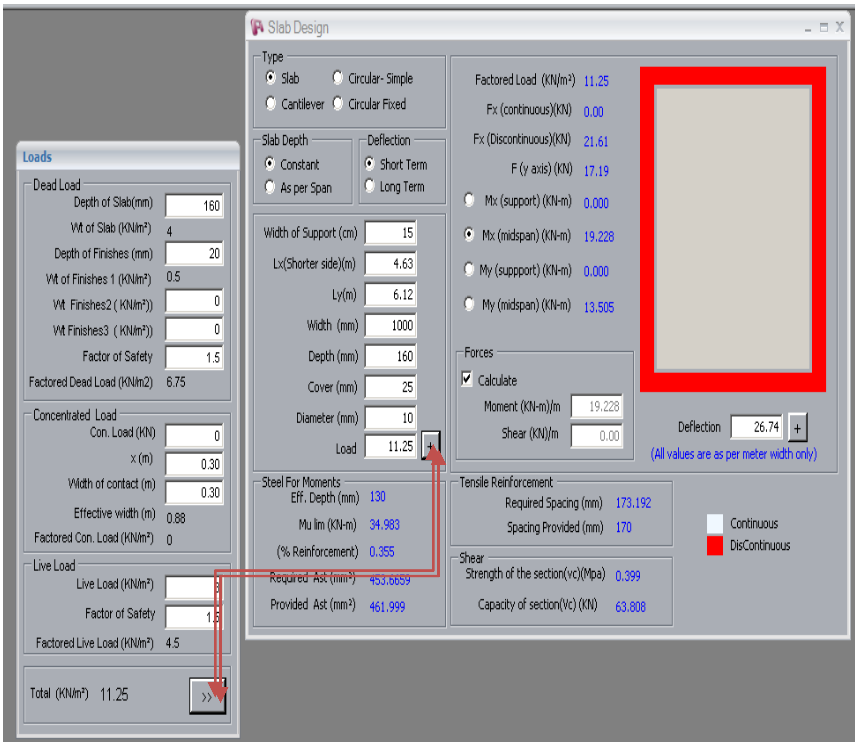


Fig 7.2.3

Now let us see how the program arrives at results.

2. Program automatically calculates the moment and shear. A comparison with manual calculation is shown below.

	Values from Program	Values by manual Calculation
Effective depth	130mm	130mm
Factored load	11.25kN/m^2	11.25kN/m^2
M_{ux}	19.228kNm	19.05kNm
M_{uy}	13.51kNm	13.51kNm
Steel for short span	10mm @170mm c/c	10mm @170mm c/c
Steel for long span	10mm @250mm c/c	10mm @230mm c/c

Here almost all values are exactly same as that of manual calculation.

7.2.2 cantilever slab design

User can set data in slab design grid (fig 7.2.4)

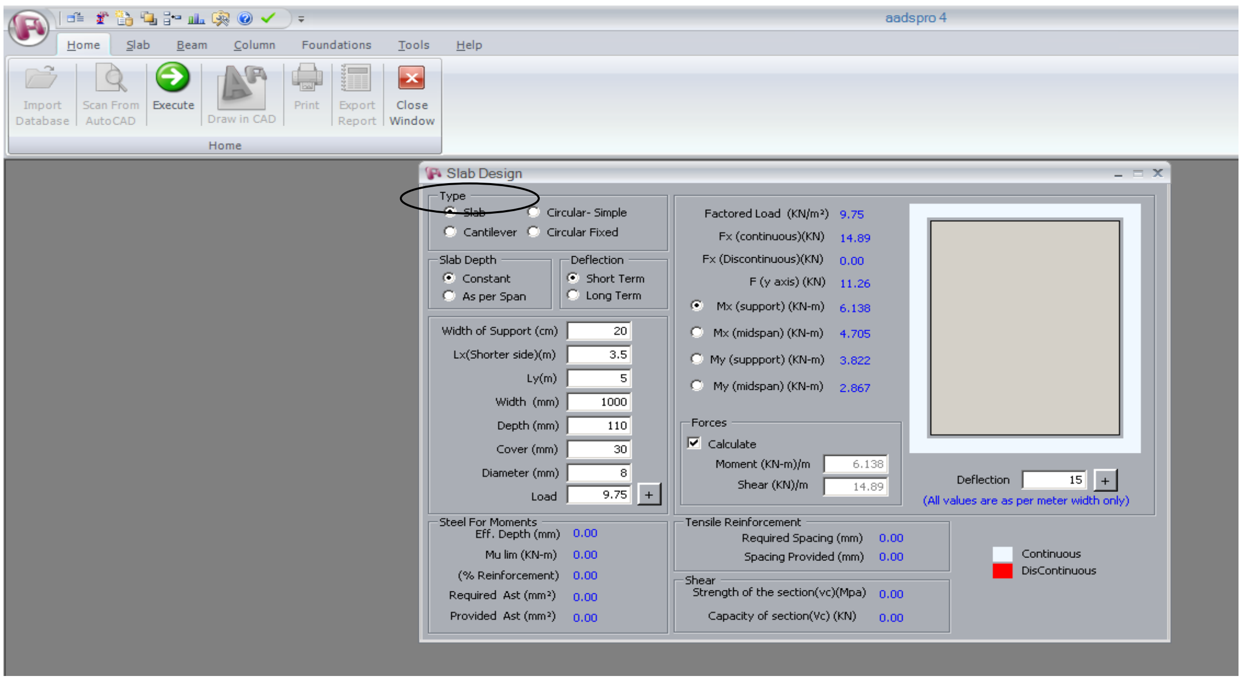


Fig 7.2.4

Example 2:

Design a cantilever slab to carry a live load of 2500 N/m^2. The overhang of the slab is 1.25 m. Use M 15 concrete and Fe 415 steel.

Solution

Provide a slab of depth 100mm with 8 mm dia. bars and clear cover of 15mm

Effective depth = 100-15-4 mm

= 81 mm

Load Calculation

Self weight of the slab (100mm) = 0.10×25 = 2.50 KN/m^2

Wt of finishes (20 mm) = 0.02×25 = 0.50 KN/m^2

Live Load (given) = 2.50 KN/m^2

Total Load = 5.50 KN/m^2

With a partial safety factor of 1.5,

Total factored load = 5.50 × 1.5 = 8.25 N/m^2

Consider a 1 metre wide strip of slab,

Max Bending Moment , M_{ux} = $\frac{wl^2}{2}$

= $\frac{8.25 \times 1.25^2}{2}$

= 6.445 KNm

Mu limit = 0.138 x fck x b x d^2

= 13.58 KNm

(M < Mu limit)

Steel for Max Bending Moment

$\frac{M}{bd^2}$ = $\frac{6.445 \times 1000}{1000 \times (81)^2}$

= 0.98

Percentage of steel

$$= 50\left[\frac{\sqrt{1 - \frac{4.6}{f_{ck}} x \frac{M_{ux}}{bd^2}}}{\frac{f_y}{f_{ck}}}\right]$$

$$= 50\left[\frac{1 - \sqrt{1 - \frac{4.6}{15} x 0.98}}{\frac{415}{15}}\right]$$

= 0.296%

Minimum percentage of steel = 0.12 %< 0.296%

Area of steel, A_{st} $= \dfrac{0.296}{100} \text{x}1000\text{x}81$

$= 239.760 \text{ mm}^2$

Spacing of 8 mm diameter bars $= \dfrac{50.24 \times 1000}{239.76} = 209.54\text{mm}$

Therefore provide 8 mm ϕ bars @ 200 mm c/c.

Now let us see how the program arrives at results. (Fig 7.2.5)

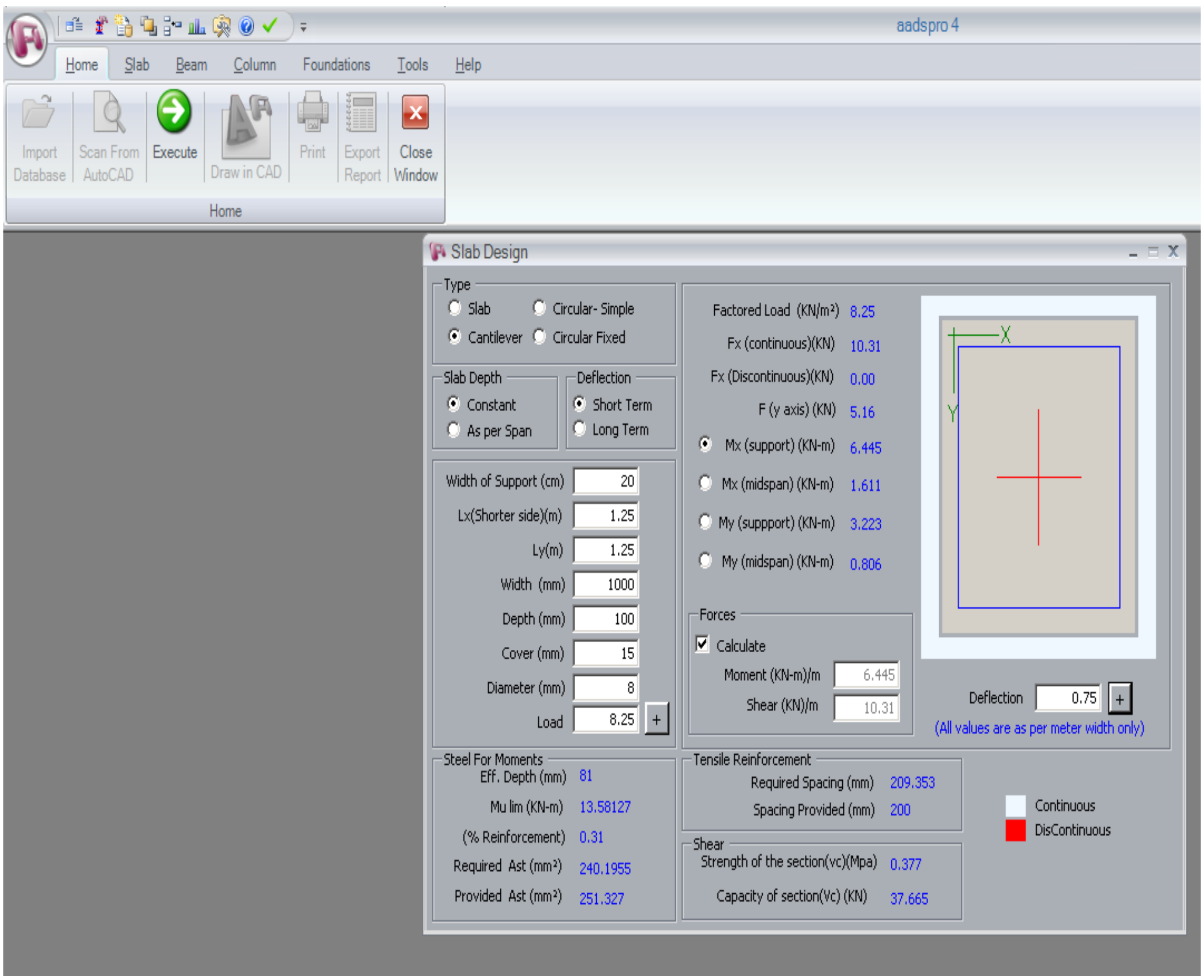


Fig 7.2.5

1. Program automatically calculates the moment and shear. A comparison with manual calculation is shown below.

	Values from Program	Values by manual Calculation
Effective depth	81mm	81mm
Factored load	8.25kN/m^2	8.25kN/m^2
M	6.445kNm	6.445kNm
Steel for Max BM	8mm @200mm c/c	8mm @200mm c/c

Here all values are exactly same as that of manual calculation.

7.3. BBS (Bar Bending Schedule) of –SLAB

This chapter provides a method for getting bar bending schedule of existing slabs. This can be used to change the existing bar bending schedule of slabs.

7.3.1 In AutoCAD

Slab detail is provided in AutoCAD as shown in the Fig 7.3.1

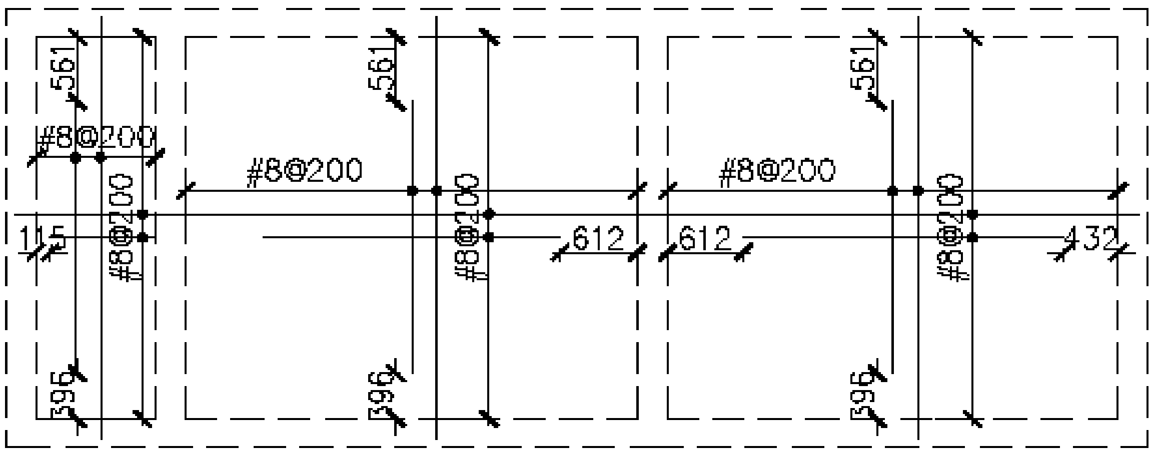


Fig 7.3.1

- There should not be any layer called 'Templayer'.
- The Z coordinate must be zero.

7.3.2 In aadspro

Select 'Slab' from the Main menu and 'Slab BBS Scan' from drop down menu Fig 7.3.2

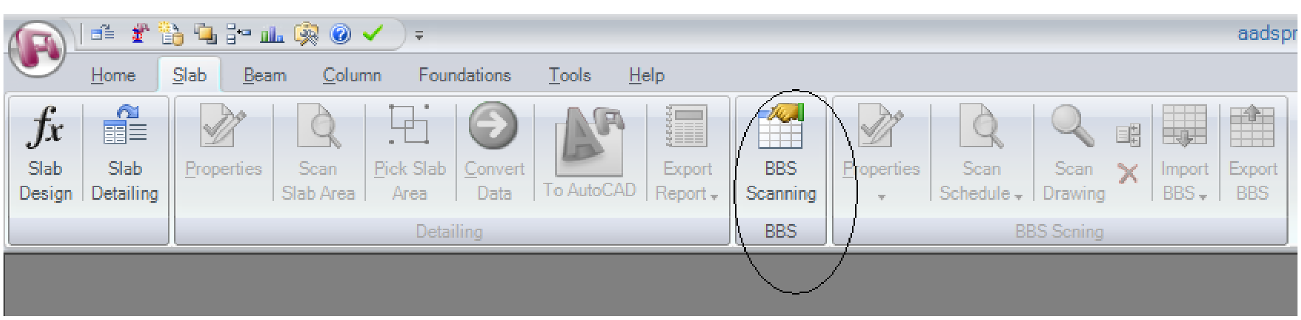


Fig 7.3.2

- Click on the **properties** from tool bar (fig 7.3.3)

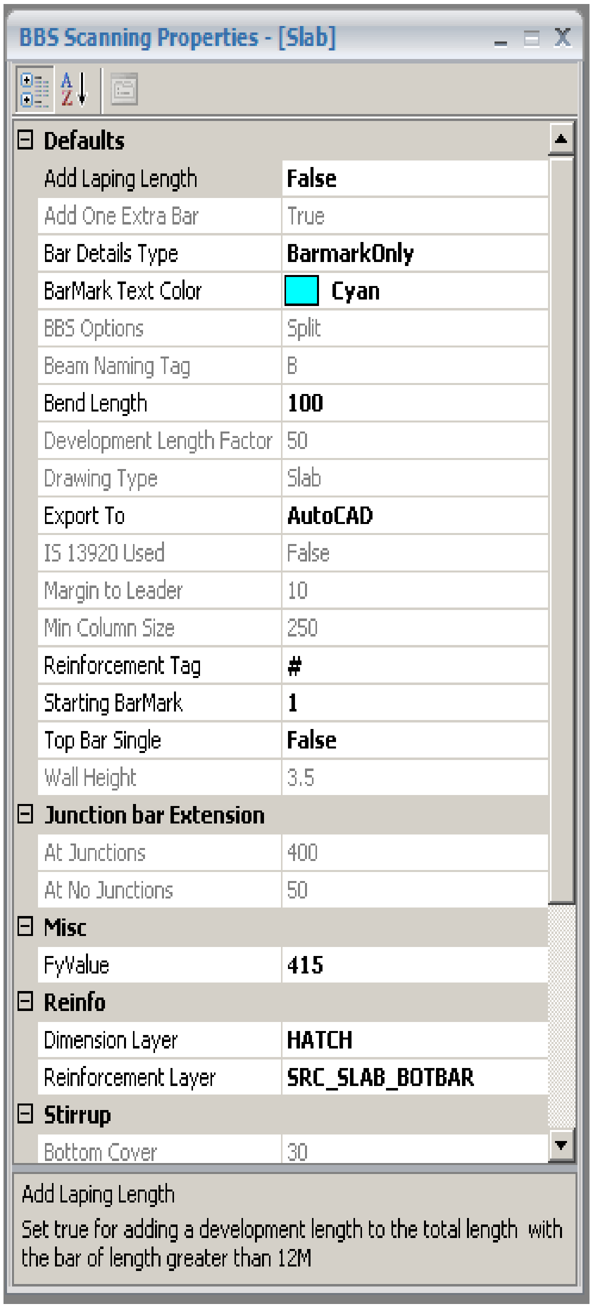
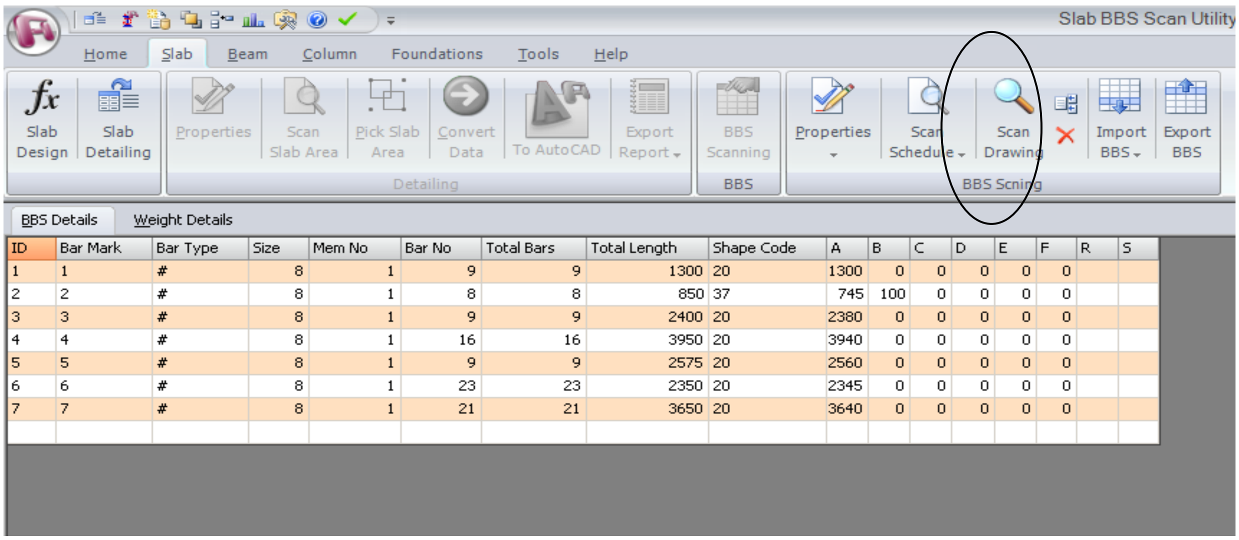


Fig 7.3.3

- ➢ Default
- Add laping length
  Set true for adding a development length to the total length with the bar of length greater than 12m.
- Bar details type
  How bar details are shown after the scanning process.
- BarMark text color
  Set suitable color for bar mark text.
- Bend length
  Length of L bend in slab detailing
- Export to
  Bar bending exporting option, 3 options are there
  1.Auto Cad
  2.Simple Excel
  3.Excel with picture.
- Reinforcement Tag
  Reinforcement tag used in drawing
- Starting Barmark
  Starting bar mark to be set for bar bending schedule.
- Top Bar single
  Set true for Top bar single in slab detailing drawing
  (only for top bar)
- ➢ Misc
- Fy value
  Set fy value
- ➢ Reinforcement
- Dimension layer- As per the project
- Reinforcement layer- As per the project

- Click on the **Scan drawing** from tool bar (fig 7.3.4)

The scan Slab Drawing button help the users to select the slab area from layout drawing in AutoCAD. Before scanning the layout the drawing must formatted to suit with aadspro scanning process.



ID	Bar Mark	Bar Type	Size	Mem No	Bar No	Total Bars	Total Length	Shape Code	A	B	C	D	E	F	R	S
1	1	#	8	1	9	9	1300	20	1300	0	0	0	0	0		
2	2	#	8	1	8	8	850	37	745	100	0	0	0	0		
3	3	#	8	1	9	9	2400	20	2380	0	0	0	0	0		
4	4	#	8	1	16	16	3950	20	3940	0	0	0	0	0		
5	5	#	8	1	9	9	2575	20	2560	0	0	0	0	0		
6	6	#	8	1	23	23	2350	20	2345	0	0	0	0	0		
7	7	#	8	1	21	21	3650	20	3640	0	0	0	0	0		

Fig 7.3.4

- Here,
  - Bar mark = Bark mark given in Result (1,2 etc as shown in Fig 5)
  - T = Type of steel.
  - Size = Bar dia size
  - Mem No = No of repetitions of the member.
  - Bar No = No of bars of same length and dia.
  - Tot Bars (Total Bars) = Member No x Bar No
  - Len(Bar)mm = Length of bar in mm
  - Shape code = Shape code depends on the code selected, British, Indian & American.
  - A, B, C etc = Representations in shape code.

SHAPE CODES

As per British code

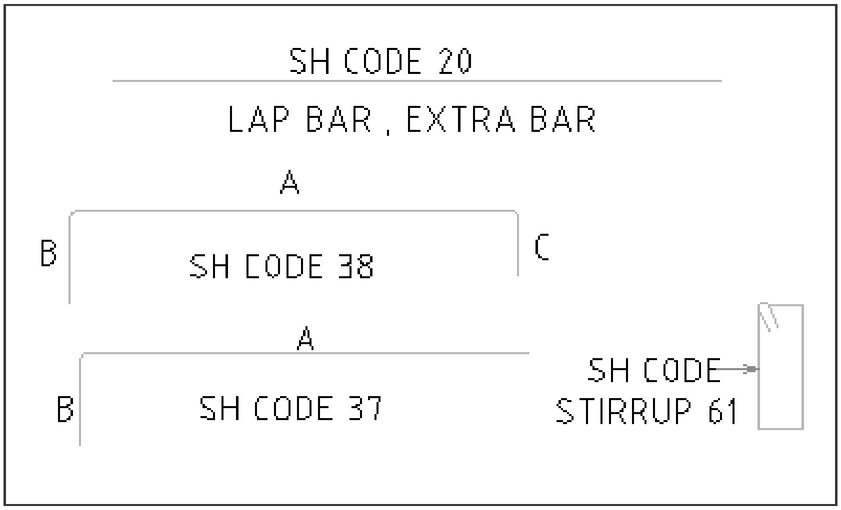


Fig 7.3.5 (a)

As per Indian code

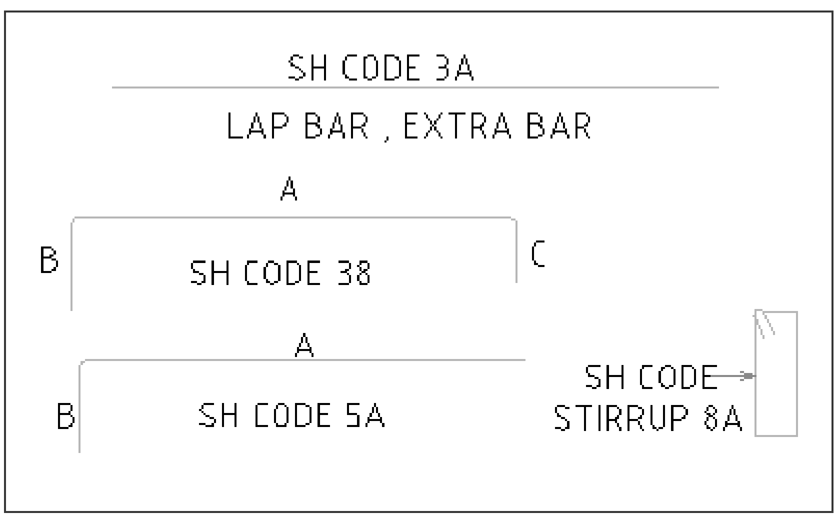


Fig 7.3.5(b)

7.3.3 Results

Bar Bending Schedule is obtained in AutoCAD (Fig 7.3.6 and Fig 7.3.7).

- 1, 2, 3 etc represents bar number for BBS.

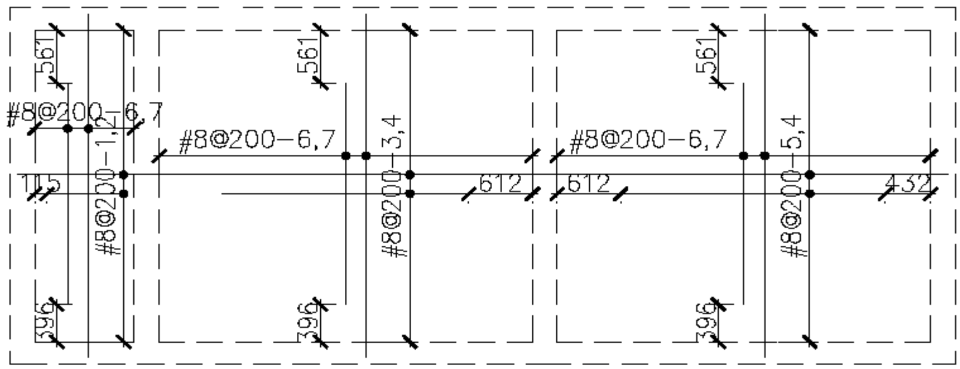


Fig 7.3.6

Bar Bending Schedule

ID	Bar Mark	Bar Type	Size	Mem No	Bar No	Total Bar	Total Length	Shape Code	A	B	C	D	E	F	R	S
1	1	#	8	1	9	9	1300	20	1300	0	0	0	0	0		1300
2	2	#	8	1	8	8	850	37	745	100	0	0	0	0		745, 100
3	3	#	8	1	9	9	2400	20	2380	0	0	0	0	0		2380
4	4	#	8	1	16	16	3950	20	3940	0	0	0	0	0		3940
5	5	#	8	1	9	9	2575	20	2560	0	0	0	0	0		2560
6	6	#	8	1	23	23	2350	20	2345	0	0	0	0	0		2345
7	7	#	8	1	21	21	3650	20	3640	0	0	0	0	0		3640

Fig 7.3.7

8. COLUMN

8.1 column detailing

This chapter provides a step-by-step tutorial for the detailing of a column of a multi-storied building.

8.1.1 Description of structure

8.1.2 Steps involved in Staad

8.1.3 Steps involved in aadspro

8.1.4 Results

8.1.1 Description of the problem

The structure is a double storied building; in which the column is to be designed. Plan of the building is shown in fig. (8.1.1).

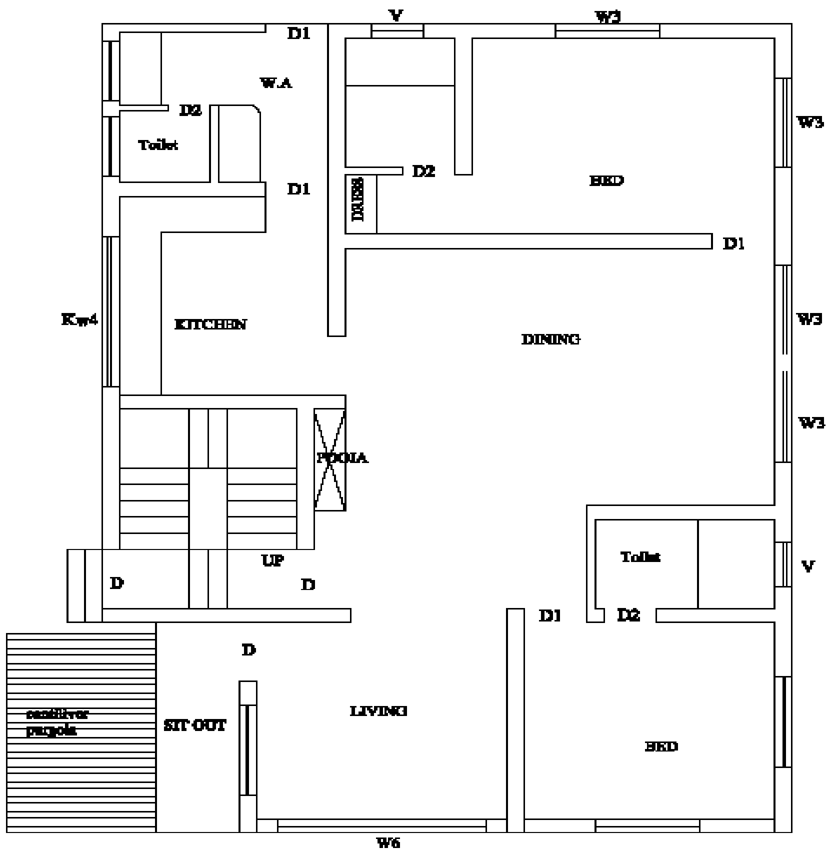


Fig.8.1.1

8.1.2 Steps involved in Staad

Staad mdb file is required for the design of column in aadspro,

Creating Staad mdb file

- Open the Staad file and analyze it.
- Select Tools >SQL Query>Advanced Query for STAAD Pro version 2006 & higher (fig.8.1.2).

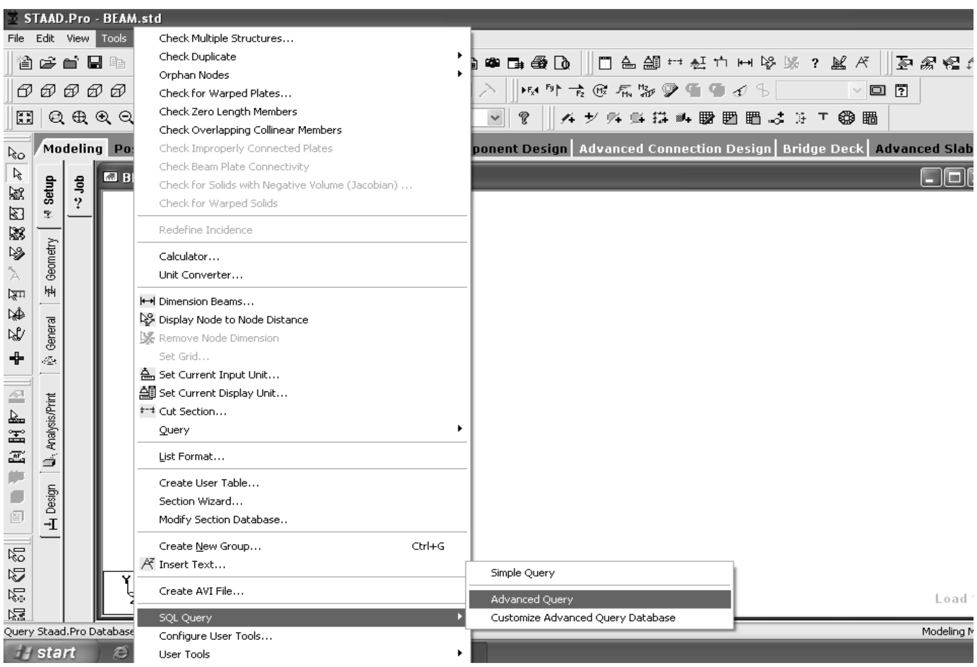


Fig.8.1.2

8.1.3 Steps involved in aadspro

Open aadspro. (AutoCAD will be open automatically)

Select ‘Column’ from main menu.

- Click on the 'Detailing' from tool bar (fig 8.1.3)

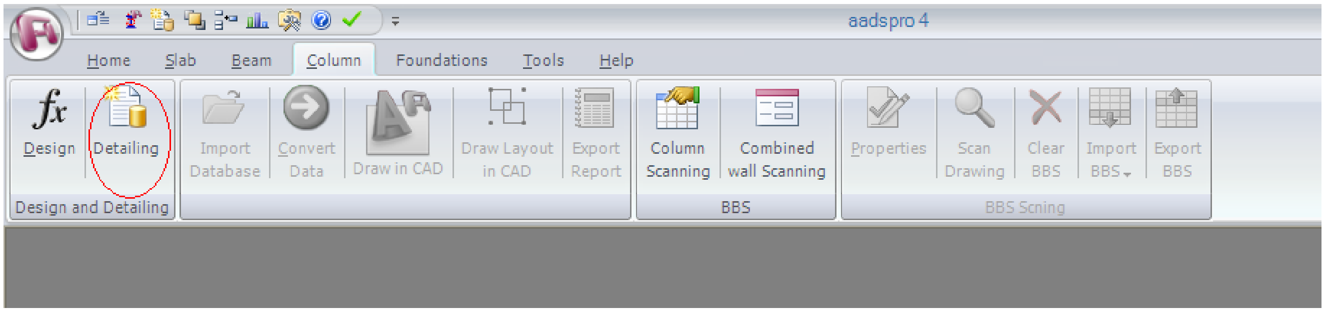
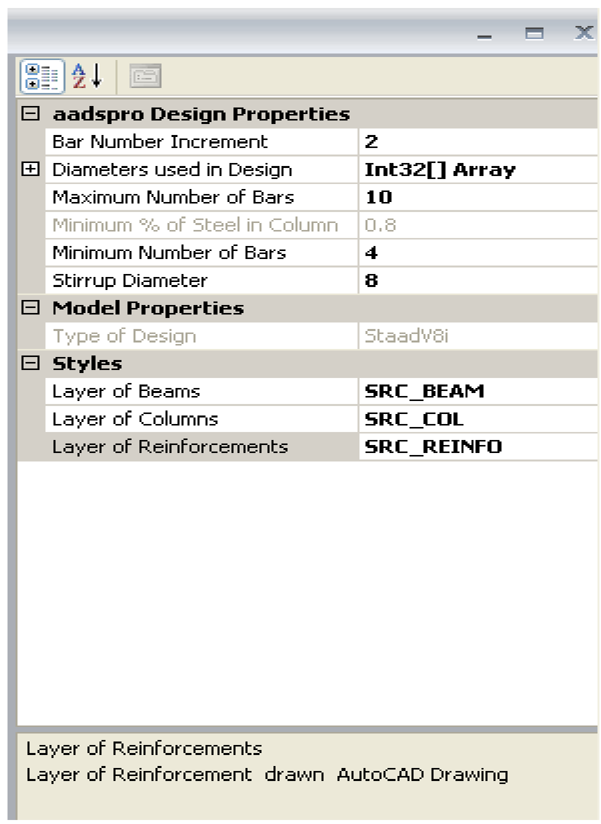


Fig 8.1.3

- User can set **properties** (fig 8.1.4)

This window helps the user to customize the detailing and design process. By setting these properties user can customize the detailing with variety of option.



User can set
Reinforcement Details

➢ AADSPRO DESIGN PROPERTIES

- Bar number Increment
  Number of bar to be incremented in design process.
- Diameters used in design
  Set reinforcement used in design
- Maximum number of bars
  Maximum number of bars used in design process.
- Minimum % of steel in column
  It's a default value, Default value can change from main menu-tools-settings- aadspro properties.
- Stirrup diameter
  Diameter of stirrup used.

Fig 8.1.4

➢ STYLES
- Layer of beams
- Layer of column
- Layer of reinforcement

As per project layout

- Click on the Import Database from tool bar (fig 8.1.5)

Open Database button allow the user to import the database file to the detailing system in aadspro .After selecting the database file aadspro check the input file and display the Level/Stories (Fig 8.1.6) in that file and prompt the user to select the desired Level/Storey.

Click Import button in the Storey window to import the data. aadspro automatically check for the STAAD Pro output file, if the selected database file is an output of STAAD Pro model.

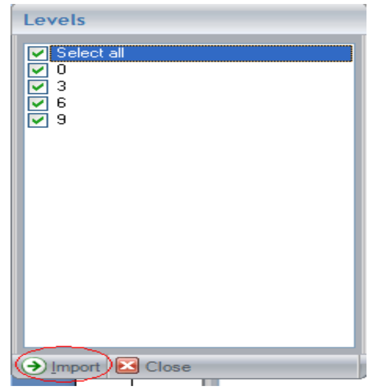


Fig 8.1.6

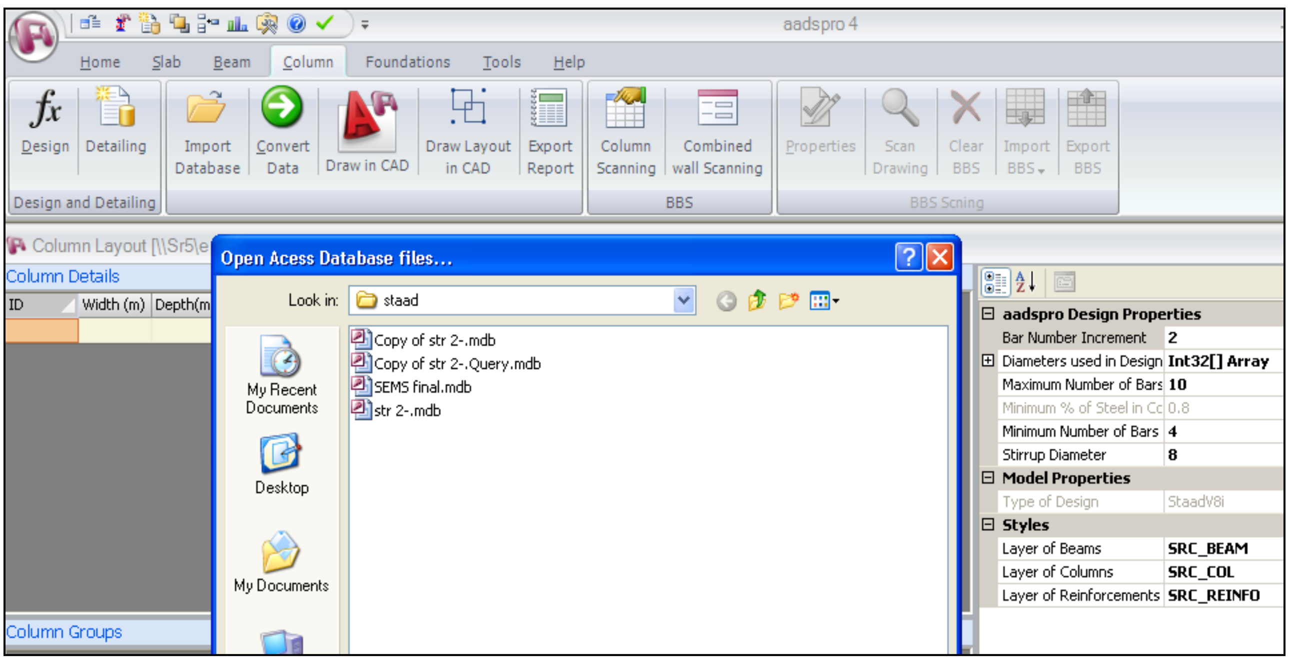


Fig 8.1.5

- Click on the Convert data from tool bar (fig 8.1.6)

The Convert data button converts the imported data into a grouped format and listed in the Column Group Grid. The Properties (Fig 8.1.6) of each column group can be viewed in the property list in the right side of the window. User can edit this property for further changes.

Aadspro will check the entire column for any invalid data, if it found any invalid data column it will be marked with red color. User can view the error message by right clicking (Fig 8.1.5) the grid and selecting the Warnings menu

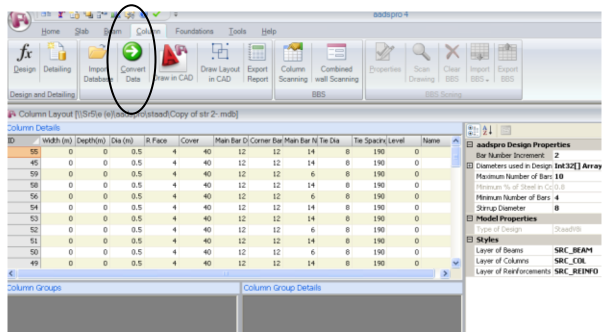


Fig 8.1.6

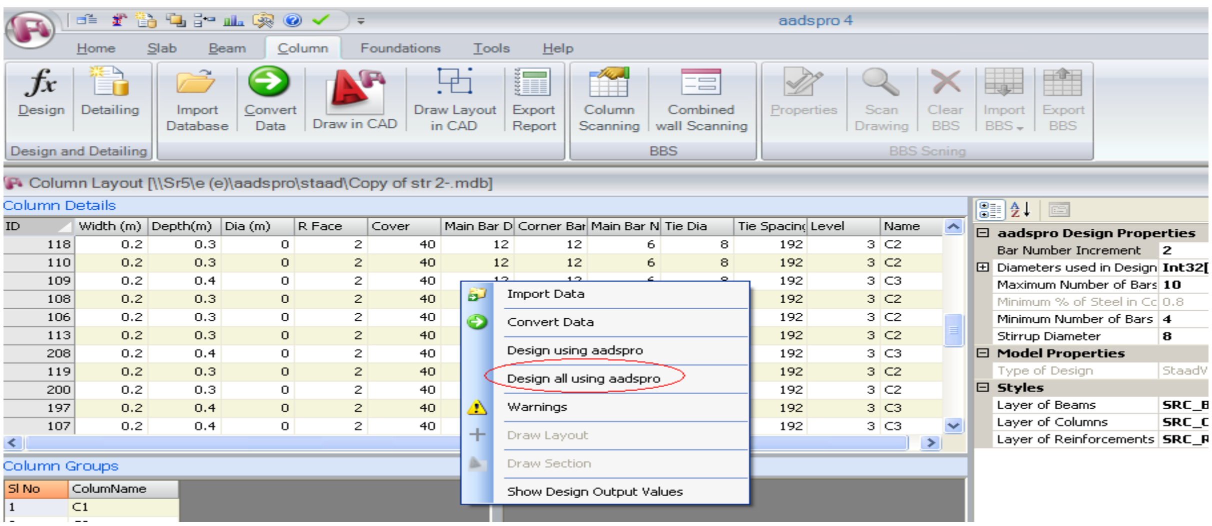


Fig 8.1.7

8.1.4 Results

- Click on the 'To Auto Cad' from tool bar

This button will draw the column schedule in AutoCAD. In this aadspro create a schedule with column name, reinforcement details, tie details and cross sectional drawing. The cross section of single column group is created by right clicking the column group name and select Draw Section menu (Fig 8.1.8)

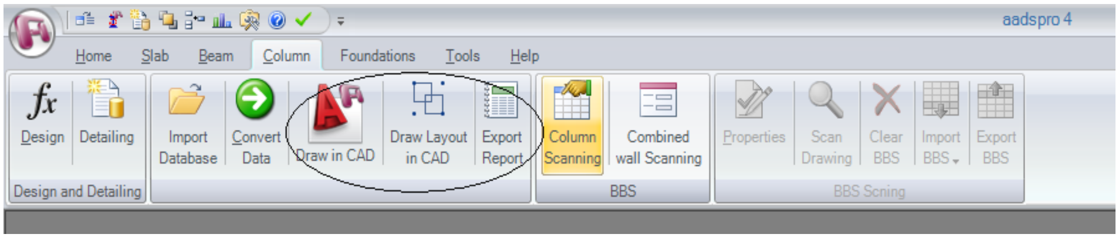


Fig 8.1.8

NO:	COL NO:	REINF	TIES	COL SECTION
2	C2	6- #12	#8-192	300X200
3	C3	6- #12	#8-192	400X200

Fig 8.1.9

- Click on the ‘Draw layout in CAD’ from tool bar

This button creates a layout of column in each floor selected while importing .aadspro prompt the user in AutoCAD to pick a point to draw the layout. (Fig 8.1.10)

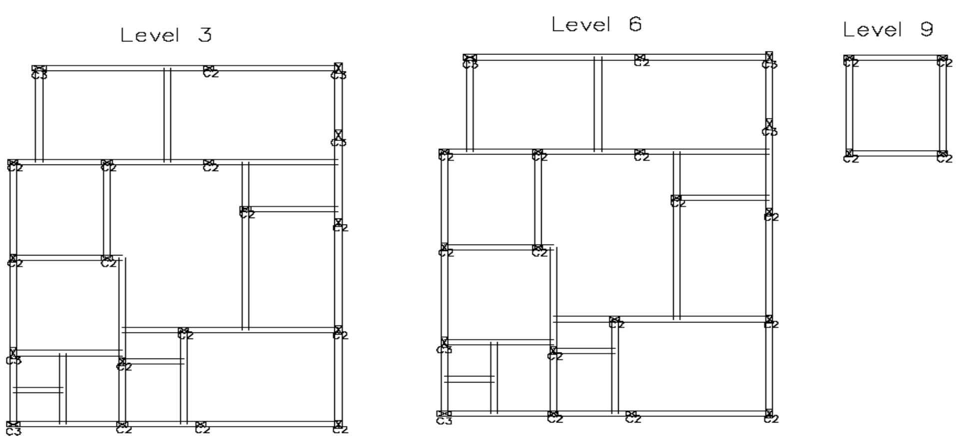


Fig 8.1.10

- Click on the ‘Export Report’ from tool bar

This button exports the design out to excel or text file the design result can viewed by right clicking the Column details grid and selecting the Show **Design Output Values**.

8.2 Column design

This module contains Design of a column section as per limit state method.

8.2.1 Steps involved in aadspro

Open aadspro. Select 'column' from the Main menu and 'Design' from drop down menu (Fig 8.2.1)

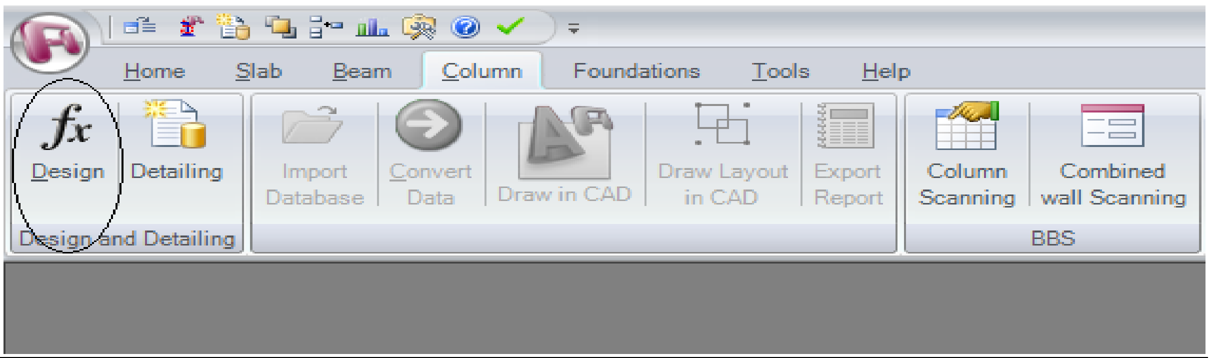


Fig 8.2.1

- User can set data in slab design grid (Fig 8.2.2)

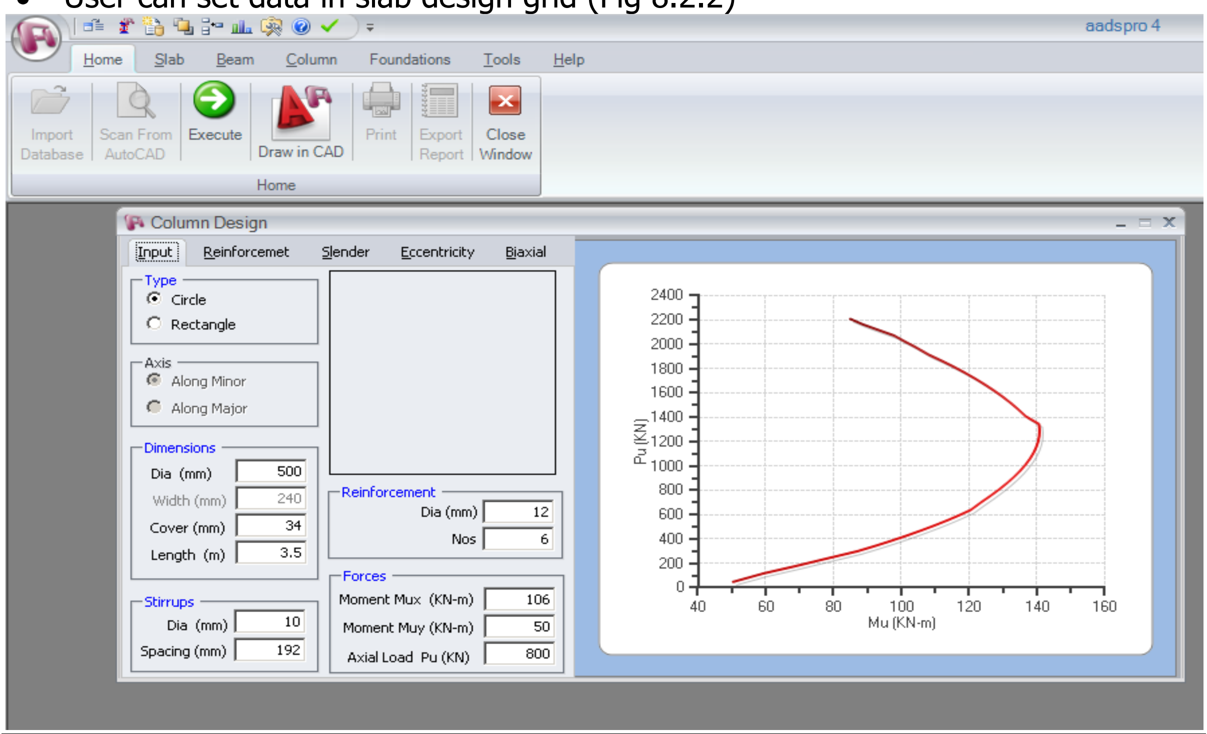


Fig 8.2.2

- Options are given for circular columns and rectangular columns. And also select the axis.
- Material properties, f_{ck} & f_y is a default value, can be change by clicking Tools-Settings-addspro properties.
- Dimensions and clear cover can be entered by typing new values
- Nos. and diameter of the bars can also change.
- Values of axial force, moment about x-direction and y-direction, etc. can be entered by typing new values. These values can be transferred to the corresponding design plate also.
- Analysis can be done by pressing the 'analysis' button. Interaction diagram is plotted in the design plate with M_u in x- axis and P_u in y-axis. Circle shown in the diagram represents the load and moment. In the analysis mode we can make this circle close to the curve either by changing the reinforcement or by changing the dimensions.

Consider the examples given below and its solutions by manual calculation.

Explanations are also attached after solutions to see how the program arrives at results.

Example1

Determine the reinforcement required for a rectangular column subjected to biaxial bending with the following data.

Width, b = 400mm

Depth, D = 600mm

Length of column, l = 3000mm

Factored load on column, P_u: = 1600KN

Factored moment acting parallel

to larger dimension, M_{ux} = 120KNm

Factored moment acting parallel

to shorter dimension, M_{uy} = 90KNm

Use M25 grade concrete and Fe 415 steel.

Calculation for minimum eccentricity;

a) Eccentricity in longer direction, e_x = $\left(\frac{l}{500}\right)+\left(\frac{D}{30}\right)$

= $\left(\frac{3000}{500}\right)+\left(\frac{600}{30}\right)$

= 26mm

b) Eccentricity in shorter direction, e_y = $\left(\frac{l}{500}\right)+\left(\frac{b}{30}\right)$

= $\left(\frac{3000}{500}\right)+\left(\frac{400}{30}\right)$

= 19.33mm

c) Minimum eccentricity limit, = 20mm

Minimum eccentricity, e_{min} = 26mm (as per IS 456-2000, Cl: 25.4)

Moment due to minimum eccentricity, M_{ax} = $P_u e_x$
= $1600 x 0.026$
= 41.60KNm

Moment due to minimum eccentricity, M_{ay} = $P_u e_y$
= $1600 x 0.0193$
= 30.88KNm

Moments due to eccentricity are less than the value of initial moments.
Then design moments are,

M_{ux} = 120KNm

M_{uy} = 90KNm

Calculation for longitudinal reinforcement:

Reinforcement is distributed equally on two sides,

Assume, percentage of reinforcement, p		=	0.84%
	p/f_{ck}	=	0.034
Dia of reinforcement,		=	16mm

Uniaxial moment capacity of the section about x-x axis;

Effective cover to the column, d'		=	40+8 =48mm
	d'/D	=	48/600
		=	0.08
	$P_u/f_{ck}bD$	=	1600x10^3/(25x400x600)
		=	0.267
Using SP 16, chart 32,	$M_u/f_{ck}bD^2$	=	0.09
	M_{ux1}	=	$0.09x25x400x600^2$
		=	325KNm

Uniaxial moment capacity of the section about y-y axis;

	d'/b	=	48/400
		=	0.12
	$P_u/f_{ck}bD$	=	1600x10^3/(25x400x600)
		=	0.267
Using SP 16, chart 33,	$M_u/f_{ck}Db^2$	=	0.07
	M_{uy1}	=	$0.07x25x600x400^2$
		=	168KNm

Calculation of P_{uz};

Using SP 16, chart 63,	Puz/Ag	=	13.5N/mm^2
	Puz	=	13.5x400x600
		=	3240KN
	Pu/ Puz	=	1600/3240
		=	0.49
	M_{ux}/M_{ux1}	=	120/325
		=	0.37
	M_{uy}/M_{uy1}	=	90/168
		=	0.54

For check;

α_n = 1.48 (with respect to Pu/ Puz)

$(M_{ux}/M_{ux1})^{\alpha n} + (M_{uy}/M_{uy1})^{\alpha n}$ = $0.37^{(1.48)} + 0.54^{(1.48)}$

= 0.63 < 1.0 (as per Cl:39.6,IS456:2000)

Hence the section is safe.

Then Provide 10no.s of 16mmDia bars.

Calculation for Transverse reinforcement:

As per IS 456-2000, Cl : 26.5.3.2.(c).(2), the diameter of tie shall not less than the following:

a) ¼ of the main Dia = 1/4x16 = 4mm
b) 5mm = 5mm

As per IS 456-2000, Cl : 26.5.3.2.(c).(1), the pitch of ties shall not exceed the following:

a) Least dimension of column = 400mm
b) 16 of smallest dia of minimum bar = 16x16
= 256mm
c) 300mm

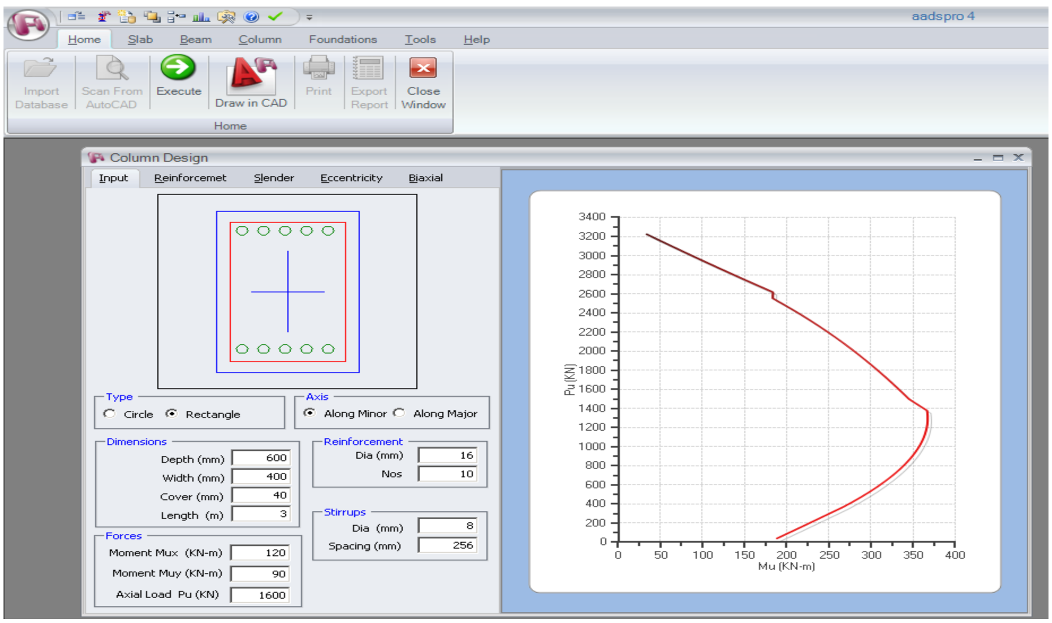
Hence the lateral ties of 8mm diameter at a spacing of 200mm c/c shall be provided as transverse reinforcement.

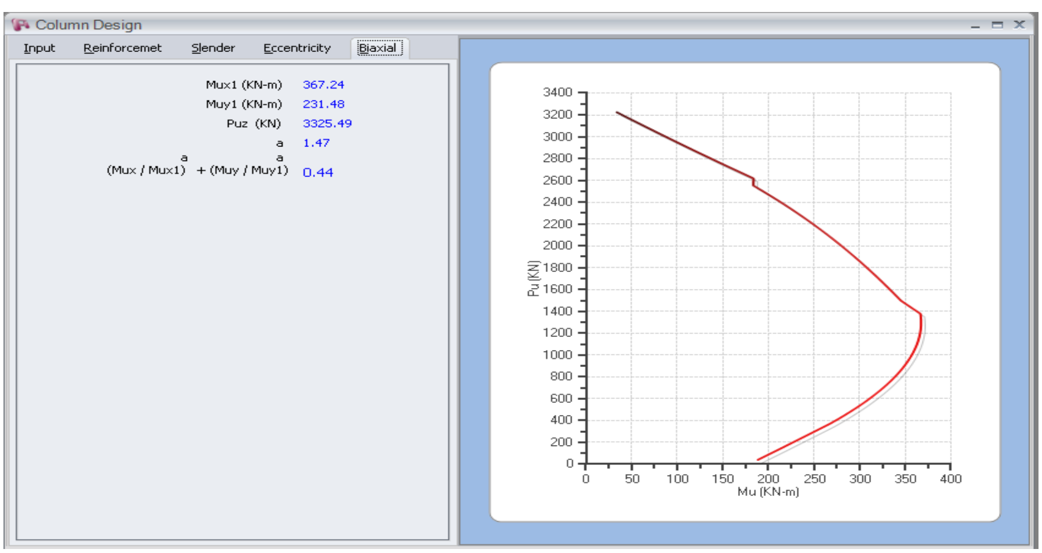
Now let us see how program arrives at results.

1. First choose Rectangle type column.
2. Select analysis to calculate the reinforcement detail of column.
3. Enter the characteristic values of concrete and steel. Here these values are f_{ck} = 25 and f_y = 415.
4. Enter the dimensions of the column. Width = 400mm, Depth = 600mm, Cover = 40mm Length = 3m.

5. Diameter of the longitudinal reinforcement can be selected by clicking on the up and down buttons. Here, select 10 no.s of 16mm dia bars as longitudinal reinforcement.
6. Enter the values of moment, M_{ux} = 120 kNm, M_{uy} = 90 kNm and axial load = 1600 kN.

Now let us see how the program arrives at results.





A comparison with manual calculation is shown below.

	Values from Program	Values by manual Calculation
Moment capacity Mux1	367.24Kn-m	325 Kn-m
Moment capacity Muy1	231.48Kn-m	168 Kn-m
Load capacity Puz	3325 KN	3240 KN

Example 2

Determine the reinforcement required for a rectangular slender column subjected to biaxial bending with the following data.

Width, b = 300mm

Depth, D = 400mm

Length of column, l = 7000mm

Factored load on column, P_u = 1500KN

Factored moment acting parallel

to larger dimension, M_{ux} = 40KNm

Factored moment acting parallel

to shorter dimension, M_{uy} = 30KNm

Use M25 grade concrete and Fe 415 steel.

Effective length for bending parallel to

larger dimension, l_{ex} = 6000mm

Effective length for bending parallel to

shorter dimension, l_{ey} = 5000mm

Slenderness ratio, l_{ex} /D = 6000/400

= 15 > 12

Slenderness ratio, l_{ey} /b = 5000/300

= 16.7 > 12

The column is slender about both the axis.

Calculation for moments due to slender:

As per Cl: 39.7.1, of IS 456-2000 Additional moments are calculated by the following:

Additional moments, M_{ax} = $\left(\frac{P_u D}{2000}\right)\left(\frac{l_{ex}}{D}\right)^2$

= $\left(\frac{1500x400}{2000}\right)15^2$

= 67.5KNm

Additional moments, M_{ay} = $\left(\frac{P_u b}{2000}\right)\left(\frac{l_{ey}}{b}\right)^2$

= $\left(\frac{1500x300}{2000}\right)16.7^2$

= 62.75KNm

Calculation for reduction factors:

Reinforcement is distributed equally on two sides,
Assume, percentage of reinforcement, p = 3.0%
p/f_{ck} = 0.12

Dia of reinforcement, = 20mm

Using SP 16, chart 63, P_{uz}/Ag = 20.5N/mm^2
P_{uz} = $20.5x300x400$

= 2460KN

About x-x axis;

Effective cover to the column, d' = 40+10=50mm

d'/D = $50/400$

= 0.12

From Table 60 of SP16, k_1 = 0.20

k_2 = 0.07

P_{bx} = $\left(k_1 + k_2 \frac{p}{f_{ck}}\right) f_{ck} bD$

= $(0.20 + (0.07x0.12))x25x300x400$

= 625.2KN

Reduction factor, k_x (As perCl: 39.7.1.1, IS 456-2000)

= $\left(\frac{P_{uz} - P_u}{P_{uz} - P_{bx}}\right)$

= $\left(\frac{2460 - 1500}{2460 - 625.2}\right)$

= 0.52

About y-y axis;

d'/b = $50/300$

= 0.16

From Table 60 of SP16, k_1 = 0.19

k_2 = 0.04

P_{by} = $\left(k_1 + k_2 \frac{p}{f_{ck}}\right) f_{ck} bD$

= $(0.19 + (0.04x0.12))x25x300x400$

= 584.4KN

Reduction factor, k_y = $\left(\frac{P_{uz} - P_u}{P_{uz} - P_{by}}\right)$

= $\left(\frac{2460-1500}{2460-584.4}\right)$

= 0.51

Modified moments, M_{ax} = $M_{ax}k_x$

= 67.5 x 0.52

= 35.1KNm

Modified moments, M_{ay} = $M_{ay}k_y$

= 62.75 x 0.51

= 32KNm

Calculation for minimum eccentricity;

As per Cl: 25.4, IS 456-2000

a) Eccentricity in longer direction, e_x = $\left(\frac{l}{500}\right)+\left(\frac{D}{30}\right)$

= $\left(\frac{7000}{500}\right)+\left(\frac{400}{30}\right)$

= 27.33mm

b) Eccentricity in shorter direction, e_y = $\left(\frac{l}{500}\right)+\left(\frac{b}{30}\right)$

= $\left(\frac{7000}{500}\right)+\left(\frac{300}{30}\right)$

= 24mm

c) Minimum eccentricity limit, = 20mm

Moment due to minimum eccentricity, M_{ax} = $P_u e_x$

= $1500 x 0.027$

= 40.50KNm > 40KNm (initial moment)

Moment due to minimum eccentricity, M_{ay} = $P_u e_y$

= $1500 x 0.024$

= 36.00KNm > 30KNm (initial moment)

Then, Total moments for which the column is to be designed are:

M_{ux} = $40.5 + 35.1$

= 75.6KNm

M_{uy} = $36 + 32$

= 68.0KNm

Calculation for longitudinal reinforcement:

Reinforcement is distributed equally on two sides,

Assume, percentage of reinforcement, p = 3%

p/f_{ck} = 0.12

Uniaxial moment capacity of the section about x-x axis;

d'/D = 0.125

$P_u/f_{ck}bD$ = $1500 x 10^3 /(25 x 300 x 400)$

= 0.50

Using SP 16, chart 33, $M_u/f_{ck}bD^2$ = 0.13

M_{ux1} = $0.13 x 25 x 300 x 400^2$

= 156KNm

Uniaxial moment capacity of the section about y-y axis;

d'/b = 0.16

$P_u/f_{ck}bD$ = 0.5

Using SP 16, chart 33, $M_u/f_{ck}Db^2$ = 0.12

M_{uy1} = $0.12 x 25 x 400 x 300^2$

= 108KNm

Puz	=	2460KN
Pu/ Puz	=	1500/2460
	=	0.61
M_{ux}/M_{ux1}	=	75.6/156
	=	0.48
M_{uy}/M_{uy1}	=	68/108
	=	0.63

For check**;**

α_n = 1.32(with respect to Pu/ Puz)

$(M_{ux}/M_{ux1})^{\alpha n} + (M_{uy}/M_{uy1})^{\alpha n}$ = $0.48^{(1.32)} + 0.63^{(1.32)}$

= 0.923 < 1.0 (as per Cl: 39.6, IS456-2000)

Hence the section is safe.

<u>Then Provide 12no.s of 20mmDia bars.</u>

Calculation for Transverse reinforcement**:**

As per IS 456-2000, Cl : 26.5.3.2.(c).(2), the diameter of tie shall not less than the following:

a) ¼ of the main Dia = 1/4x20 = 5mm

b) 5mm = 5mm

As per IS 456-2000, Cl : 26.5.3.2.(c).(1), the pitch of ties shall not exceed the following:

a) Least dimension of column = 300mm

b) 16 of smallest dia of minimum bar = 16x20

= 320mm

d) 300mm

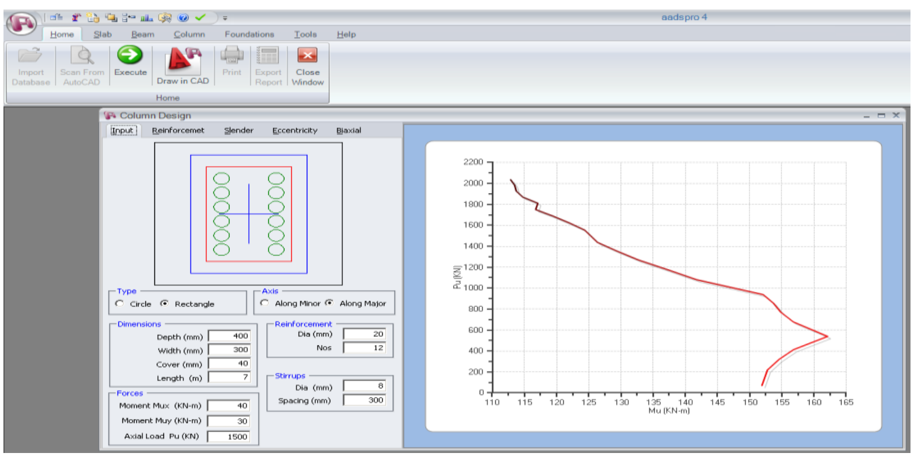
<u>Hence the lateral ties of 8mm diameter at a spacing of 200mm c/c shall be provided as transverse reinforcement.</u>

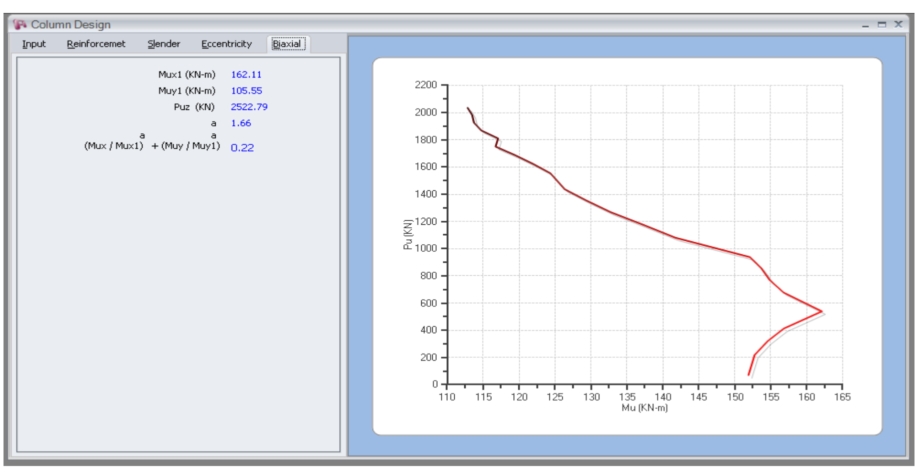
Now let us see how program arrives at results.

1. First choose Rectangle type column.

2. Select analysis to calculate the reinforcement detail for column.
3. Enter the characteristic values of concrete and steel. Here these values are f_{ck} = 25 and f_y = 415.
4. Enter the dimensions of the column. Width = 300mm, Depth = 400mm, Cover = 40mm Length = 7m.
5. Diameter of the longitudinal reinforcement can be selected by clicking on the up and down buttons. Here, select 12 no.s of 20mm dia bars as longitudinal reinforcement.
6. Enter the values of moment, M_{ux} = 40 kNm, M_{uy} =30 kNm and axial load = 1500 kN.

Now let us see how the program arrives at results.





A comparison with manual calculation is shown below.

	Values from Program	Values by manual Calculation
Moment capacity Mux1	162.11 Kn-m	156 Kn-m
Moment capacity Muy1	105.55 Kn-m	108 Kn-m
Load capacity Puz	2522.79 KN	2460 KN

8.3 BBS (Bar Bending Schedule) SCAN-COLUMN

This chapter provides a method for getting the bar bending schedule of existing columns. This can be used to change the existing bar bending schedule of columns.

8.3.1 In AutoCAD

Column detail is provided in AutoCAD as shown in the fig 8.3.1, classify the columns in different zone.

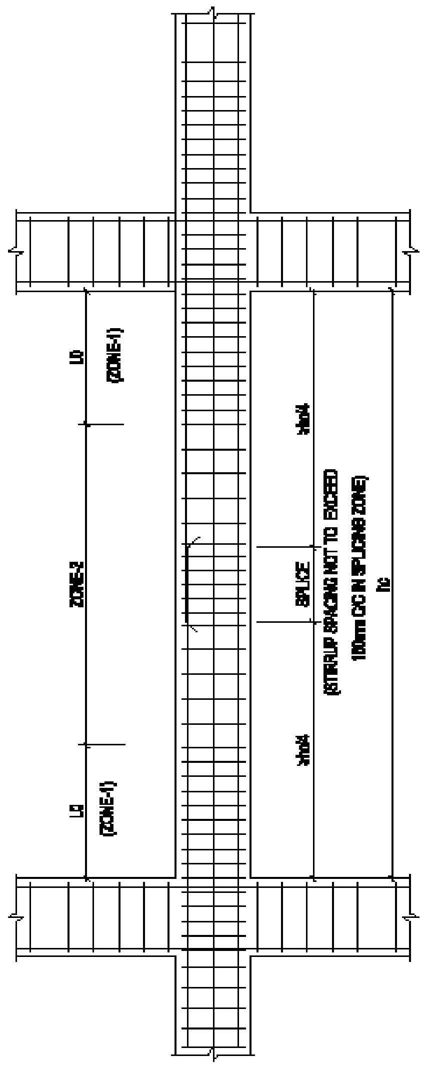


Fig 8.3.1

- Make the reinforced concrete details for each zones (Fig 8.3.2)
- All layers of bars should be in same layer.

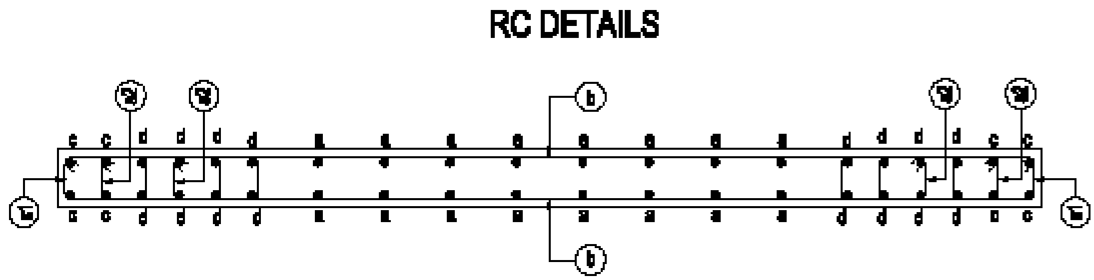


Fig 8.3.2

- Make the reinforced concrete details in a table format for each zones also the longitudinal and distribution bar details.(Fig 8.3.3).

Zone 1 details

Y10 AT 100 C/C	s1
Y10 AT 100 C/C	s2

Zone 2 details

Y10 AT 150 C/C	s1
Y10 AT 150 C/C	s2

Long: and Distribution bars details

Y10 AT 250 C/C	a
Y10 AT 250 C/C	b
8-Y16	c
16-Y12	d

Fig 8.3.3

8.3.2 In aadspro

Select 'Column' from the Main menu

- Click on the 'Column Scanning' from tool bar (fig1.2.1)

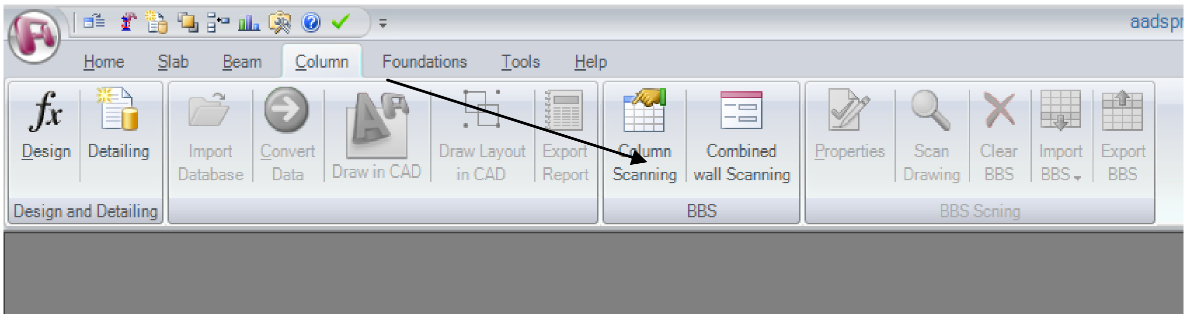


Fig 1.2.1

- User can set properties (fig1.2.2)

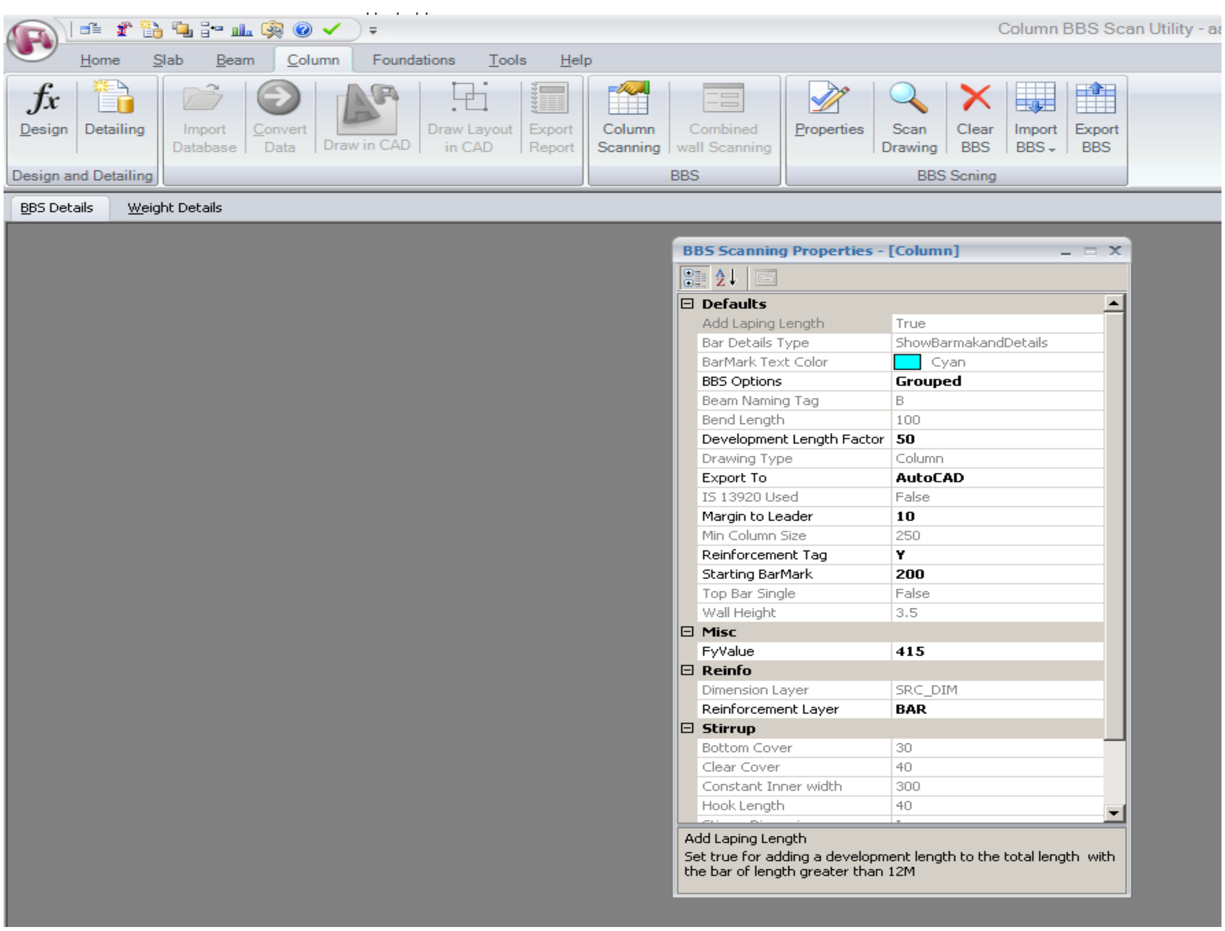


Fig 1.2.2

- BBS option

  This option is to confirm that how should be the BBS report displayed . It means the type of BBS report. It can be grouped or split.
- Development length factor

  Development length factor
- Export to
- To which software you want to export the bbs report. There are three options
  1. Auto Cad
  2. Simple Excel
  3. Excel with picture

- Margin to leader

  Setting the tolerance of the leader position with the margin
- Reinforcement Tag.

  Reinforcement tag provided in drawing.
- Starting bar mark

  Bar mark means the id of the reinforcement to represent it in the drawing to separate each bar from one other . Starting letter of this id need to be set here
- Reinforcement Layer

  Set the Layer of reinforcement in which the whole reinforce bars are arranged. And it is to verify that all reinforcement lines are assigned to this layer

1.2.3- Click on the 'Scan Drawing' from tool bar.

All steps coming in auto cad command line.

Step 1: Select Zone 1 Lateral ties details schedules.

Step 2: Select Zone 2 Lateral ties details schedules.

Step 3: Select longitudinal details schedules.

Step 4: Enter the height of Zone 1.

Step 5: Enter the height of Zone 2. (Fig.a)

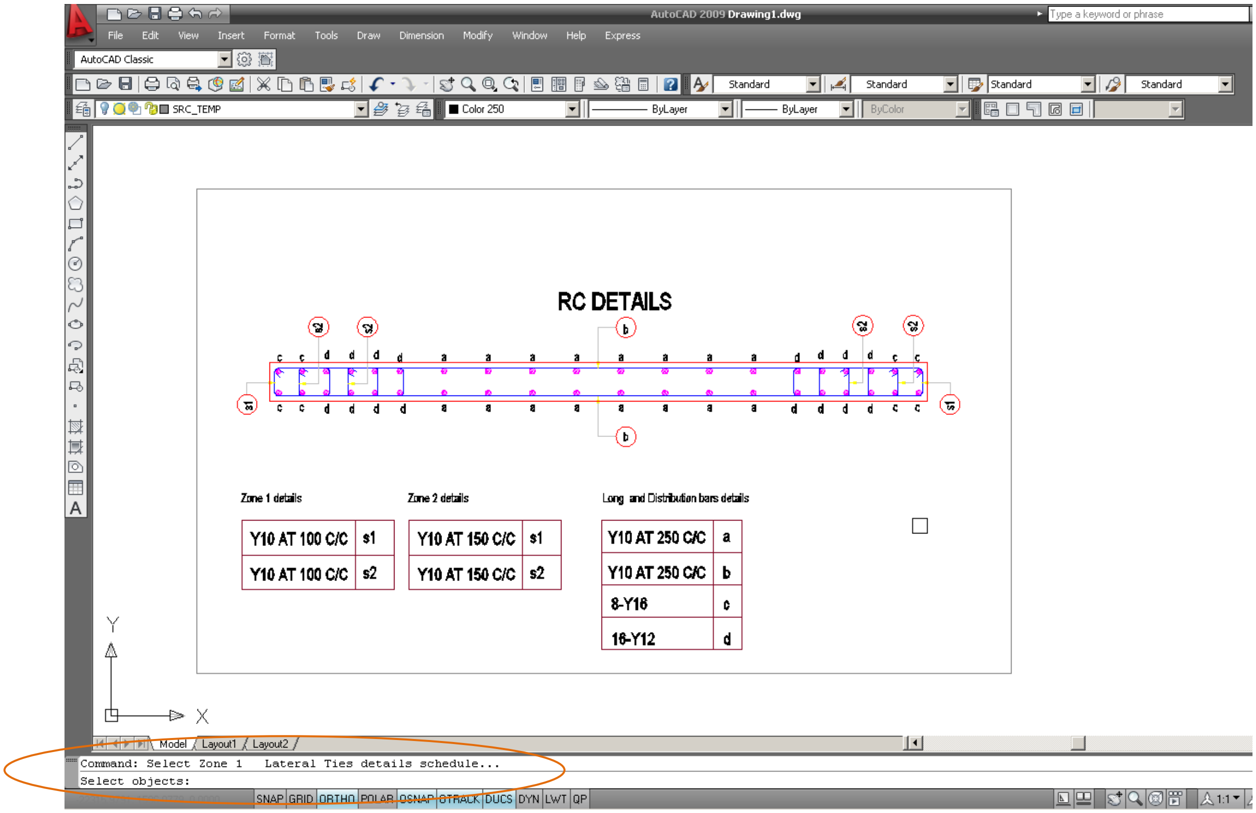


Fig.a

Step 6: Then the aadspro reinforcement details table is coming, close the table.(Fig b)

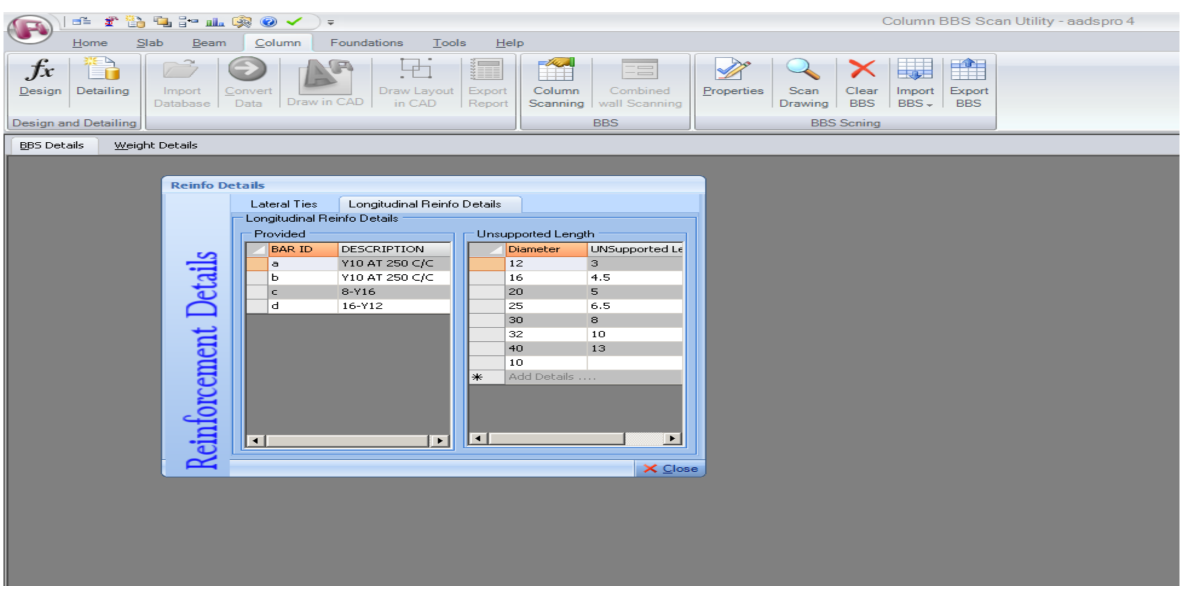
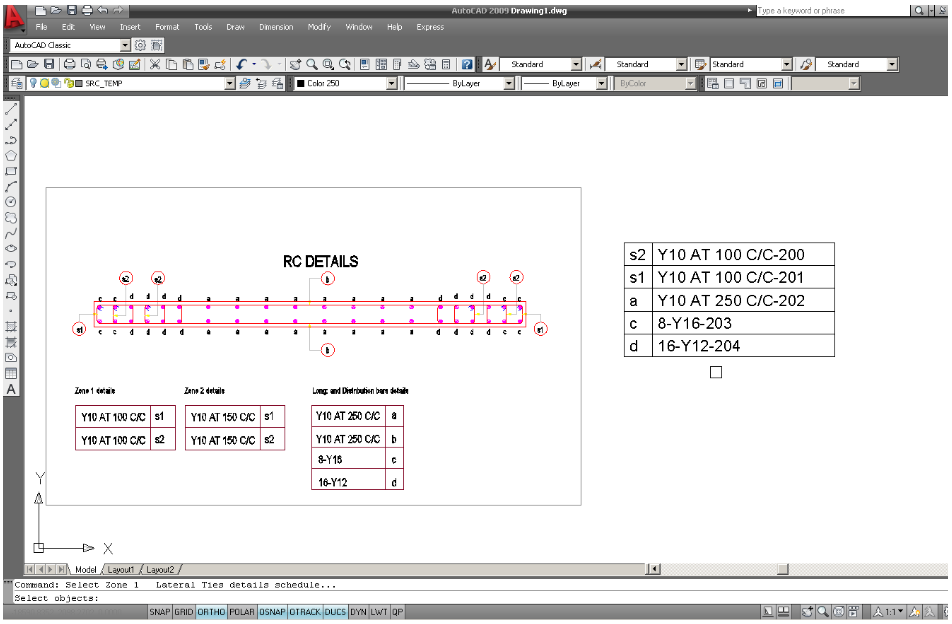


Fig.b

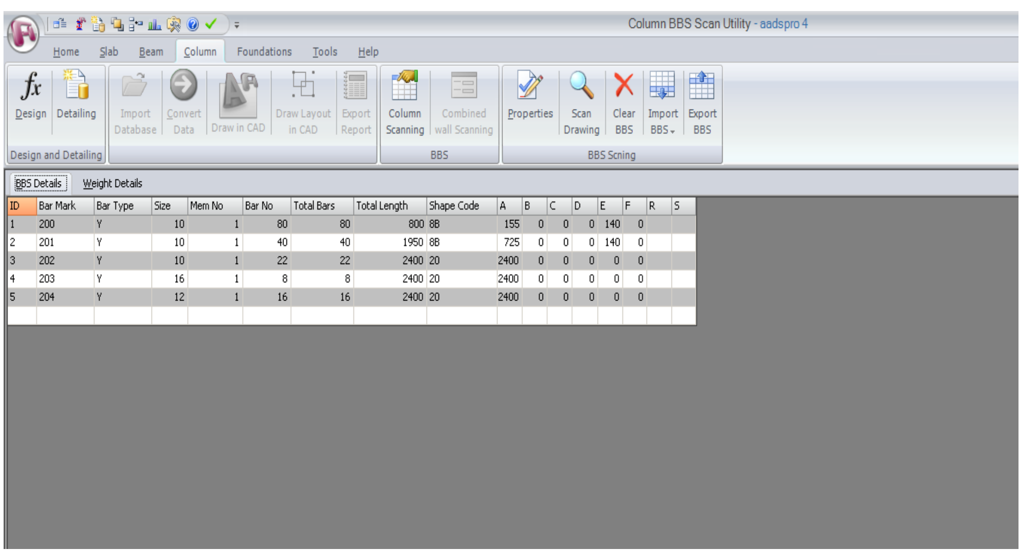
Step 7: Select the object.

Step 8: Select a point to draw schedule. (Fig: c)



(Fig .c)

After that bar bending schedules and weight details obtained from the aadspro (Fig .d)



ID	Bar Mark	Bar Type	Size	Mem No	Bar No	Total Bars	Total Length	Shape Code	A	B	C	D	E	F	R	S
1	200	Y	10	1	80	80	800	8B	155	0	0	0	140	0		
2	201	Y	10	1	40	40	1950	8B	725	0	0	0	140	0		
3	202	Y	10	1	22	22	2400	20	2400	0	0	0	0	0		
4	203	Y	16	1	8	8	2400	20	2400	0	0	0	0	0		
5	204	Y	12	1	16	16	2400	20	2400	0	0	0	0	0		

Fig .d

1.2.4- Click on the 'Export BBS' from tool bar (fig1.2.4), It give bar bending schedule.

Bar Bending Schedule

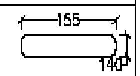
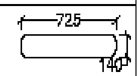
Bar Mark	Bar Type	Size	Mem No	Bar No	Total Bars	Total Length	Shape Code	A	B	C	D	E	F	R	S	
200	Y	10	1	80	80	800	88	155	0	0	0	140	0			
201	Y	10	1	40	40	1950	88	725	0	0	0	140	0			
202	Y	10	1	22	22	2400	20	2400	0	0	0	0	0			
203	Y	16	1	8	8	2400	20	2400	0	0	0	0	0			
204	Y	12	1	16	16	2400	20	2400	0	0	0	0	0			

Fig 1.2.4

- Here,
  - Bar mark = Bark mark given in Result (200,201 etc as shown in Fig 1.2.4)
  - Y = Type of steel.
  - Size = Bar dia size
  - Mem No = No of repetitions of the member.
  - Bar No = No of bars of same length and dia.
  - Tot Bars (Total Bars) = Mem No x Bar No
  - Len(Bar)mm = Length of bar in mm
  - Shape code = Shape code depends on the code selected, British, Indian & American.
  - A, B, C etc = Representations in shape code.

SHAPE CODES

As per Indian code

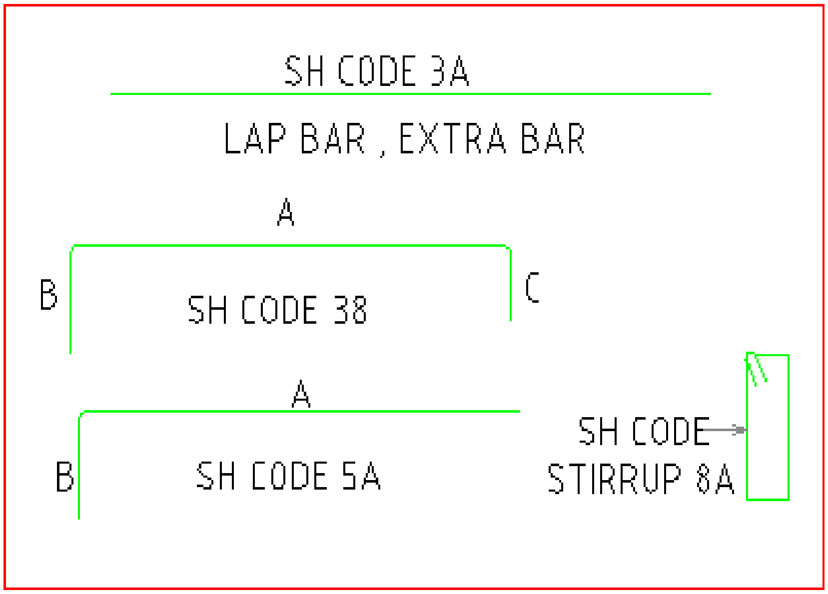


Fig 4(b)

As per British code

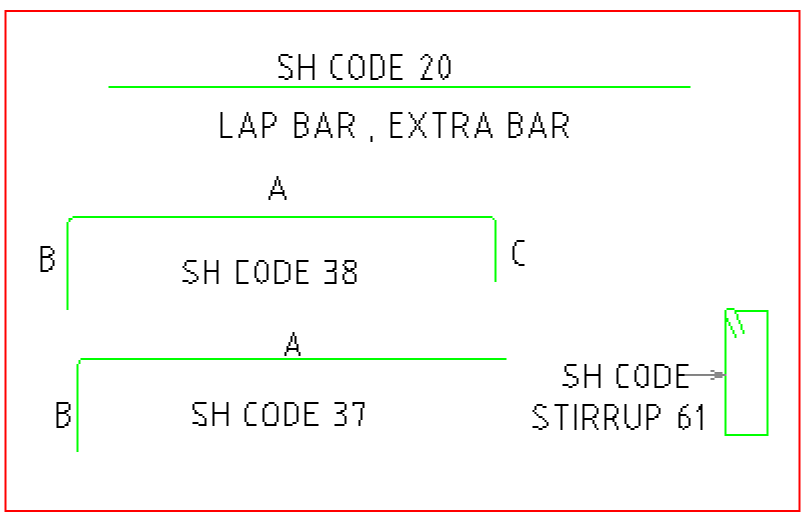


Fig 4(a)