Govt. & UGC Approved

UNIVERSITY OF GLOBAL VILLAGE (UGV), BARISHAL

THE UNIVERSITY FOR HI-TECH AND HUMANITY

SAFE

(Foundation Design)

Content of Laboratory Course

Prepared By

Somen Saha

Lecturer

Department of Civil Engineering

University of Global Village (UGV), Barishal

Program: B.Sc. in Civil Engineering

Govt. & UGC Approved

UNIVERSITY OF GLOBAL VILLAGE (UGV), BARISHAL

THE UNIVERSITY FOR HI-TECH AND HUMANITY

BASIC COURSE INFORMATION

Course Title	SAFE (Foundation Design)
Course Code	CE 0732-3202
Credits	01
CIE Marks	30
SEE Marks	20
Exam Hours	2 hours (Semester Final Exam)
Level	6th Semester

SAFE (Foundation Design)

COURSE CODE: CE 0732-3202

CREDIT: 01
CIE MARKS: 30

SEE MARKS: 20

CLO 01 **Differentiate** between different foundation types like isolated footings, combined footings, mat foundations, and pile foundations.
CLO 02 **Create** accurate 3D models of foundations in CSI SAFE, incorporating soil properties and loading conditions.
CLO 03 **Conduct** structural analysis of foundations to determine stresses, deformations, and safety factors. Design foundations to meet relevant codes and standards.
CLO4 **Analyze** the results of the analysis to assess the performance of the foundation and identify potential issues.

Sl.	Course Contents	Hours	CLOs
1	Introduction to csi SAFE software & foundation -An **introduction** to the csi SAFE software, a powerful tool for the analysis and design of foundations and geotechnical structures.	10	CLO 1
2	Design of Isolated Single Column Footing -**Design and analysis** of isolated single column footings using the SAFE software, including load combination checks and detailing.	10	CLO2,CLO 3, CLO4
3	Design of Combined Footing -**Application** of the SAFE software for structural analysis and design of combined footings to support multiple columns, considering soil-structure interaction and load combinations.	35	CLO2,CLO 3, CLO4
4	Design of Mat Foundation -**Principles and practices** for designing mat foundations, covering soil-structure interaction, load distribution, structural analysis, detailing, and construction considerations.	30	CLO2,CLO 3, CLO4

5	**Design of Pile and Pile cap** **-Exploration** of principles and methodologies for designing piles and pile caps for various loading conditions	30	**CLO2,CLO 3, CLO4**

References

1. SAFE tutorial by ISO SAF

Prepared By
Somen Saha
Lecturer
Department of Civil Engineering
University of Global Village (UGV), Barishal

ASSESSMENT PATTERN

CIE- Continuous Internal Evaluation (30 Marks)

SEE- Semester End Examination (20 Marks)

SEE- Semester End Examination (40 Marks) (should be converted in actual marks (20))

Bloom's Category	**Tests**
Remember	**05**
Understand	**07**
Apply	**08**
Analyze	**07**
Evaluate	**08**
Create	**05**

CIE- Continuous Internal Evaluation (100 Marks) (should be converted in actual marks (30))

Bloom's Category Marks (out of 100)	Lab Final (30)	Lab Report (10)	**Continuous lab performance** (30)	Presentation & Viva (10)	External Participation in Curricular/**Final Project Exhibition** (10)
Remember/**Imitation**	05		05	02	Attendance 10
Understand/**manipulation**	05	05	05	03	
Apply/**Precision**	05		05		
Analyze/**Articulation**	05		05		
Evaluate/**Naturalisation**	05	05	05		
Create	05		05	05	

Week	Topic	Teaching Learning Strategy	Assessment Strategy	CLOs
1	Introduction to csi SAFE software & foundation	Lecture, Oral presentation	Lab Test, Quiz and Report	CLO 1
2-4	Design of Isolated Single Column Footing	Lecture, Discussion	Lab Test, Quiz and Report	CLO 2
5-6	Design of Combined Footing	Lecture, Discussion	Lab Test, Quiz and Report	CLO 3
7-9	Design of Mat Foundation	Lecture, Discussion	Lab Test, Quiz and Report	CLO 3
9-11	Design of Pile and Pile cap	Lecture, Discussion	Lab Test, Quiz and Report	CLO 3
12-13	**Practice, Review/Reserved Day**	Lecture, Discussion	Lab Test, Quiz and Report	CLO 4
14-15	Lab Report Assessment, Self study	Lecture, Discussion	Lab Test, Quiz and Report	CLO 4
16-17	Lab Test, Viva, Quiz, Overall Assessment, Skill Development Test (Competency)	Lecture, Discussion	Lab Test, Quiz and Report	CLO 4

Introduction to csi SAFE software & foundation

Week 1

1. Introduction

Foundations are components of a structure which ultimately transfer loads from the superstructure safely into the soil. Foundations are alternatively named as *footings*. Depending on the mechanism and the depth of the soil to which the loads are transferred, foundations can be broadly categorized into shallow foundations and deep foundations. Shallow foundations are those where the foundation loads are transferred to a soil at shallow depth and where the load transfer mechanism is predominantly through resistance of the soil at the bottom of the foundations. The depth of shallow foundations is usually less than their largest dimension. Foundations which transfer the superstructure load to a soil layer found a large depth are said to be deep foundations. The resistance soil force on deep foundations is generated through two mechanisms; namely, tip resistance and side friction. This tutorial is entirely about shallow foundations.

Depending on the number and arrangement of columns they support and their geometry, shallow foundations can be classified into isolated, combined, strip and mat.

Isolated footings (or as sometimes called spread or single footings) are footings which support only one column. The load from the column may be only concentrated load, only moment or a combination of concentrated load and moment. The shape of the footings may be square, rectangular or circular.

Combined footings are footings which support a number of columns along a single row. The number of columns is usually two. However, more than two columns can also be present on combined footings all aligned along a single row. The ratio of the longer side to the shorter side of these foundations shouldn't be considerably large. The shape of combined foundations can be either rectangular or trapezoidal.

Strip foundations are those foundations where the shorter dimension of the foundation is negligible as compared to the longer dimension. The load on strip foundations may come either from a number of columns are from a long wall.

Mat foundations are foundations which support many columns arranged in more than one row i.e. in two dimensions. These foundations may support all the columns of the entire building or a number of columns from portion of the building.

During this tutorial, it is assumed that the user has prior knowledge about the theoretical design process of shallow foundations.

Design of isolated Single Column Footing

Week 2-3

2. Isolated (Single) Footing

In this part of the tutorial, an isolated footing will be designed in three different ways. First, a square isolated foundation will be designed using the built-in model of the software itself. Then a rectangular isolated foundation will be designed using grids. Lastly, a circular foundation will be designed by importing the geometry from AutoCAD. In all sections, a design problem will be given first and the procedures which will be followed to carry out the design process in SAFE program will be detailed.

2.1. From the Built-in Model

STEP 1:

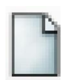
Design a square isolated footing with the following parameters:

- *Dimension of the footing*: 2.5mX2.5m
- *Dimension of the column*: 0.5mX0.5m
- *Ultimate bearing capacity of the soil* : 100kN/m^2
- *Maximum allowable settlement of the foundation*: 10mm
- *Concentrated load (DL)* : 300kN
- *Concentrated load (LL)*: 140kN
- *Grade of concrete*: C-25 (25MPa 28-day characteristics cube strength)
- *Grade of reinforcement bar (rebar)*: S-300 (300MPa characteristics yield strength)
- *Overall thickness of the foundation*: 500mm
- *Concrete cover* : 50mm
- *There are no bending moments on the column*

Creating the Model

To create a footing model from the built-in program, there are three ways.

a. Go to **File** menu and click on **New Model …**

b. Click on the icon . This icon is the first among the list of icons just below the menu bar.

c. Simply click on the **Ctrl** and **N** keys simultaneously (**Ctrl+N**).

Performing one of the above three ways results in the popping-up of the following **'New Model Initialization'** window.

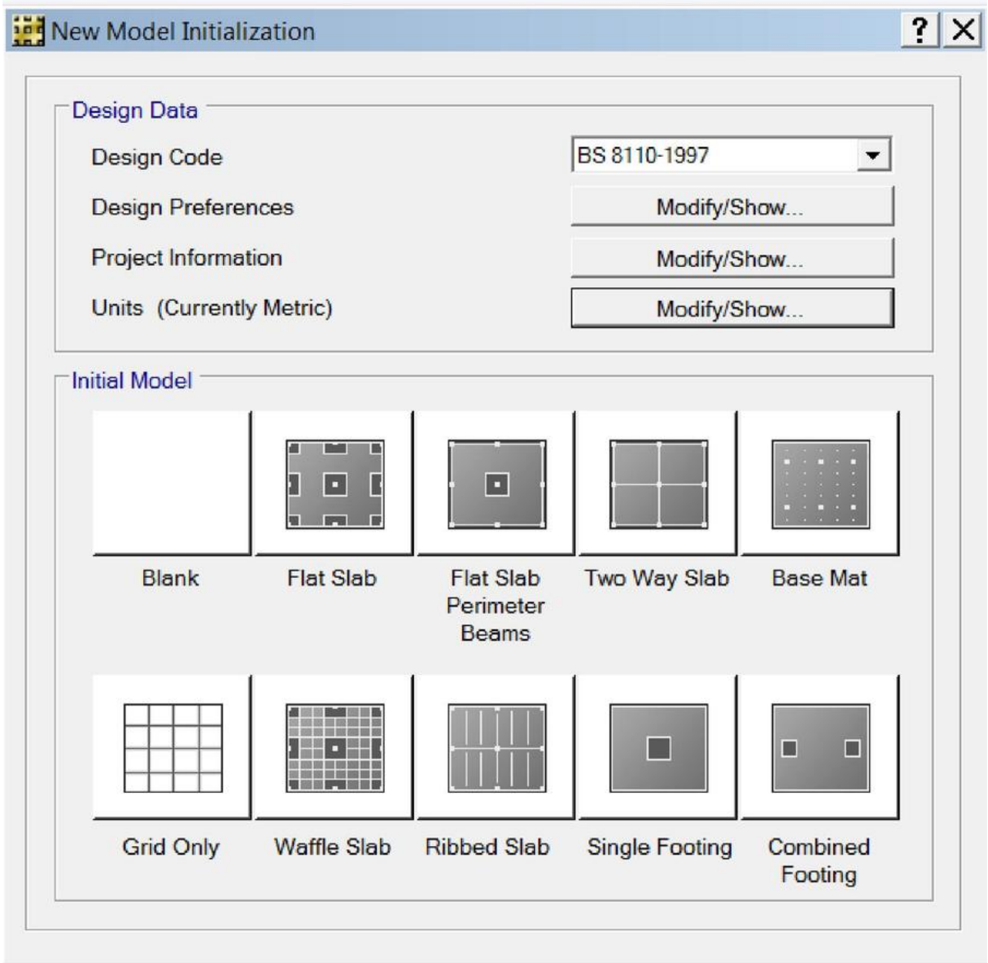
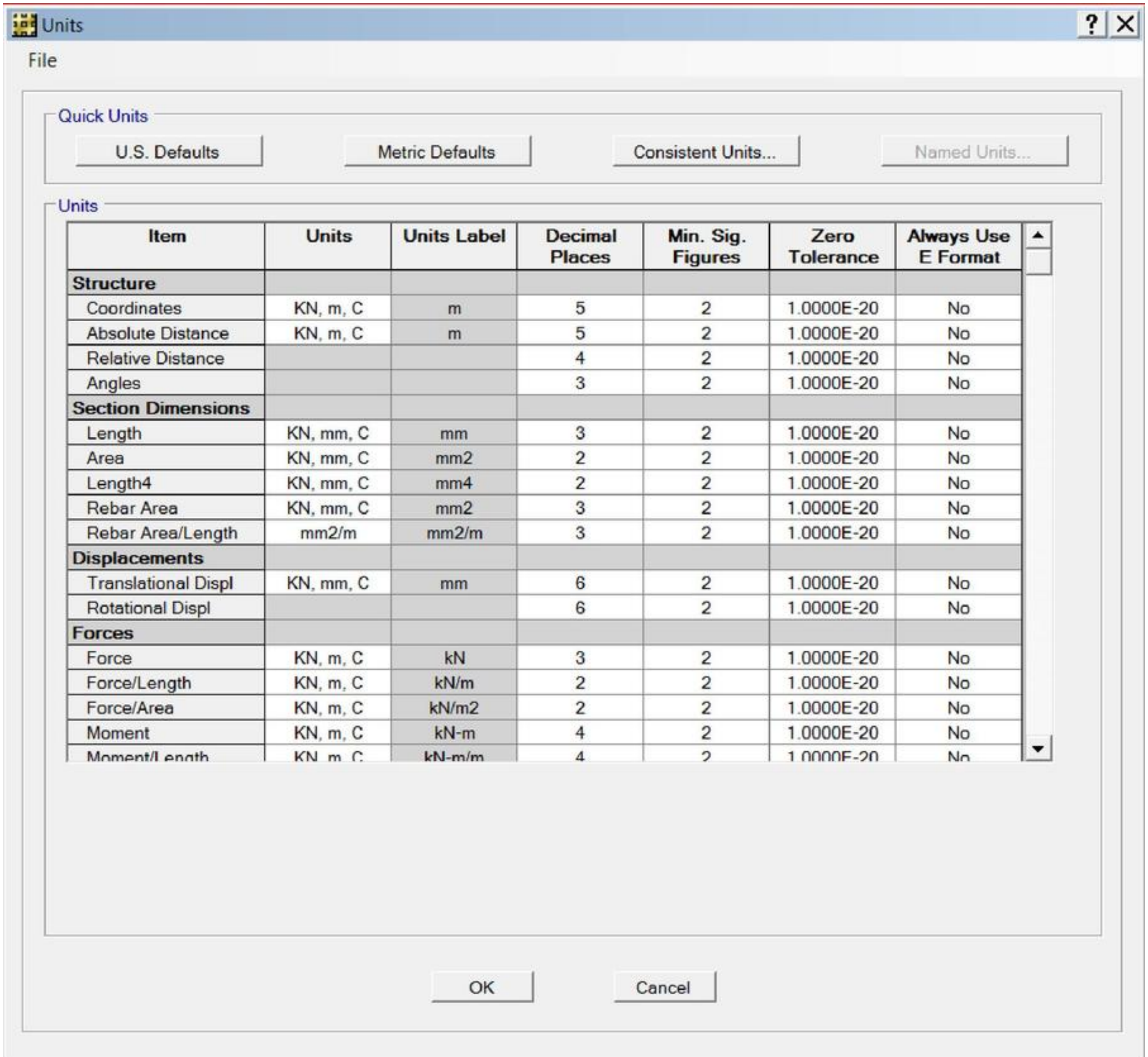


Figure 1

This window consists of two parts: the '**Design Data**' part and the '**Initial Model**' part. In the '**Design Data**', there are '**Design Code**', '**Design Preferences**', '**Project Information**' and '**Units**'.

The '**Design Code**' consists of a list of selected design codes from many countries. Depending on the suitability of the code for your country, you may select one among the list of codes. The design code which follows similar design philosophy to my country code is BS 8110-1997. So, this design code is selected.

After selecting the design code, it is better to first select the units to be used in the design and analysis process. These units can be selected by clicking on the '**Modify/Show**' button in front of '**Units**'. The following window pops up when the button is clicked. To choose metric units, click on '**Metric Defaults**' button. The click results is metric units for best practices to be selected. It can be observed that, even though all the units are metric, they are not consistent. To select a preferred consistent unit, click on the '**Consistent Units**' button. In this '**Units**' window, the decimal places and minimum number of significant digits for any quantity can also be modified.



Item	Units	Units Label	Decimal Places	Min. Sig. Figures	Zero Tolerance	Always Use E Format
Structure						
Coordinates	KN, m, C	m	5	2	1.0000E-20	No
Absolute Distance	KN, m, C	m	5	2	1.0000E-20	No
Relative Distance			4	2	1.0000E-20	No
Angles			3	2	1.0000E-20	No
Section Dimensions						
Length	KN, mm, C	mm	3	2	1.0000E-20	No
Area	KN, mm, C	mm2	2	2	1.0000E-20	No
Length4	KN, mm, C	mm4	2	2	1.0000E-20	No
Rebar Area	KN, mm, C	mm2	3	2	1.0000E-20	No
Rebar Area/Length	mm2/m	mm2/m	3	2	1.0000E-20	No
Displacements						
Translational Displ	KN, mm, C	mm	6	2	1.0000E-20	No
Rotational Displ			6	2	1.0000E-20	No
Forces						
Force	KN, m, C	kN	3	2	1.0000E-20	No
Force/Length	KN, m, C	kN/m	2	2	1.0000E-20	No
Force/Area	KN, m, C	kN/m2	2	2	1.0000E-20	No
Moment	KN, m, C	kN-m	4	2	1.0000E-20	No
Moment/Length	KN, m, C	kN-m/m	4	2	1.0000E-20	No

Figure 2

By clicking on the '**Modify/Show**' button in front of the '**Project Information**', information regarding the project, the company and the model can be entered.

By clicking on the '**Modify/Show**' button in front of the '**Design Preferences**', the user preferences regarding the design code, concrete cover for slabs and beams, post-tensioning can be changed. The following **'Design Preferences'** window pops-up when the button is clicked. The '**Min. Cover Slabs**' tab of the window may be of interest for this particular problem as footings will be modelled as slabs.

Thus, click on this tab. This is the tab where the concrete cover and preferred rebar size will be entered.

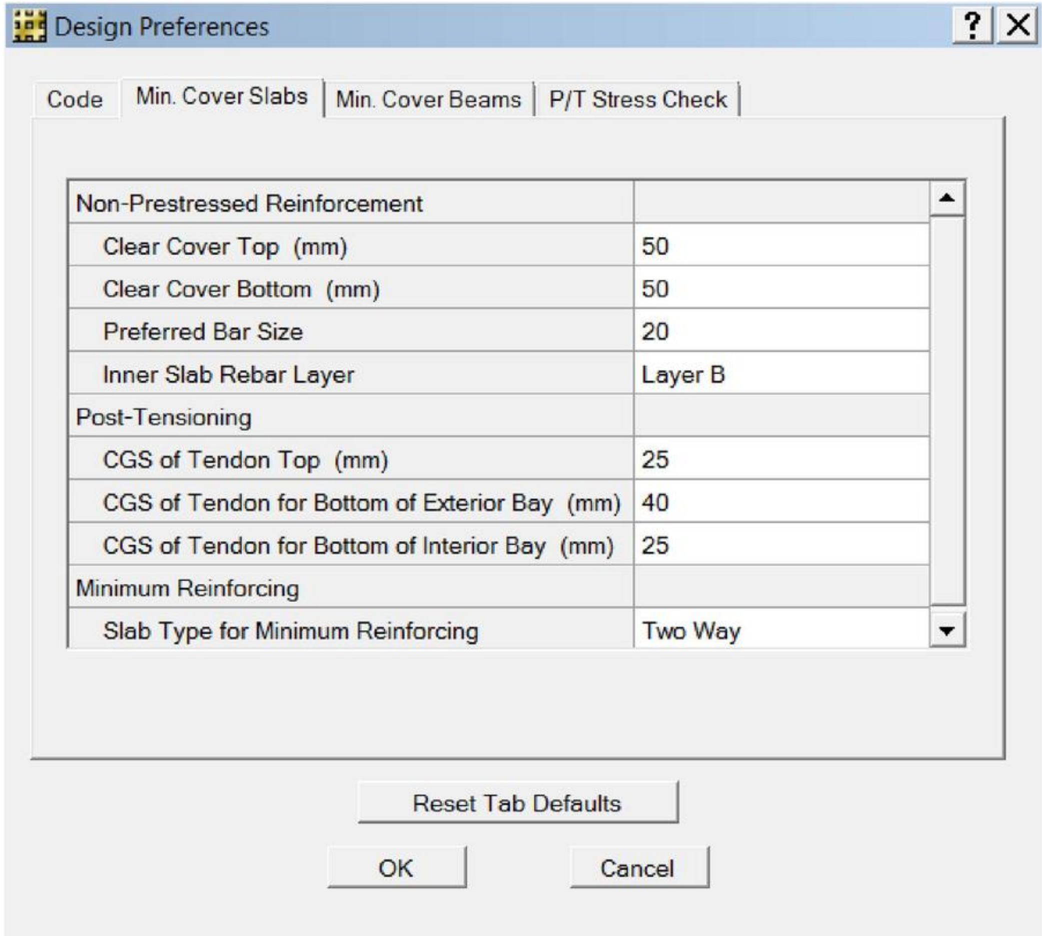


Figure 3.

In the 'Min. Cover Slabs' tab, for '**Non-Prestressed Reinforcement**' , both the '**Clear Cover Top**' and '**Clear Cover Bottom**' should be set to 50mm as the concrete cover in this design problem stated to be 50mm. The '**Preferred Bar Size**' can be set to any reasonable value. Here, the '**Preferred Bar Size**' is set to 20 which is the diameter of the rebar in mm which will be used as the main reinforcement in the foundation. Leave the rest at they are.

Once everything is set correctly in the '**Design Data**' box, go to the '**Initial Model**' box and select the suitable model. Since, an isolated (single) footing will be designed click on the big square button just above '**Single Footing**'. The button itself shows a plan view of an isolated footing. When the button is clicked, the following **'Single Footing'** window will pop up. In this window, there are three boxes: '**Plan Dimensions'**, '**Load**' and '**Properties**'.

In the '**Plan Dimensions**' box, the footing and column dimensions will be entered. While entering the footing dimension, it is the distance from the center of the column to each of the edge which will be entered not the actual dimension of the footing. Since the column is located centrally on a 2.5mX 2.5m square footing, the distance from the center of the column to all the edges of the footing will be 1.25m. Thus, set all the values of '**Left Edge Distance**', '**Right Edge Distance**', '**Top Edge Distance**' and '**Bottom Edge Distance**' to be 1.25m. These values may be accordingly changed depending on the location of the column on the footing.

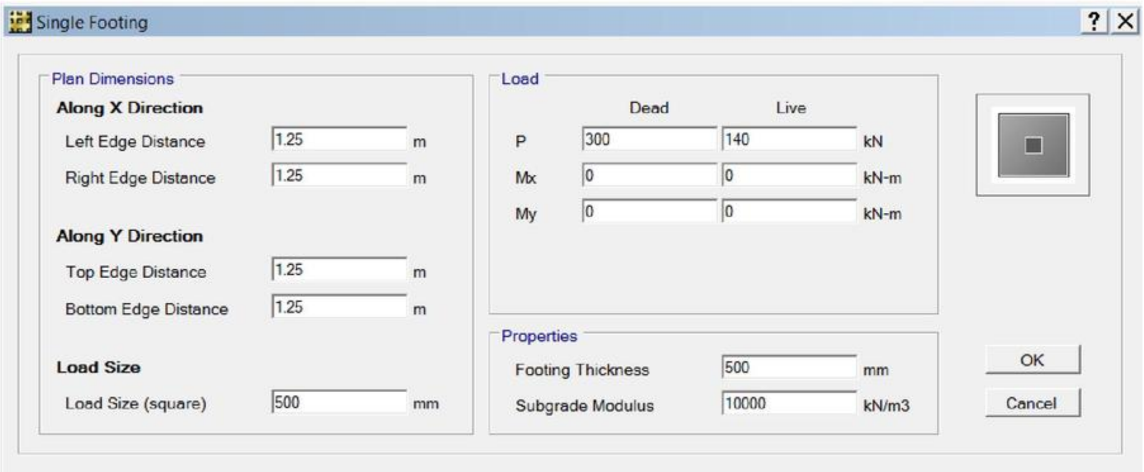


Figure 4.

The text field in front of the '**Load Size (Square)**' is the field where the column size for square columns will be entered. Since in this design problem, the column is of dimensions 500mmX500mm, enter 500 in the text field as the unit is already in mm. For column shapes other than square, just enter a preliminary column dimension and you can change both the shape and size of columns at a later time.

In the '**Load**' box, enter the concentrated loads and moments in the column depending on their type as '**Dead**' or '**Live**'. In this particular problem, there is 300kN concentrated dead load and 140kN concentrated live load with no moments. Thus, enter these values as shown in Fig. 4.

In the '**Properties**' box, the footing thickness and the subgrade modulus of the soil will be entered. The footing thickness of the footing is given to be 500mm. Thus, enter this value as shown in fig. 4. However this value may be revised depending on the punching shear requirements at a later time. The subgrade modulus of a soil is the ratio of the increment of contact pressure to the corresponding change in settlement. In the absence of a supportive test to determine its value, it can also be safely approximated by the ratio of the bearing capacity of the soil to the maximum permissible settlement in the soil. In this particular problem, the subgrade modulus can be approximated by $100kN/m^2/10mm = 10000kN/m^3$. Thus enter this value in the text field in front of the 'Subgrade Modulus'. Then press the '**OK**' button. This exits the '**Single Footing**' window and a plan view of the foundation will be displayed on the main screen. You can zoom in and zoom out using the wheel of your mouse or using the tool bar under the menu bar.

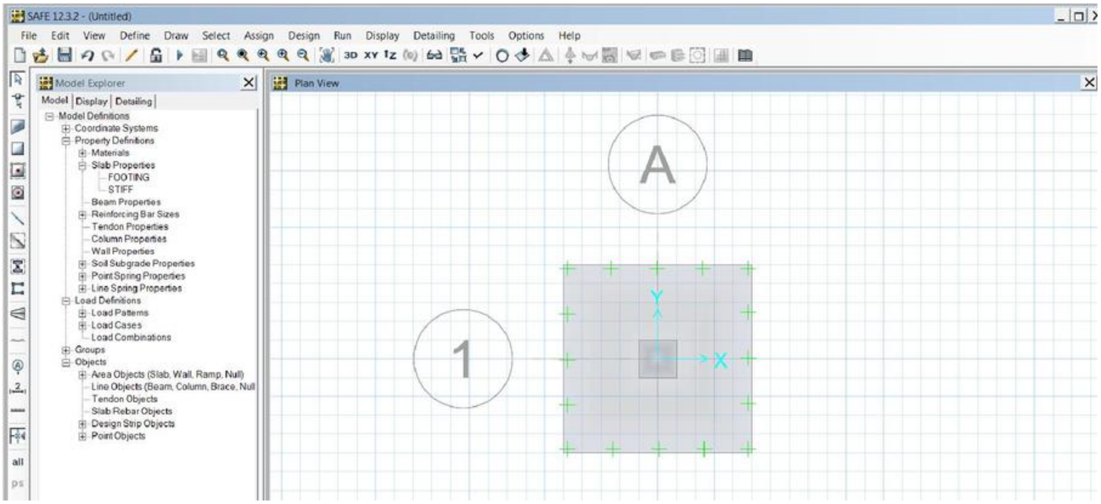


Figure 5

The column shape and dimensions can be changed at this stage. The column is shown at the center of the plan view in Fig.5 with somewhat darker color. Changing the originally square column to rectangular column is a pretty easy process and there are different options to do that. The simplest one is to draw the column from the '**Draw**' menu. For this, delete the existing column with unwanted shape and go to '**Draw**' menu and click on '**Quick Draw Areas Around Points**'. Or simply click on the equivalent icon  on the left hand side tool bar and the following window will pop up.

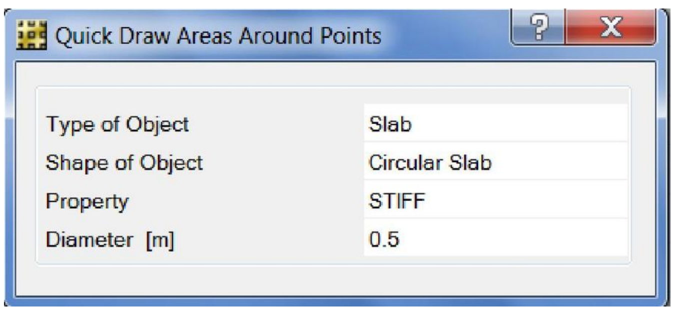


Figure 6

In this window, change the '**Shape of Object**' to the shape that you want and the property to '**STIFF**', enter the dimensions and click on the footing where you want the column.

The other way to change the shape of columns is to left click on any edge, then right click on the left clicked edge and a '**Point Object Information**' window pops up. In the '**Geometry**' tab of this window, you can change the X,Y,Z coordinates of the corner to any desired value. Through similar process on other corners of the column, the shape of the column can be converted into any desired quadrilateral or triangular shape. To change the column into other shapes and sizes, hover over the column in the plan view and right click on any point inside the column. On the right click, the following window pops up.

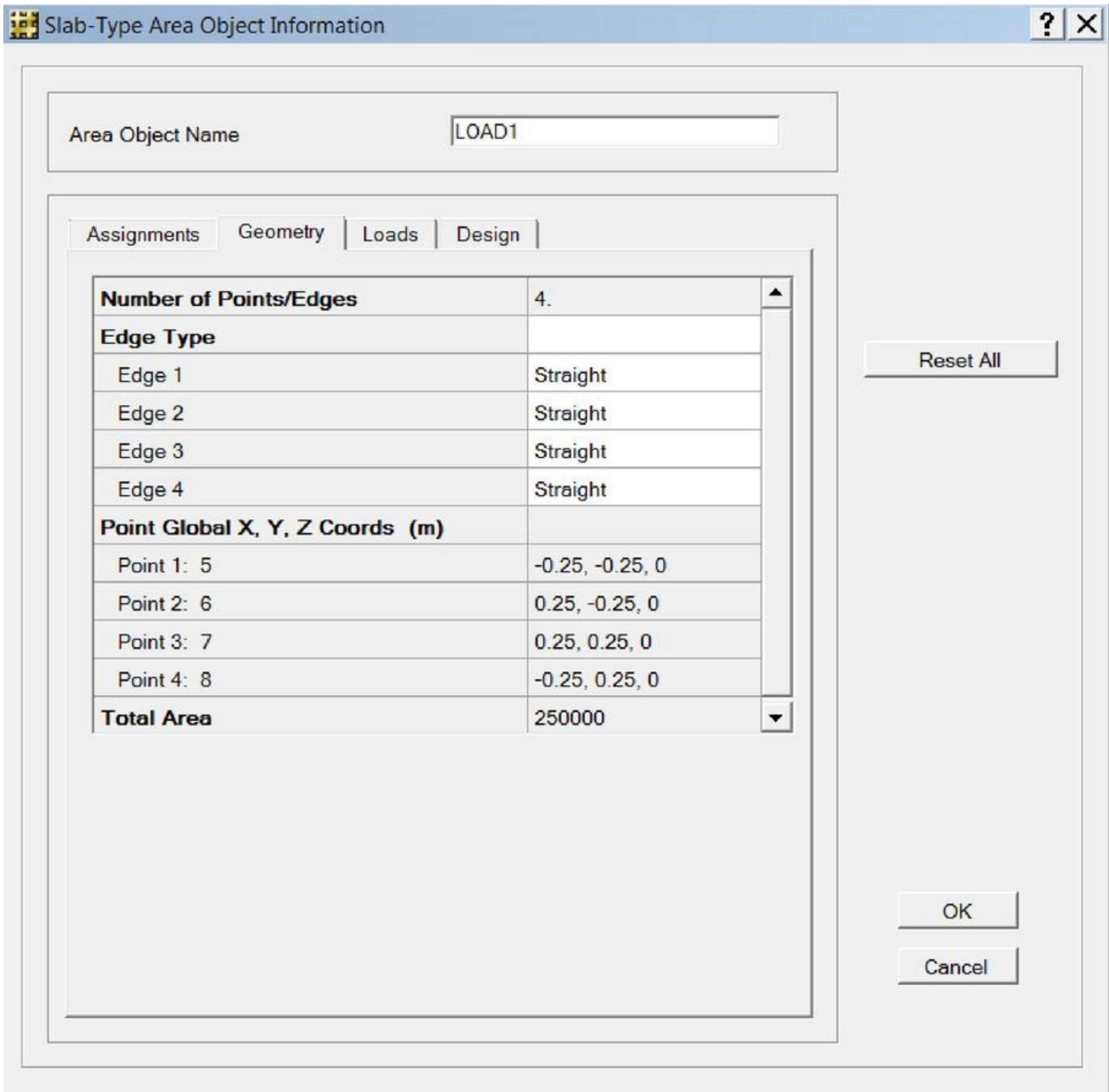


Figure 7

Then by clicking on the '**Geometry**' tab, any edge of the originally 'Straight' side of the square can be changed to some selected geometric shapes.

STEP 2: Defining Material Properties

The materials which are involved in the design of the footing should be defined before the analysis. These materials are the concrete, the reinforcement bar (rebar) and the soil which is already defined. Thus, the concrete property and the rebar property will be defined here.

The material definition can be carried out in two ways.

The first one is through the '**Define**' menu in the menu bar. The other one through the model explorer on the left hand side of the home window. To define material properties through the former method, click on '**Define**' menu and again click on '**Materials…**' resulting in the following window depending on prior material definitions.

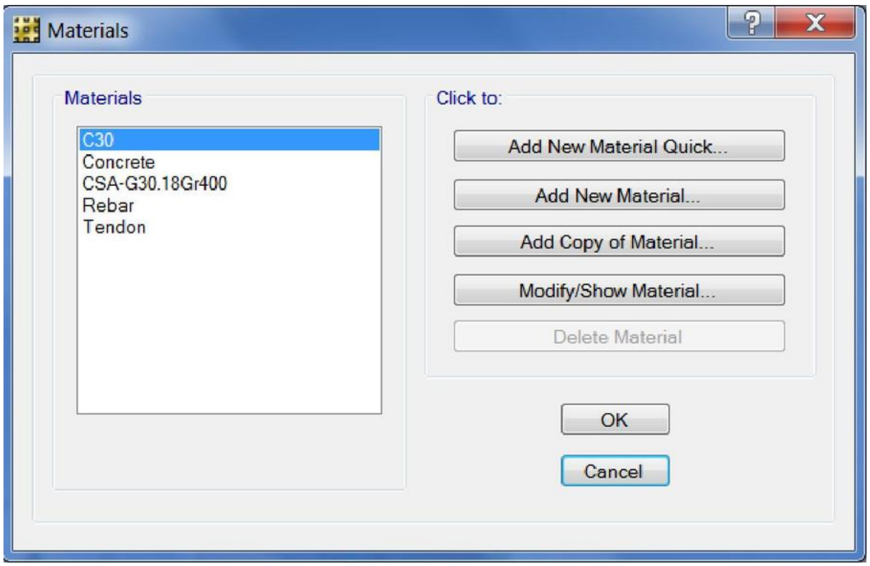


Figure 8

The list in the '**Materials**' box may not be exactly as it appears in your widow. However, that doesn't bring any change in the outcome of the design process as you can customize this list any time.

The '**Add New Material Quick…**' button allows you to define materials quickly from a list of pre-defined materials. The '**Add New Material**' button allows you to define materials by changing their properties. The '**Add Copy of Material**' button allows you to define a material with same property as an already defined material. The '**Modify/Show Material**' button displays the property of an already defined material with the possibility of modification. The '**Delete Material**' button, when active, deletes a defined material property.

For this particular problem, we will define concrete and rebar properties through the '**Add New Material**' button. Thus, click on this button. This can also be achieved by right clicking on '**Materials**' on the model explorer and selecting '**New Material**'. The following window pops up in ether cases.

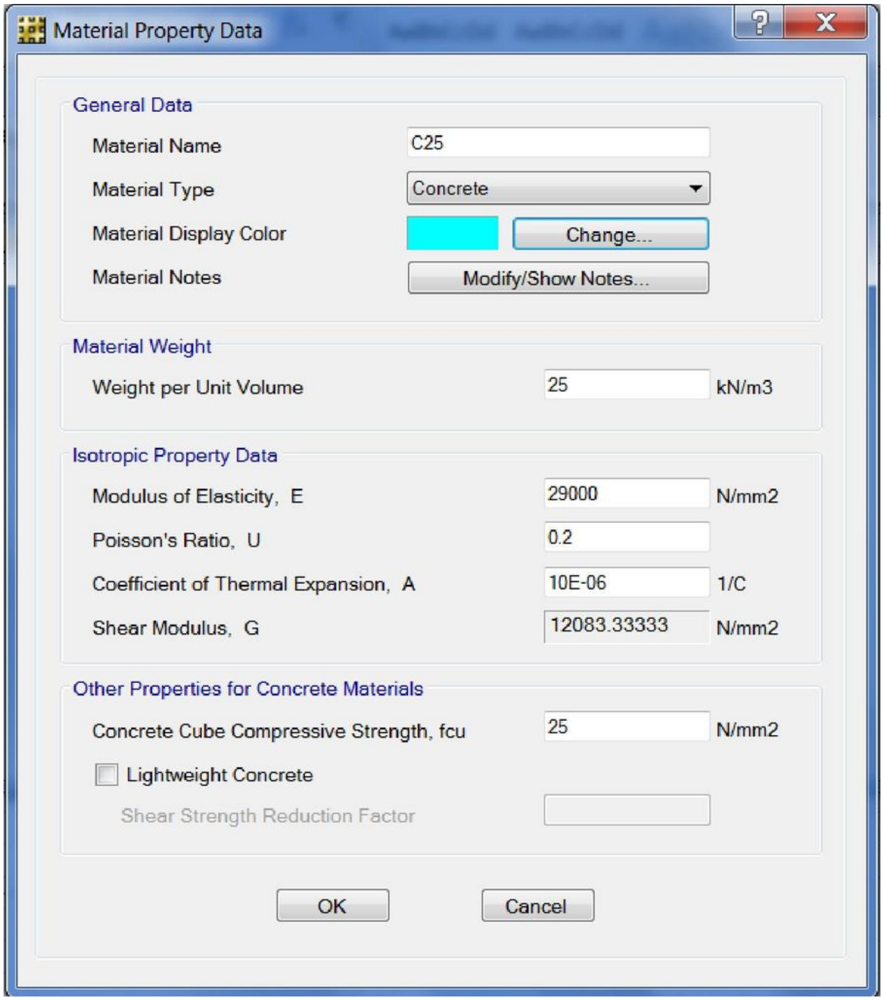


Figure 9

This is the window where material properties are defined. First, let us define the concrete property which will be used in the design. Put any name for the material in the text field in front of '**Material Name**'. But, it is important to make sure that the concrete with the defined material name is assigned for the footing. For this particular problem, let the name of the concrete to be used be '**C25**'. Since we are defining a concrete, the '**Material Type**' should be set to '**Concrete**'.

The unit weight of reinforced concrete may vary depending of the design code of your country. Thus, enter the unit weight of reinforced concrete stipulated in your country code in the text field in front of '**Weight per Unit Volume**'. For this particular example, we will use 25kN/m^3.

For C-25 concrete, the modulus of elasticity according to BS 8110-1197 is around 29GPa. Thus, enter this value in the text field in front of '**Modulus of Elasticity, E**'. If you selected another design code in step 1 while creating the model, you should refer to actual value of this parameter from the code and enter it accordingly. Be aware of the units though.

The values of Poisson's ratio and coefficient of thermal expansion may also be defined in the design code and should be entered accordingly. For this particular problem, a value of 0.2 for '**Poisson's Ratio, U**' and a value of 10x10^{-6}/^{o}C for '**Coefficient of Thermal Expansion, A**' will be entered. The '**Shear Modulus, G**' will be automatically calculated in an uneditable text field.

The grade of concrete for this particular problem is C-25 which is a concrete with 28 day characteristics cube compressive strength of 25MPa. The concrete designation may be different for different country codes but the concept is the same. Therefore, enter 25 in the text field in front of '**Concrete Cube Compressive Strength, fcu**'.

If a lightweight concrete is used, check on '**Lightweight Concrete**' and enter the corresponding '**Shear Strength Reduction Factor**' in the space provided.

When you press on '**OK**', a concrete material with the above properties will be added to the list of materials. This material will be assigned for the footing before the analysis.

After defining the concrete property, the program returns to the window shown in Fig. 5. To define a rebar property, we will follow the same procedure as we followed while defining the concrete property. Since a new rebar property will be defined, click on the '**Add New Material…**' button. A '**Material Property Data**' window pops up and when you change the '**Material Type**' to '**Rebar**', the window appears to look like the following.

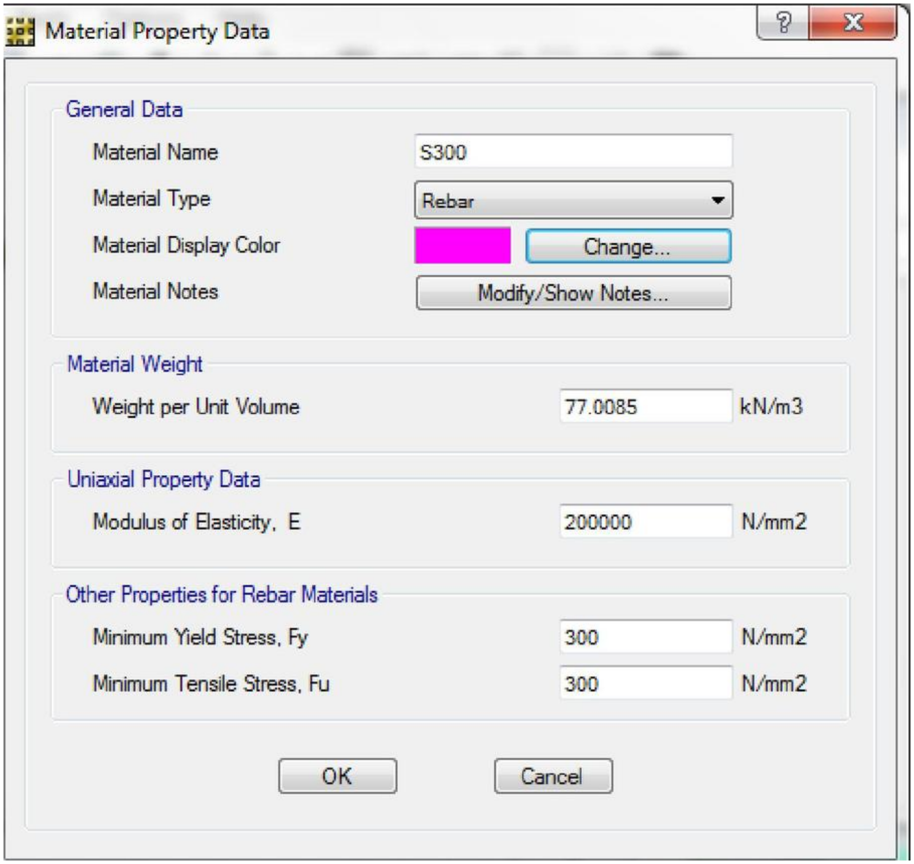


Figure 10

Change the '**Material Name**' to any name you want. Here, we name it '**S300**'. The material type should be '**Rebar**'. The weight per unit volume of steel is stipulated in the design code. For BS 8110-1197, the weight per unit volume is 77.0085kN/m^3. Thus, enter this value in the text field in front of '**Weight per Unit Volume**'. The modulus of elasticity for reinforcement bars according to the same design code is 200GPa. Thus enter this value in the text field in front of '**Modulus of Elasticity, E**' considering the unit.

In the '**Other Properties for Rebar Materials**' box, two quantities are mentioned: minimum yield stress and minimum tensile stress for the reinforcing material. The values of these parameters will be specified in the design code which you defined earlier. If the code assumes that the rebar material exhibits elastic perfectly plastic behavior, the values of these two quantities will be the same. The grade of steel to be used for this particular example is S-300. The yield stress for this type of reinforcement bar is 300MPa. Since the design code of my country assumes that rebars exhibit elastic perfectly plastic behavior, the minimum tensile stress will also be 300MPa. Thus enter 300 in both text fields in front of the '**Minimum yield**

stress, Fy' and '**Minimum Tensile Stress, Fu**'. Then press '**OK**' in both '**Material Property Data**' and '**Materials**' windows concluding the material definition step.

STEP 3: Defining Footing and Column Properties

After defining the material properties, the footing and column properties can be defined. This definition can take place in two ways: from the menu bar and from the model explorer. In SAFE software, footings are modelled as 'footings' and foundation columns are modelled as 'stiff'.

To define footing and column properties from the menu bar, go to '**Define**' menu and click on '**Slab Properties…**'. The following window will pop up.

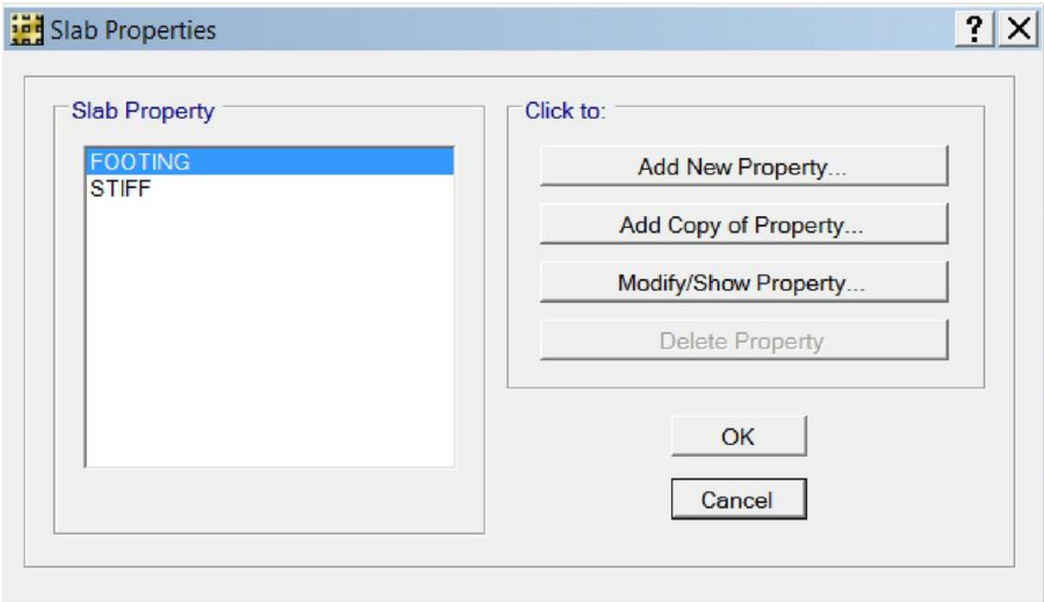


Figure 11

In the '**Slab Property**' box, two foundation components are listed. The names in this box for your particular design may be different from the list of names in the box. The '**FOOTING**' corresponds to the bottom slab portion of the foundation and '**STIFF**' corresponds to the foundation column. The '**Add New Property…**' button prompts the user to enter new properties for the footing and foundation column while the '**Add Copy of Property…**' copies the property of an existing slap. The '**Modify/Show Property…**' allows the user to show the property of an existing component with the possibility of modification. When the '**Delete Property**' button is active, it allows the user to delete an existing slab property.

In this case, we use the '**Modify/Show Property…**' button. However, one can also use the '**Add New Property…**' button. To use '**Modify/Show Property…**'

button, highlight the foundation component to be modified like the '**FOOTING**' is highlighted in Fig. 8 and click on this button. Since '**FOOTING**' is highlighted, we will be modifying the footing property. The following window appears after the click.

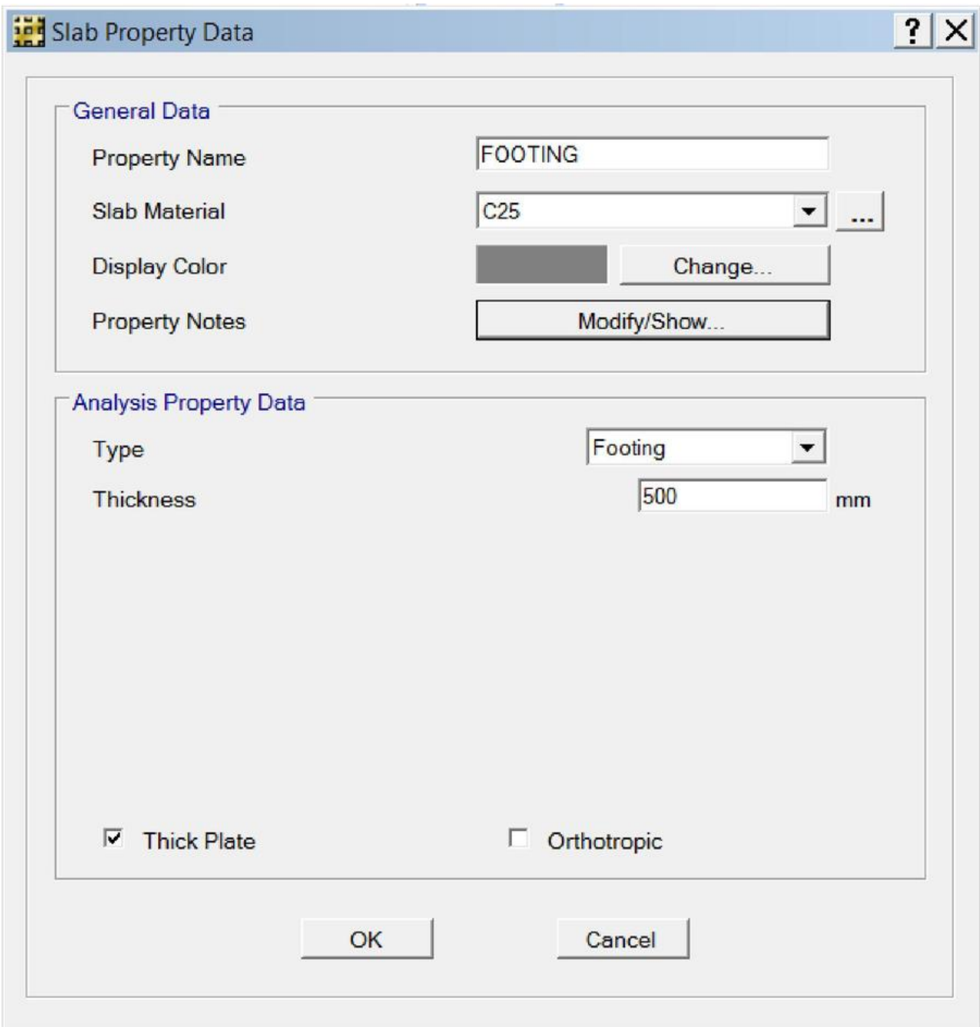


Figure 12

The '**Property Name**' will be automatically assigned to '**FOOTING**'. The '**Slab Material**' should be set to the concrete grade which is defined in step 2. Since the name of the concrete material defined in step 2 for this particular problem is '**C25**', a material with this name should be selected from the list.

In the '**Analysis Property Data**' box, '**Type**' should be set to '**Footing**' as we are modifying the footing property. The '**Thickness**' value should be set to the same value as the one which is used while creating the model. Since the thickness of the footing is 500mm, this value is entered in the text field corresponding to '**Thickness**'. As footings are modelled as thick plates, check the '**Thick Plate**' option. The '**Orthotropic**' check box is selected when a footing with irregular dimension is to be used.

When you press '**OK**', the '**Slab Property Data**' window will be exited and the '**Slab Properties**' window in Fig. 11 gets activated. Now, the property of the foundation column will be modified. To do this, highlight '**STIFF**' in the '**Slab Property**' box and click on the '**Modify/Slab Property…**' button. The following window appears after the click.

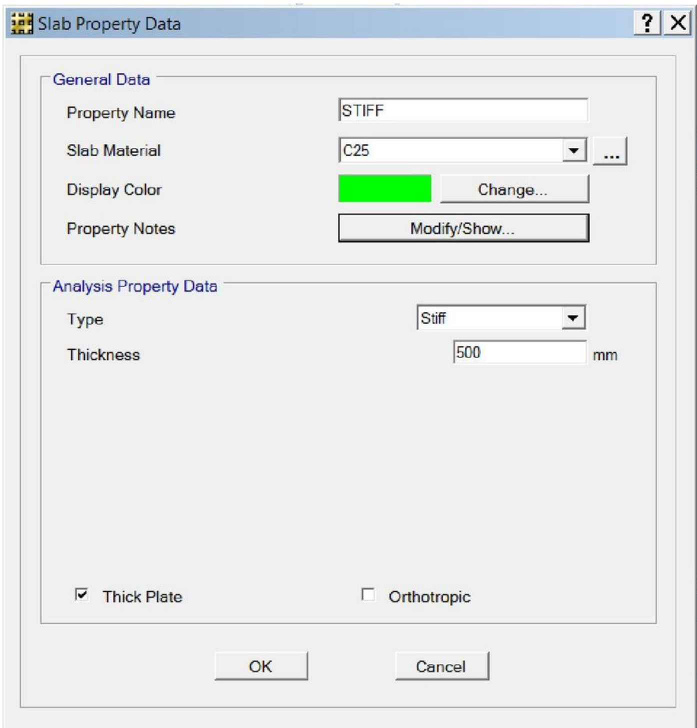


Figure 13

The '**Property Name**' will be automatically assigned to '**STIFF**'. The '**Slab Material**' should be set to the concrete grade which is defined in step 2. Since the name of the concrete material defined in step 2 for this particular problem is '**C25**', a material with this name should be selected from the list.

In the '**Analysis Property Data**' box, '**Type**' should be set to '**Stiff**' as we are modifying the footing property. The '**Thickness**' value should be set to the same value as the one which is used while creating the model. Since the thickness of the foundation column and the footing are equal, the thickness of the '**Stiff**' is 500mm, this value is entered in the text field corresponding to '**Thickness**'. As foundation columns are modelled as thick plates, check the '**Thick Plate**' option. The '**Orthotropic**' check box is selected when a column with irregular dimension is to be used.

When you press '**OK**', the '**Slab Property Data**' window will be exited and the '**Slab Properties**' window in Fig. 8 gets activated. Again press '**OK**' and exit the window for defining the footing and foundation column.

STEP 4: Defining Load Patterns, Load Cases and Load Combinations

The loads on the foundation should be defined accordingly before the analysis. First, the load pattern should be defined. This can be done from the '**Define**' menu or from the '**Model Explorer**'. This time, we will do it from the model explorer. In the model explorer, expand '**Load Definitions**' and you will see '**Load Patterns**'. When you expand '**Load Patterns**', you will see '**DEAD**' and '**LIVE**'. At the end, the model explorer appears to look like:

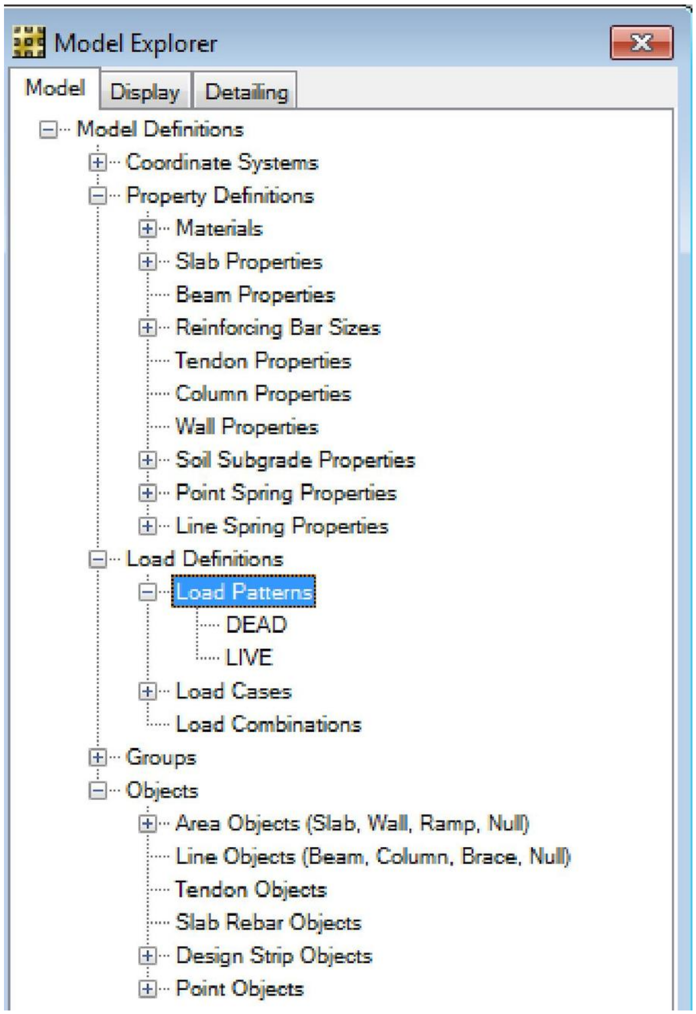


Figure 14

Now, right click on '**Load Patterns**' and click on '**New Load Pattern**' and the following window pops up.

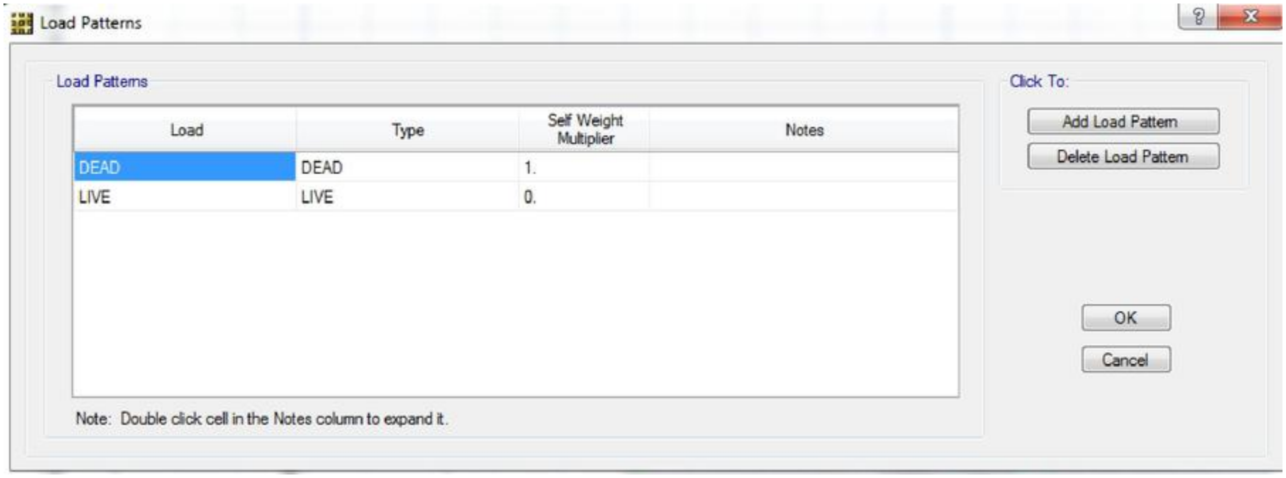


Figure 15

In the '**Load Patterns**' window, two load patterns are already defined: '**DEAD**' and '**LIVE**'. The '**Type**' of the '**Load**' should also be changed accordingly. There are many options for type of loading. The '**Type**' for dead loads should be set to '**DEAD**' and for live loads '**LIVE**'. If there are other types of load patterns on the foundation such as earth quake load, you can add the load pattern with the '**Add Load Pattern**' button. You can also delete any undesirable load pattern using the '**Delete Load Pattern**' button. Since in this example, we have only dead and live loads, we will leave the existing load patterns as they are. The '**Self Weight Multiplier**' value should also be changed accordingly. This value imparts the option whether to include or exclude the self-weight of the foundation in addition to external loads. If the self-weight of the foundation is already included as an external dead load or if you want to exclude the effect of self-weight from the analysis, the value under '**Self Weight Multiplier**' should be set to zero. In this example, we will consider the self-weight as an additional load to the external dead load. Thus, the value under '**Self Weight Multiplier**' for the '**DEAD**' load is one. For the '**LIVE**' load, it will be zero. Press '**OK**' and the window will be exited.

After this, the load cases will be defined. Load cases are used to dictate the way the loads are applied (statically or dynamically) or the way the structure responds (linearly or non-linearly) for the defined load patterns. To define a load case, go to '**Define**' menu and click on '**Load Cases…**'. The following window will pop up after the click.

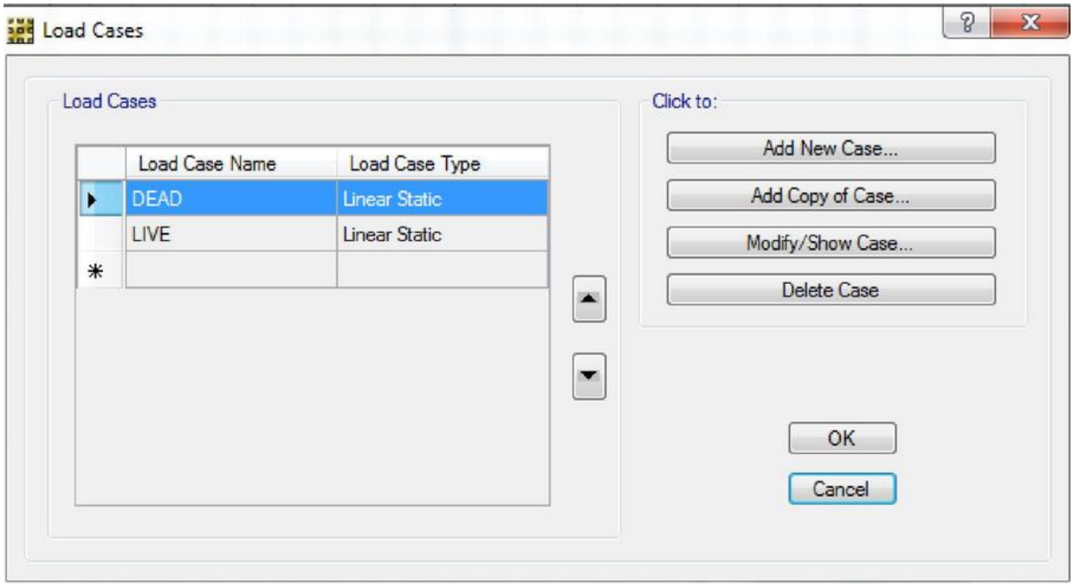


Figure 16

The load patterns which were defined earlier will automatically appear in the list of '**Load Case Name**' of the '**Load Cases**' window. The '**Load Case Type**' column shows the way in which each load pattern will be applied during analysis. If you want to modify this, highlight the load pattern for which you are going to change the load case type and click on the '**Modify/Show Case…**' button. If you click this button, the following window will appear. In this window, you can change the way the load is applied from the '**Load Case Type**' box. The way the structure responds can also be selected from the '**Analysis Type**' box. This problem '**Static**' is for the '**Load Case Type**' and '**Linear**' is selected for the '**Analysis Type**' since the load is static and the foundation responds linearly. The scale factor for the dead load in the '**Loads Applied**' box will be left as one. Press '**OK**' and exit the window.

The load case type for the live load should also be '**Linear Static**'. Otherwise, it should be changed by clicking the '**Modify/Show Case…**' button to linear static case. If both the load case types are as desired click '**OK**' and exit the '**Load Cases**' window.

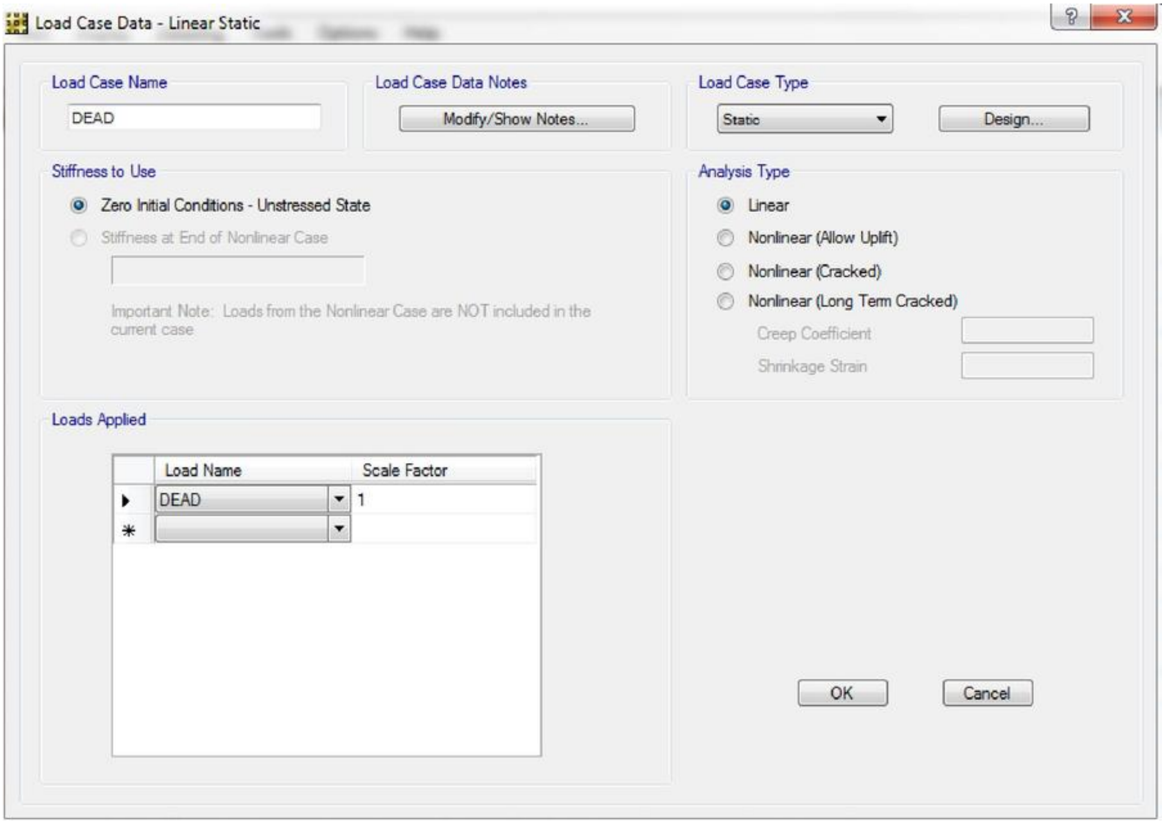


Figure 17

Definition of the load combinations will be the next step. Two load combinations will be considered in this example for ultimate limit state and serviceability limit state. To define load combinations, go to '**Define**' menu and click on '**Load Combinations…**' and the following window pops up.

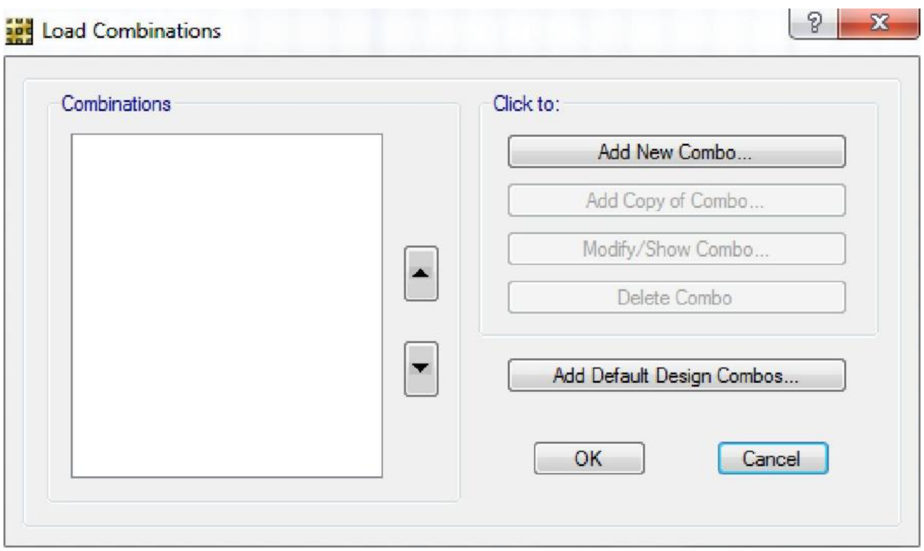


Figure 18

New load combinations will be added through the '**Add New Combo…**' button. Here, two load combinations will be added; one for the ultimate limit state the other for the serviceability limit state. When the '**Add New Combo…**' button is clicked, the following window appears.

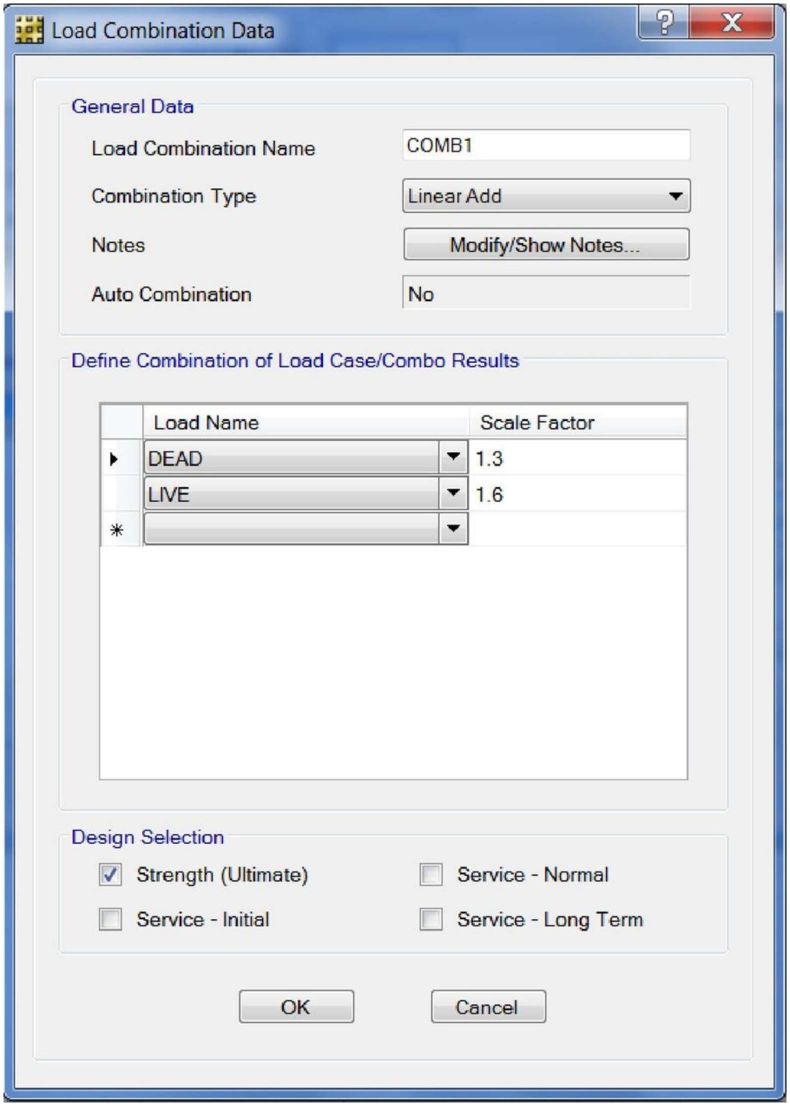


Figure 19

In the '**Load Combination Data**' window, any name can be given for the load combination. The '**Combination Type**' should be set to '**Linear Add**' as the component loads (dead and live) will be added linearly. However, there are also other options from the drop down menu in front of '**Combination Type**'. In the window shown in Fig. 19, the two loads (dead and live) should be activated below the '**Load Name**' column of the '**Define Combination of Load Case/Combo Results**' box. The values in the '**Scale Factor**' column correspond to the partial

safety factors for load for the failure mode under consideration. These partial safety factors are specified in the design code of your country. In the design code of my country, the partial safety factor for dead loads for ultimate limit sate case is 1.3 and for serviceability limit state is 1 for the cases where there are only dead and live loads. For such load patterns, the partial safety factor for live loads for ultimate limit sate case is 1.6 and for serviceability limit state is 1. Thus, enter a value of 1.3 in front of '**DEAD**' and 1.6 in front of '**LIVE**' in '**Scale Factor**' column. The failure condition which is being under consideration can be defined by selecting and deselecting the check boxes in the '**Design Selection**' box. Since the above scale factors are for the ultimate limit state, check on '**Strength (Ultimate)**'. The '**Load Combination Data**' window for the serviceability limit state looks like the following window.

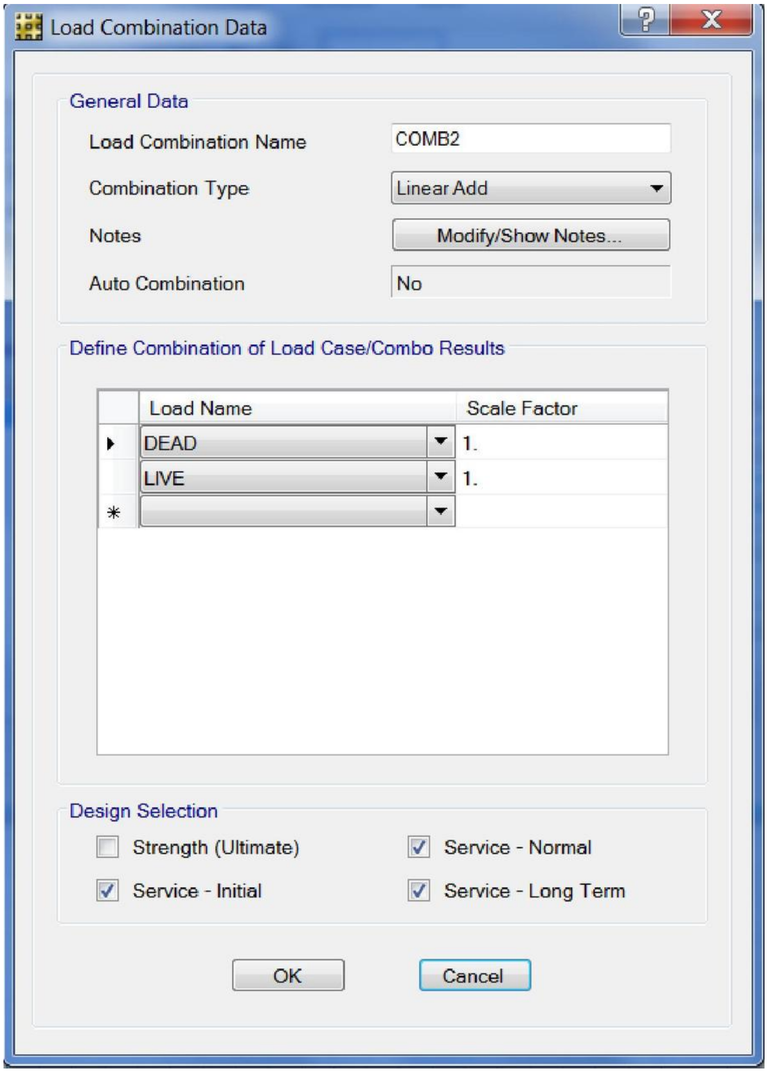


Figure 20

STEP 6: Drawing Design Strips

Design strips determine the way in which different quantities related to the reinforcement calculation are calculated. Forces are integrated across the design strips. Thus, the larger the width of coverage of the design strips within the given structure, the higher will be the calculated values of the bending moments and shear forces. Thus, an optimum width of strip is required compromising the safety and economical requirements. The width of the design strip will be specified in the design code. According to the code of my country, the width of design strips for isolated foundations is 1m. Thus, a one meter design strip will be drawn in both X and Y directions on the foundation. These design strips in X and Y direction are usually defined in SAFE software as layer A and layer B.

Before drawing design strips, existing design strips, if any, should be deleted. To do this, follow the following command: '**Select**'>'**Select**'>'**Properties**'>'**Design Strip Layers**'. This results in the following '**Select Design Strip Layers**' window.

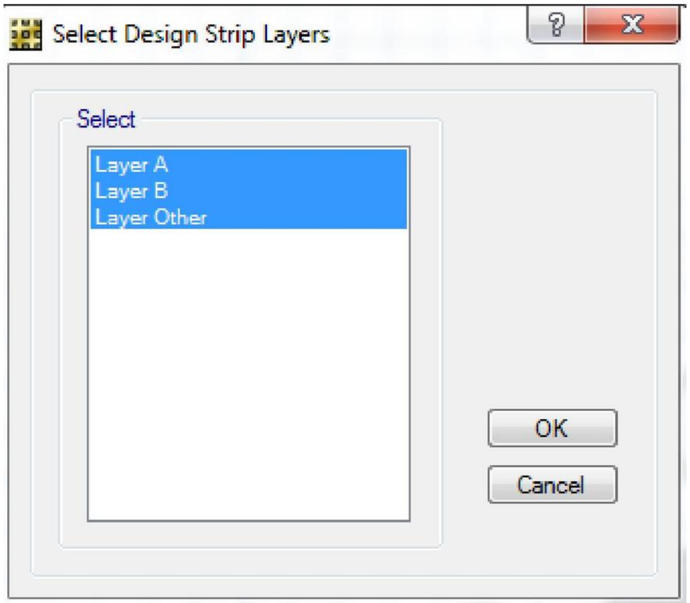


Figure 21

In this window, select all layers and press '**OK**' and the window will be exited while any existing design strips are selected. Then press the '**Delete**' key. This deletes any existing design strips. After this, new layers of design strips in both X and Y directions will be drawn.

To draw the design strip, go to the '**Draw**' menu and click on '**Design Strips**' and the following window pops up.

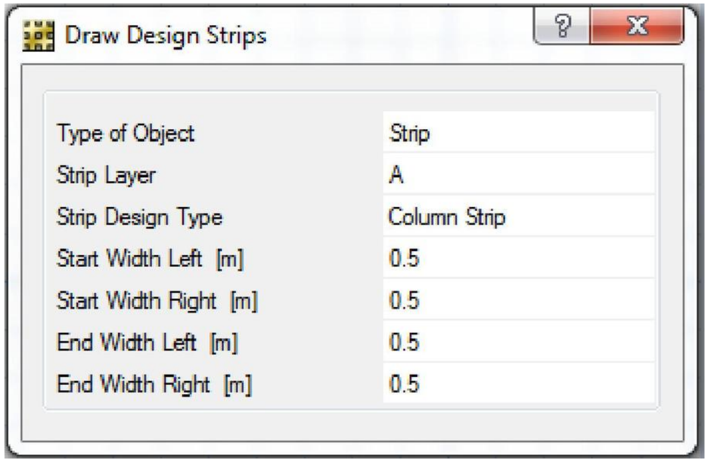


Figure 22

In this window, the '**Strip Layer**' should be selected to be either '**A**' or '**B**'. But if '**A**' is for design strip in X direction '**B**' should be for Y direction and vice versa. Since we are drawing a strip around the column to consider maximum moment and shear forces, the '**Strip Design Type**' should be set to '**Column Strip**'. The width of the strip will be 1m in both directions and since the column will be centrally located, all the widths will be set to 0.5m. To draw the design strip in the X direction, without closing the window, left click at the center of one side of the footing on the plan view parallel to the Y axis and again left click at the center of the parallel side and right click. This creates a design strip in X direction. In doing so, if you can't snap to the center of the side of the footing, you can modify the snap options by clicking on the '**Snap Options…**' command from the '**Draw**' menu and adjusting the options which you want to snap to. The design strip in Y direction can also be drawn in a similar procedure after changing the '**Strip Layer**' to '**B**'.

You can display the design strips by setting the display options by clicking on '**Set Display Options…**' from the '**View**' menu or by simultaneously clicking on '**Ctrl**' and '**W**' keys or by just clicking on the set display options icon ✓ from the tool bar below the menu bar. This results in the following window:

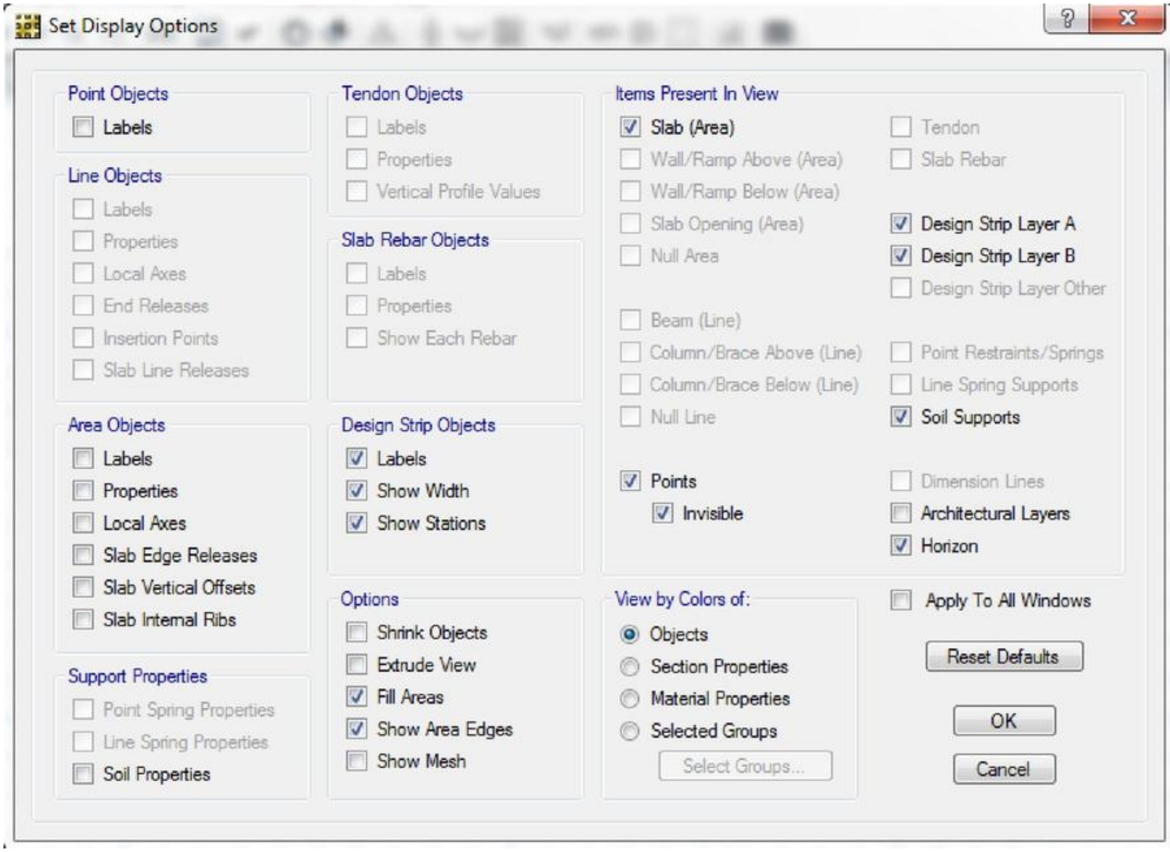


Figure 23

Then, check on '**Labels**', '**Show Width**' and '**Show Stations**' in the '**Design Strip Objects**' box and press '**OK**' and the following window appears displaying the design strips in the two directions.

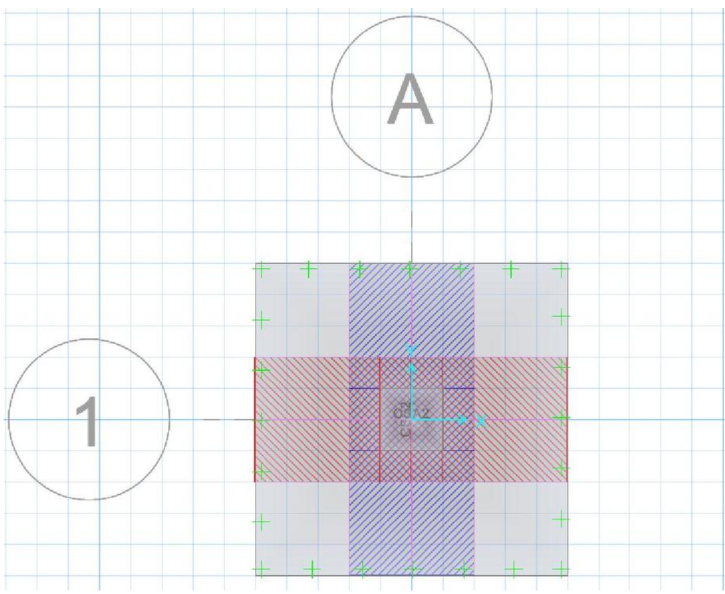


Figure 24

STEP 7: Assigning Slab Data, Support Data and Load Data

The slab data, support data, and load data which are defined in the previous steps should be assigned to the corresponding structural component. The way to do this is first to select the component and next to assign the slab property, support property or loads accordingly. The selection can be done by clicking on the component from the plan view or through the '**Select**' menu. The latter option assures that the component is selected exactly as selection by clicking may result in incorrect selection. Thus, all the selections here will be done from the '**Select**' menu. The assignments will be discussed as follows:

a. Assigning slab data to the footing

Select the footing through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**'Slab Properties…**'. Then select '**Footing**' and press '**OK**'. Then assign the footing property through the following strings of commands '**Assign**'>'**Slab Data**'>'**Properties…**'. Then select '**FOOTING**' and press '**OK**'.

b. Assigning slab data to the foundation column

Select the foundation column through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**'Slab Properties…**'. Then select '**Stiff**' and press '**OK**'. Then assign the footing property through the following strings of commands '**Assign**'>'**Slab Data**'>'**Properties…**'. Then select '**STIFF**' and press '**OK**'.

c. Assigning support data to the footing

Select the footing through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**'Slab Properties…**'. Then select '**Footing**' and press '**OK**'. Then assign the footing property through the following strings of commands '**Assign**'>'**Support Data**'>'**Soil Properties…**'. Then select '**SOIL**' and press '**OK**'.

d. Assigning reinforcement data to the design strips

Select each design strip through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**Design Strip Layers…**'. Then select '**A**' or '**B**' (one at a time) and press '**OK**'. When you right click on the selected strip layer, the '**Slab-Type Area Object Information**' window pops up. In the '**Design**' tab of this window, set the '**Rebar Material**' to '**S300**' and press '**OK**'. Do this for both strips.

e. Assigning load on the foundation column

To assign load on the foundation column, right click on the point at the center of the foundation column and a '**Point Object Information**' window pops up. In this window, click on the '**Loads**' tab. Then click on '**Reset All**' button to delete any existing loads.

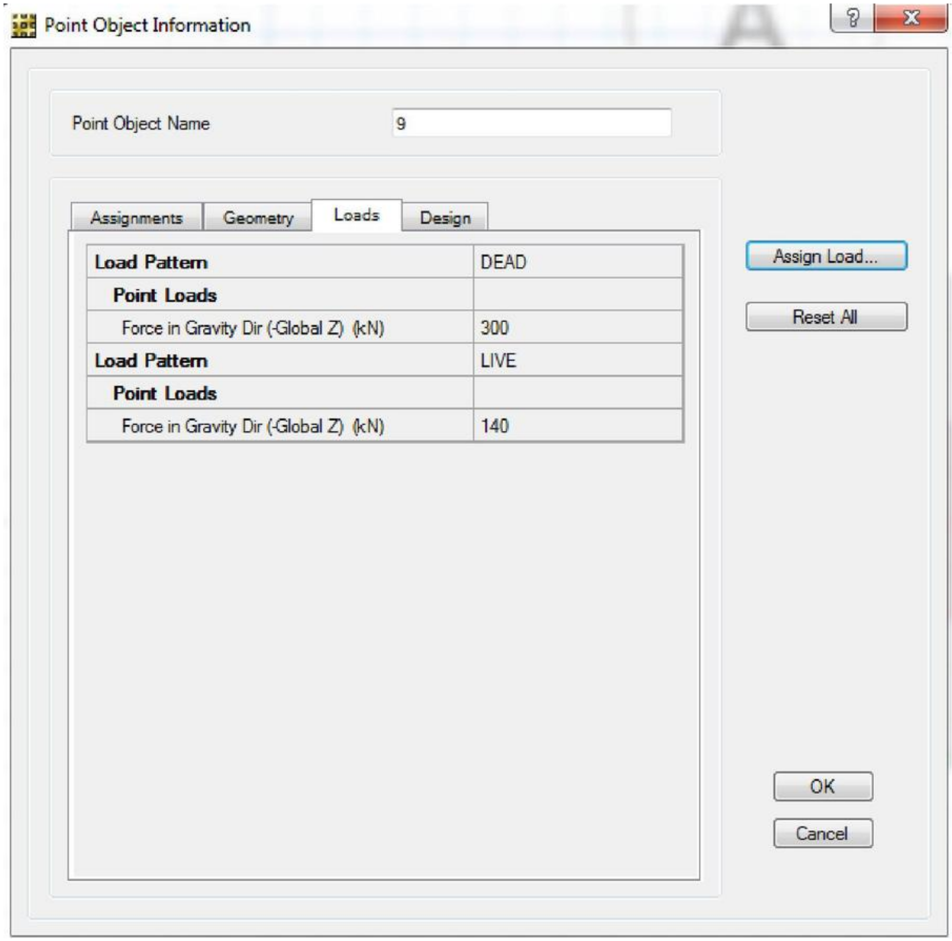


Figure 25

The dead load and the live load can be assigned through the '**Assign Load…**' button. The procedure is: click on '**Assign Load…**' button, then select '**Force Loads**' then press '**OK**' then select either '**DEAD**' or '**LIVE**' depending on which load you want to enter its value then enter its value accordingly and in the right direction (axis), then select '**Add to Existing Loads**' and press '**OK**'. While doing this, the foundation column dimensions should be entered in the '**Size of Load for Punching Shear**' box of '**Point Loads**' window. You can also delete any forces which are wrongly entered through '**Delete Existing Loads**' radio button option. In this example, since the dead load is 300kN and the live load is 140kN, these values are entered in the Gravity Direction.

STEP 8: Running the Analysis

After this, the analysis can be run. But, make sure that the footing and the foundation column are assigned with the correct rebar material. To do this, right click anywhere in the plan view of the footing and the '**Slab-Type Area Object Information**' window will pop-up.

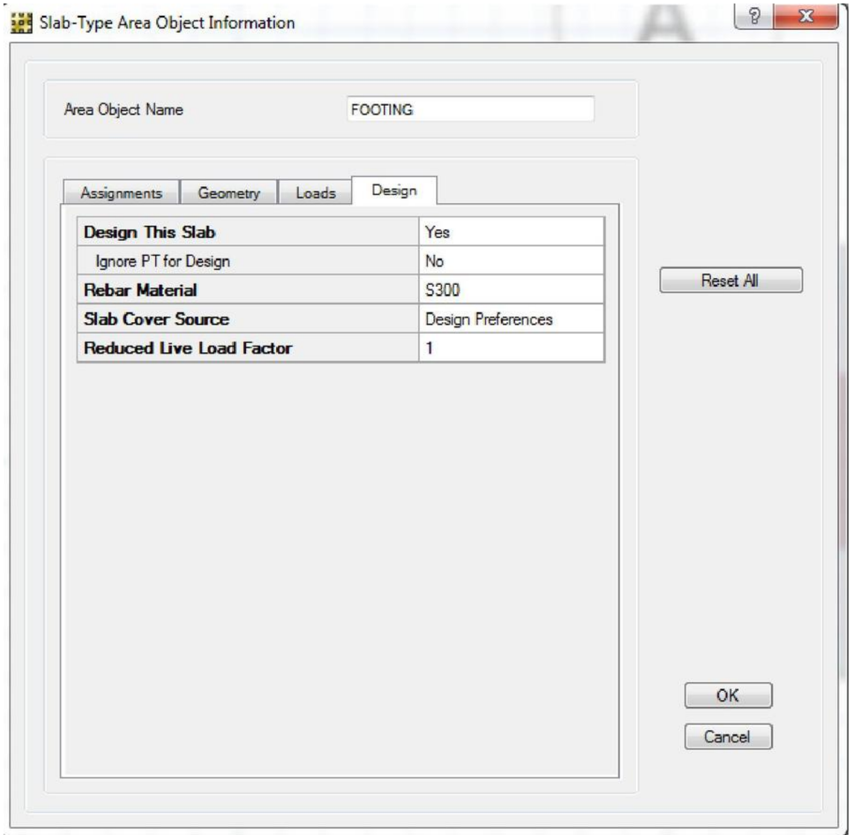


Figure 26

In the '**Design**' tab of this window, set the value of '**Rebar Material**' to '**S300**'. Do this for the column foundation, and both design strips as well. After this, go to the '**Run**' menu and click on the '**Run Analysis**' command.

STEP 9: Displaying the Output

Once the analysis is run, the output will be displayed. Particularly, the punching shear design is of great importance as the footing cannot be designed without the punching shear requirement being adequately satisfied. To do this, go to the '**Run**' menu and click on '**Run Analysis & Design**' command or simply click on the '**F5**' key. When you do this, you will be prompted to save the model, if you haven't already don this. When you save the model, the following window showing the displacement of the soil in a banded figure will be displayed.

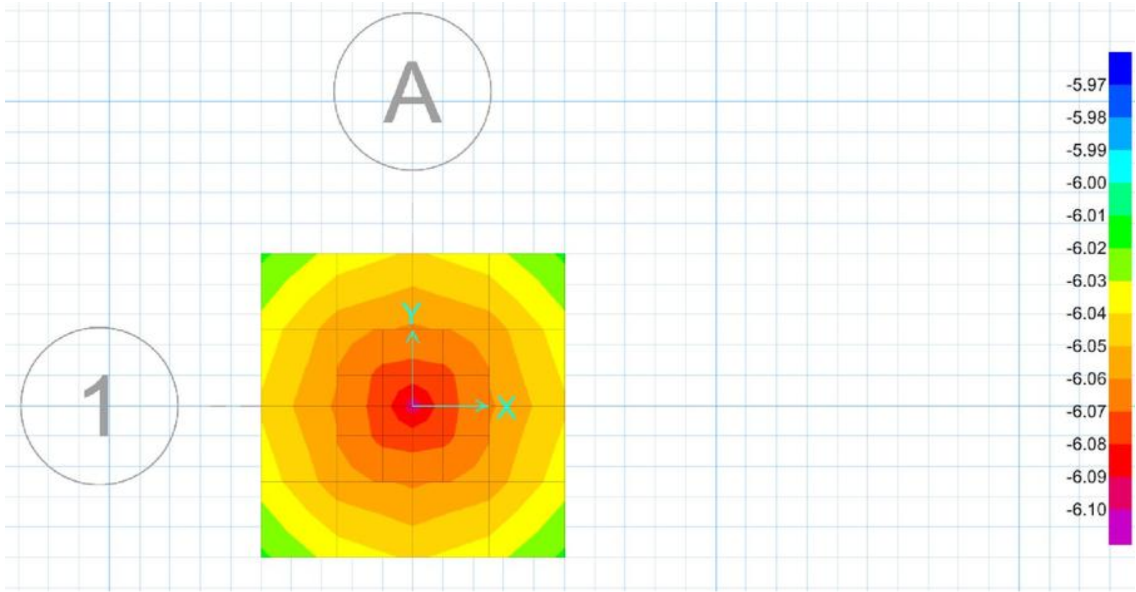


Figure 27

To display the punching shear ratio, go to the '**Display**' menu and click on '**Show Punching Shear Design**'. After this, the punching shear ratio will be displayed in the plan view around the foundation column as in the following figure.

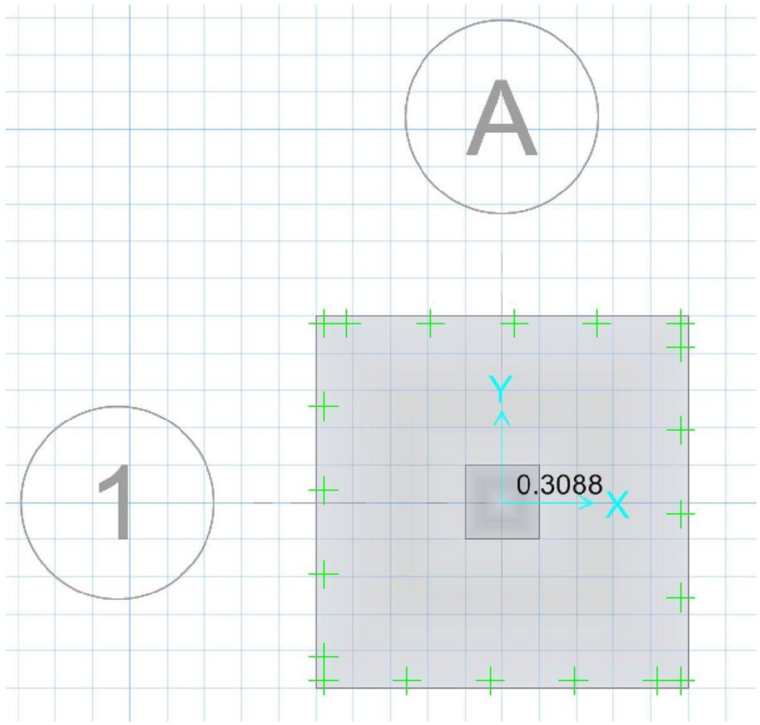


Figure 28

As can be noticed from the figure, the punching shear ratio is 0.3088. Generally, a punching shear ratio less than one indicates the concrete thickness is adequate to resist punching shear and a value greater than one indicates that the punching shear capacity is exceeded somewhere along the critical section. For economical design, it is recommended to keep the punching shear ratio between 0.95 and 1 as very small values of punching shear ratio means excess concrete thickness is used. However, if the punching shear ratio is greater than one, the thickness of the concrete should be increased and the foundation should be re-designed. A detailed quantitative description of the foundation design can also be obtained by right clicking on the plan view shown in Fig 28 as shown below. Several trial may be made by zooming in and out to get the quantitative description.

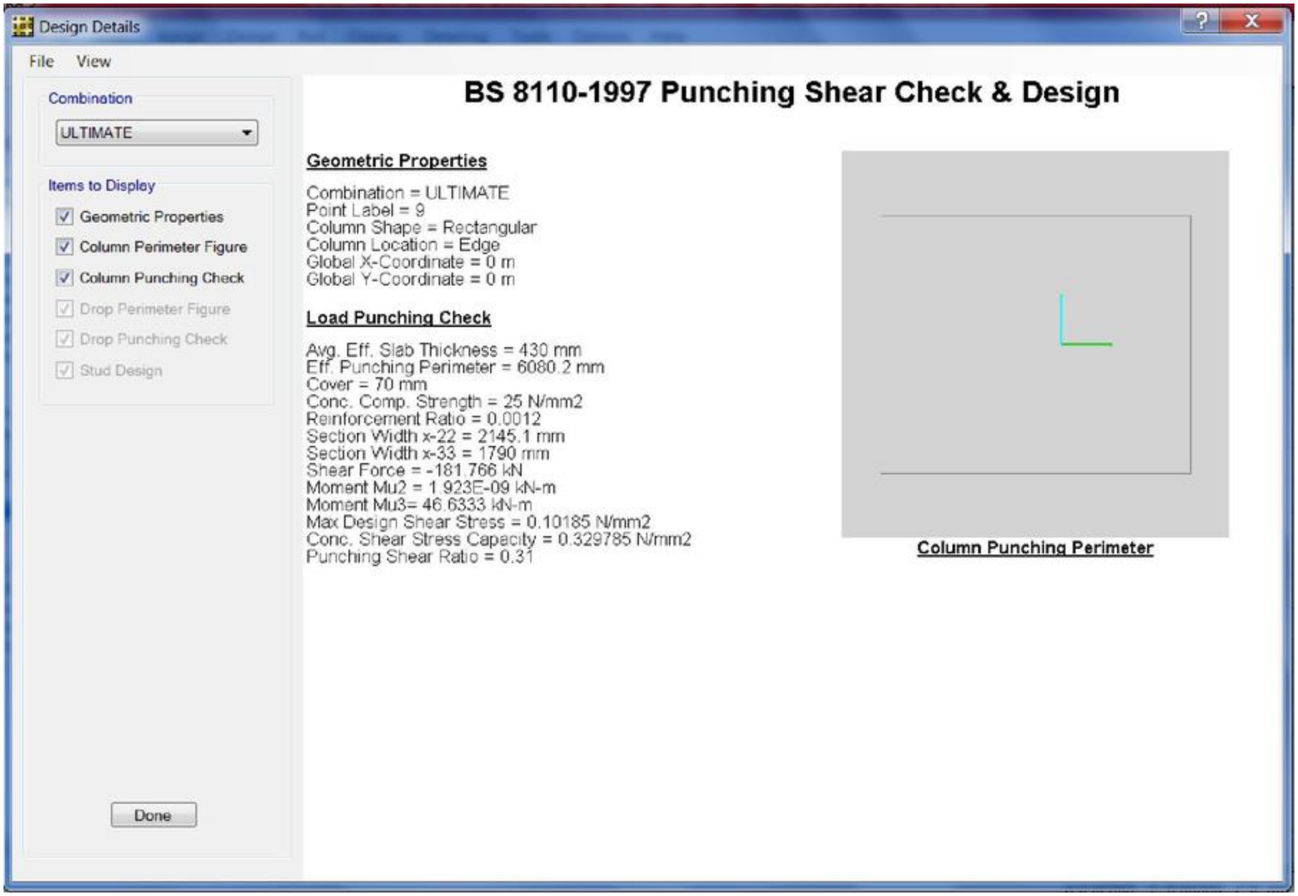


Figure 29

Through the '**Display**' menu, relevant quantities can be displayed on the screen. For instance, the '**Display**'>'**Show Strip Forces**' command or by simply clicking the '**F8**' key, the following window will be displayed.

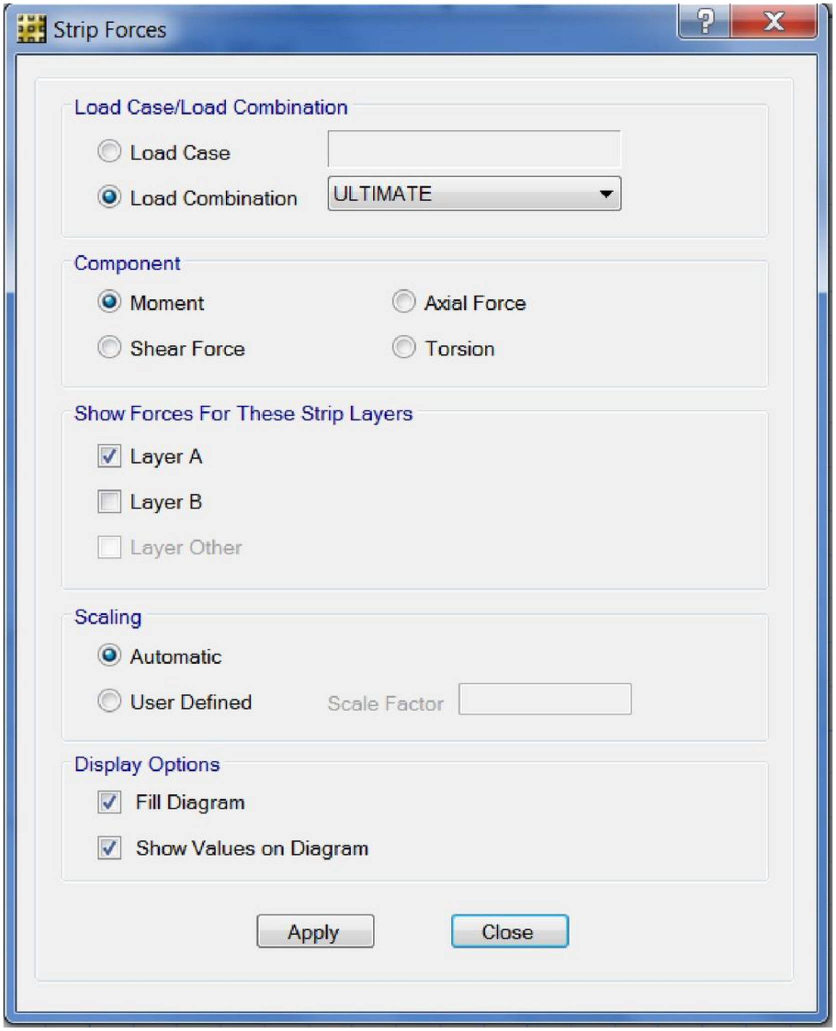


Figure 30

In this window, the '**Load Case/Load Combination**' box provides with radio buttons to select for which particular load case or load combination that we want to display the output. The '**Component**' box contains four radio buttons to select which quantity to display. The '**Show Forces For These Strip Layers**' box allows us to select the strip layer for which the quantity is displayed. Both strip layers can be selected at the same time. From the '**Scaling**' box, we can select whether automatic scaling or user defined scaling is used while displaying the diagram. The '**Display Options**' box allows us to fill or not to fill the diagram and to display or not to display the values on the diagram. For the preferences shown in Fig. 30, the following diagram will be displayed.

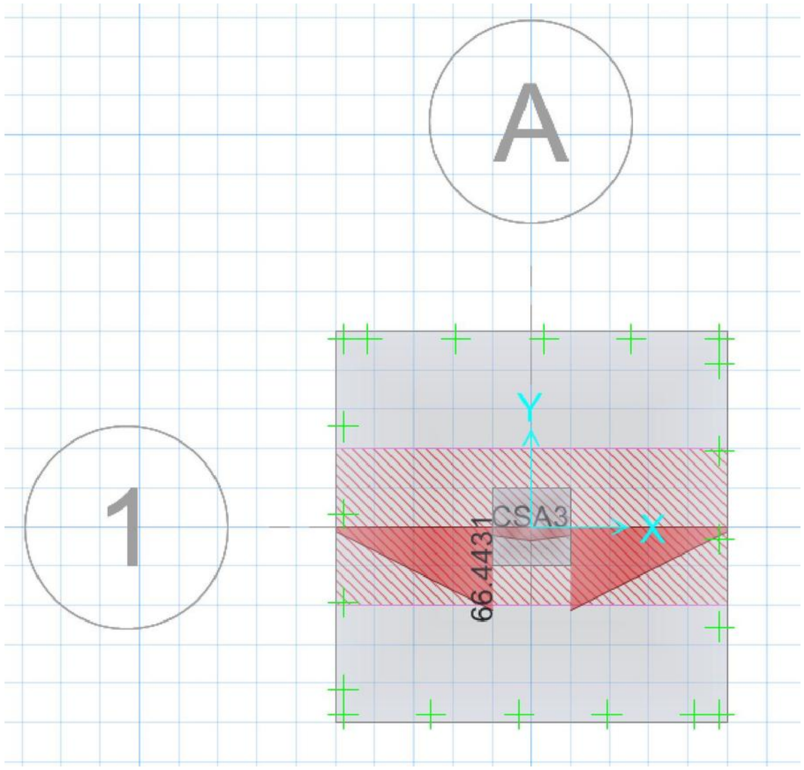


Figure 31

The '**Display**'>'**Show Slab Design…**' command results in the window shown in Fig 32. In this window, several options can be set in order to display the footing design in the way we wanted to. The '**Choose Display Type**' box allows us to select the '**Design Basis**' between '**Strip Based**' and '**Finite Element Based**'. Unless some differences in the way the design is displayed, there is no difference in the amount of reinforcement between these selections. Through the '**Display Type**' in the same box, it can be selected whether to display flexural reinforcement or shear reinforcement. This box also allows us to impose or not to impose minimum reinforcement during the design. The '**Rebar Location Shown**' box allows us to select which reinforcement, top or bottom or both, to be displayed. The '**Reinforcing Display Type**' box allows us to set the manner in which the amount of reinforcement is displayed. The option whether to show the reinforcing envelop diagram and the reinforcing extent can be set by the check boxes in the '**Reinforcing Diagram**' window. The strip layer direction for which the amount of reinforcement is displayed can be chosen from the '**Choose Strip Direction**' box. The '**Display Options**' box allows us whether to display output in filled diagram or not and whether the values at controlling stations will be displayed or not. If we want to display the amount of reinforcement above some specified reinforcement bar area or spacing, we can use the options in the '**Show Rebar**

Above Specified Value' box. When the '**Typical Uniform Reinforcing Specified Below**' radio button is selected, the '**Typical Uniform Reinforcing'** box get activated. In this box, we can set a specific value above which the reinforcement amount will be displayed. The reinforcement diagram output, for the options set in Fig. 32, will be shown below in fig. 33.

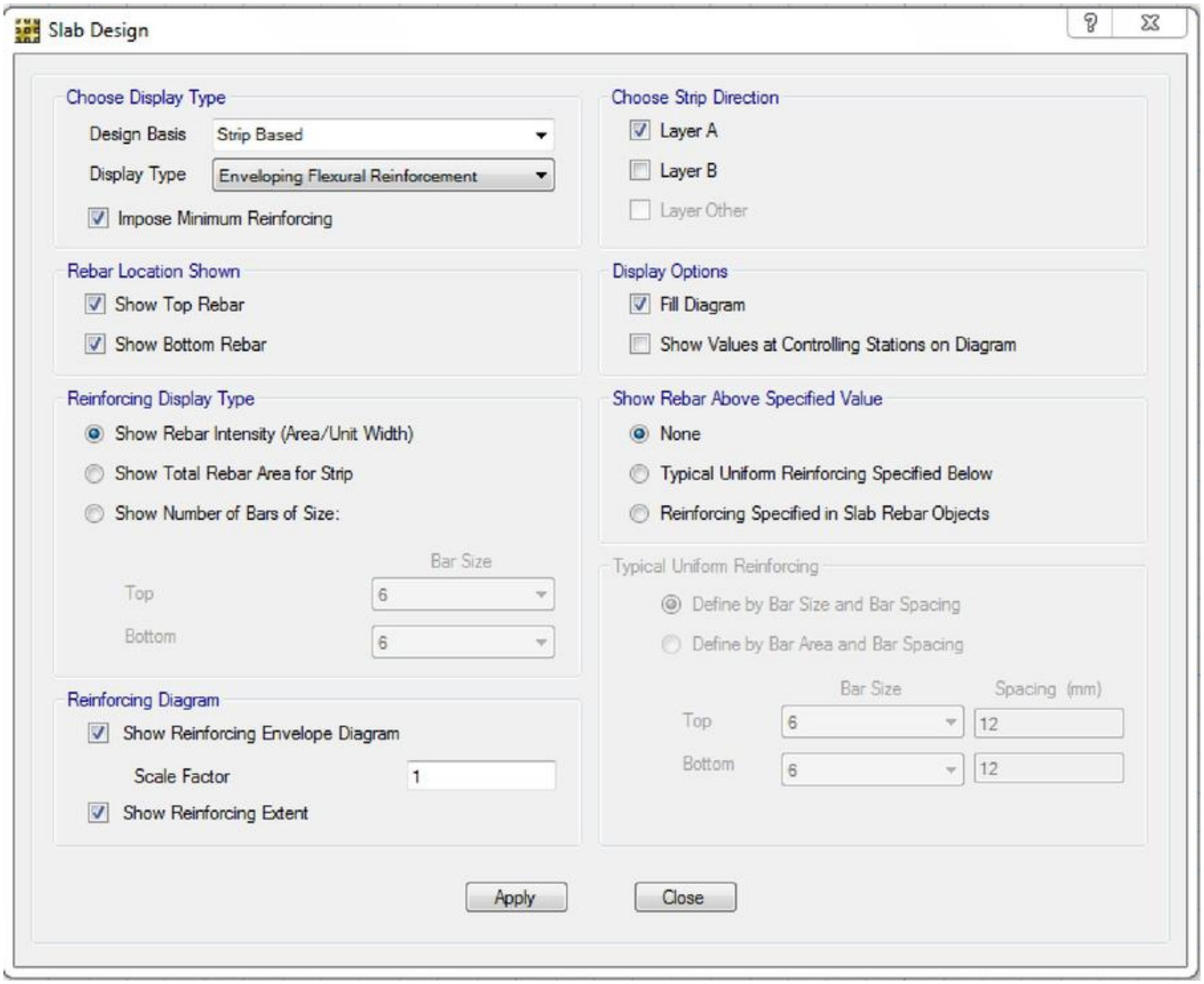


Figure 32

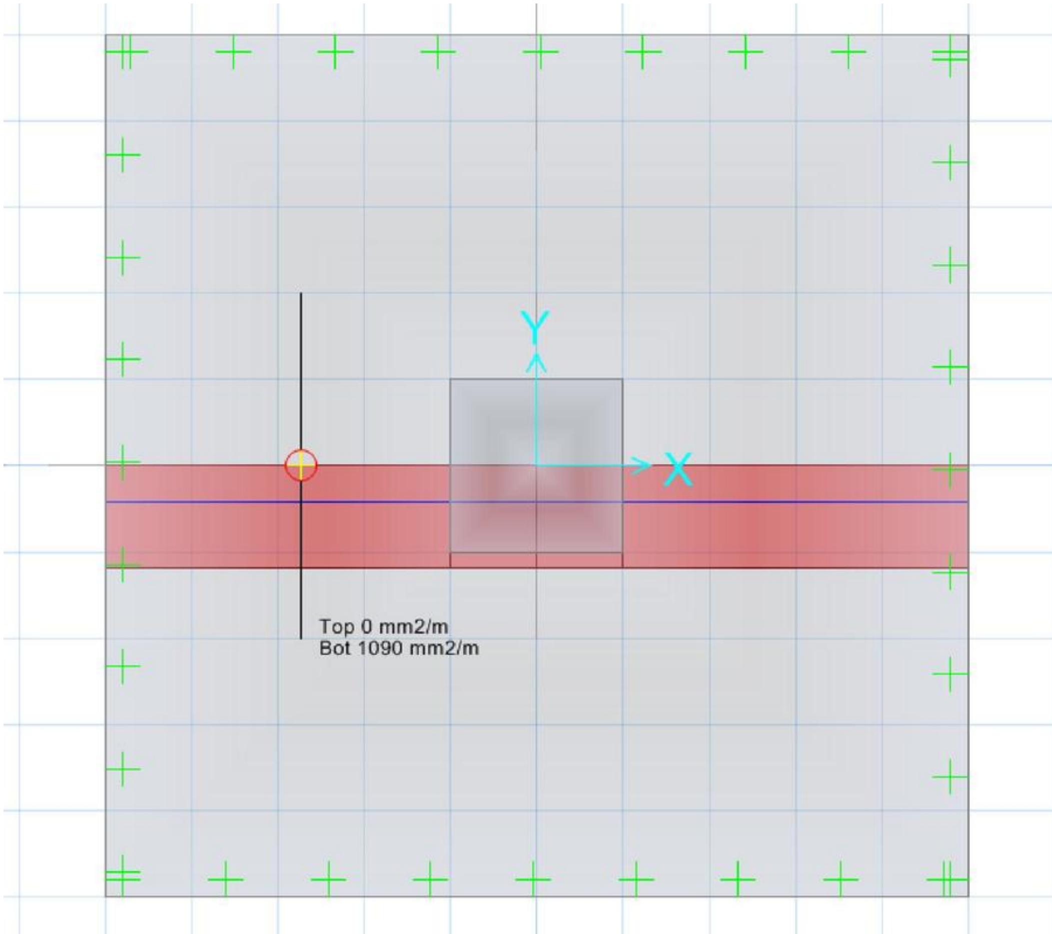


Figure 33

As can be noticed from the footing design diagram in fig. 33, the top reinforcement across strip layer A is $0mm^2/m$ for top reinforcement and $1090mm^2/m$ for bottom reinforcement.

The design outputs can also be displayed in tabular format by clicking on the '**Show Tables…**' menu item from the '**Display**' menu or by just clicking on the equivalent icon  from the tool bar below the menu bar and the following window will pop up.

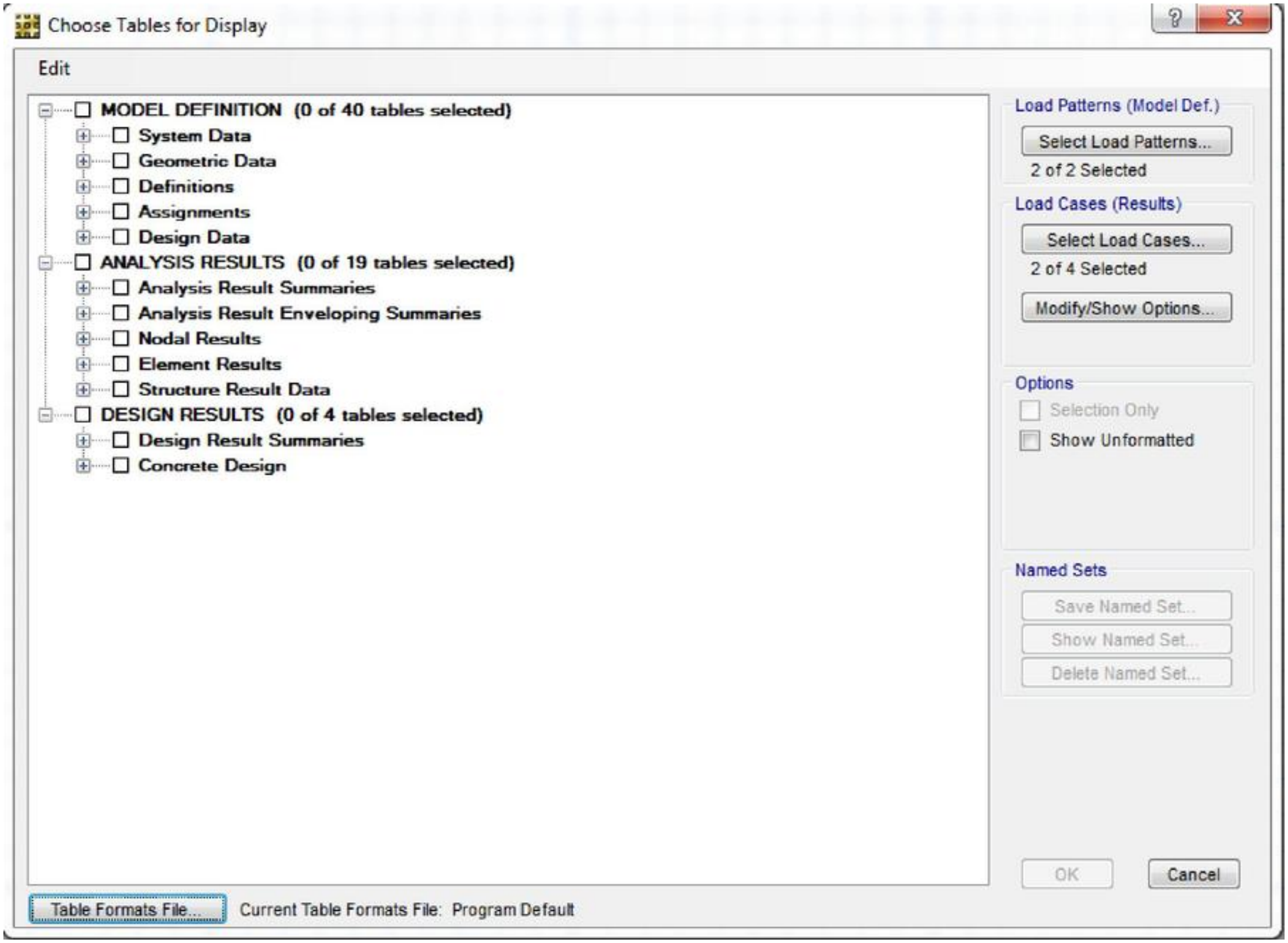


Figure 34

In this window, we can select any of the model definitions or analysis results or design results and press '**OK**' to display the quantity which we want to have a look at. By using the right hand side buttons in the window, the load patterns and the load cases can be selected.

STEP 10: Detailing

After running the analysis and after checking that the results are reasonable, the detailing will be done. However, before running the detailing, the detailing preferences can be set from the '**Detailing**' menu. From the '**Detailing Preferences…**', the likes of dimensional units and material quantity units can be selected. From '**Slab/Mat Detailing Preferences…**', the likes of rebar curtailment options, the rebar detailing options, rebar selection rules and preferred rebar sizes can be selected. The '**Drawing Sheet Set-up…**' menu allows us to set-up the contents of the drawing sheet. The '**Drawing Format Properties…**' allows us to set some formats in which the output displayed.

To run the detailing, go to '**Run**' menu and click on '**Run Detailing…**' or simultaneously press '**Shift**' and '**F5**' keys or just click on the run detailing icon  from the tool bar just below the menu bar. Then, the '**Run Detailing Options**' window pops up so that we set the detailing options. Set the detailing options which you want and click '**OK**'.

Once the detailing is run, the detailing can be displayed. The detailing display options can be best accessed from the '**Model Explorer**'. When expanded in full, the '**Detailing**' tab of the model explorer looks like:

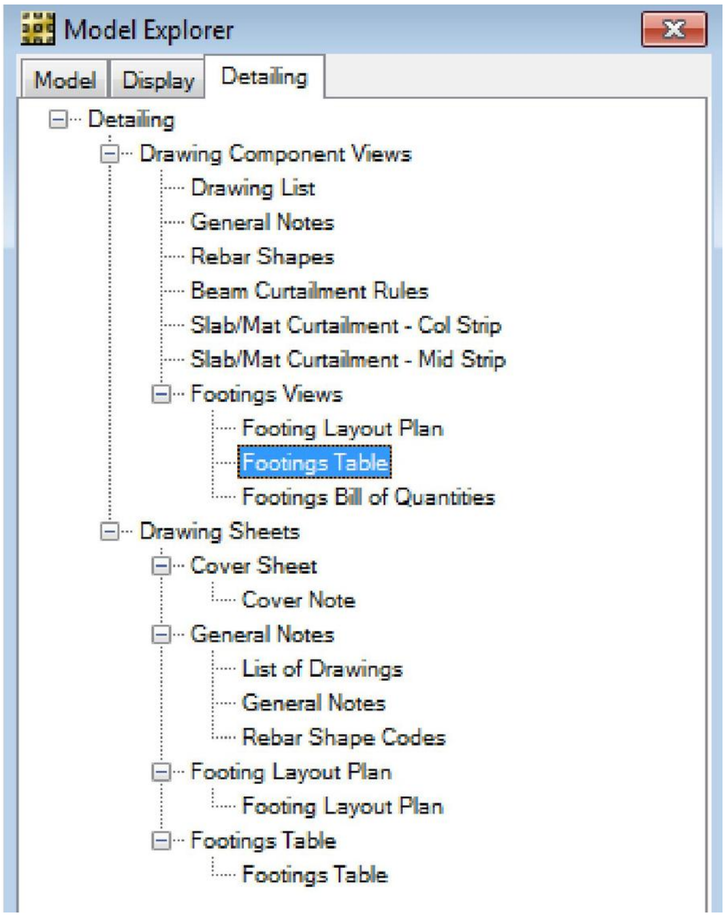


Figure 35

By clicking on any of the options from the detailing tab, a desired detailing can be displayed. For instance, by double clicking on the '**Footing Table**', the following detail of footing can be shown.

FOOTINGS TABLE

SR. NO.	TYPE	NOS	LX	LY	T	REBARS-A	REBARS-B
1	F1	1	2.500 M	2.500 M	0.500 M	8-10	9-10

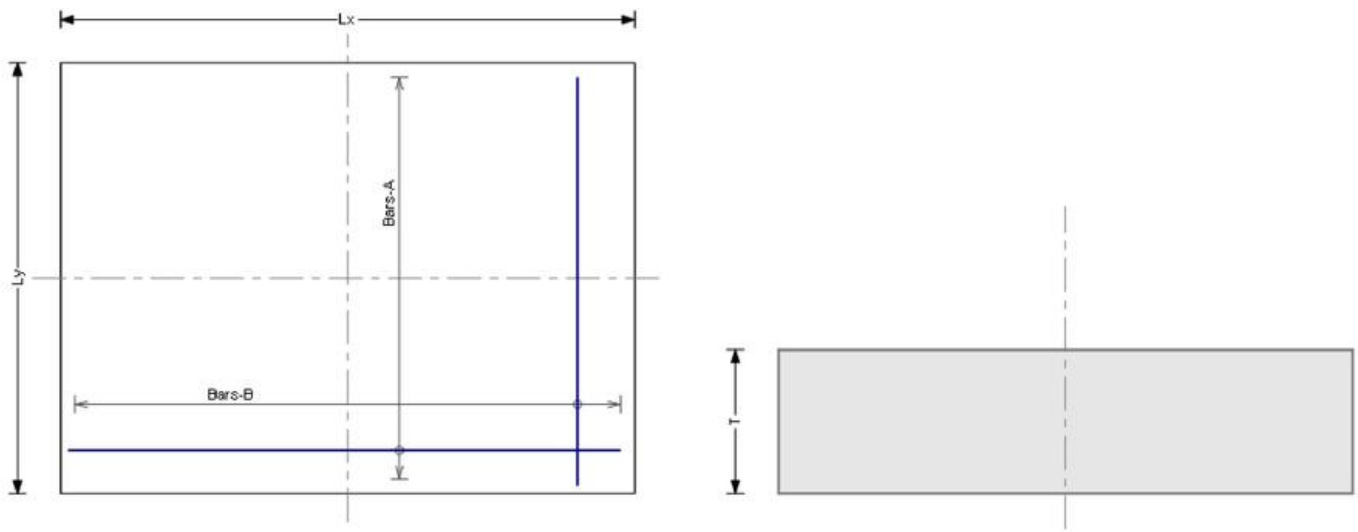


Figure 36

Apart from this detail, other details can also be shown.

STEP 11: Reporting

The last step of foundation design is reporting. Before creating the report, the report preferences should be set up. To do this, go to the 'File' menu and click on 'Report Set-up…' and the following window pops up.

In this '**Report Setup Data**' window, the user preferences regarding the reporting such as the report output type, the report page orientation and the report items can be set along with the load patterns and load combinations. Once the preference is set, the report can be created by clicking on '**Create Report**' command in the '**File**' menu. The '**Advanced Report Writer**' command in the same menu can be used to set some advanced reporting formats.

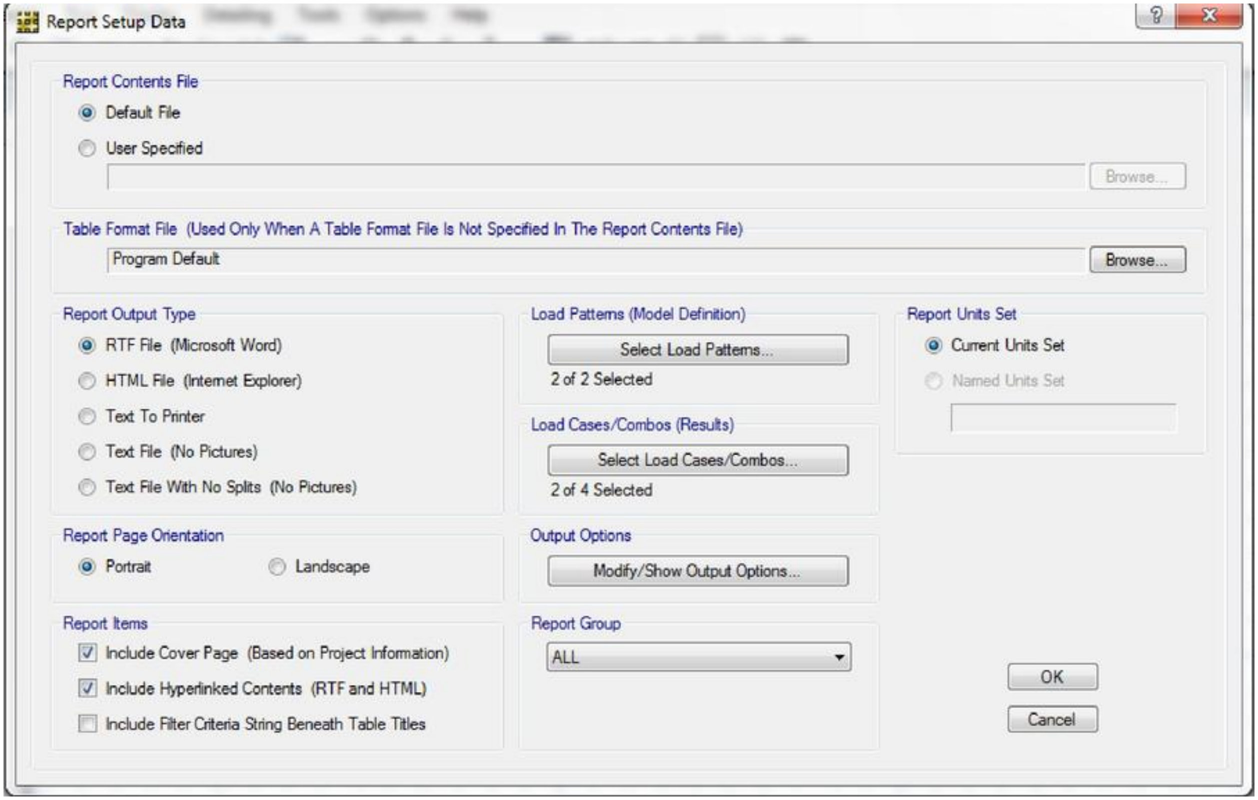


Figure 37

This concludes the tutorial for the design of single footing using the built-in model.

2.2. Using Grids

Design a rectangular isolated footing with the following parameters:

- *Dimension of the footing*: 2mX3m
- *Dimension of the column*: 0.4mX0.6m
- *Allowable bearing capacity of the soil* : 100kN/m^2
- *Maximum allowable settlement of the foundation*: 10mm
- *Concentrated load (DL)* : 275kN
- *Concentrated load (LL)*: 175kN
- *Bending moment (dead) in the long direction (My, DL)*:50kNm
- *Bending moment (live) in the long direction (My, LL)*:25kNm
- *Grade of concrete*: C-30 (30MPa 28-day characteristics cube strength)
- *Grade of reinforcement bar (rebar)*: S-400 (400MPa characteristics yield strength)
- *Overall thickness of the foundation*: 500mm
- *Concrete cover* : 50mm

STEP 1: Creating the Model

To create a footing model by using a grid, there are three ways. Go to '**File'** menu and click on '**New Model …'** or just click on the icon  . This icon is the first among the list of icons just below the menu bar or simply click on the '**Ctrl'** and '**N'** keys simultaneously (**Ctrl+N**).

Performing one of the above three ways results in the popping-up of the following **'New Model Initialization'** window.

This window consists of two parts: the '**Design Data**' part and the '**Initial Model**' part. In the '**Design Data**', there are '**Design Code**', '**Design Preferences**', '**Project Information**' and '**Units**'.

The '**Design Code**' consists of a list of selected design codes from many countries. Depending on the suitability of the code for your country, you may select one among the list of codes. The design code which follows similar design philosophy to my country code is BS 8110-1997. So, this design code is selected.

After selecting the design code, it is better to first select the units to be used in the design and analysis process. These units can be selected by clicking on the '**Modify/Show**' button in front of '**Units**'. The following window pops up when the button is clicked. To choose metric units, click on '**Metric Defaults**' button. The click results in metric units for best practices to be selected. It can be observed that, even though all the units are metric, they are not consistent. To select a preferred consistent unit, click on the '**Consistent Units**' button. In this '**Units**' window, the decimal places and minimum number of significant digits for any quantity can also be modified.

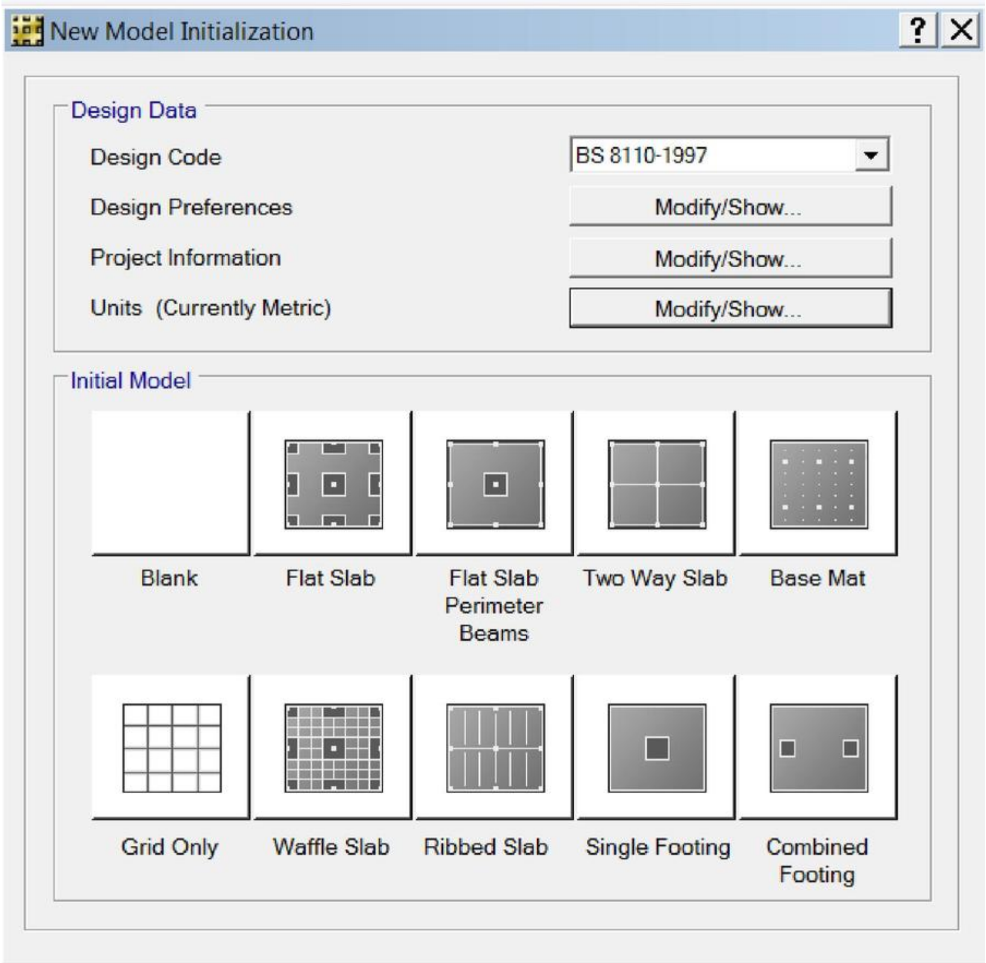
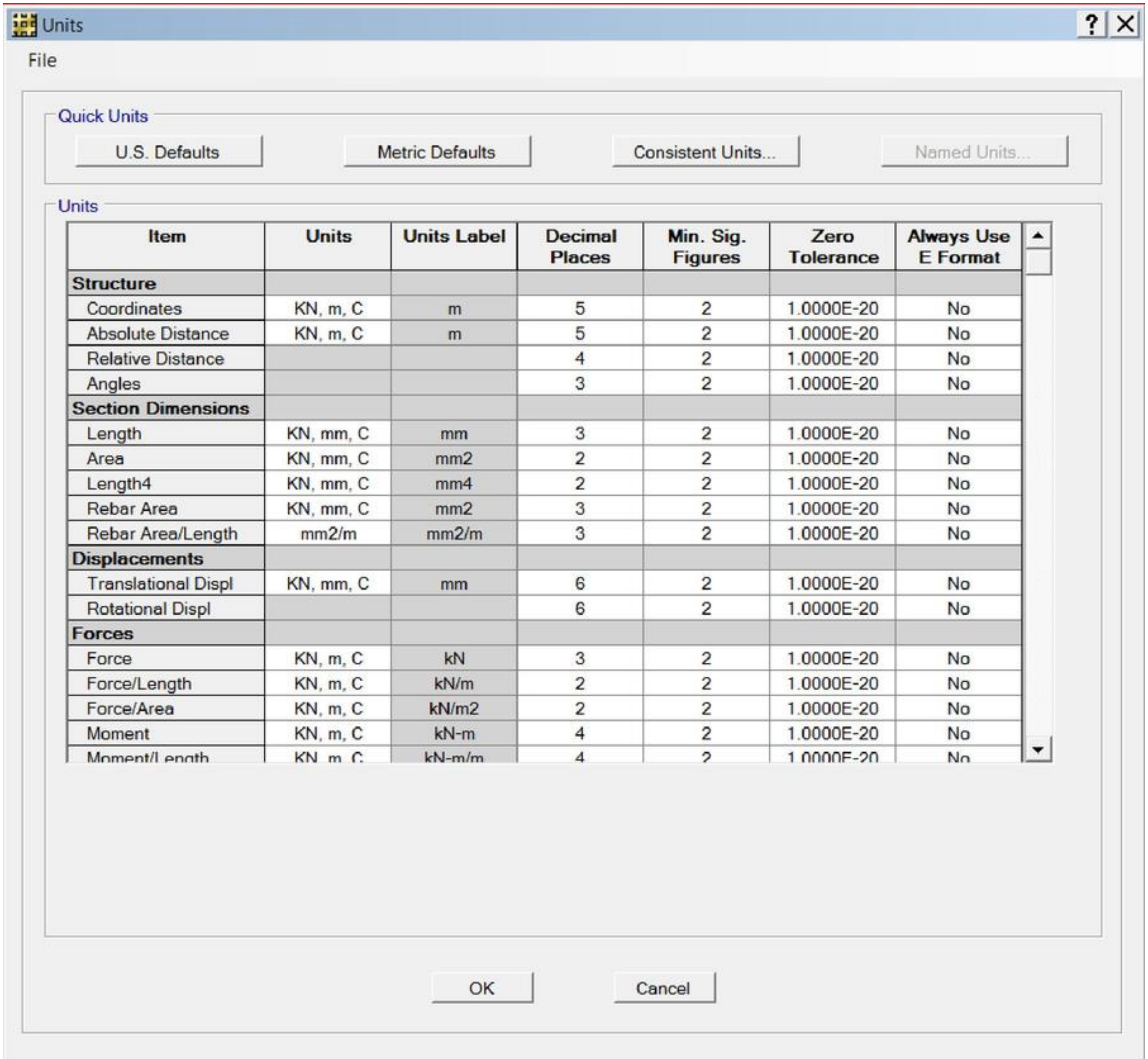


Figure 38

By clicking on the '**Modify/Show**' button in front of the '**Project Information**', information regarding the project, the company and the model can be entered.

By clicking on the '**Modify/Show**' button in front of the '**Design Preferences**', the user preferences regarding the design code, concrete cover for slabs and beams, post-tensioning can be changed. The following **'Design Preferences'** window pops-up when the button is clicked. The '**Min. Cover Slabs**' tab of the window may be of interest for this particular problem as footings will be modelled as slabs. Thus, click on this tab. This is the tab where the concrete cover and preferred rebar size will be entered.



Item	Units	Units Label	Decimal Places	Min. Sig. Figures	Zero Tolerance	Always Use E Format
Structure						
Coordinates	KN, m, C	m	5	2	1.0000E-20	No
Absolute Distance	KN, m, C	m	5	2	1.0000E-20	No
Relative Distance			4	2	1.0000E-20	No
Angles			3	2	1.0000E-20	No
Section Dimensions						
Length	KN, mm, C	mm	3	2	1.0000E-20	No
Area	KN, mm, C	mm2	2	2	1.0000E-20	No
Length4	KN, mm, C	mm4	2	2	1.0000E-20	No
Rebar Area	KN, mm, C	mm2	3	2	1.0000E-20	No
Rebar Area/Length	mm2/m	mm2/m	3	2	1.0000E-20	No
Displacements						
Translational Displ	KN, mm, C	mm	6	2	1.0000E-20	No
Rotational Displ			6	2	1.0000E-20	No
Forces						
Force	KN, m, C	kN	3	2	1.0000E-20	No
Force/Length	KN, m, C	kN/m	2	2	1.0000E-20	No
Force/Area	KN, m, C	kN/m2	2	2	1.0000E-20	No
Moment	KN, m, C	kN-m	4	2	1.0000E-20	No
Moment/Length	KN, m, C	kN-m/m	4	2	1.0000E-20	No

Figure 39

In the '**Min. Cover Slabs**' tab, for '**Non-Prestressed Reinforcement**' , both the '**Clear Cover Top**' and '**Clear Cover Bottom**' should be set to 50mm as the concrete cover in this design problem stated to be 50mm. The '**Preferred Bar Size**' can be set to any reasonable value. Here, the '**Preferred Bar Size**' is set to 20 which is the diameter of the rebar in mm which will be used as the main reinforcement in the foundation. Leave the rest at they are.

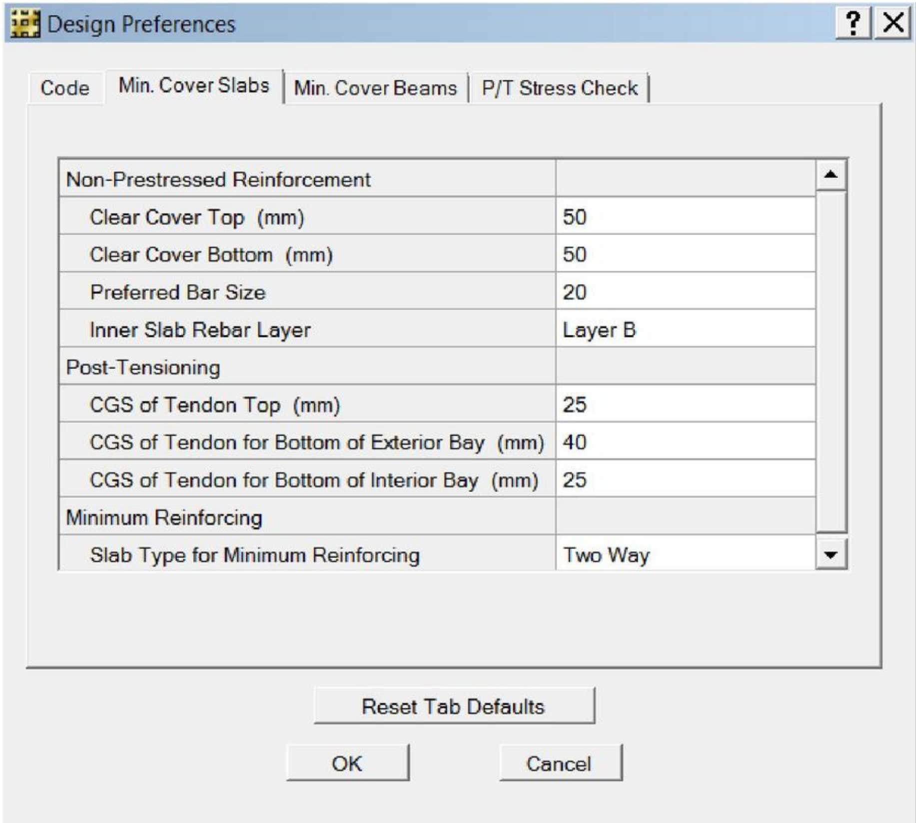


Figure 40.

Once everything is set correctly in the '**Design Data**' box, go to the '**Initial Model**' box and select the suitable model. Since an isolated (single) footing will be designed from grids click on the big square button just above '**Grid Only**'. The button itself shows crossing grids. When the button is clicked, the following **'Coordinate System Definition'** window will pop up.

The footing in this example is rectangular with centrally located column. The grid which will be created will be boundaries of the outer edges of the footing and the outer edges of the foundation column. As a result, there is uneven spacing between the grids. To create such grids with uneven spacing, it is better to create it by clicking the '**Edit Grid**' button. When you click this button, a '**Coordinate System**' window will pop up. In the '**Display Grid Data as**' box, two radio buttons are there to choose between '**Ordinate**' and '**Spacing**'. These are the options how the location of different grids is defined. If the '**Ordinate**' option is selected, the X and Y-coordinates of the grids will be entered. If the '**Spacing**' option is selected, the spacing between the grids will be entered. If the grids are irregular in shape with non-vertical and non-horizontal grids, they can be defined in the '**General Grid Data**' box by entering the XY coordinates of the starting and ending points. Regular grids can also be defined by this option too.

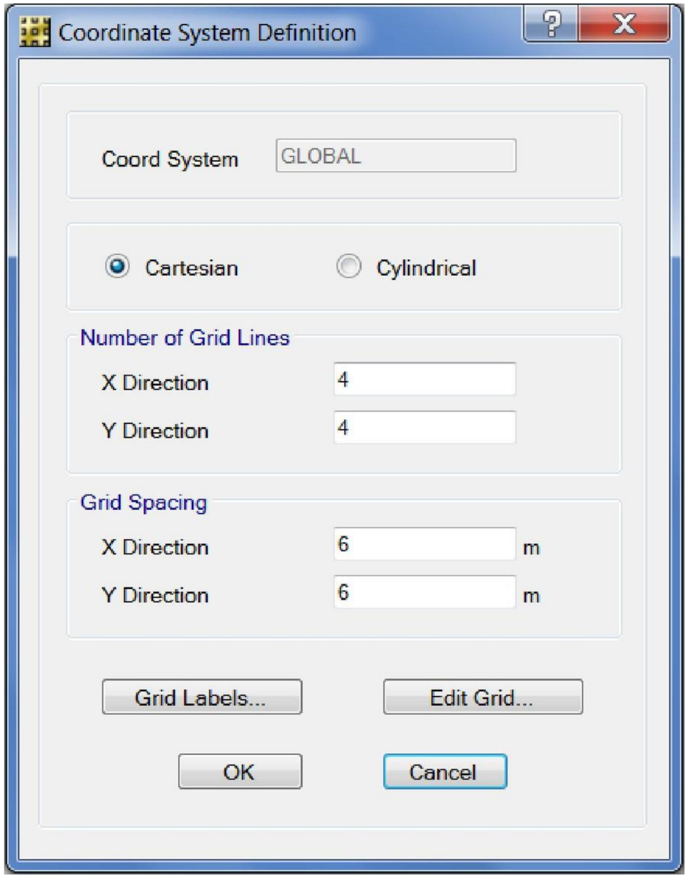


Figure 41

In this example, the length of the footing is 3m and the column dimension is 0.6m in x direction. The distance between the left face of the footing and the left face of the foundation column is 1.2m. The same is true for the distance between the right face of the footing and the right face of the foundation column. To limit the footing and the column in x-direction, the grids should have x-coordinates of 0, 1.2, 1.8 and 3.

The length of the slab is 2m and the column dimension is 0.4m in y direction. The distance between the bottom face of the footing and the bottom face of the foundation column is 0.8m. The same is true for the distance between the top face of the footing and the top face of the foundation column. To limit the footing and the column in y-direction, the grids should have x-coordinates of 0, 0.8, 1.2 and 2.

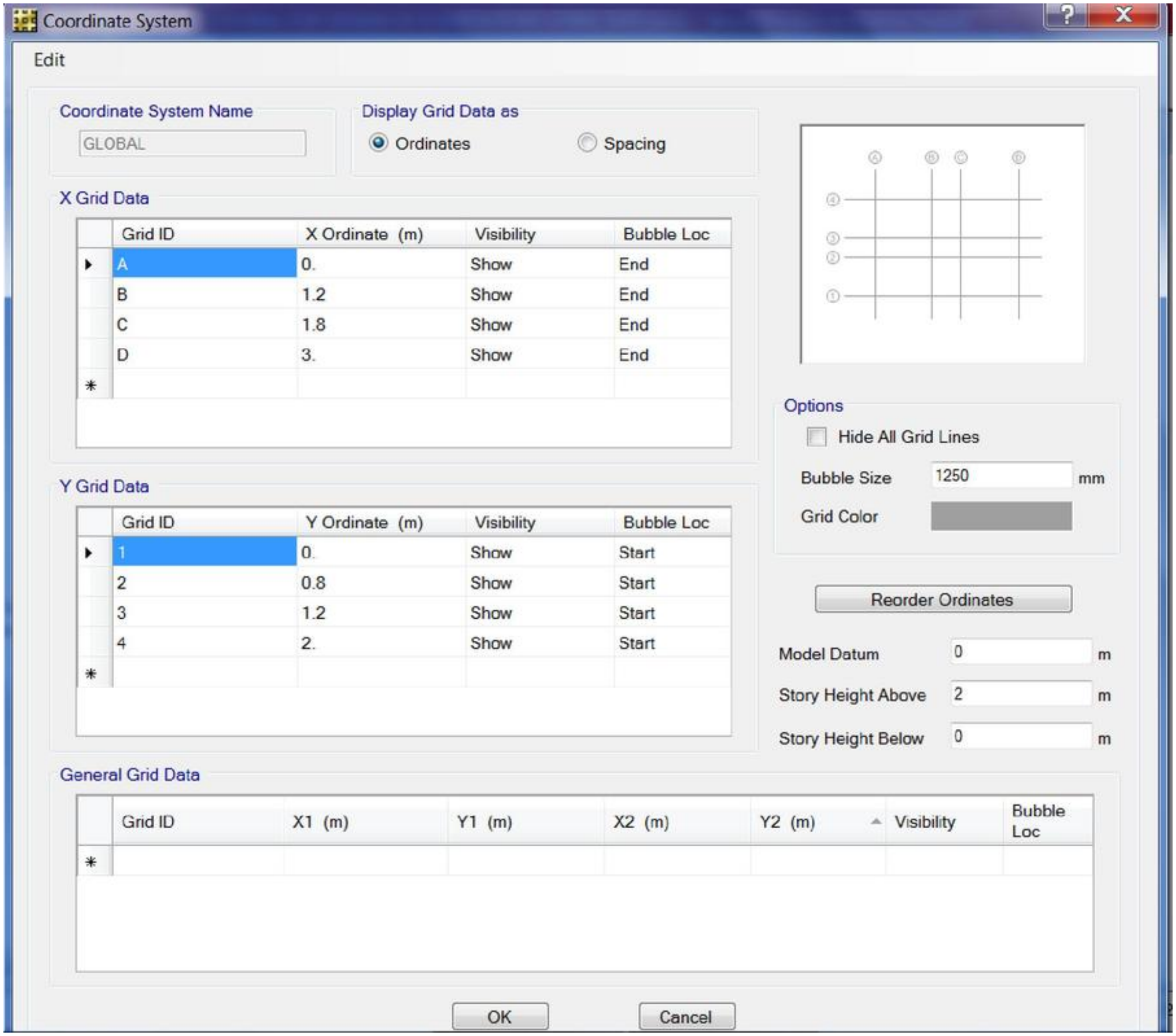


Figure 42

The '**Model Datum**' is the level of the model in z-direction. Any value can be set to this. Set the '**Story Height Above**' value to some reasonable non zero value, say 2, and set the '**Story Height Below**' to 0 as there will be no story below the footing. Then press '**OK**' and the grids will be displayed on the '**Plan View**' window as below:

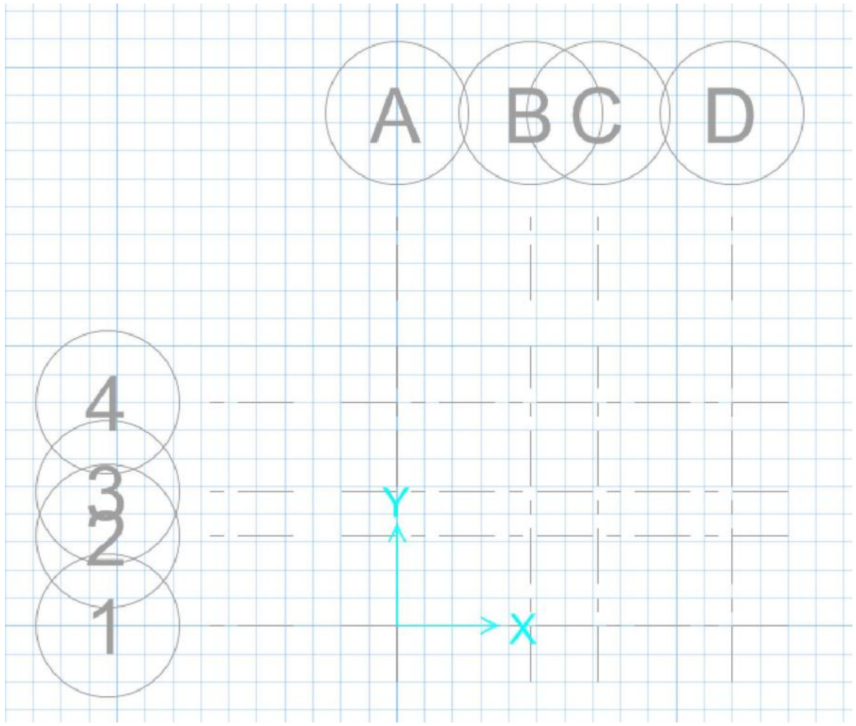


Figure 43

STEP 2: Defining Material Properties

The materials which are involved in the design of the footing should be defined before the analysis. These materials are the concrete, the reinforcement bar (rebar) and the soil support. Thus, these properties will be defined here.

The material definition can be carried out in two ways.

The first one is through the '**Define**' menu in the menu bar. The other one through the model explorer on the left hand side of the home window. To define material properties through the former method, click on '**Define**' menu and again click on '**Materials…**' resulting in the following window depending on prior material definitions.

The list in the '**Materials**' box may not be exactly as it appears in your widow. However, that doesn't bring any change in the outcome of the design process as you can customize this list any time.

The '**Add New Material Quick…**' button allows you to define materials quickly from a list of pre-defined materials. The '**Add New Material**' button allows you to define materials by changing their properties. The '**Add Copy of Material**' button allows you to define a material with same property as an already defined material. The '**Modify/Show Material**' button displays the property of an already defined material with the possibility of modification. The '**Delete Material**' button, when active, deletes a defined material property.

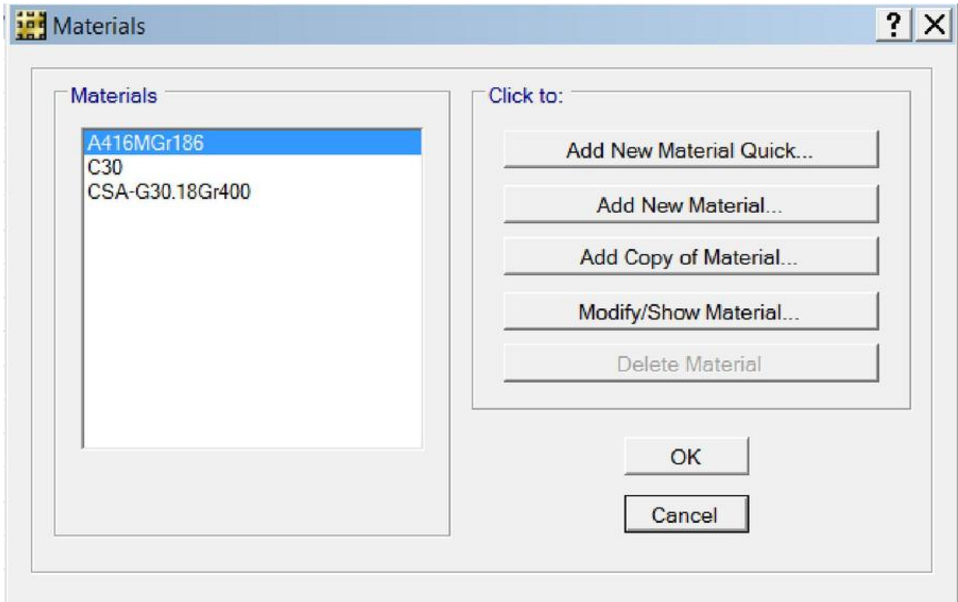


Figure 44

For this particular problem, we will modify the existing '**C30**' concrete using the '**Modify/Show Material**' button and define a new rebar property through the '**Add New Material**' button.

Highlight '**C30**' in the '**Materials**' box and click on the '**Modify/Show Material**' button. This can also be achieved by right clicking on '**Materials**' on the model explorer. The following window will pop up.

This is the window where concrete properties will be modified. Put any name for the material in the text field in front of '**Material Name**'. But, it is important to make sure that the concrete with the defined material name is assigned for the footing. For this particular problem, let us keep the name of the concrete as '**C30**'. Since we are defining a concrete, the '**Material Type**' should be set to '**Concrete**'.

The unit weight of reinforced concrete may vary depending of the design code of your country. Thus, enter the unit weight of reinforced concrete stipulated in your country code in the text field in front of '**Weight per Unit Volume**'. For this particular example, we will use 25kN/m^3.

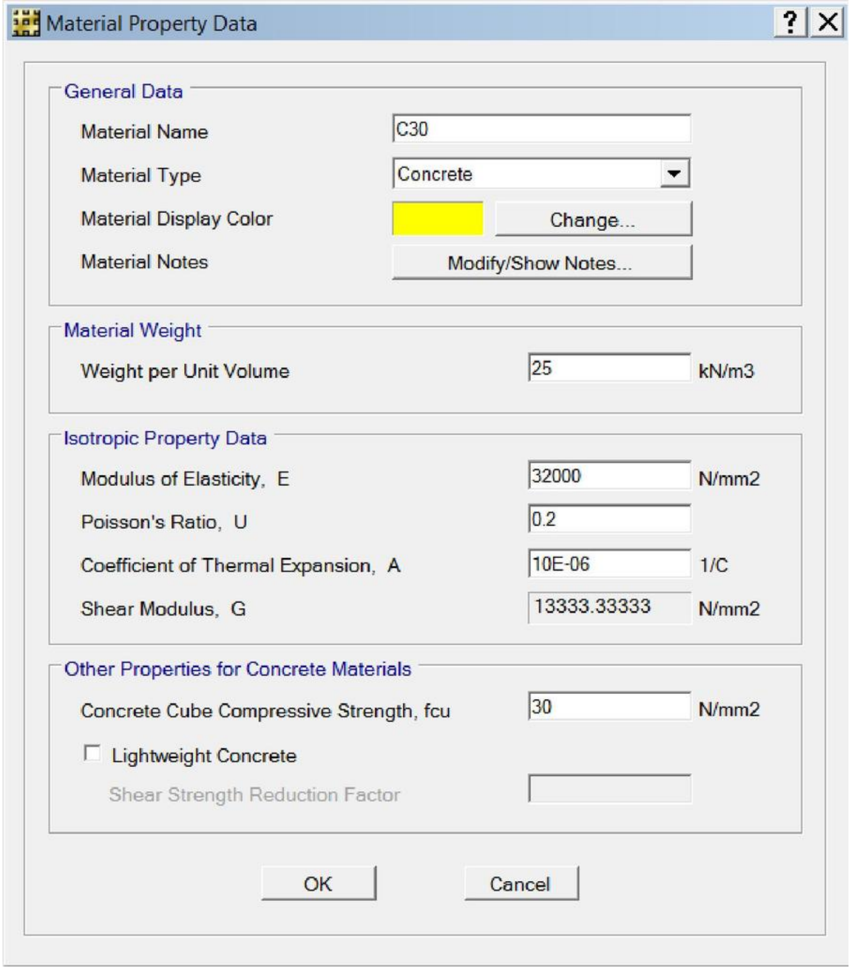


Figure 45

For C-30 concrete, the modulus of elasticity according to BS 8110-1197 is around 32GPa. Thus, enter this value in the text field in front of '**Modulus of Elasticity, E**'. If you selected another design code in step 1 while creating the model, you should refer to actual value of this parameter from the code and enter it accordingly. Be aware of the units though.

The values of Poisson's ratio and coefficient of thermal expansion may also be defined in the design code and should be entered accordingly. For this particular problem, a value of 0.2 for '**Poisson's Ratio, U**' and a value of $10 \times 10^{-6}/°C$ for '**Coefficient of Thermal Expansion, A**' will be entered. The '**Shear Modulus, G**' will be automatically calculated in an un-editable text field.

The grade of concrete for this particular problem is C-30 which is a concrete with 28 day characteristics cube compressive strength of 30MPa. The concrete designation may be different for different country codes but the concept is the

same. Therefore, enter 30 in the text field in front of '**Concrete Cube Compressive Strength, fcu**'.

If a lightweight concrete is used, check on '**Lightweight Concrete**' and enter the corresponding '**Shear Strength Reduction Factor**' in the space provided.

When you press on '**OK**', a concrete material with the above properties will be added to the list of materials. This material will be assigned for the footing before the analysis.

After modifying the concrete property, the program returns to the window shown in Fig. 44. To define a rebar property, we will follow the same procedure as we followed while defining the concrete property. Since a new rebar property will be defined, click on the '**Add New Material…**' button. A '**Material Property Data**' window pops up and when you change the '**Material Type**' to '**Rebar**', the window appears to look like the following.

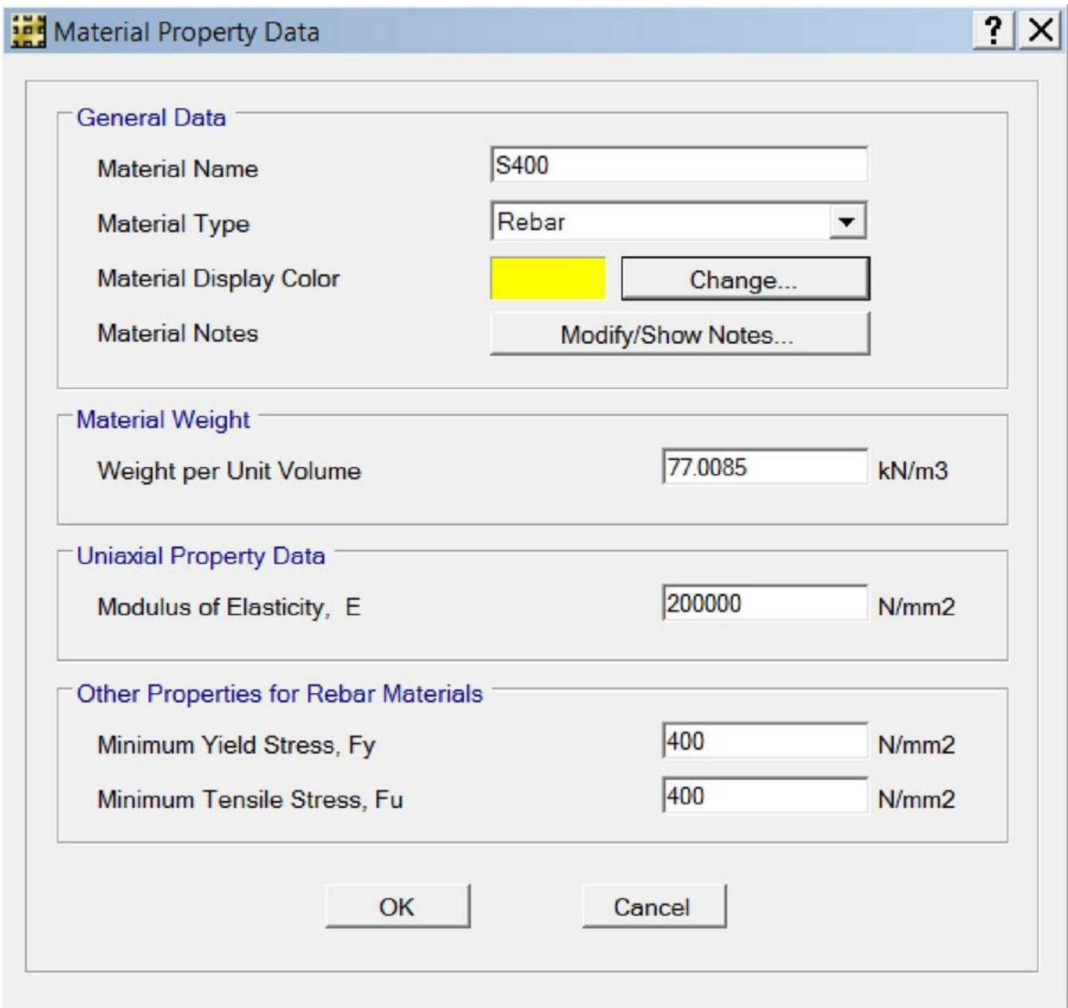


Figure 46

Change the '**Material Name**' to any name you want. Here, we name it '**S400**'. The material type should be '**Rebar**'. The weight per unit volume of steel is stipulated in the design code. For BS 8110-1197, the weight per unit volume is 77.0085kN/m^3. Thus, enter this value in the text field in front of '**Weight per Unit Volume**'. The modulus of elasticity for reinforcement bars according to the same design code is 200GPa. Thus enter this value in the text field in front of '**Modulus of Elasticity, E**' considering the unit.

In the '**Other Properties for Rebar Materials**' box, two quantities are mentioned: minimum yield stress and minimum tensile stress for the reinforcing material. The values of these parameters will be specified in the design code which you defined earlier. If the code assumes that the rebar material exhibits elastic perfectly plastic behavior, the values of these two quantities will be the same. The grade of steel to be used for this particular example is S-400. The yield stress for this type of reinforcement bar is 400MPa. Since the design code of my country assumes that rebars exhibit elastic perfectly plastic behavior, the minimum tensile stress will also be 400MPa. Thus enter 400 in both text fields in front of the '**Minimum yield stress, Fy**' and '**Minimum Tensile Stress, Fu**'. Then press '**OK**' in both '**Material Property Data**' and '**Materials**' windows concluding the material definition step.

The other property which should be defined is the soil support. To define the soil properties, go to the '**Define**' menu and click on '**Soil Subgrade Properties**' menu item and the following window appears.

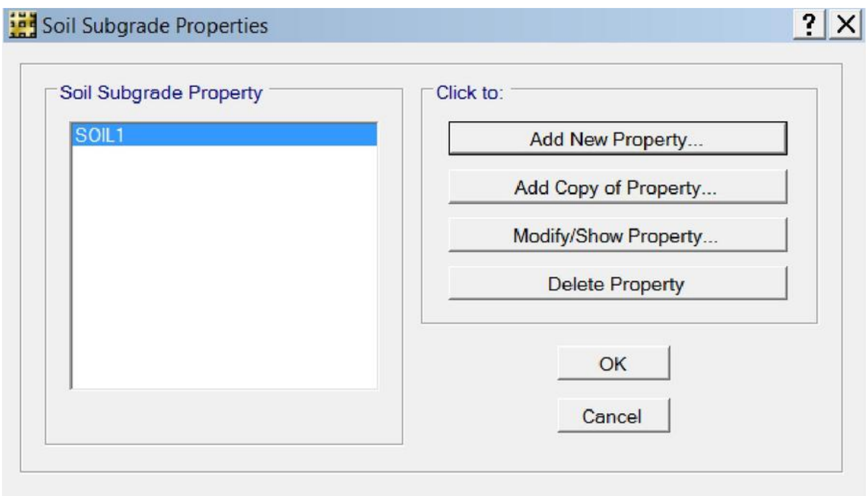


Figure 47

By using the buttons in this window, a new soil property or copy of soil property can be added. An existing soil property can also be modified or deleted. For this problem, let us add a new soil property by using the '**Add New Property…**' button. Thus, click on this button and the following window pops up.

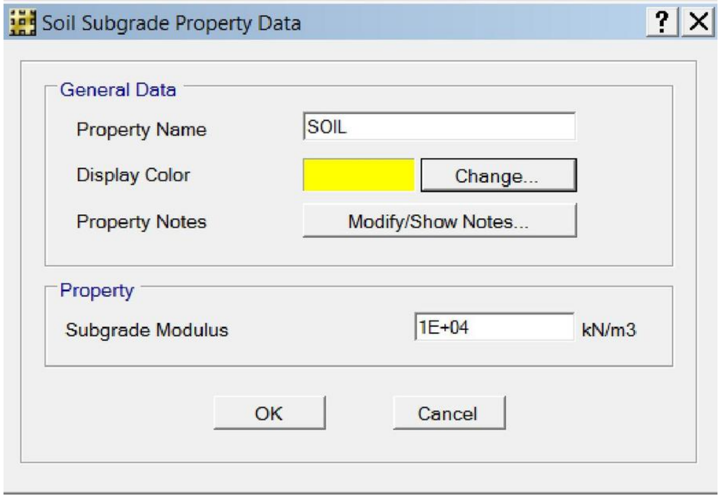


Figure 48

In this window, set the property name to '**SOIL**' and change the subgrade modulus to 10,000kN/m^3. Because, the subgrade modulus which can be assume by the ratio of the bearing capacity by the allowable settlement for this particular example is so. Then press '**OK**' in both the '**Soil Subgrade Property Data**' and '**Soil Subgrade Properties**' windows.

STEP 3: Defining Footing and Column Properties

After defining the material properties, the footing and column properties can be defined. This definition can take place in two ways: from the menu bar and from the model explorer. In SAFE software, footings are modelled as 'footings' and foundation columns are modelled as 'stiff'.

To define footing and column properties from the menu bar, go to '**Define**' menu and click on '**Slab Properties…**'. The following window will pop up.

The '**Add New Property…**' button prompts the user to enter new properties for the footing and foundation column while the '**Add Copy of Property…**' copies the property of an existing slap. The '**Modify/Show Property…**' allows the user to show the property of an existing component with the possibility of modification. When the '**Delete Property**' button is active, it allows the user to delete an existing slab property.

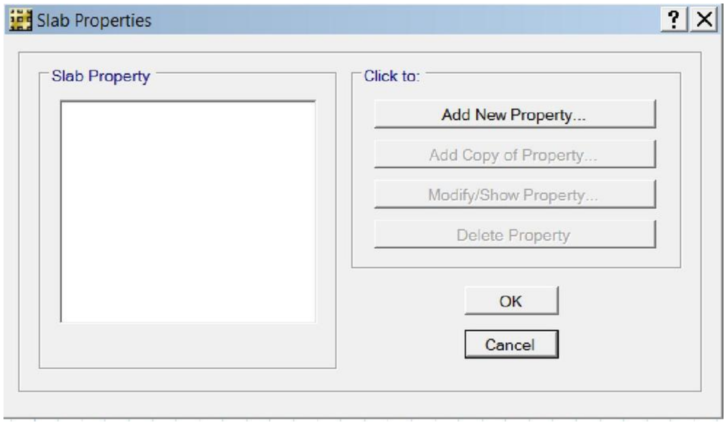


Figure 49

In this case, we use the '**Add New Property…**' button to add new slab properties for the footing and foundation column. Click the button and the following window will pop up.

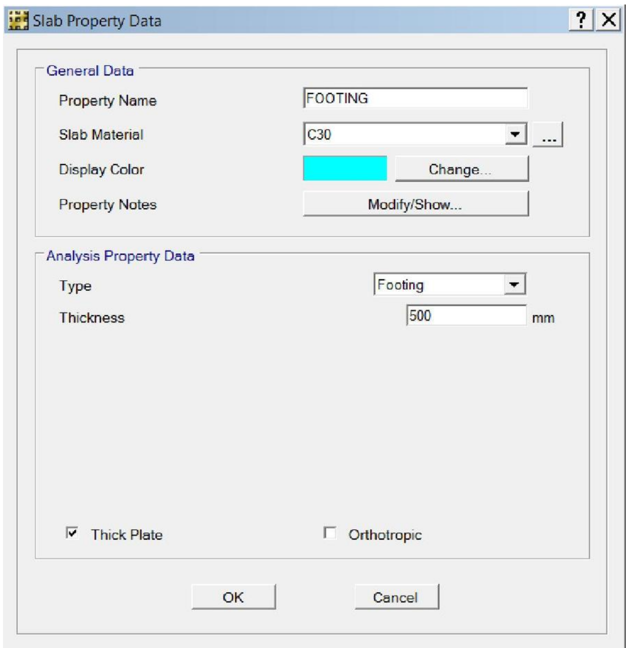


Figure 50

The '**Property Name**' can be assigned with any name; but, in this case, the property name which will be used is '**FOOTING**'. The '**Slab Material**' should be set to the concrete grade which is defined in step 2. Since the name of the concrete material defined in step 2 for this particular problem is '**C30**', a material with this name should be selected from the list.

In the '**Analysis Property Data**' box, '**Type**' should be set to '**Footing**'. The '**Thickness**' value should be set to the thickness of the footing defined in the example. Since the thickness of the footing is 500mm, this value is entered in the text field corresponding to '**Thickness**'. As footings are modelled as thick plates, check the '**Thick Plate**' option. The '**Orthotropic**' check box is selected when a footing with irregular dimension is to be used.

When you press '**OK**', the '**Slab Property Data**' window will be exited and the '**Slab Properties**' window gets activated. Now, the property of the foundation column will be added. To do this, again click on the '**Add New Property…**' button. The following window appears after the click.

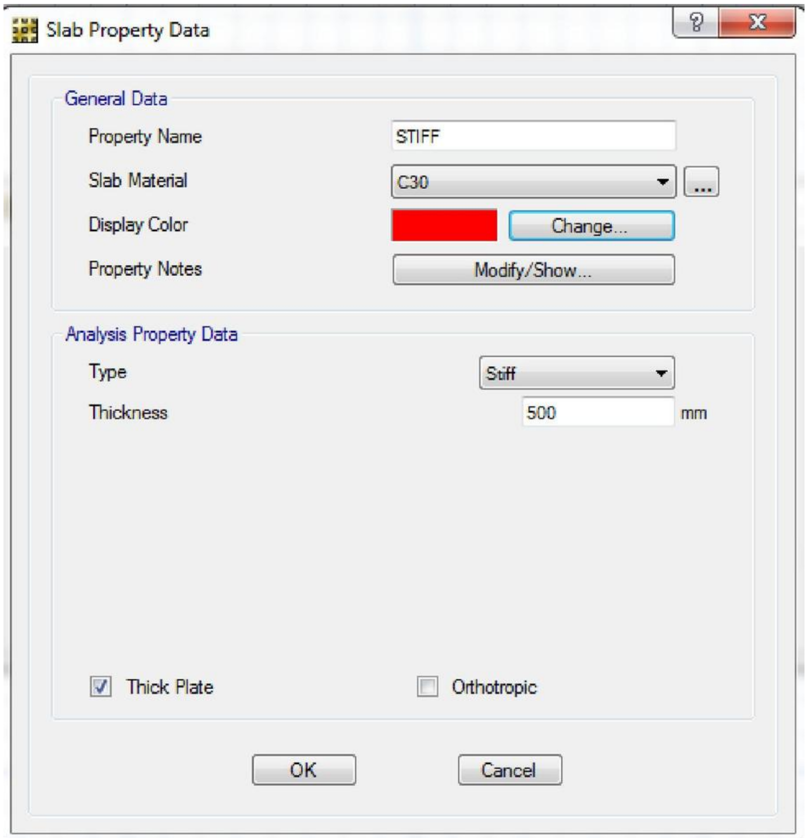


Figure 51

The '**Property Name**' can be any name but we use '**STIFF**'. The '**Slab Material**' should be set to the concrete grade which is defined in step 2. Since the name of the concrete material defined in step 2 for this particular problem is '**C30**', a material with this name should be selected from the list.

In the '**Analysis Property Data**' box, '**Type**' should be set to '**Stiff**' as we are defining the property of column. The '**Thickness**' value should be the foundation column thickness which is equal to the footing thickness. Thus, a value of 500 will be entered in the text field corresponding to '**Thickness**'. As foundation columns are modelled as thick plates, check the '**Thick Plate**' option. The '**Orthotropic**' check box is selected when a column with irregular dimension is to be used.

When you press '**OK**', the '**Slab Property Data**' window will be exited and the '**Slab Properties**' window gets activated. Again press '**OK**' and exit the window for defining the footing and foundation column.

STEP 4: Defining Load Patterns, Load Cases and Load Combinations

The loads on the foundation should be defined accordingly before the analysis. First, the load pattern should be defined. This can be done from the '**Define**' menu or from the '**Model Explorer**'. This time, we will do it from the model explorer. In the model explorer, expand '**Load Definitions**' and you will see '**Load Patterns**'. When you expand '**Load Patterns**', you will see '**DEAD**' and '**LIVE**'. At the end, the model explorer appears to look like:

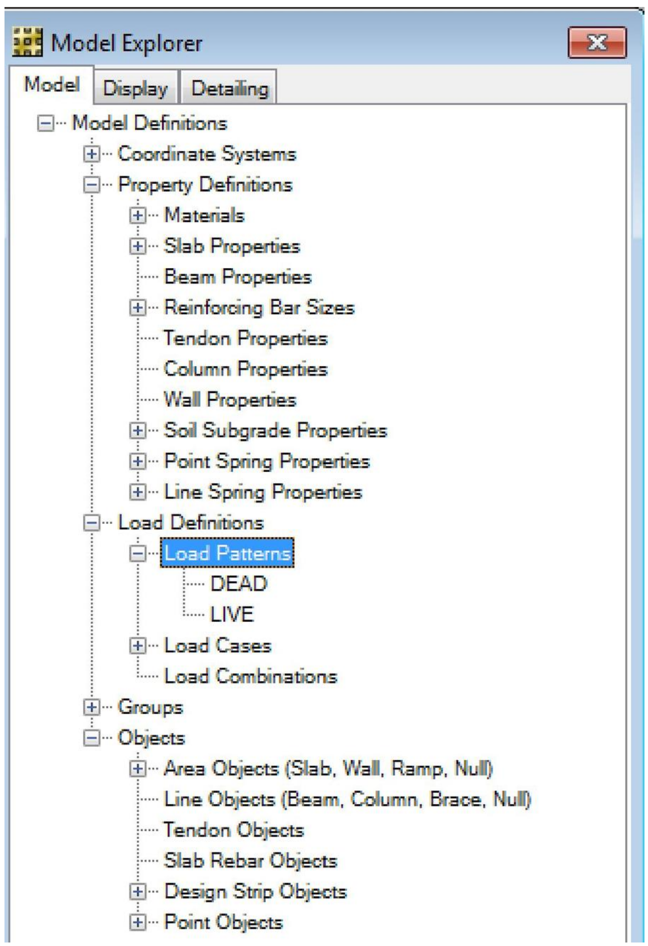


Figure 52

Now, right click on '**Load Patterns**' and click on '**New Load Pattern**' and the following window pops up.

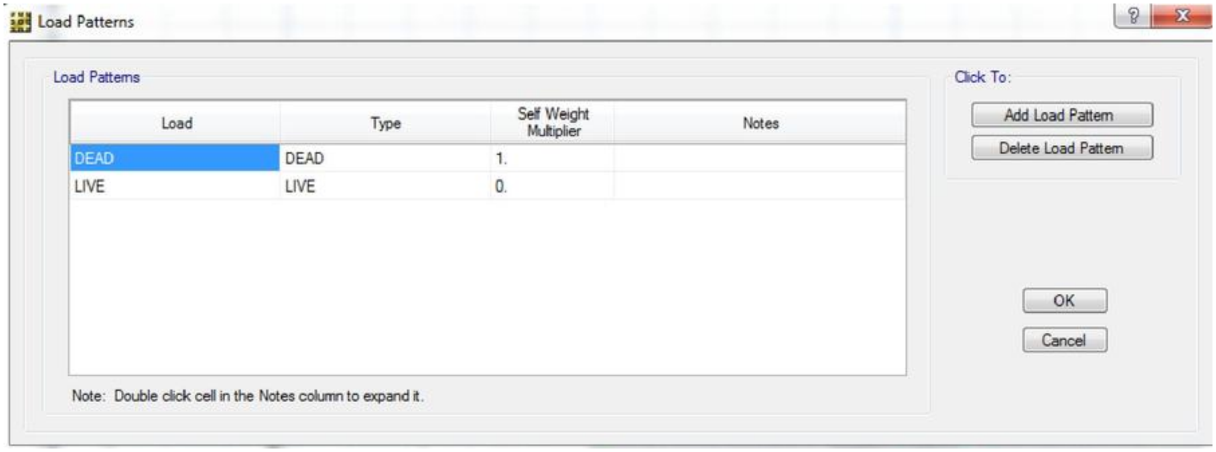


Figure 53

In the '**Load Patterns**' window, two load patterns are already defined: '**DEAD**' and '**LIVE**'. The '**Type**' of the '**Load**' should also be changed accordingly. There are many options for type of loading. The '**Type**' for dead loads should be set to '**DEAD**' and for live loads '**LIVE**'. If there are other types of load patterns on the foundation such as earth quake load, you can add the load pattern with the '**Add Load Pattern**' button. You can also delete any undesirable load pattern using the '**Delete Load Pattern**' button. Since in this example, we have only dead and live loads, we will leave the existing load patterns as they are. The '**Self Weight Multiplier**' value should also be changed accordingly. This value imparts the option whether to consider or ignore the self-weight of the foundation in addition to external loads. If the self-weight of the foundation is already included as an external dead load or if you want to exclude the effect of self-weight from the analysis, the value under '**Self Weight Multiplier**' should be set to zero. In this example, we will consider the self-weight as an additional load to the external dead load. Thus, the value under '**Self Weight Multiplier**' for the '**DEAD**' load is one. For the '**LIVE**' load, it will be zero. Press '**OK**' and the window will be exited.

After this, the load cases will be defined. Load cases are used to dictate the way the loads are applied (statically or dynamically) or the way the structure responds (linearly or non-linearly) for the defined load patterns. To define a load case, go to '**Define**' menu and click on '**Load Cases…**'. The following window will pop up after the click.

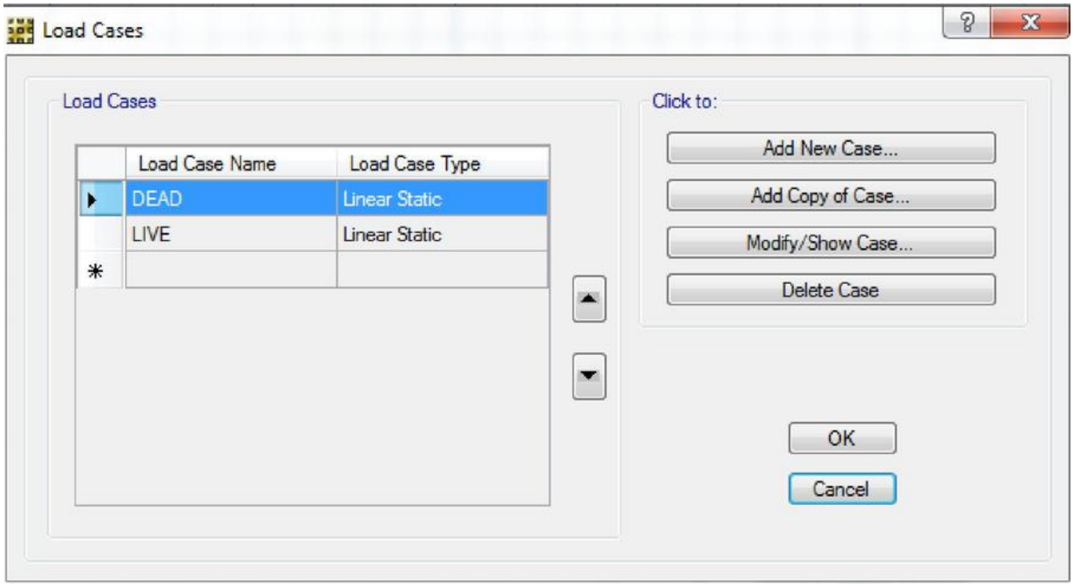


Figure 54

The load patterns which were defined earlier will automatically appear in the list of '**Load Case Name**' of the '**Load Cases**' window. The '**Load Case Type**' column shows the way in which each load pattern will be applied during analysis. If you want to modify this, highlight the load pattern for which you are going to change the load case type and click on the '**Modify/Show Case…**' button. If you click this button, the following window will appear. In this window, you can change the way the load is applied from the '**Load Case Type**' box. The way the structure responds can also be selected from the '**Analysis Type**' box. This problem '**Static**' is for the '**Load Case Type**' and '**Linear**' is selected for the '**Analysis Type**' since the load is static and the foundation responds linearly. The scale factor for the dead load in the '**Loads Applied**' box will be left as one. Press '**OK**' and exit the window.

The load case type for the live load should also be 'Linear Static'. Otherwise, it should be changed by clicking the '**Modify/Show Case…**' button to linear static case. If both the load case types are as desired click '**OK**' and exit the '**Load Cases**' window.

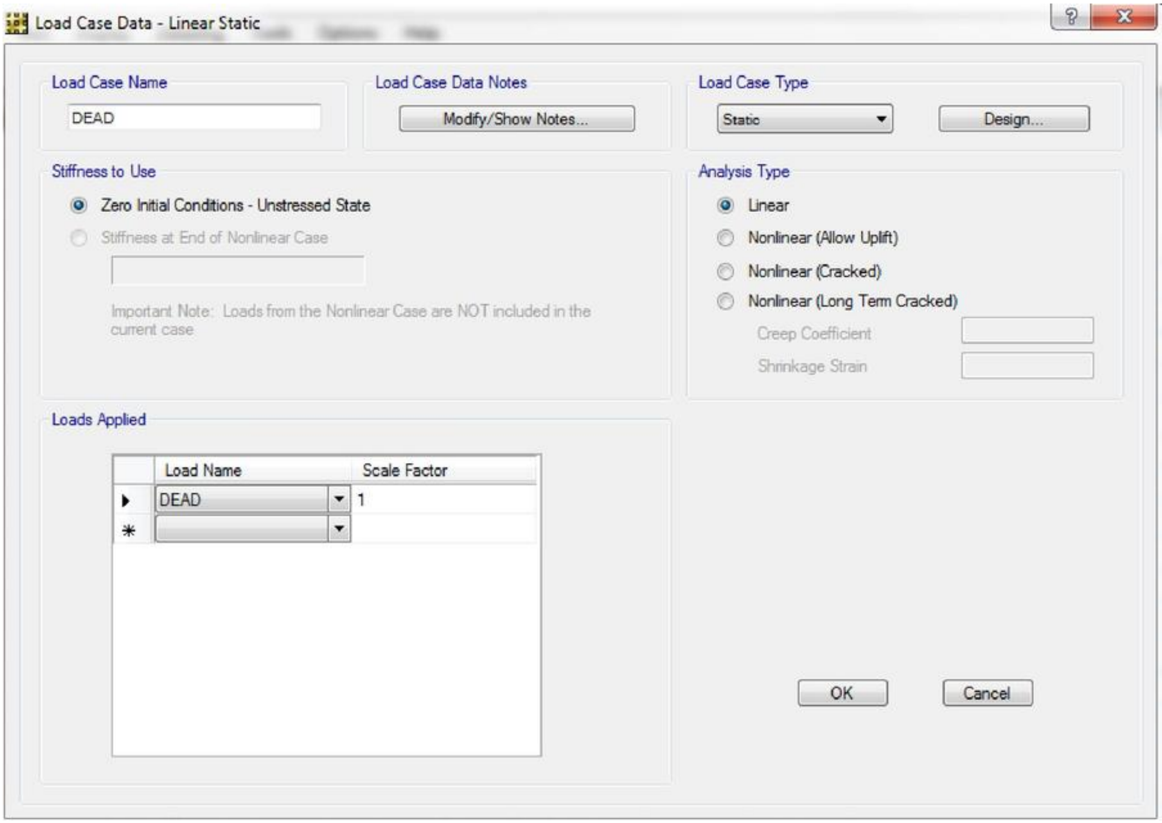


Figure 55

Definition of the load combinations will be the next step. Two load combinations will be considered in this example for ultimate limit state and serviceability limit state. To define load combinations, go to '**Define**' menu and click on '**Load Combinations…**' and the following window pops up.

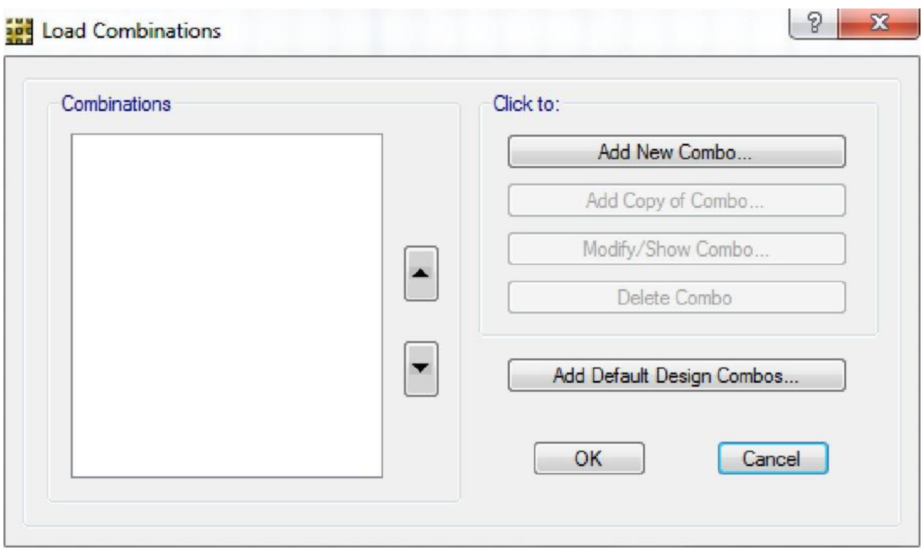


Figure 56

New load combinations will be added through the '**Add New Combo…**' button. Here, two load combinations will be added; one for the ultimate limit state the other for the serviceability limit state. When the '**Add New Combo…**' button is clicked, the following window appears.

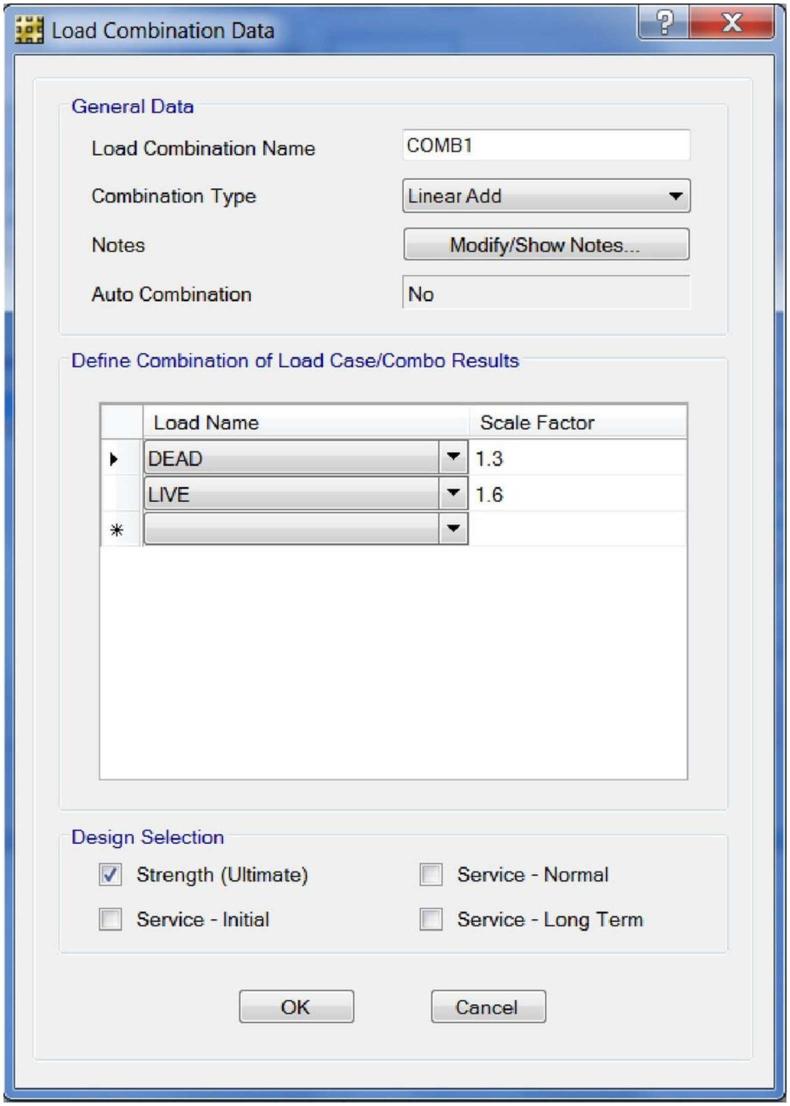


Figure 57

In the '**Load Combination Data**' window, any name can be given for the load combination. The '**Combination Type**' should be set to '**Linear Add**' as the component loads (dead and live) will be added linearly. However, there are also other options from the drop down menu in front of '**Combination Type**'. In the window shown in Fig. 57, the two loads (dead and live) should be activated below the '**Load Name**' column of the '**Define Combination of Load Case/Combo Results**' box. The values in the '**Scale Factor**' column correspond to the partial

safety factors for load for the failure mode under consideration. These partial safety factors are specified in the design code of your country. In the design code of my country, the partial safety factor for dead loads for ultimate limit sate case is 1.3 and for serviceability limit state is 1 for the cases where there are only dead and live loads. For such load patterns, the partial safety factor for live loads for ultimate limit sate case is 1.6 and for serviceability limit state is 1. Thus, enter a value of 1.3 in front of '**DEAD**' and 1.6 in front of '**LIVE**' in '**Scale Factor**' column. The failure condition which is being under consideration can be defined by selecting and deselecting the check boxes in the '**Design Selection**' box. Since the above scale factors are for the ultimate limit state, check on '**Strength (Ultimate)**'. The '**Load Combination Data**' window for the serviceability limit state looks like the following window.

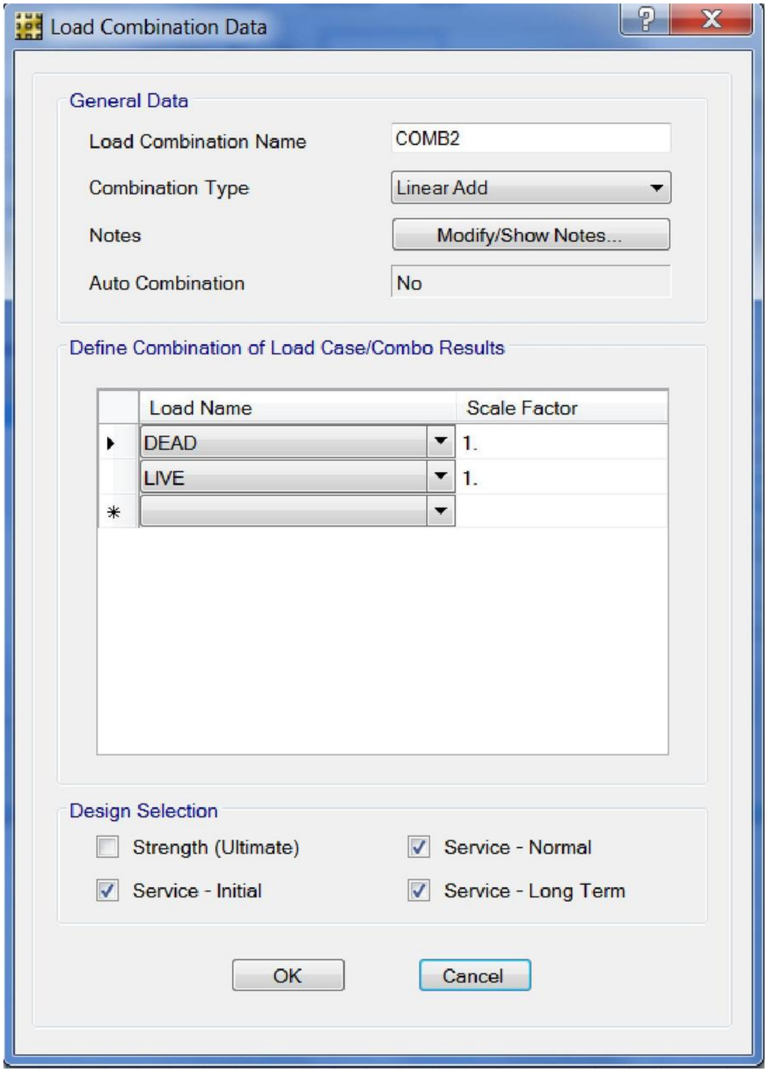


Figure 58

STEP 6: Drawing the Footing Components and Design Strips

The footing, the foundation column and a point where the loads will be applied on the foundation column should now be drawn on the grid.

i. *Drawing the footing*

Since the footing will be drawn around points and since the footing is rectangular, go to the '**Draw**' menu and click on '**Draw Rectangular Slabs/Areas**' or simply click on the equivalent icon  on the left hand side tool bar. The following window will appear on the screen after the click.

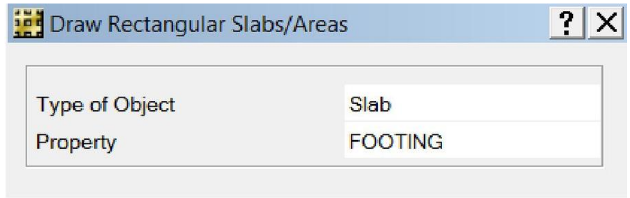


Figure 59

Making sure that the '**Property**' is set to '**FOOTING**', first click on the grid where one corner of the footing will be located. Then, without releasing the click, drag into the opposite corner of the footing to draw the footing.

ii. *Drawing the foundation column*

While the window in fig 59 is active, change the '**Property**' to '**STIFF**' and click on the grid where one corner of the foundation column will be located. Then, without releasing the click, drag into the opposite corner of the foundation column to draw the foundation column.

iii. *Drawing the point on the foundation column where the load will be applied*

Go to '**Draw**' menu and click on '**Draw Points**', then click on the mid-point of the footing and the point will be created. If the cursor could not snap to the midpoint, you can adjust the '**Snap Options**' from the '**Draw**' menu.

After this, the design strips will be drawn. Design strips determine the way in which different quantities related to the reinforcement calculation are calculated. Forces are integrated across the design strips. Thus, the larger the width of coverage of the design strips within the given structure, the higher will be the calculated values of the bending moments and shear forces. Thus, an optimum width of strip is required compromising the safety and economical requirements. The width of the design strip will be specified in the design code. According to the code of my country, the width of design strips for isolated foundations is 1m. Thus, a one meter design strip will be drawn in both X and Y directions on the foundation. These design strips in X and Y direction are usually defined in SAFE software as layer A and layer B.

To draw the design strip, go to the '**Draw**' menu and click on '**Design Strips**' or simply click on the equivalent icon from the left hand sided tool bar and the following window pops up.

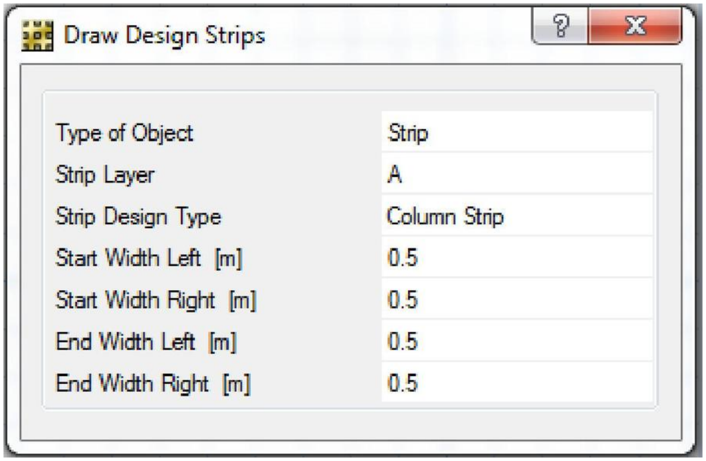


Figure 60

In this window, the '**Strip Layer**' should be selected to be either '**A**' or '**B**'. But if '**A**' is for design strip in X direction '**B**' should be for Y direction and vice versa. Since we are drawing a strip around the column to consider maximum moment and shear forces, the '**Strip Design Type**' should be set to '**Column Strip**'. The width of the strip will be 1m in both directions and since the column will be centrally located, all the widths will be set to 0.5m. To draw the design strip in the X direction, without closing the window, left click at the center of one side of the footing on the plan view parallel to the Y axis and again left click at the center of the parallel side and right click. This creates a design strip in X direction. In doing so, if you can't snap to the center of the side of the footing, you can modify the snap options by clicking on the '**Snap Options…**' command from the '**Draw**' menu and adjusting the options which you want to snap to. The design strip in Y direction can also be drawn in a similar procedure after changing the '**Strip Layer**' to '**B**'.

You can display the design strips by setting the display options by clicking on '**Set Display Options…**' from the '**View**' menu or by simultaneously clicking on '**Ctrl**' and '**W**' keys or by just clicking on the set display options icon from the tool bar below the menu bar. This results in the following window:

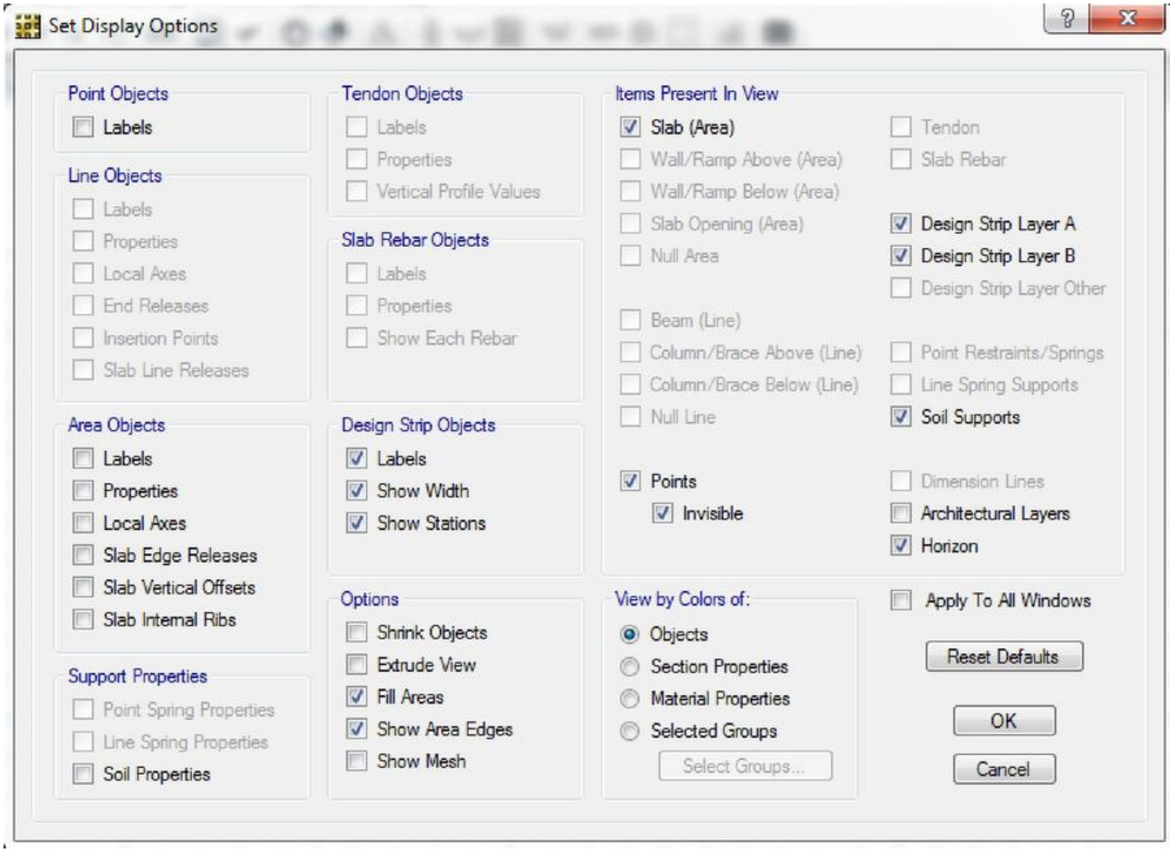


Figure 61

Then, check on ‘**Labels**’, ‘**Show Width**’ and ‘**Show Stations**’ in the ‘**Design Strip Objects**’ box and press ‘**OK**’ and after drawing dimension lines by using the command ‘**Draw**’>’**Draw Dimension Lines**’, the following window appears displaying the design strips in the two directions.

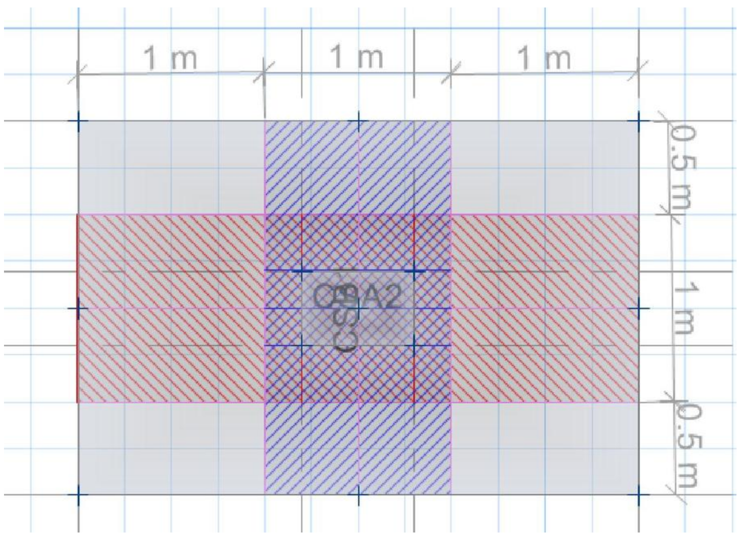


Figure 62

STEP 7: Assigning Slab Data, Support Data and Load Data

The slab data, support data, and load data which are defined in the previous steps should be assigned to the corresponding structural component. The way to do this is first to select the component and next to assign the slab property, support property or loads accordingly. The selection can be done by clicking on the component from the plan view or through the '**Select**' menu. The latter option assures that the component is selected exactly as selection by clicking may result in incorrect selection. Thus, all the selections here will be done from the '**Select**' menu. The assignments will be discussed as follows:

a. *Assigning support data to the footing*

   Select the footing through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**'Slab Properties…**'. Then select '**FOOTING**' and press '**OK**'. Then assign the footing property through the following strings of commands '**Assign**'>'**Support Data**'>'**Soil Properties…**'. Then select '**SOIL**' and press '**OK**'.

b. *Assigning reinforcement data to the design strips*

   Select each design strip through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**Design Strip Layers…**'. Then select '**A**' or '**B**' (one at a time) and press '**OK**'. When you right click on the selected strip layer, the '**Slab-Type Area Object Information**' window pops up. In the '**Design**' tab of this window, set the '**Rebar Material**' to '**S400**' and press '**OK**'. Do this for both strips.

c. *Assigning load on the foundation column*

   To assign load on the foundation column, right click on the point at the center of the foundation column and a '**Point Object Information**' window pops up. In this window, click on the '**Loads**' tab.

   The dead load and the live load can be assigned through the '**Assign Load…**' button. The procedure is: click on '**Assign Load…**' button, then select '**Force Loads**' then press '**OK**' then select either '**DEAD**' or '**LIVE**' depending on which loads you want to enter their values then enter their values accordingly (both the concentrated load and the bending moment) and in the right direction (axis), then select '**Add to Existing Loads**' and press '**OK**'. While doing this, the foundation column dimensions should be entered in the '**Size of Load for Punching Shear**' box of '**Point Loads**' window.

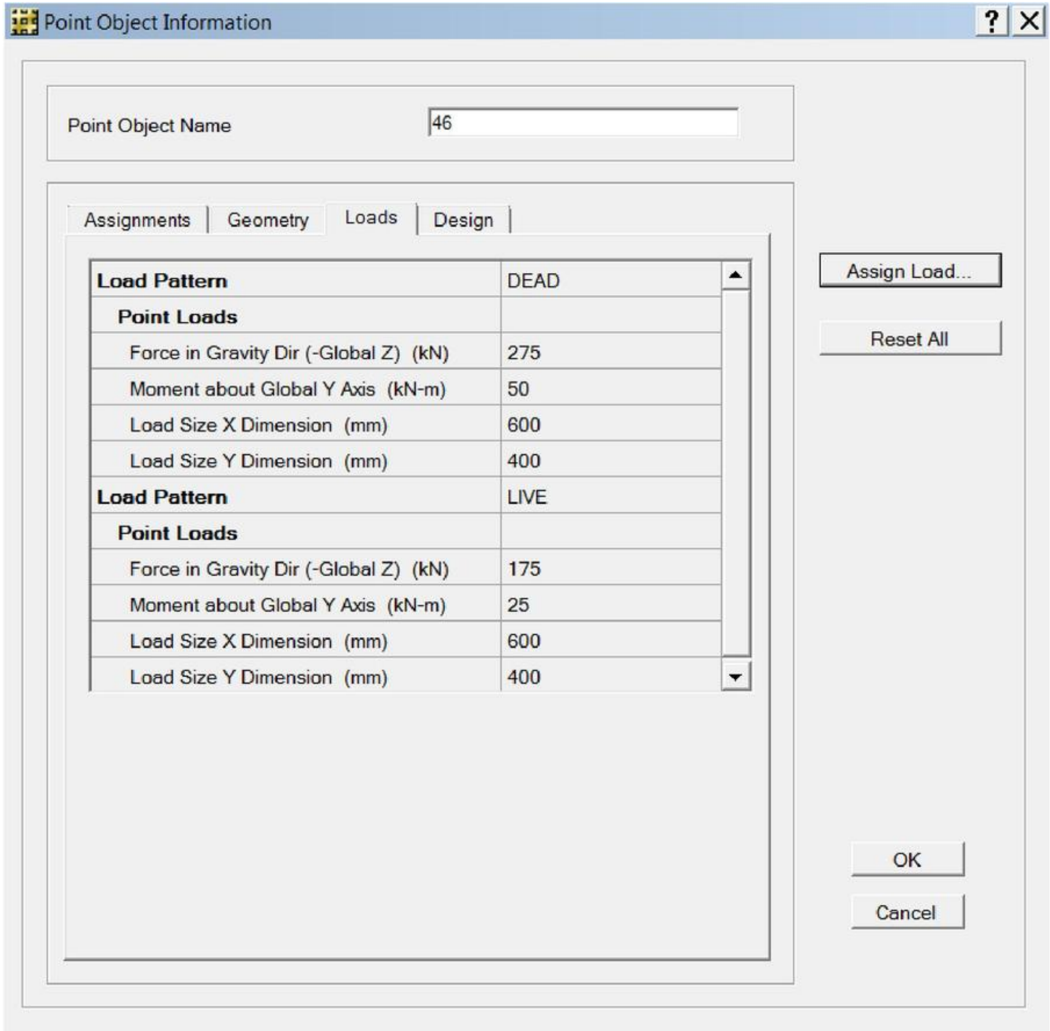


Figure 63

You can also delete any forces which are wrongly entered through '**Delete Existing Loads**' radio button option. In this example, since the dead concentrated load is 275kN and the dead bending moment is 50kNm about y-axis and since the live concentrated load is 175kN and the live bending moment is 25kNm about y-axis, these values are entered in the Gravity Direction for the concentrated load and along y-axis for the bending moment.

STEP 8: Running the Analysis

After this, the analysis can be run. But, make sure that the footing and the foundation column are assigned with the correct rebar material. To do this, right click anywhere in the plan view of the footing and the '**Slab-Type Area Object Information**' window will pop-up.

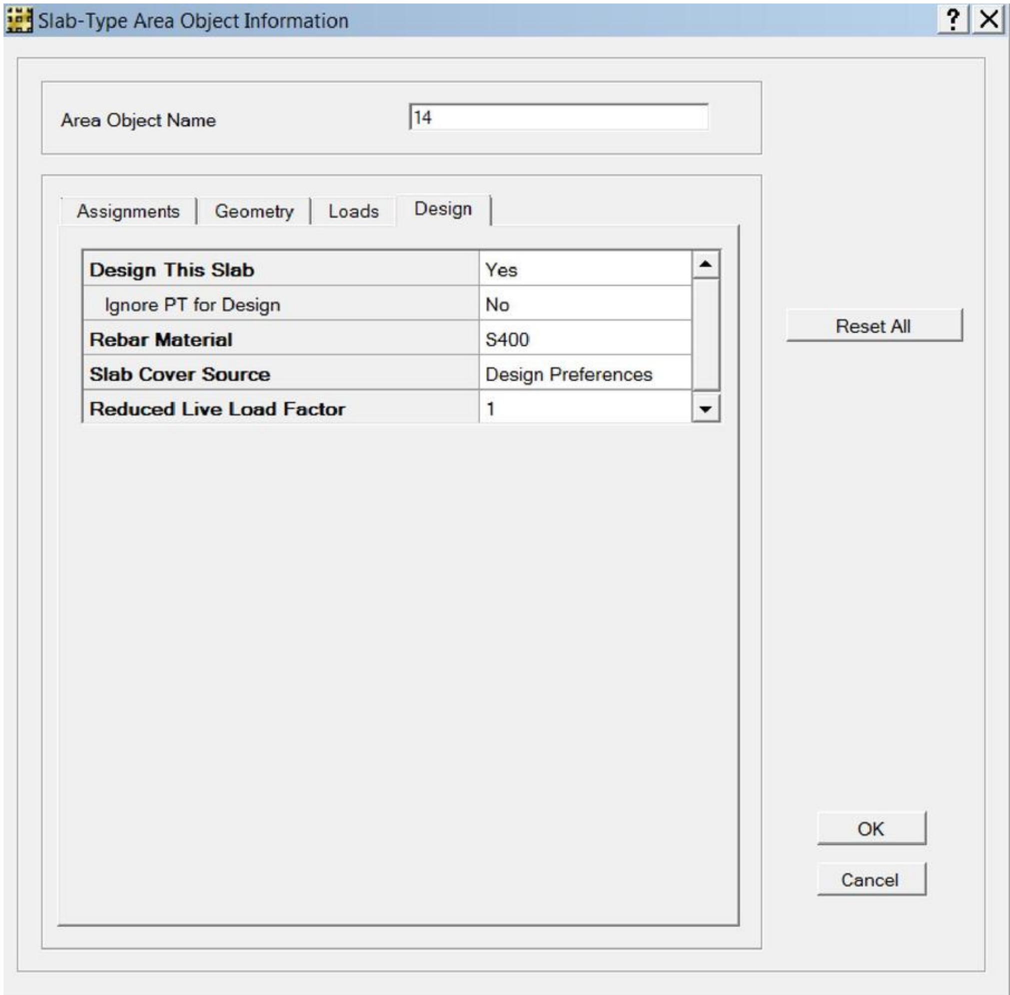


Figure 64

In the '**Design**' tab of this window, set the value of '**Rebar Material**' to '**S400**'. Do this for the column foundation, and both design strips as well. After this, go to the '**Run**' menu and click on the '**Run Analysis**' command.

STEP 9: Displaying the Output

Once the analysis is run, the output will be displayed. Particularly, the punching shear design is of great importance as the footing cannot be designed without the punching shear requirement being adequately satisfied. To do this, go to the '**Run**' menu and click on '**Run Analysis & Design**' command or simply click on the '**F5**' key. When you do this, you will be prompted to save the model, if you haven't already don this. When you save the model, the following window showing the displacement of the soil in a banded figure will be displayed.

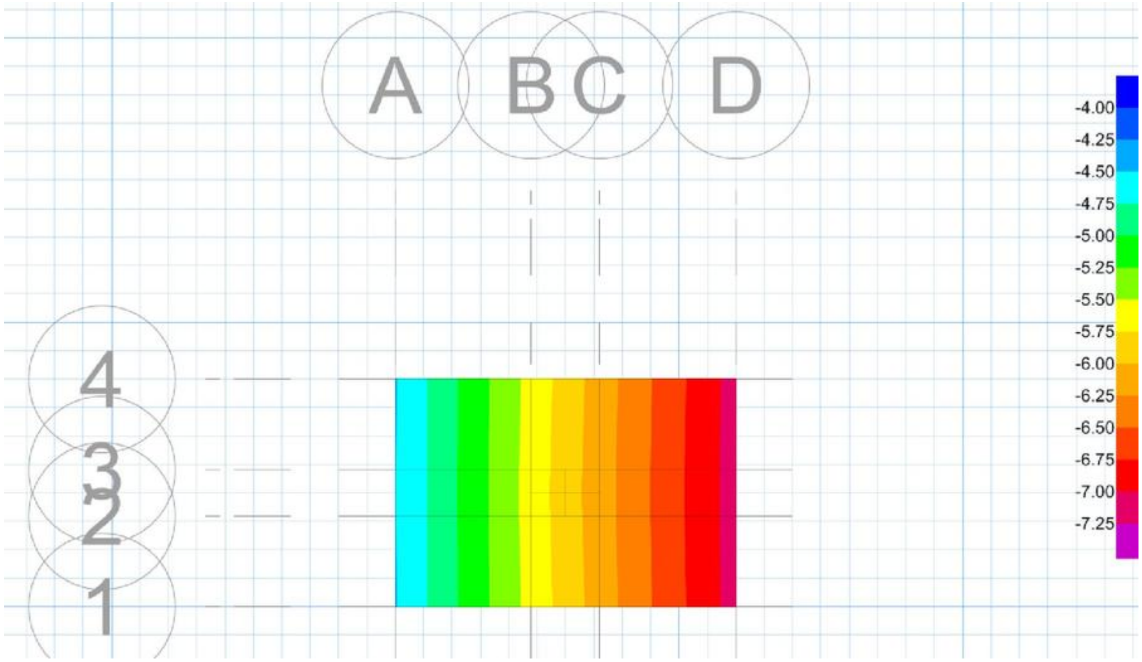


Figure 65

To display the punching shear ratio, go to the '**Display**' menu and click on '**Show Punching Shear Design**'. After this, the punching shear ratio will be displayed in the plan view around the foundation column as in the following figure.

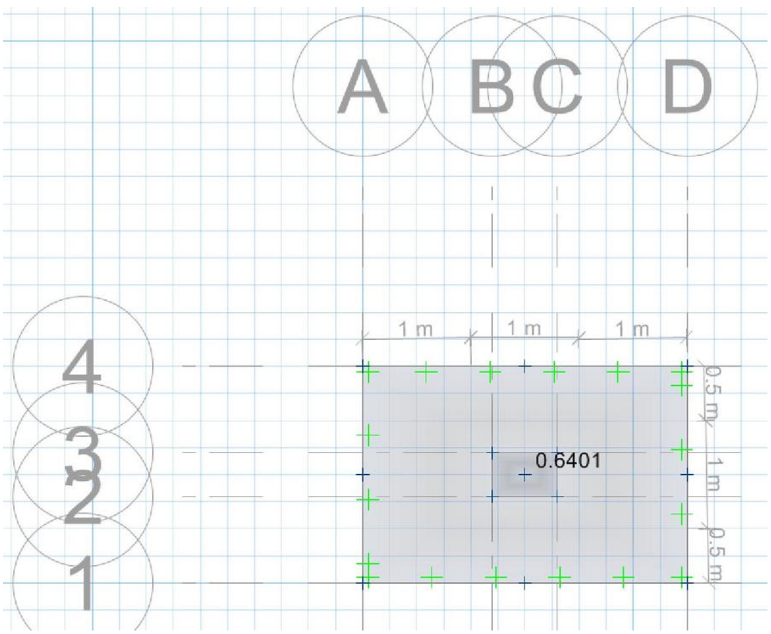


Figure 66

As can be noticed from the figure, the punching shear ratio is 0.6401. Generally, a punching shear ratio less than one indicates the concrete thickness is adequate to resist punching shear and a value greater than one indicates that the punching shear capacity is exceeded somewhere along the critical section. For economical design, it is recommended to keep the punching shear ratio between 0.95 and 1 as very small values of punching shear ratio means excess concrete thickness is used. However, if the punching shear ratio is greater than one, the thickness of the concrete should be increased and the foundation should be re-designed. A detailed quantitative description of the foundation design can also be obtained by right clicking on the plan view shown in Fig 67 as shown below. Several trial may be made by zooming in and out to get the quantitative description.

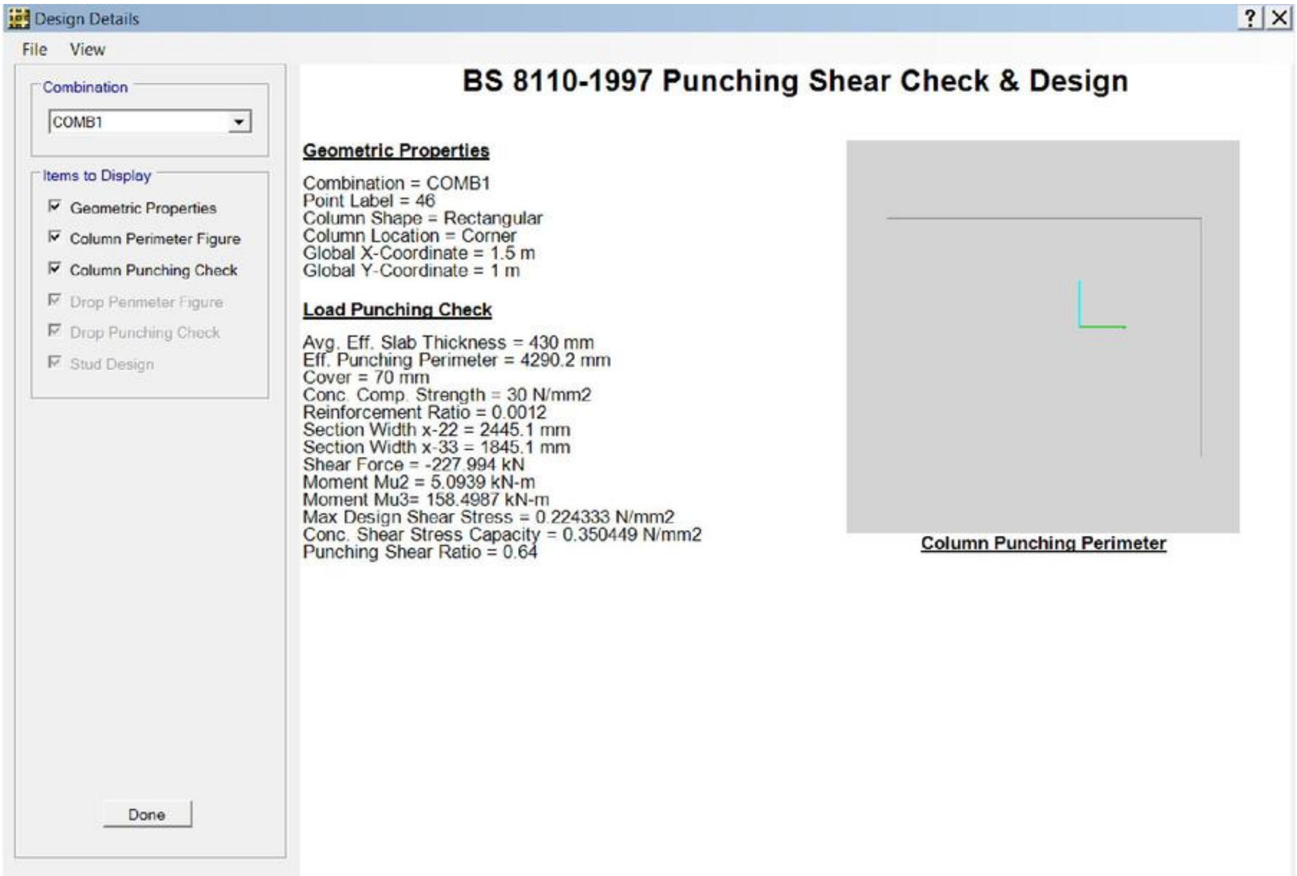


Figure 67

Through the '**Display**' menu, relevant quantities can be displayed on the screen. For instance, the '**Display**'>'**Show Strip Forces**' command or by simply clicking the '**F8**' key, the following window will be displayed.

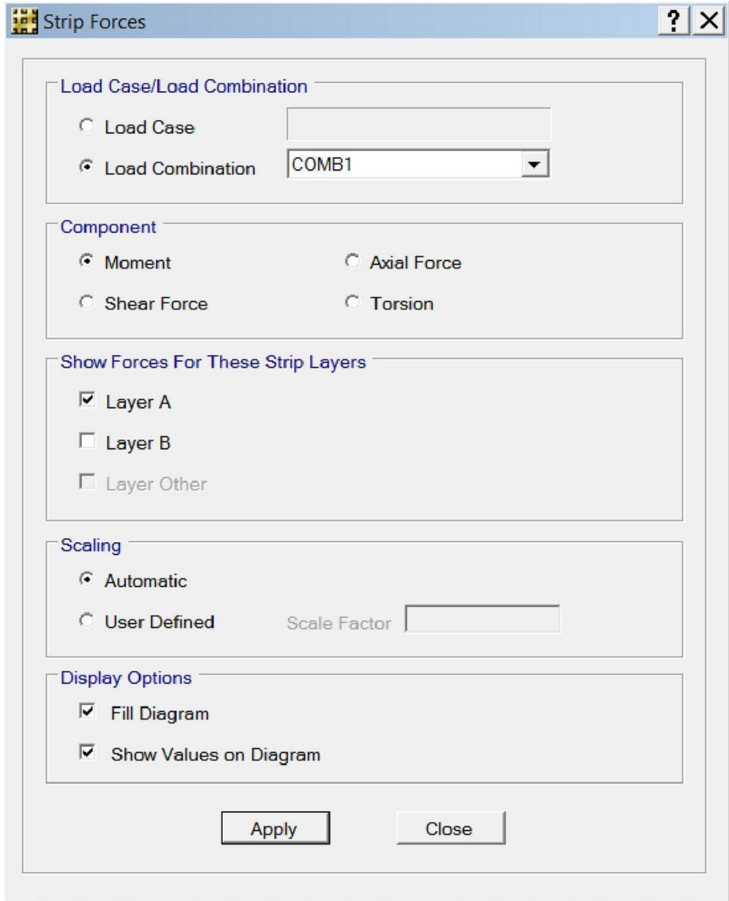


Figure 68

In this window, the '**Load Case/Load Combination**' box provides with radio buttons to select for which particular load case or load combination that we want to display the output. The '**Component**' box contains four radio buttons to select which quantity to display. The '**Show Forces For These Strip Layers**' box allows us to select the strip layer for which the quantity is displayed. Both strip layers can be selected at the same time. From the '**Scaling**' box, we can select whether automatic scaling or user defined scaling is used while displaying the diagram. The '**Display Options**' box allows us to fill or not to fill the diagram and to display or not to display the values on the diagram. For the preferences shown in Fig. 68, the following diagram will be displayed.

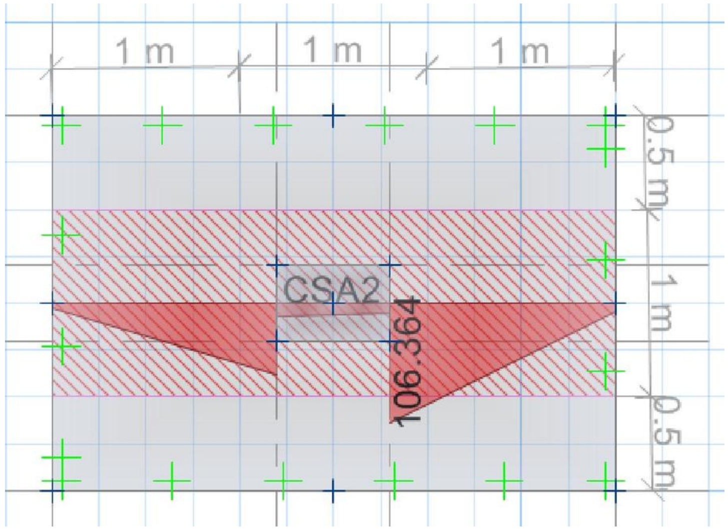


Figure 69

The '**Display**'>'**Show Slab Design…**' command results in the window shown in Fig 70. In this window, several options can be set in order to display the footing design in the way we wanted to. The '**Choose Display Type**' box allows us to select the '**Design Basis**' between '**Strip Based**' and '**Finite Element Based**'. Unless some differences in the way the design is displayed, there is no difference in the amount of reinforcement between these selections. Through the '**Display Type**' in the same box, it can be selected whether to display flexural reinforcement or shear reinforcement. This box also allows us to impose or not to impose minimum reinforcement during the design. The '**Rebar Location Shown**' box allows us to select which reinforcement, top or bottom or both, to be displayed. The '**Reinforcing Display Type**' box allows us to set the manner in which the amount of reinforcement is displayed. The option whether to show the reinforcing envelop diagram and the reinforcing extent can be set by the check boxes in the '**Reinforcing Diagram**' window. The strip layer direction for which the amount of reinforcement is displayed can be chosen from the '**Choose Strip Direction**' box. The '**Display Options**' box allows us whether to display output in filled diagram or not and whether the values at controlling stations will be displayed or not. If we want to display the amount of reinforcement above some specified reinforcement bar area or spacing, we can use the options in the '**Show Rebar Above Specified Value**' box. When the '**Typical Uniform Reinforcing Specified Below**' radio button is selected, the '**Typical Uniform Reinforcing'** box get activated. In this box, we can set a specific value above which the reinforcement amount will be displayed. The reinforcement diagram output, for the options set in Fig. 70, will be shown below in fig. 71.

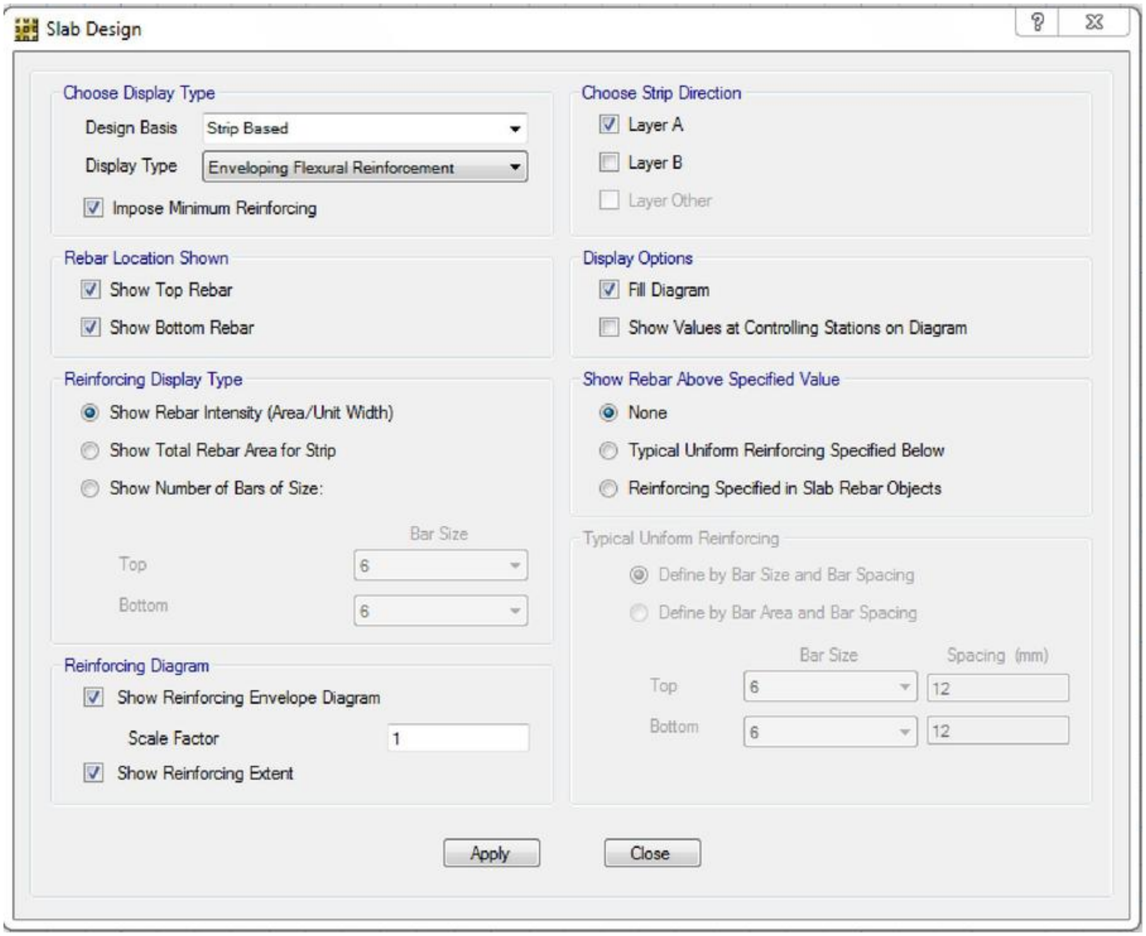


Figure 70

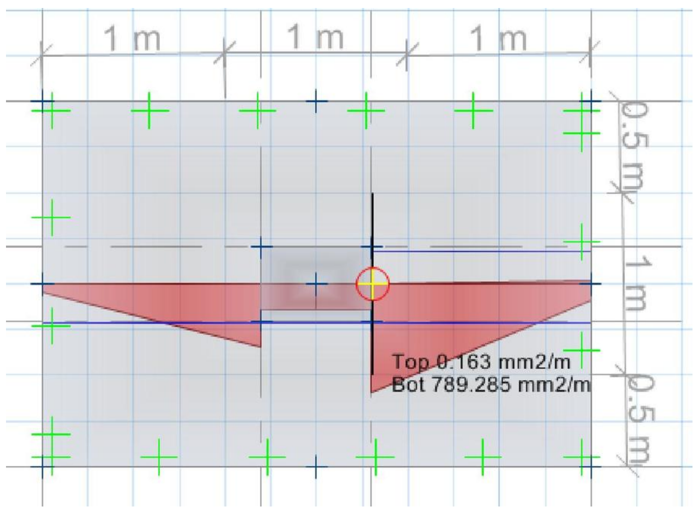


Figure 71

As can be noticed from the footing design diagram in fig. 71, the top reinforcement across strip layer A is around 0mm^2/m for top reinforcement and 790mm^2/m for bottom reinforcement.

The design outputs can also be displayed in tabular format by clicking on the '**Show Tables…**' menu item from the '**Display**' menu or by just clicking on the equivalent icon  from the tool bar below the menu bar and the following window will pop up.

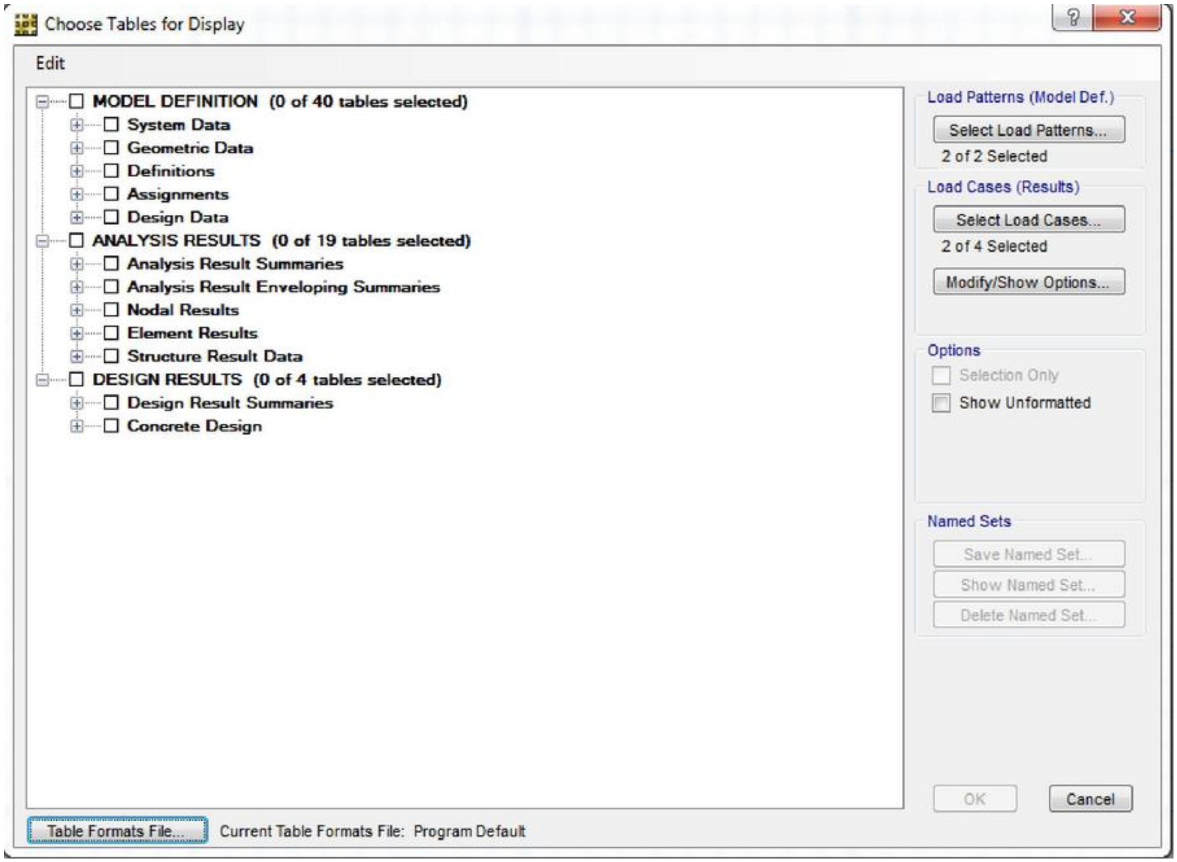


Figure 72

In this window, we can select any of the model definitions or analysis results or design results and press '**OK**' to display the quantity which we want to have a look at. By using the right hand side buttons in the window, the load patterns and the load cases can be selected.

STEP 10: Detailing

After running the analysis and after checking that the results are reasonable, the detailing will be done. However, before running the detailing, the detailing preferences can be set from the '**Detailing**' menu. From the '**Detailing Preferences…**', the likes of dimensional units and material quantity units can be selected. From '**Slab/Mat Detailing Preferences…**', the likes of rebar curtailment options, the rebar detailing options, rebar selection rules and preferred

rebar sizes can be selected. The '**Drawing Sheet Set-up…**' menu allows us to set-up the contents of the drawing sheet. The '**Drawing Format Properties…**' allows us to set some formats in which the output displayed.

To run the detailing, go to '**Run**' menu and click on '**Run Detailing…**' or simultaneously press '**Shift**' and '**F5**' keys or just click on the run detailing icon  from the tool bar just below the menu bar. Then, the '**Run Detailing Options**' window pops up so that we set the detailing options. Set the detailing options which you want and click '**OK**'.

Once the detailing is run, the detailing can be displayed. The detailing display options can be best accessed from the '**Model Explorer**'. When expanded in full, the '**Detailing**' tab of the model explorer looks like:

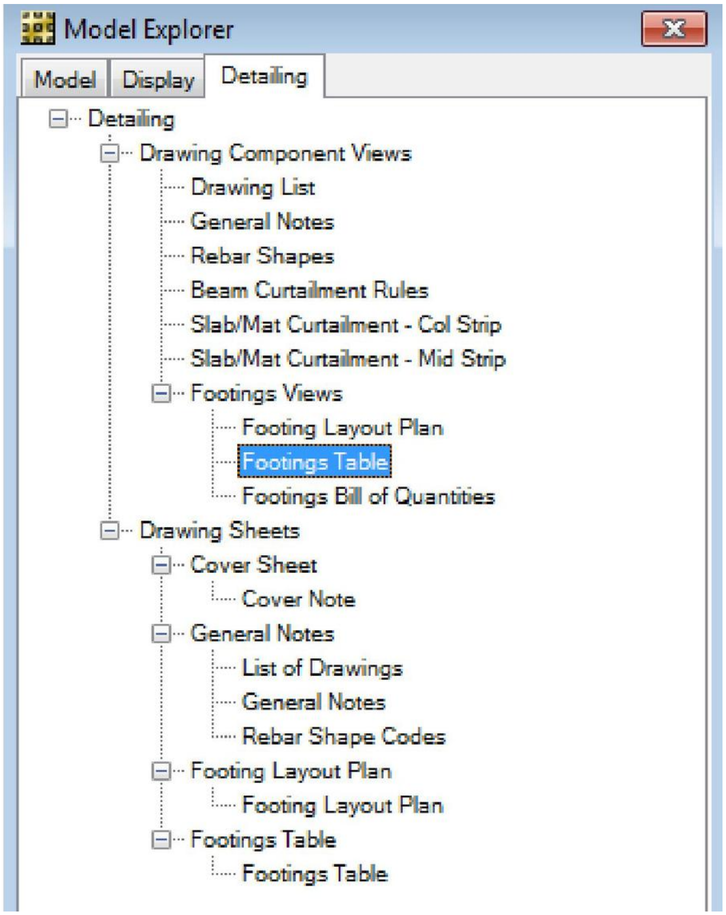


Figure 73

By clicking on any of the options from the detailing tab, a desired detailing can be displayed. For instance, by double clicking on the '**Footing Table**', the following detail of footing can be shown.

FOOTINGS TABLE

SR. NO.	TYPE	NOS	LX	LY	T	REBARS-A	REBARS-B
1	F1	1	3.000 M	2.000 M	0.500 M	10-10	8-8

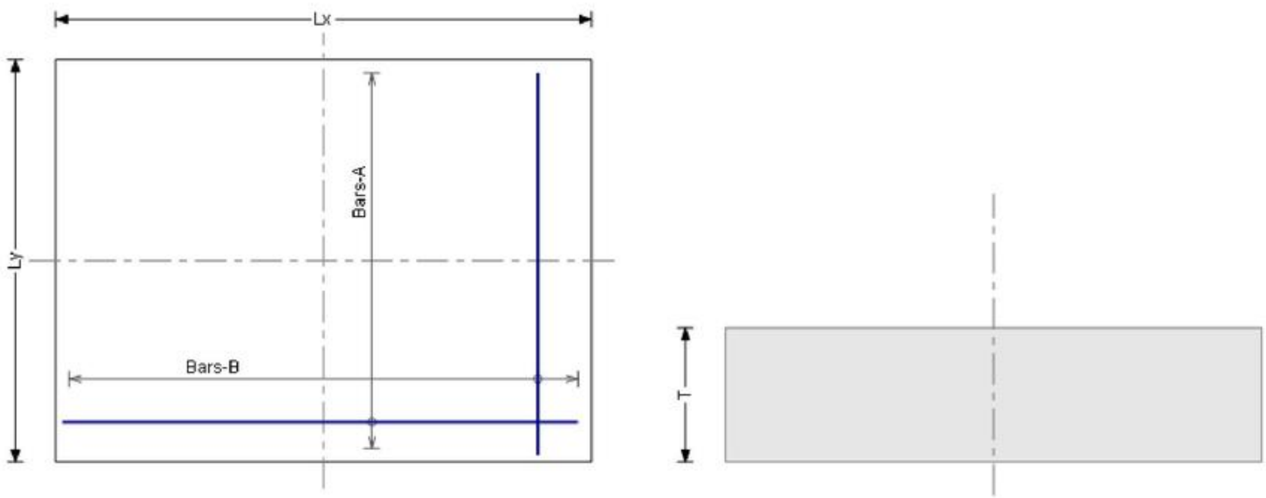


Figure 74

In this detail, the diameter of the reinforcement which is used is 10mm. If you want to change the diameter….

Apart from this detail, other details can also be shown.

STEP 11: Reporting

The last step of foundation design is reporting. Before creating the report, the report preferences should be set up. To do this, go to the 'File' menu and click on 'Report Set-up…' and the following window pops up.

In this '**Report Setup Data**' window, the user preferences regarding the reporting such as the report output type, the report page orientation and the report items can be set along with the load patterns and load combinations. Once the preference is set, the report can be created by clicking on '**Create Report**' command in the '**File**' menu. The '**Advanced Report Writer**' command in the same menu can be used to set some advanced reporting formats.

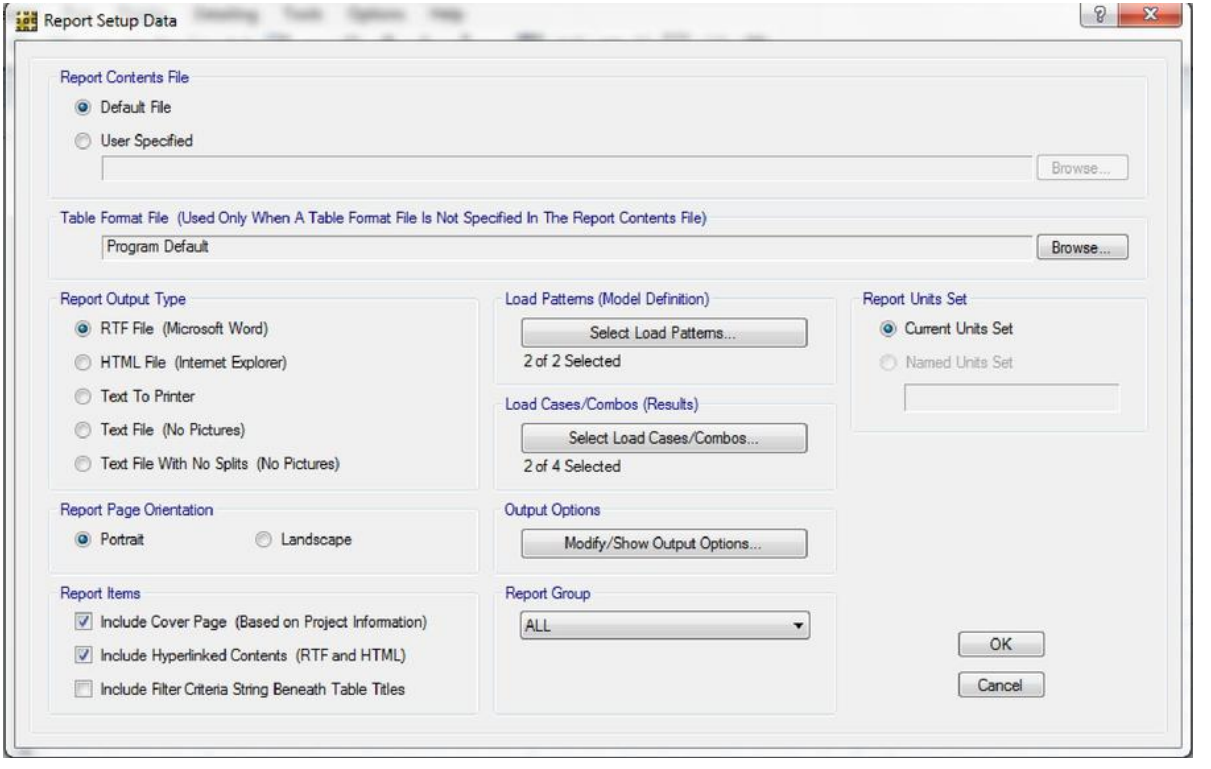


Figure 75

This concludes the tutorial for the design of single footing using the grids.

2.3. From AutoCAD

STEP 1:

Design a circular isolated footing with the following parameters:

- *Radius of the footing*: 1.5m
- *Radius of the column*: 0.3m
- *Allowable bearing capacity of the soil* : 150kN/m^2
- *Maximum allowable settlement of the foundation*: 10mm
- *Concentrated load (DL)* : 500kN
- *Concentrated load (LL)*: 250kN
- *Grade of concrete*: C-20 (20MPa 28-day characteristics cube strength)
- *Grade of reinforcement bar (rebar)*: S-460 (460MPa characteristics yield strength)
- *Overall thickness of the foundation*: 600mm
- *Concrete cover* : 50mm

Creating the Model

To create a footing model from an AutoCAD file, first draw the plan view of the foundation on AutoCAD. In this case, it will be two concentric circles one with a radius of 0.3m (foundation column) and another with a radius of 1.2m (footing). When you draw the plan view on AutoCAD, make sure that you are drawing it on a new layer. If you want the origin of the SAFE mode to coincide with the center of the circular foundation, make the center of the circle coincide with the global origin of the AutoCAD file when you draw it. Then save the AutoCAD file in DXF format. To save an AutoCAD file in DXF format, go to '**File**' menu in the AutoCAD file and click on '**Save As'**. At the bottom of the upcoming window, just below the text field where you will enter the file name, you will see '**Files of type**' with a drop-down menu list. From the drop down menu list, select the option which has '**(.dxf)**' at the end. Then enter the file name and save it at any location in your computer where you can easily remember close it. This saves the file in DXF format.

Once you saved the AutoCAD file in DXF format, open the SAFE software and when you use the command '**File**'>'**Import**'>'**.DXF/.DWG Architectural Plan**' or when you simultaneously click on '**Ctrl**'+'**Shift**'+'**I**' keys, and you will be prompted to open a file. Open the DXF file from the location where you saved it and the following window will pop up.

In this window, just change the '**CAD Drawing Units**' to '**m**' since in the AutoCAD file is the model is drawn in meter units. Then press '**OK**'.

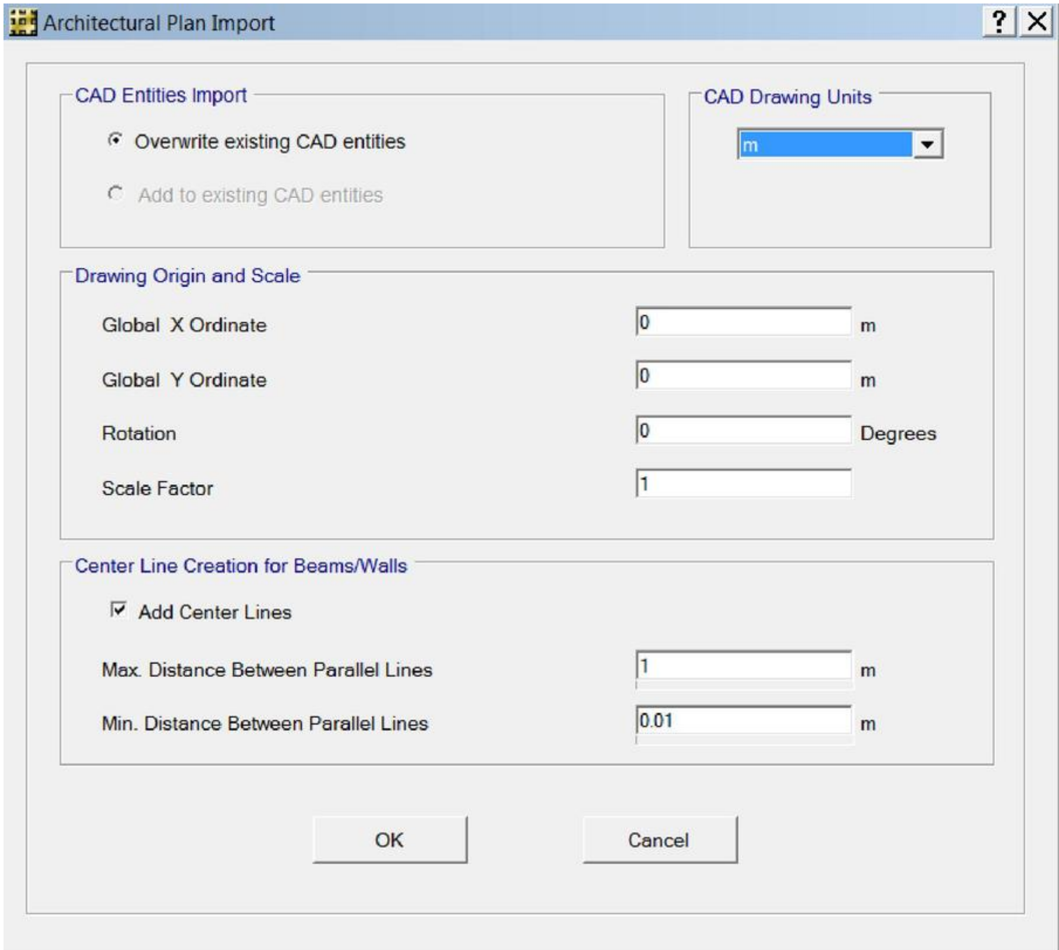


Figure 76

Then, the following model appears in the plan view.

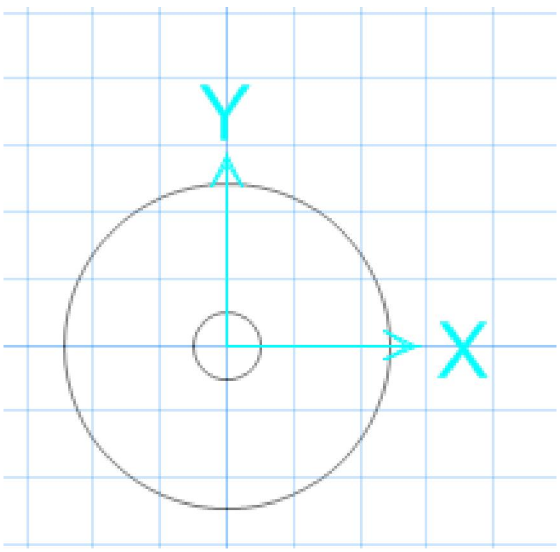


Figure 77

After the model is created, the preferences regarding the units, the design codes and concrete cover should be entered. To change the units, click on '**Units…**' button which is found in the right bottom corner of the main window and the following widow will pop up. In this window, click on '**Metric Defaults**' and press '**OK**' as we will be using metric units. If you want to use U.S. units click on '**U.S. Defaults**' and if you want to use a particular metric or U.S. unit consistently click on '**Consistent Units…**' and select that particular unit which you want to use consistently.

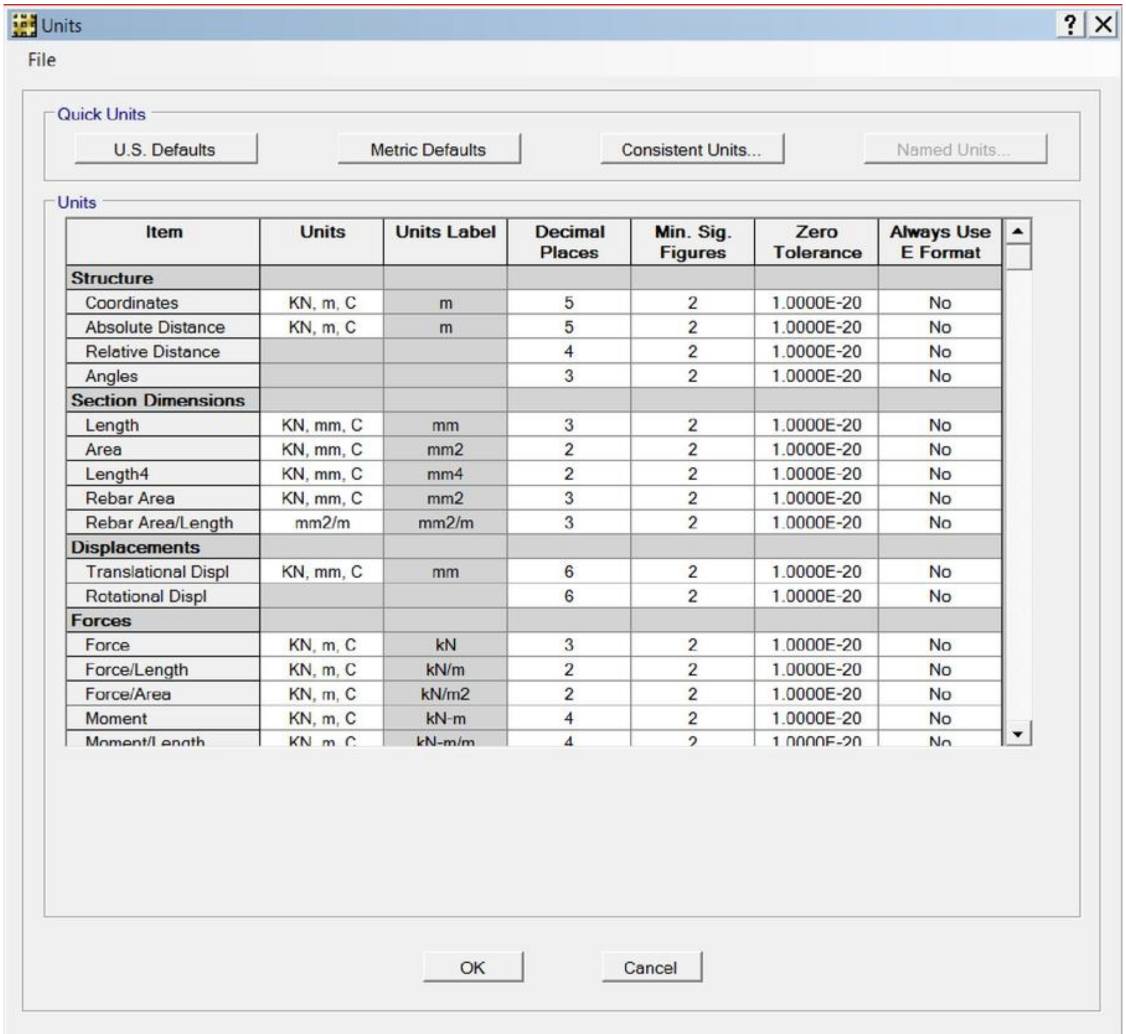


Figure 80

To select the design code and other design preferences, go to '**Design**' menu and click on '**Design Preferences…**' and the following window will pop up. In the '**Code**' tab of this window, change the '**Design Code**' to '**BS 8110-1997**' or any other code which you want to design the circular foundation with. In the '**Min. Cover Slabs**' tab, for '**Non-Prestressed Reinforcement**' , both the '**Clear Cover**

Top' and '**Clear Cover Bottom**' should be set to 50mm as the concrete cover in this design problem stated to be 50mm. The '**Preferred Bar Size**' can be set to any reasonable value. Here, the '**Preferred Bar Size**' is set to #14 which is the bar number which will be used as the main reinforcement in the foundation. Leave the rest as they are and press '**OK**'.

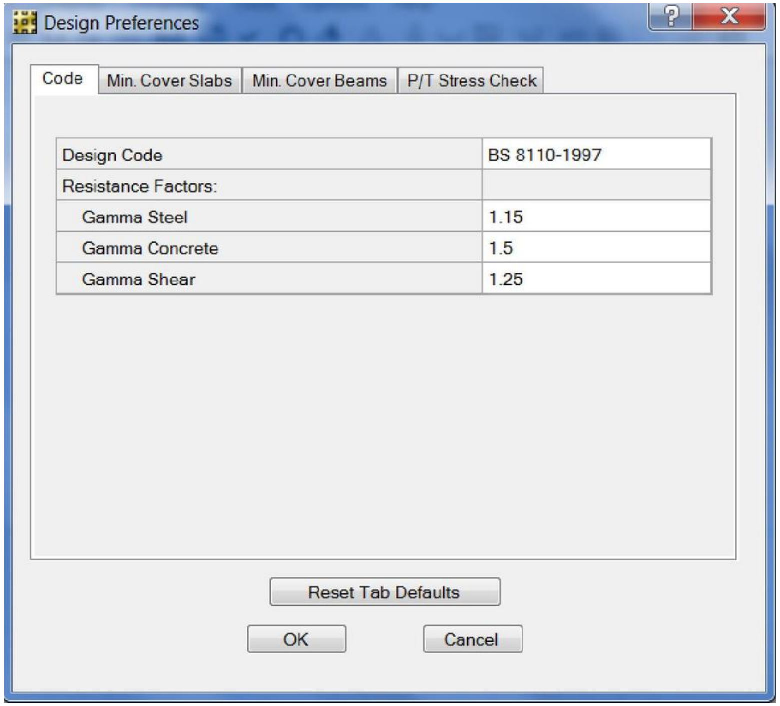


Figure 81.

STEP 2: Defining Material Properties

The materials which are involved in the design of the footing should be defined before the analysis. These materials are the concrete, the reinforcement bar (rebar) and the soil support. Thus, these properties will be defined here.

The material definition can be carried out in two ways.

The first one is through the '**Define**' menu in the menu bar. The other one through the model explorer on the left hand side of the home window. To define material properties through the former method, click on '**Define**' menu and again click on '**Materials…**' resulting in the following window depending on prior material definitions.

The list in the '**Materials**' box may not be exactly as it appears in your widow. However, that doesn't bring any change in the outcome of the design process as you can customize this list any time.

The '**Add New Material Quick…**' button allows you to define materials quickly from a list of pre-defined materials. The '**Add New Material**' button allows you to define materials by changing their properties. The '**Add Copy of Material**' button allows you to define a material with same property as an already defined material. The '**Modify/Show Material**' button displays the property of an already defined material with the possibility of modification. The '**Delete Material**' button, when active, deletes a defined material property.

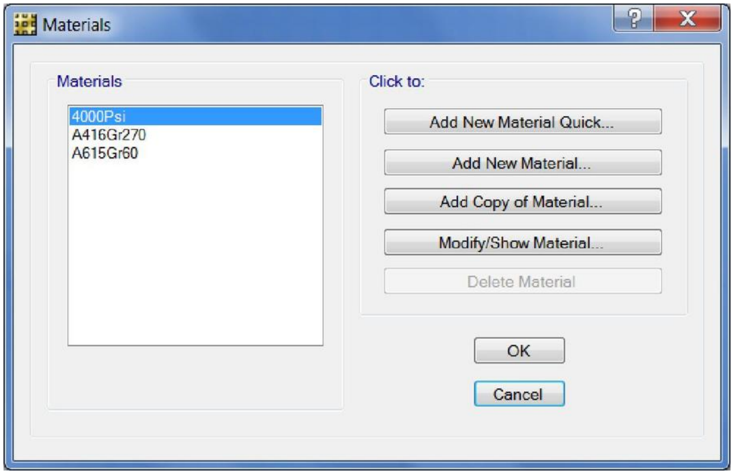


Figure 82

For this particular problem, we will add new concrete and a rebar materials using the '**Add New Material…**' button. Thus click on this button and the following window will pop up.

This is the window where concrete properties will be modified. Put any name for the material in the text field in front of '**Material Name**'. But, it is important to make sure that the concrete with the defined material name is assigned for the footing. For this particular problem, let us use the name of the concrete as '**C20**'. Since we are defining a concrete, the '**Material Type**' should be set to '**Concrete**'.

The unit weight of reinforced concrete may vary depending of the design code of your country. Thus, enter the unit weight of reinforced concrete stipulated in your country code in the text field in front of '**Weight per Unit Volume**'. For this particular example, we will use 25kN/m^3.

For C-20 concrete, the modulus of elasticity according to BS 8110-1197 is around 27GPa. Thus, enter this value in the text field in front of '**Modulus of Elasticity, E**'. If you selected another design code in step 1 while creating the model, you should refer to actual value of this parameter from the code and enter it accordingly. Be aware of the units though.

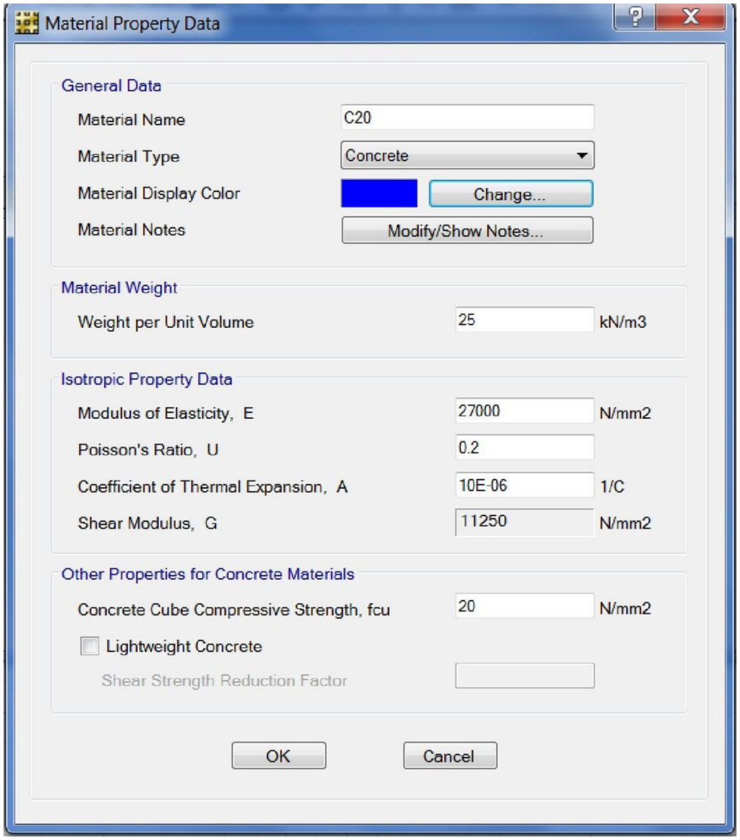


Figure 83

The values of Poisson's ratio and coefficient of thermal expansion may also be defined in the design code and should be entered accordingly. For this particular problem, a value of 0.2 for '**Poisson's Ratio, U**' and a value of $10x10^{-6}/^{o}C$ for '**Coefficient of Thermal Expansion, A**' will be entered. The '**Shear Modulus, G**' will be automatically calculated in an un-editable text field.

The grade of concrete for this particular problem is C-20 which is a concrete with 28 day characteristics cube compressive strength of 20MPa. The concrete designation may be different for different country codes but the concept is the same. Therefore, enter 20 in the text field in front of '**Concrete Cube Compressive Strength, fcu**'.

If a lightweight concrete is used, check on '**Lightweight Concrete**' and enter the corresponding '**Shear Strength Reduction Factor**' in the space provided.

When you press on '**OK**', a concrete material with the above properties will be added to the list of materials. This material will be assigned for the footing before the analysis.

After adding a new concrete property, the program returns to the window shown in Fig. 82 To define a rebar property, we will follow the same procedure as we followed while defining the concrete property. Since a new rebar property will be defined, click on the '**Add New Material…**' button. A '**Material Property Data**' window pops up and when you change the '**Material Type**' to '**Rebar**', the window appears to look like the following.

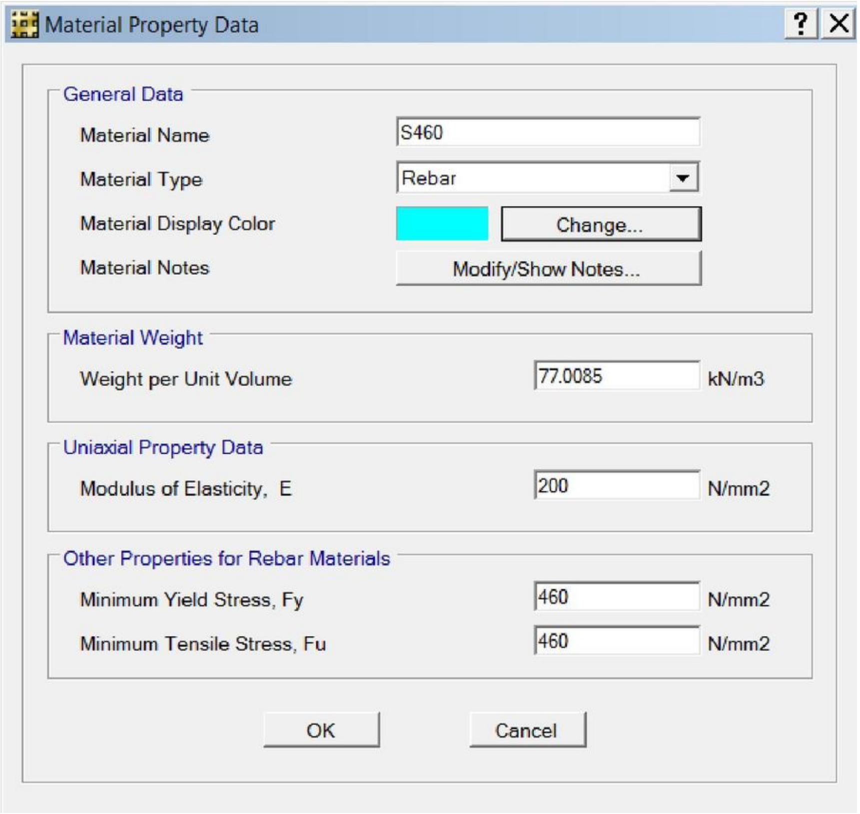


Figure 84

Change the '**Material Name**' to any name you want. Here, we name it '**S460**'. The material type should be '**Rebar**'. The weight per unit volume of steel is stipulated in the design code. For BS 8110-1197, the weight per unit volume is 77.0085kN/m^3. Thus, enter this value in the text field in front of '**Weight per Unit Volume**'. The modulus of elasticity for reinforcement bars according to the same design code is 200GPa. Thus enter this value in the text field in front of '**Modulus of Elasticity, E**' considering the unit.

In the '**Other Properties for Rebar Materials**' box, two quantities are mentioned: minimum yield stress and minimum tensile stress for the reinforcing material. The values of these parameters will be specified in the design code which you defined earlier. If the code assumes that the rebar material exhibits elastic perfectly plastic behavior, the values of these two quantities will be the same. The grade of steel to be used for this particular example is S-460. The yield stress for this type of reinforcement bar is 460MPa. Since the design code of my country assumes that rebars exhibit elastic perfectly plastic behavior, the minimum tensile stress will also be 460MPa. Thus enter 460 in both text fields in front of the '**Minimum yield stress, Fy**' and '**Minimum Tensile Stress, Fu**'. Then press '**OK**' in both '**Material Property Data**' and '**Materials**' windows concluding the material definition step.

The other property which should be defined is the soil support. To define the soil properties, go to the '**Define**' menu and click on '**Soil Subgrade Properties**' menu item and the following window appears.

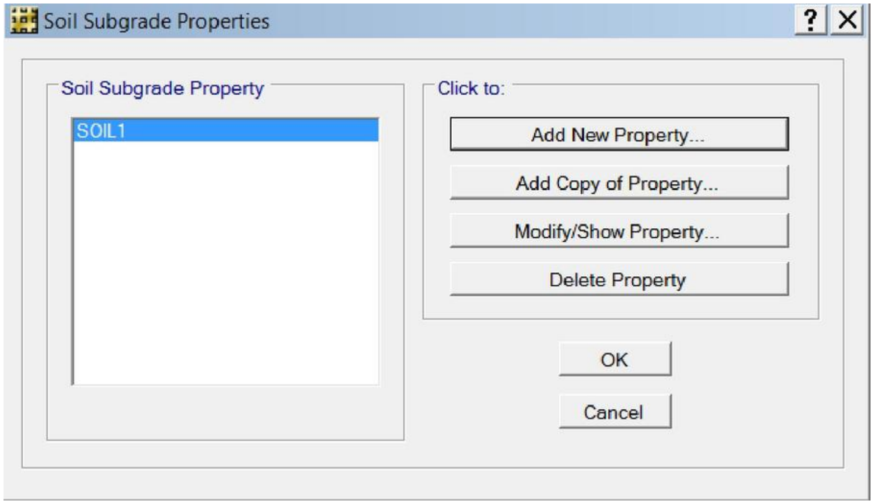


Figure 85

By using the buttons in this window, a new soil property or copy of soil property can be added. An existing soil property can also be modified or deleted. For this problem, let us add a new soil property by using the '**Add New Property…**' button. Thus, click on this button and the following window pops up.

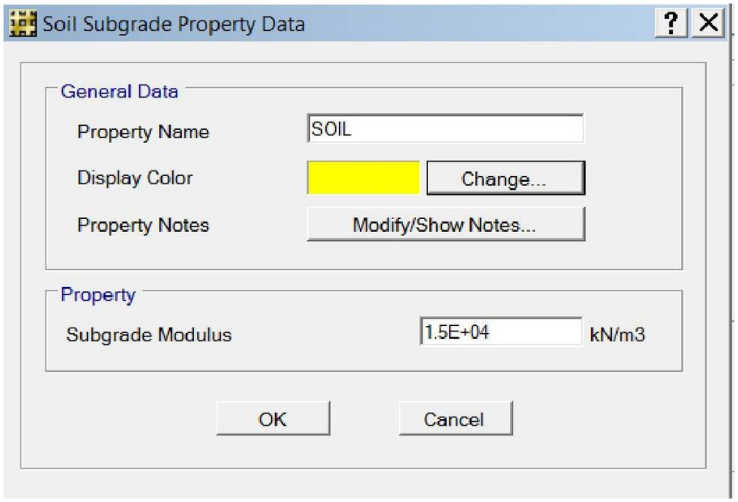


Figure 86

In this window, set the property name to '**SOIL**' and change the subgrade modulus to 15,000kN/m^3. Because, the subgrade modulus which can be assume by the ratio of the bearing capacity by the allowable settlement for this particular example is so. Then press '**OK**' in both the '**Soil Subgrade Property Data**' and '**Soil Subgrade Properties**' windows.

STEP 3: Defining Footing and Column Properties

After defining the material properties, the footing and column properties can be defined. This definition can take place in two ways: from the menu bar and from the model explorer. In SAFE software, footings are modelled as 'footings' and foundation columns are modelled as 'stiff'.

To define footing and column properties from the menu bar, go to '**Define**' menu and click on '**Slab Properties…**'. The following window will pop up.

The '**Add New Property…**' button prompts the user to enter new properties for the footing and foundation column while the '**Add Copy of Property…**' copies the property of an existing slap. The '**Modify/Show Property…**' allows the user to show the property of an existing component with the possibility of modification. When the '**Delete Property**' button is active, it allows the user to delete an existing slab property.

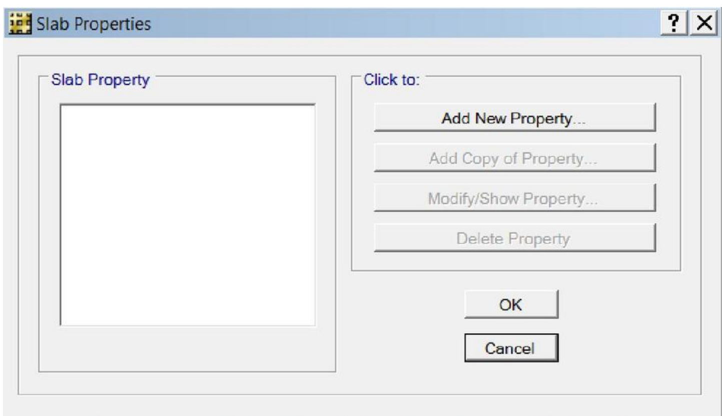


Figure 87

In this case, we use the '**Add New Property…**' button to add new slab properties for the footing and foundation column. Click the button and the following window will pop up.

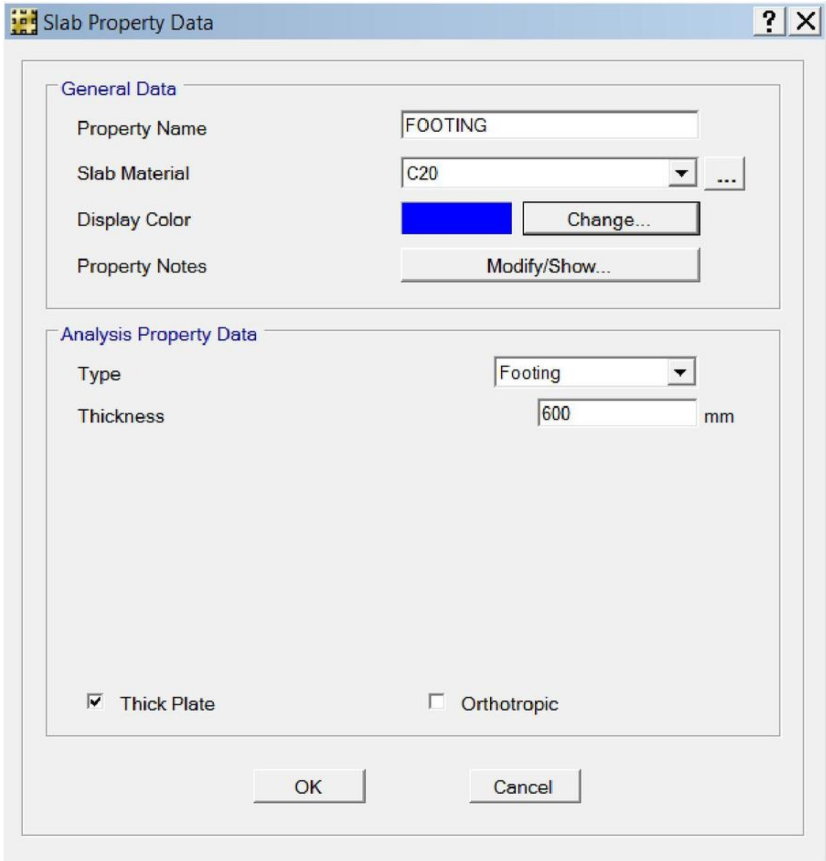


Figure 88

The '**Property Name**' can be assigned with any name; but, in this case, the property name which will be used is '**FOOTING**'. The '**Slab Material**' should be set to the concrete grade which is defined in step 2. Since the name of the concrete material defined in step 2 for this particular problem is '**C20**', a material with this name should be selected from the list.

In the '**Analysis Property Data**' box, '**Type**' should be set to '**Footing**'. The '**Thickness**' value should be set to the thickness of the footing defined in the example. Since the thickness of the footing is 600mm, this value is entered in the text field corresponding to '**Thickness**'. As footings are modelled as thick plates, check the '**Thick Plate**' option. The '**Orthotropic**' check box is selected when a footing with irregular dimension is to be used.

When you press '**OK**', the '**Slab Property Data**' window will be exited and the '**Slab Properties**' window gets activated. Now, the property of the foundation column will be added. To do this, again click on the '**Add New Property…**' button. The following window appears after the click.

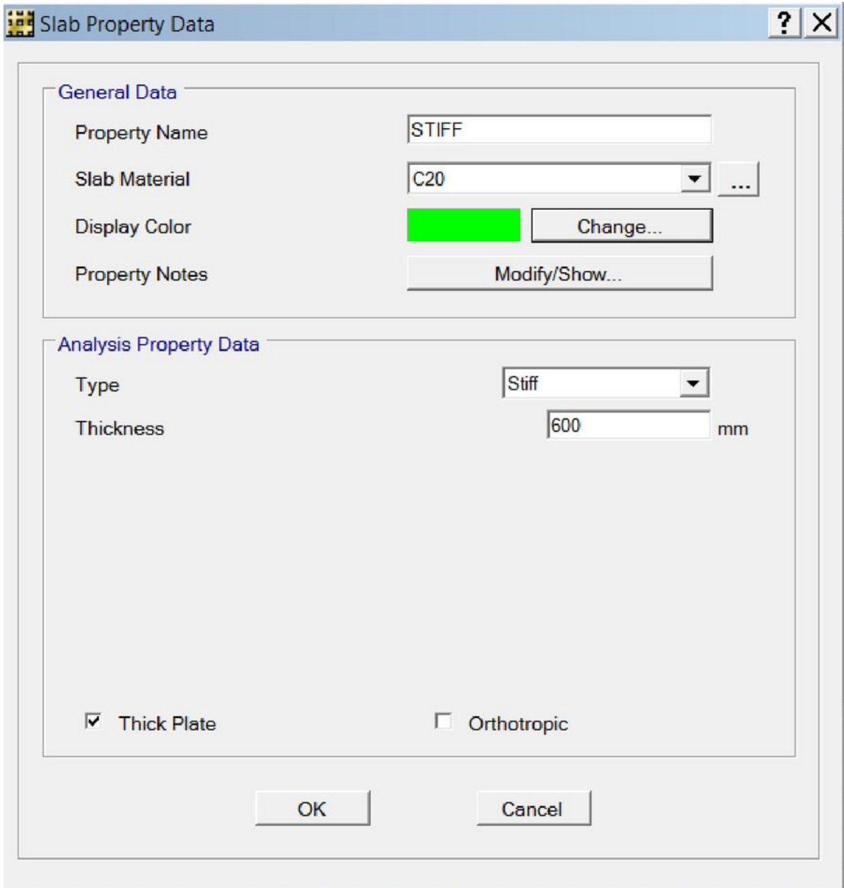


Figure 89

The '**Property Name**' can be any name but we use '**STIFF**'. The '**Slab Material**' should be set to the concrete grade which is defined in step 2. Since the name of the concrete material defined in step 2 for this particular problem is '**C20**', a material with this name should be selected from the list.

In the '**Analysis Property Data**' box, '**Type**' should be set to '**Stiff**' as we are defining the property of column. The '**Thickness**' value should be the footing thickness which is equal to 600mm. As foundation columns are modelled as thick plates, check the '**Thick Plate**' option. The '**Orthotropic**' check box is selected when a column with irregular dimension is to be used.

When you press '**OK**', the '**Slab Property Data**' window will be exited and the '**Slab Properties**' window gets activated. Again press '**OK**' and exit the window for defining the footing and foundation column.

STEP 4: Defining Load Patterns, Load Cases and Load Combinations

The loads on the foundation should be defined accordingly before the analysis. First, the load pattern should be defined. This can be done from the '**Define**' menu or from the '**Model Explorer**'. This time, we will do it from the model explorer. In the model explorer, expand '**Load Definitions**' and you will see '**Load Patterns**'. When you expand '**Load Patterns**', you will see '**DEAD**' and '**LIVE**'. At the end, the model explorer appears to look like:

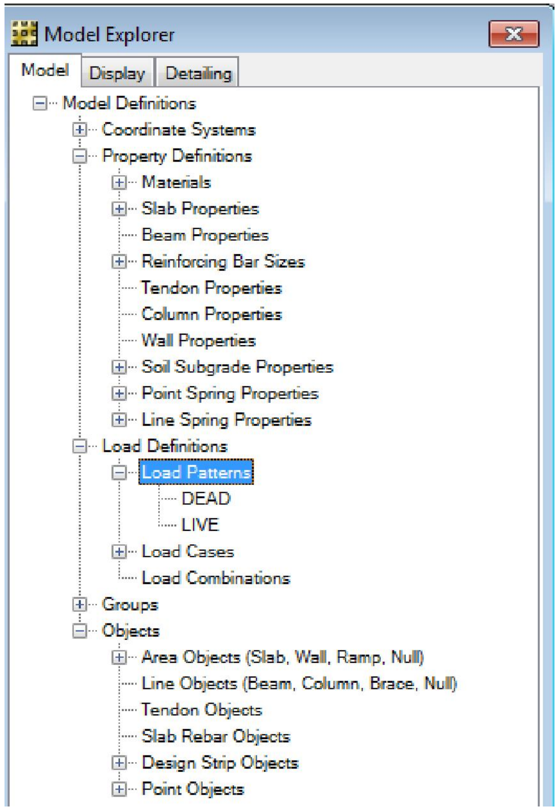


Figure 90

Now, right click on '**Load Patterns**' and click on '**New Load Pattern**' and the following window pops up.

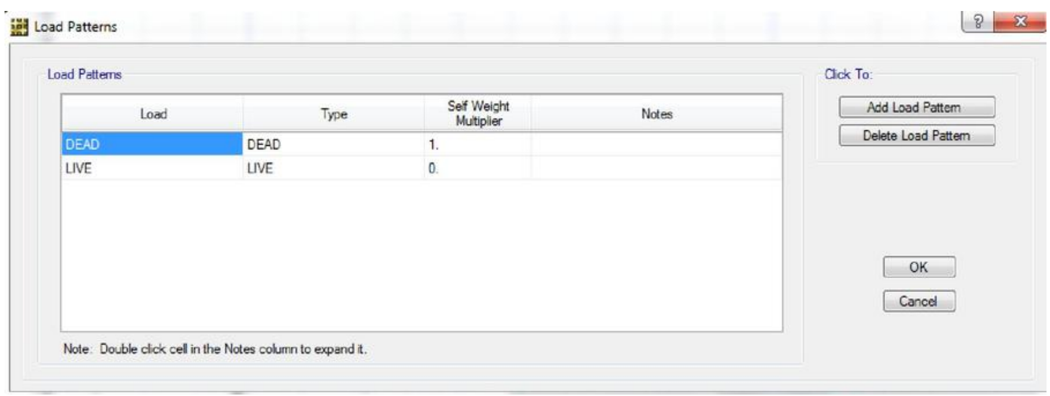


Figure 91

In the '**Load Patterns**' window, two load patterns are already defined: '**DEAD**' and '**LIVE**'. The '**Type**' of the '**Load**' should also be changed accordingly. There are many options for type of loading. The '**Type**' for dead loads should be set to '**DEAD**' and for live loads '**LIVE**'. If there are other types of load patterns on the foundation such as earth quake load, you can add the load pattern with the '**Add Load Pattern**' button. You can also delete any undesirable load pattern using the '**Delete Load Pattern**' button. Since in this example, we have only dead and live loads, we will leave the existing load patterns as they are. The '**Self Weight Multiplier**' value should also be changed accordingly. This value imparts the option whether to consider or ignore the self-weight of the foundation in addition to external loads. If the self-weight of the foundation is already included as an external dead load or if you want to exclude the effect of self-weight from the analysis, the value under '**Self Weight Multiplier**' should be set to zero. In this example, we will consider the self-weight as an additional load to the external dead load. Thus, the value under '**Self Weight Multiplier**' for the '**DEAD**' load is one. For the '**LIVE**' load, it will be zero. Press '**OK**' and the window will be exited.

After this, the load cases will be defined. Load cases are used to dictate the way the loads are applied (statically or dynamically) or the way the structure responds (linearly or non-linearly) for the defined load patterns. To define a load case, go to '**Define**' menu and click on '**Load Cases…**'. The following window will pop up after the click.

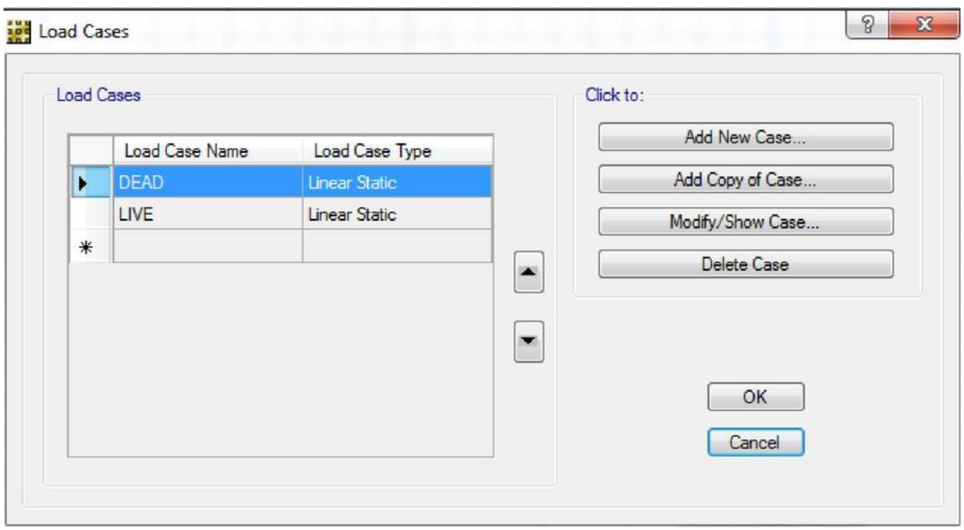


Figure 92

The load patterns which were defined earlier will automatically appear in the list of '**Load Case Name**' of the '**Load Cases**' window. The '**Load Case Type**' column shows the way in which each load pattern will be applied during analysis. If you want to modify this, highlight the load pattern for which you are going to change the load case type and click on the '**Modify/Show Case…**' button. If you click this button, the following window will appear. In this window, you can change the way the load is applied from the '**Load Case Type**' box. The way the structure responds can also be selected from the '**Analysis Type**' box. This problem '**Static**' is for the '**Load Case Type**' and '**Linear**' is selected for the '**Analysis Type**' since the load is static and the foundation responds linearly. The scale factor for the dead load in the '**Loads Applied**' box will be left as one. Press '**OK**' and exit the window.

The load case type for the live load should also be 'Linear Static'. Otherwise, it should be changed by clicking the '**Modify/Show Case…**' button to linear static case. If both the load case types are as desired click '**OK**' and exit the '**Load Cases**' window.

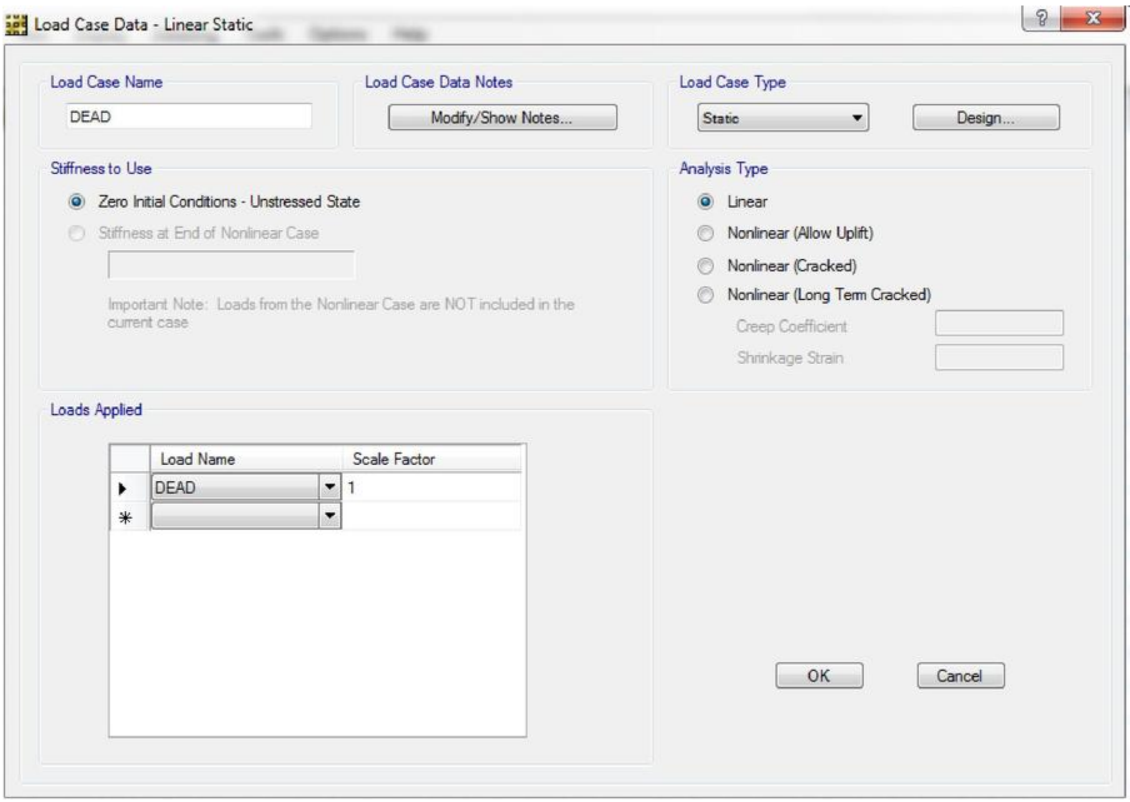


Figure 93

Definition of the load combinations will be the next step. Two load combinations will be considered in this example for ultimate limit state and serviceability limit state. To define load combinations, go to '**Define**' menu and click on '**Load Combinations…**' and the following window pops up.

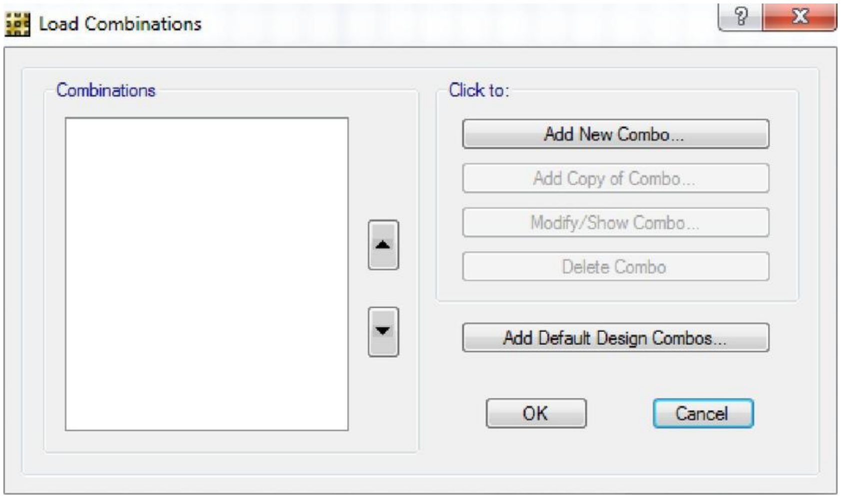


Figure 94

New load combinations will be added through the '**Add New Combo…**' button. Here, two load combinations will be added; one for the ultimate limit state the other for the serviceability limit state. When the '**Add New Combo…**' button is clicked, the following window appears.

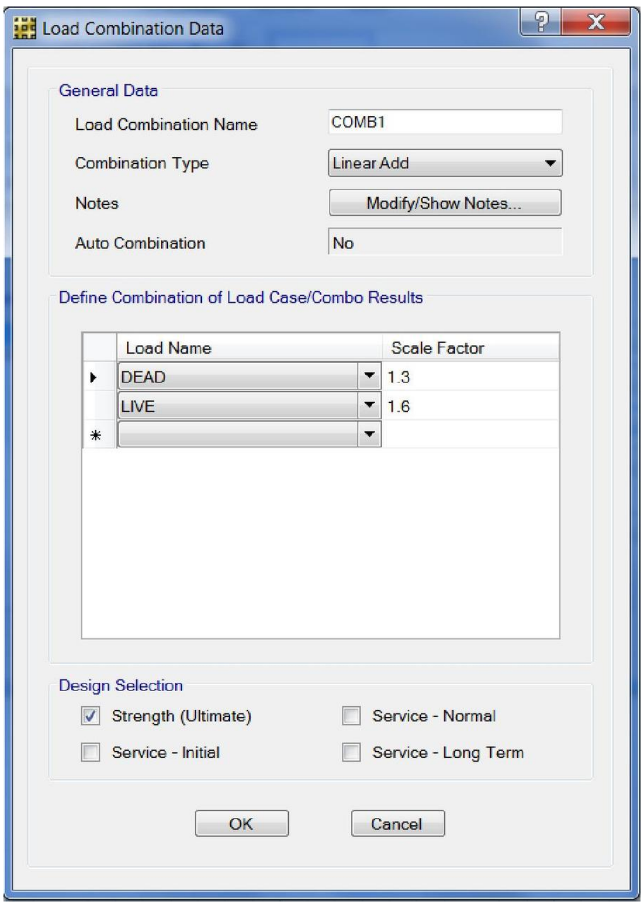


Figure 95

In the '**Load Combination Data**' window, any name can be given for the load combination. The '**Combination Type**' should be set to '**Linear Add**' as the component loads (dead and live) will be added linearly. However, there are also other options from the drop down menu in front of '**Combination Type**'. In the window shown in Fig. 95, the two loads (dead and live) should be activated below the '**Load Name**' column of the '**Define Combination of Load Case/Combo Results**' box. The values in the '**Scale Factor**' column correspond to the partial safety factors for load for the failure mode under consideration. These partial safety factors are specified in the design code of your country. In the design code of my country, the partial safety factor for dead loads for ultimate limit sate case is 1.3 and for serviceability limit state is 1 for the cases where there are only dead and live loads. For such load patterns, the partial safety factor for live loads for ultimate limit sate case is 1.6 and for serviceability limit state is 1. Thus, enter a value of 1.3 in front of '**DEAD**' and 1.6 in front of '**LIVE**' in '**Scale Factor**' column. The failure condition which is being under consideration can be defined by selecting and deselecting the check boxes in the '**Design Selection**' box. Since the above scale factors are for the ultimate limit state, check on '**Strength (Ultimate)**'. The '**Load Combination Data**' window for the serviceability limit state looks like the following window.

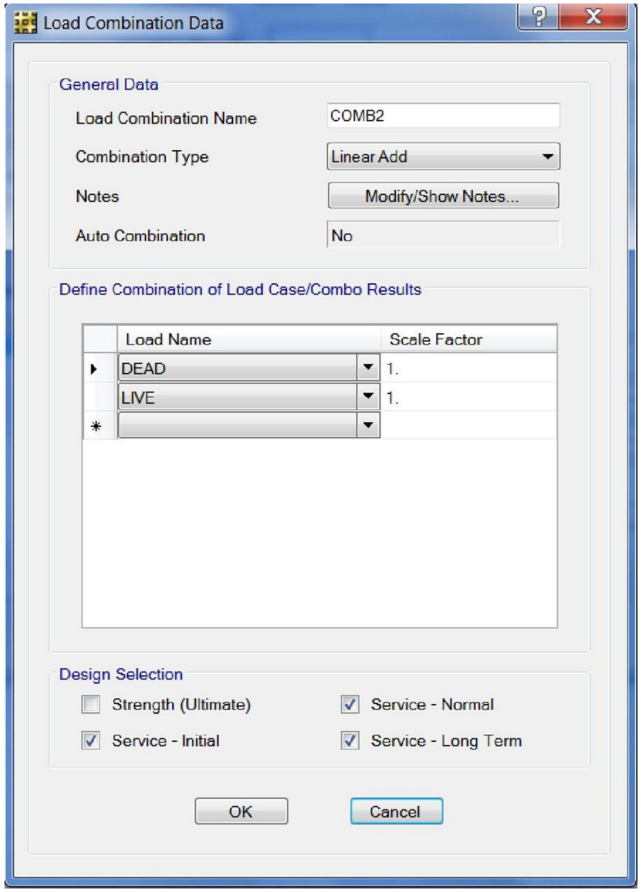


Figure 96

STEP 6: Drawing the Footing Components and Design Strips

The footing, the foundation column and a point where the loads will be applied on the foundation column should now be drawn on the grid.

i. Drawing the footing

Since the footing will be drawn around points and since the footing is circular, go to the '**Draw**' menu and click on '**Quick Draw Areas Around Points**' or simply click on the equivalent icon  on the left hand side tool bar. The following window will appear on the screen after the click.

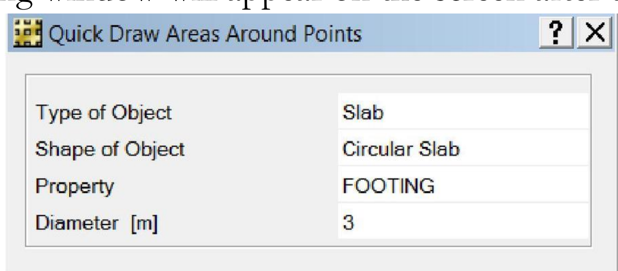


Figure 97

Since the footing has circular shape, change the '**Shape of Object**' to '**Circular Slab**'. Make sure that the '**Property**' is set to '**FOOTING**'. The value of the '**Diameter [m]**' should be set to '**3**' as the footing diameter is 3m.Then, with great care take the cursor to exactly the center of the footing, and just make one left click. This draws the footing.

ii. *Drawing the foundation column*

While the window in fig 97 is active, change the '**Property**' to '**STIFF**' and the '**Diameter [m]**' to 0.6. Then, with great care take the cursor to exactly the center of the column and just make one lift click. This draws the foundation column.

iii. *Drawing the point on the foundation column where the load will be applied*

Go to '**Draw**' menu and click on '**Draw Points**', then click on the mid-point of the footing and the point will be created. If the cursor could not snap to the midpoint, you can adjust the '**Snap Options**' from the '**Draw**' menu.

After this, the design strips will be drawn. Design strips determine the way in which different quantities related to the reinforcement calculation are calculated. Forces are integrated across the design strips. Thus, the larger the width of coverage of the design strips within the given structure, the higher will be the calculated values of the bending moments and shear forces. Thus, an optimum width of strip is required compromising the safety and economical requirements. The width of the design strip will be specified in the design code. According to the code of my country, the width of design strips for isolated foundations is 1m. Thus, a one meter design strip will be drawn in both X and Y directions on the foundation. These design strips in X and Y direction are usually defined in SAFE software as layer A and layer B.

To draw the design strip, go to the '**Draw**' menu and click on '**Design Strips**' or simply click on the equivalent icon  from the left hand sided tool bar and the following window pops up.

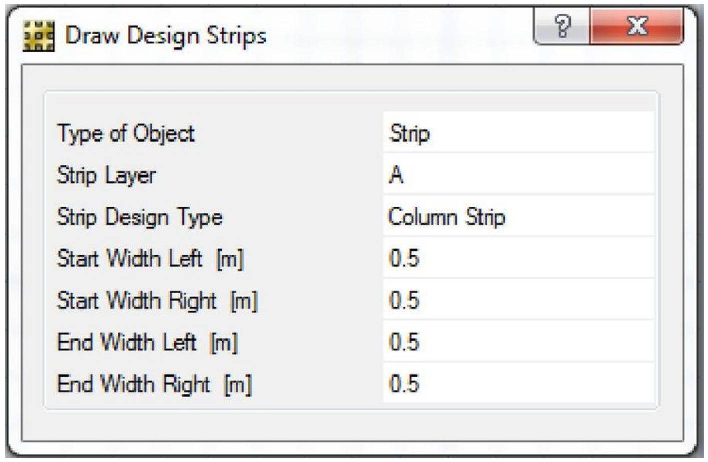


Figure 98

In this window, the '**Strip Layer**' should be selected to be either '**A**' or '**B**'. But if '**A**' is for design strip in X direction '**B**' should be for Y direction and vice versa. Since we are drawing a strip around the column to consider maximum moment and shear forces, the '**Strip Design Type**' should be set to '**Column Strip**'. The width of the strip will be 1m in both directions and since the column will be centrally located, all the widths will be set to 0.5m. To draw the design strip in the X direction, without closing the window, left click at the extreme left side of the footing on the plan view and again left click at the extreme right side of the footing and right click. This creates a design strip in X direction. In doing so, if you can't snap to the extreme points on the footing, you can modify the snap options by clicking on the '**Snap Options…**' command from the '**Draw**' menu and adjusting the options which you want to snap to. The design strip in Y direction can also be drawn in a similar procedure after changing the '**Strip Layer**' to '**B**'.

You can display the design strips by setting the display options by clicking on '**Set Display Options…**' from the '**View**' menu or by simultaneously clicking on '**Ctrl**' and '**W**' keys or by just clicking on the set display options icon from the tool bar below the menu bar. This results in the following window:

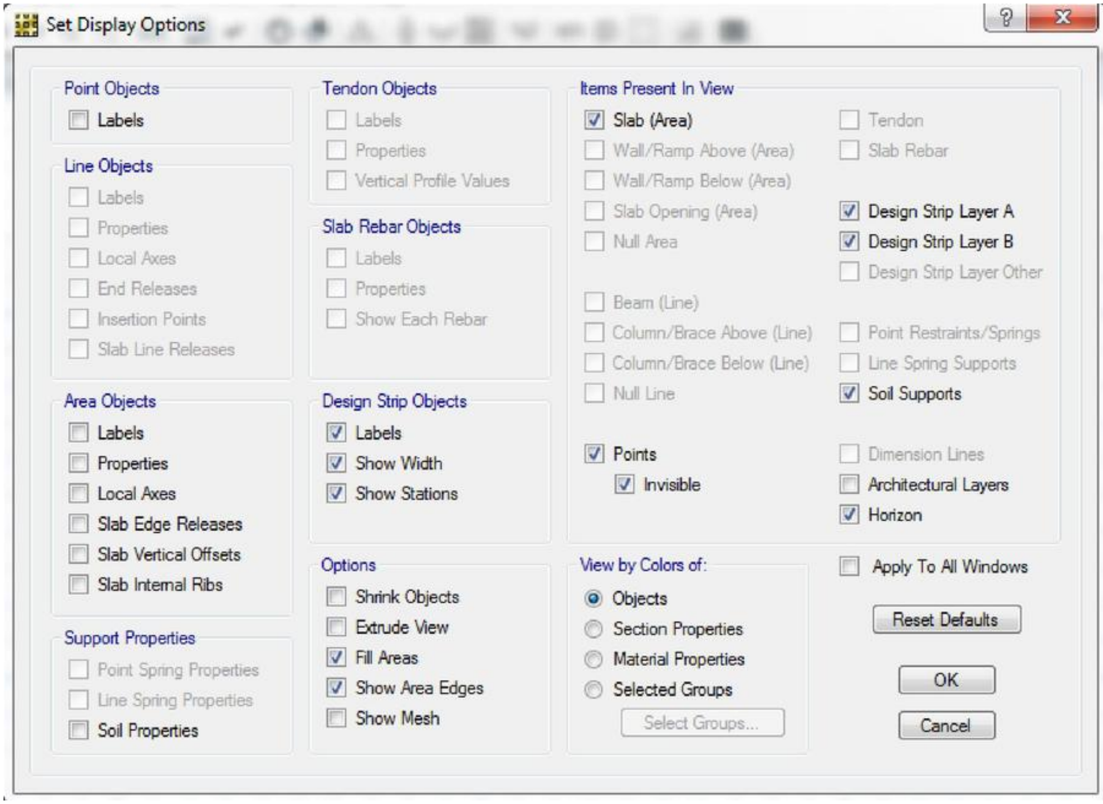


Figure 99

Then, check on ‘**Labels**’, ‘**Show Width**’ and ‘**Show Stations**’ in the ‘**Design Strip Objects**’ box and press ‘**OK**’ and after drawing dimension lines by using the command ‘**Draw**’>’**Draw Dimension Lines**’, the following window appears displaying the design strips in the two directions.

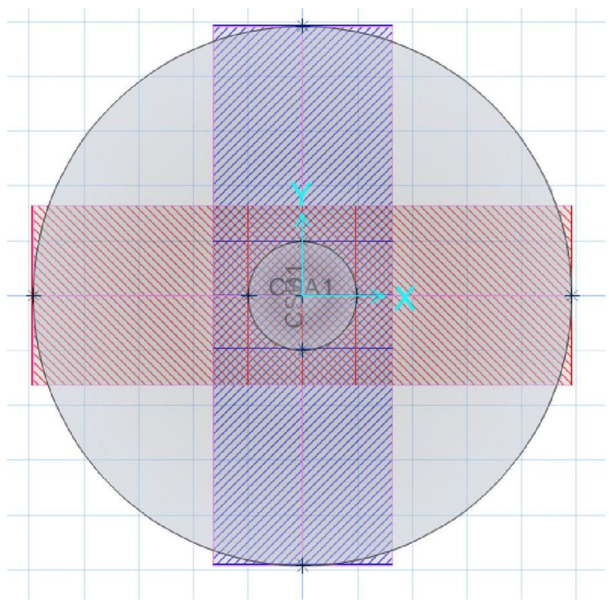


Figure 100

STEP 7: Assigning Slab Data, Support Data and Load Data

The slab data, support data, and load data which are defined in the previous steps should be assigned to the corresponding structural component. The way to do this is first to select the component and next to assign the slab property, support property or loads accordingly. The selection can be done by clicking on the component from the plan view or through the '**Select**' menu. The latter option assures that the component is selected exactly as selection by clicking may result in incorrect selection. Thus, all the selections here will be done from the '**Select**' menu. The assignments will be discussed as follows:

a. *Assigning support data to the footing*
Select the footing through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**'Slab Properties…**'. Then select '**FOOTING**' and press '**OK**'. Then assign the footing property through the following strings of commands '**Assign**'>'**Support Data**'>'**Soil Properties…**'. Then select '**SOIL**' and press '**OK**'.

b. *Assigning reinforcement data to the design strips*
Select each design strip through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**Design Strip Layers…**'. Then select '**A**' or '**B**' (one at a time) and press '**OK**'. When you right click on the selected strip layer, the '**Slab-Type Area Object Information**' window pops up. In the '**Design**' tab of this window, set the '**Rebar Material**' to '**S460**' and press '**OK**'. Do this for both strips.

c. *Assigning load on the foundation column*
To assign load on the foundation column, right click on the point at the center of the foundation column and a '**Point Object Information**' window pops up. In this window, click on the '**Loads**' tab.

The dead load and the live load can be assigned through the '**Assign Load…**' button. The procedure is: click on '**Assign Load…**' button, then select '**Force Loads**' then press '**OK**' then select either '**DEAD**' or '**LIVE**' depending on which loads you want to enter their values then enter their values accordingly (both the concentrated load and the bending moment) and in the right direction (axis), then select '**Add to Existing Loads**' and press '**OK**'. While doing this, the foundation column dimensions should be entered in the '**Size of Load for Punching Shear**' box of '**Point Loads**' window.

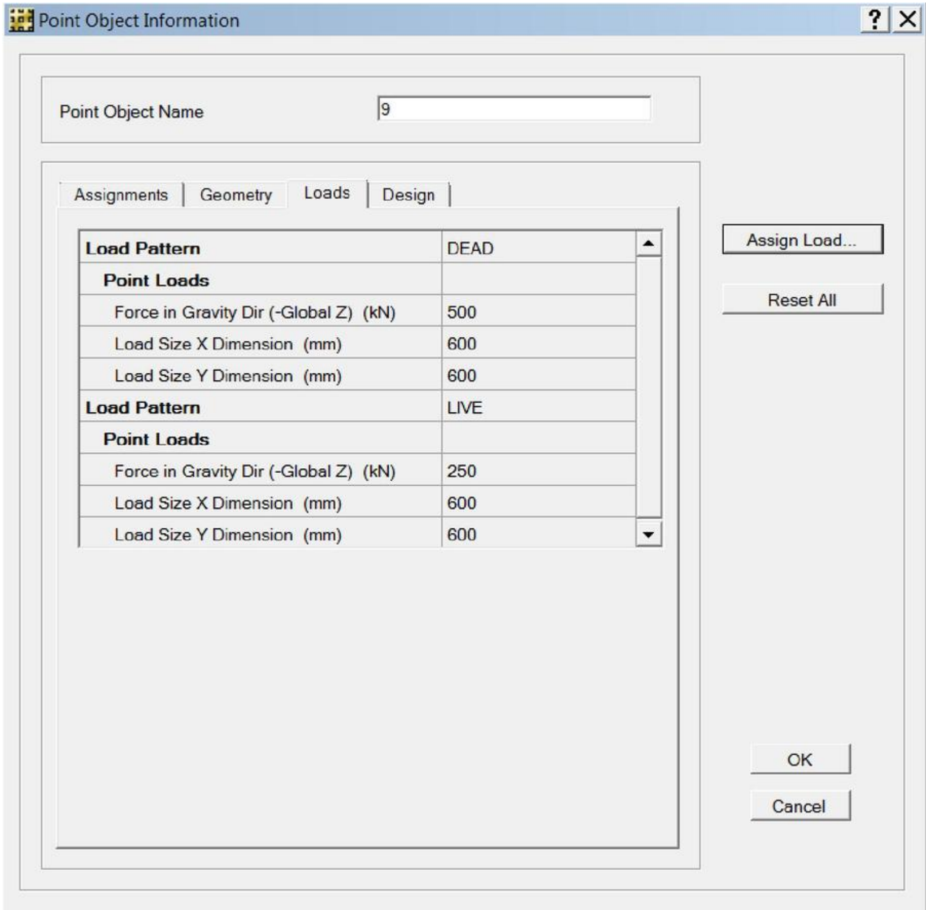


Figure 101

You can also delete any forces which are wrongly entered through ‘**Delete Existing Loads**’ radio button option. In this example, since the dead concentrated load is 500kN and since the live concentrated load is 250kN, these values are entered in the Gravity Direction for the concentrated load and along y-axis for the bending moment.

STEP 8: Running the Analysis

After this, the analysis can be run. But, make sure that the footing and the foundation column are assigned with the correct rebar material. To do this, right click anywhere in the plan view of the footing and the ‘**Slab-Type Area Object Information**’ window will pop-up.

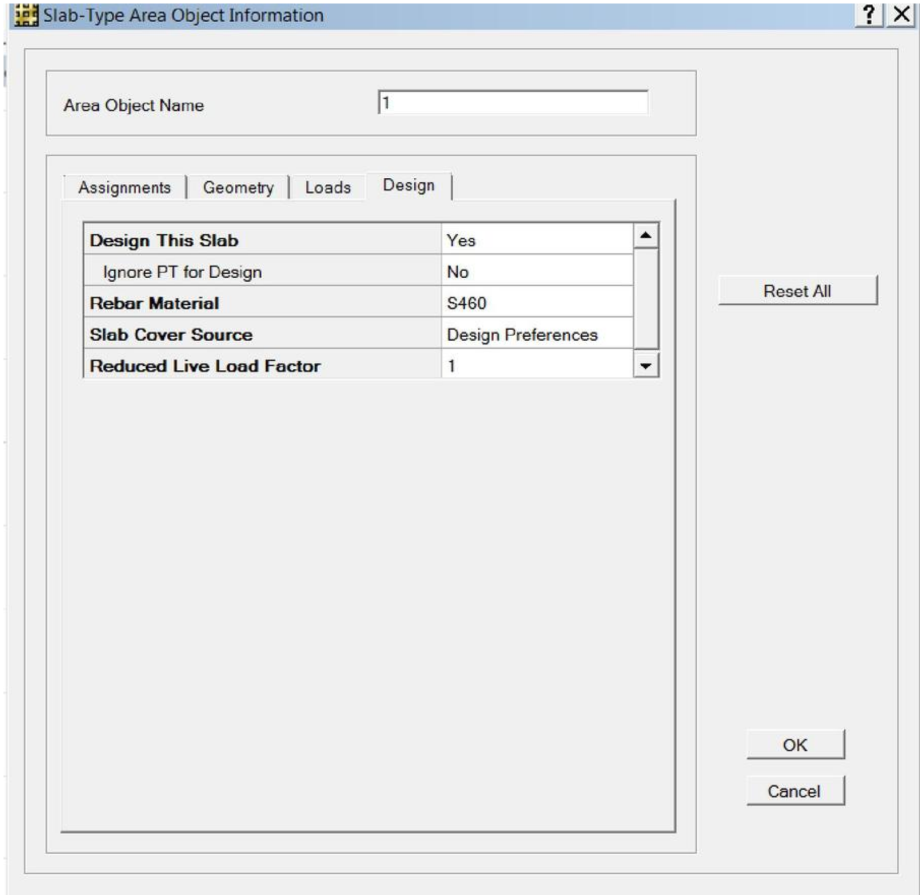


Figure 102

In the '**Design**' tab of this window, set the value of '**Rebar Material**' to '**S460**'. Do this for the column foundation, and both design strips as well. After this, go to the '**Run**' menu and click on the '**Run Analysis**' command.

STEP 9: Displaying the Output

Once the analysis is run, the output will be displayed. Particularly, the punching shear design is of great importance as the footing cannot be designed without the punching shear requirement being adequately satisfied. To do this, go to the '**Run**' menu and click on '**Run Analysis & Design**' command or simply click on the '**F5**' key. When you do this, you will be prompted to save the model, if you haven't already don this. When you save the model, the following window showing the displacement of the soil in a banded figure will be displayed.

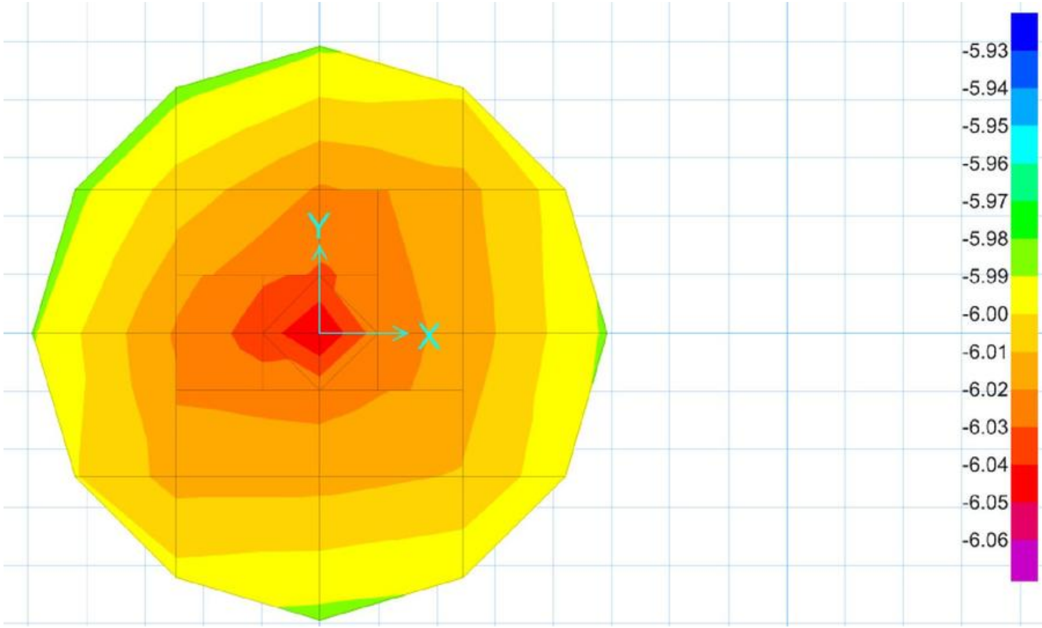


Figure 103

To display the punching shear ratio, go to the '**Display**' menu and click on '**Show Punching Shear Design**'. After this, the punching shear ratio will be displayed in the plan view around the foundation column as in the following figure.

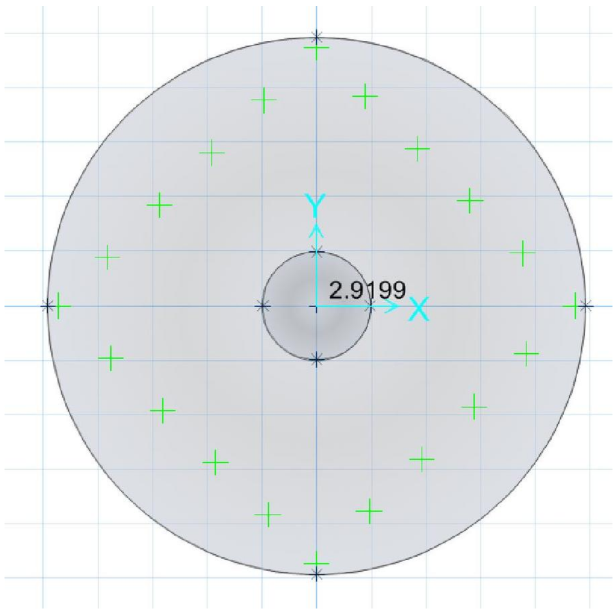


Figure 104

As can be noticed from the figure, the punching shear ratio is 2.9199. Generally, a punching shear ratio less than one indicates the concrete thickness is adequate to resist punching shear and a value greater than one indicates that the punching shear capacity is exceeded somewhere along the critical section. For economical design, it is recommended to keep the punching shear ratio between 0.95 and 1 as very small values of punching shear ratio means excess concrete thickness is used. However, if the punching shear ratio is greater than one, like in this example, the thickness of the concrete should be increased or the grade of the concrete should be increased and the foundation should be re-designed. A detailed quantitative description of the foundation design can also be obtained by right clicking on the plan view shown in Fig 104 as shown below. Several trial may be made by zooming in and out to get the quantitative description.

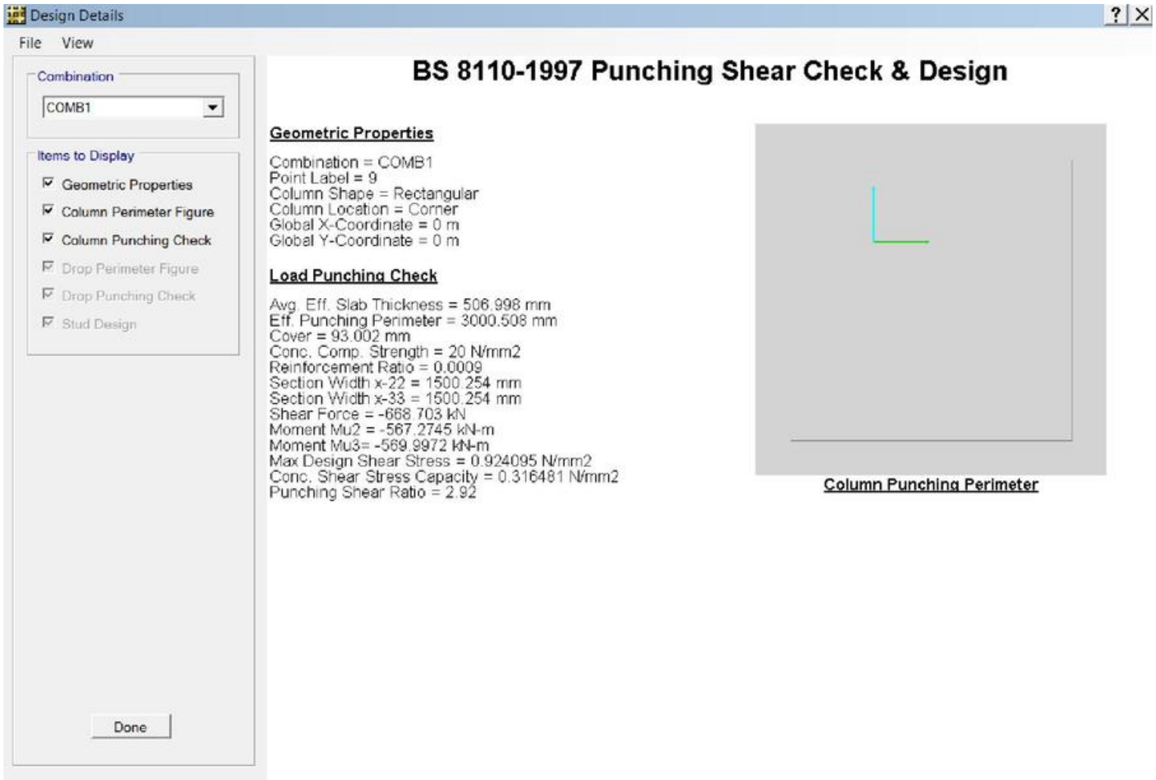


Figure 105

Through the ‘**Display**’ menu, relevant quantities can be displayed on the screen. For instance, the ‘**Display**’>’**Show Strip Forces**’ command or by simply clicking the ‘**F8**’ key, the following window will be displayed.

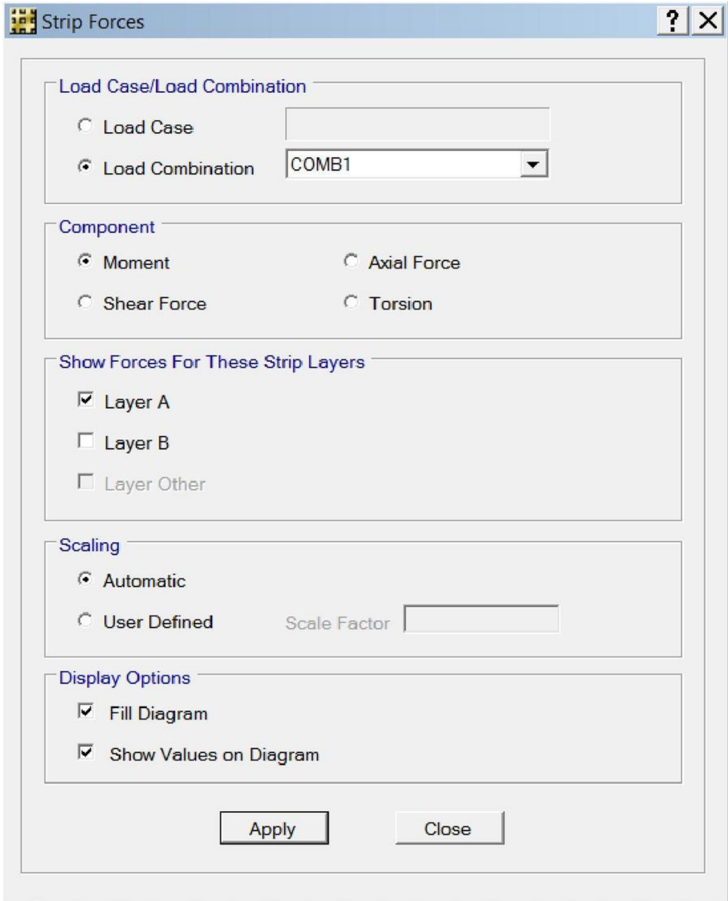


Figure 106

In this window, the '**Load Case/Load Combination**' box provides with radio buttons to select for which particular load case or load combination that we want to display the output. The '**Component**' box contains four radio buttons to select which quantity to display. The '**Show Forces For These Strip Layers**' box allows us to select the strip layer for which the quantity is displayed. Both strip layers can be selected at the same time. From the '**Scaling**' box, we can select whether automatic scaling or user defined scaling is used while displaying the diagram. The '**Display Options**' box allows us to fill or not to fill the diagram and to display or not to display the values on the diagram. For the preferences shown in Fig. 106, the following diagram will be displayed.

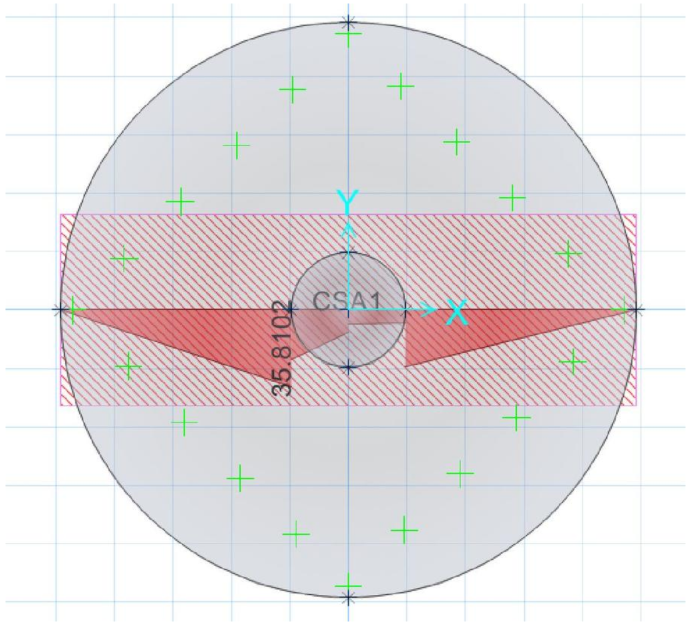


Figure 107

The '**Display**'>'**Show Slab Design…**' command results in the window shown in Fig 108. In this window, several options can be set in order to display the footing design in the way we wanted to. The '**Choose Display Type**' box allows us to select the '**Design Basis**' between '**Strip Based**' and '**Finite Element Based**'. Unless some differences in the way the design is displayed, there is no difference in the amount of reinforcement between these selections. Through the '**Display Type**' in the same box, it can be selected whether to display flexural reinforcement or shear reinforcement. This box also allows us to impose or not to impose minimum reinforcement during the design. The '**Rebar Location Shown**' box allows us to select which reinforcement, top or bottom or both, to be displayed. The '**Reinforcing Display Type**' box allows us to set the manner in which the amount of reinforcement is displayed. The option whether to show the reinforcing envelop diagram and the reinforcing extent can be set by the check boxes in the '**Reinforcing Diagram**' window. The strip layer direction for which the amount of reinforcement is displayed can be chosen from the '**Choose Strip Direction**' box. The '**Display Options**' box allows us whether to display output in filled diagram or not and whether the values at controlling stations will be displayed or not. If we want to display the amount of reinforcement above some specified reinforcement bar area or spacing, we can use the options in the '**Show Rebar Above Specified Value**' box. When the '**Typical Uniform Reinforcing Specified Below**' radio button is selected, the '**Typical Uniform Reinforcing'** box get activated. In this box, we can set a specific value above which the

reinforcement amount will be displayed. The reinforcement diagram output, for the options set in Fig. 108, will be shown below in fig. 109.

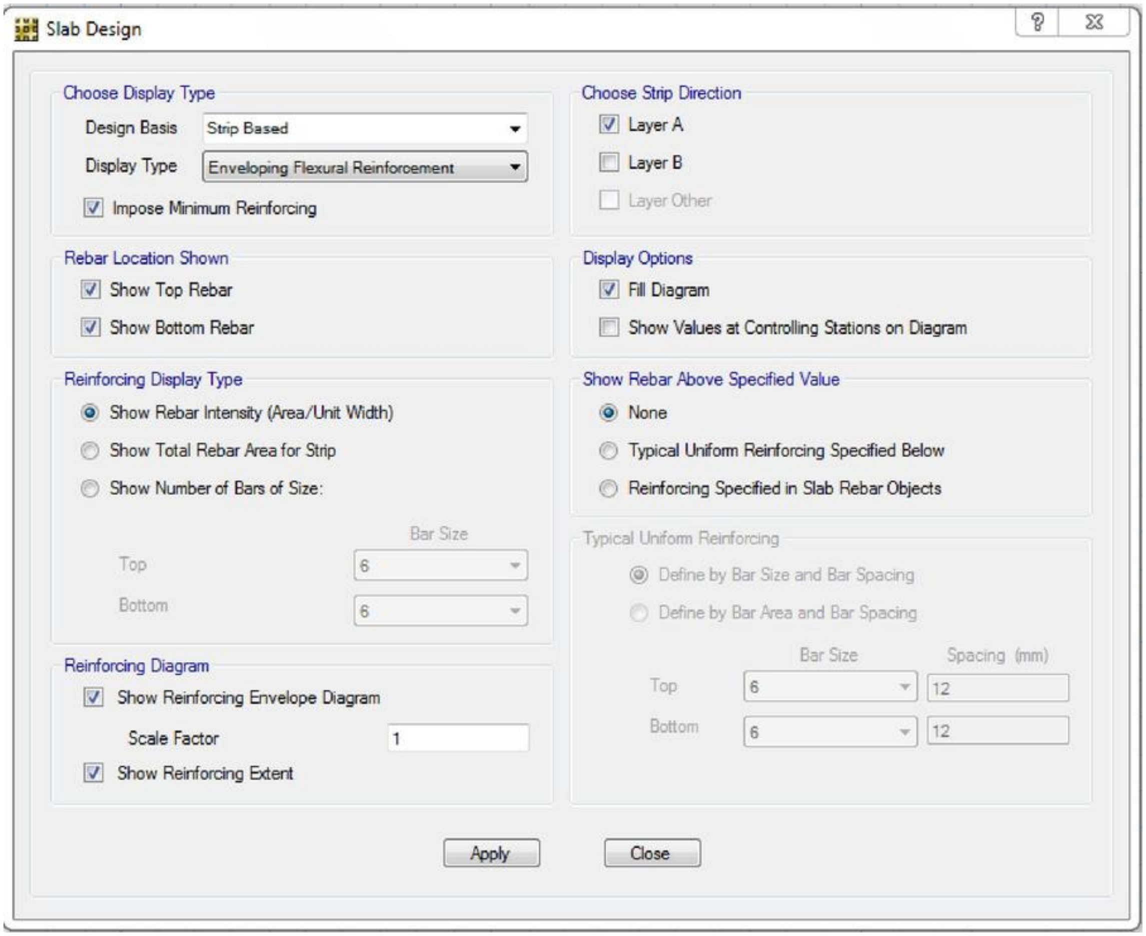


Figure 108

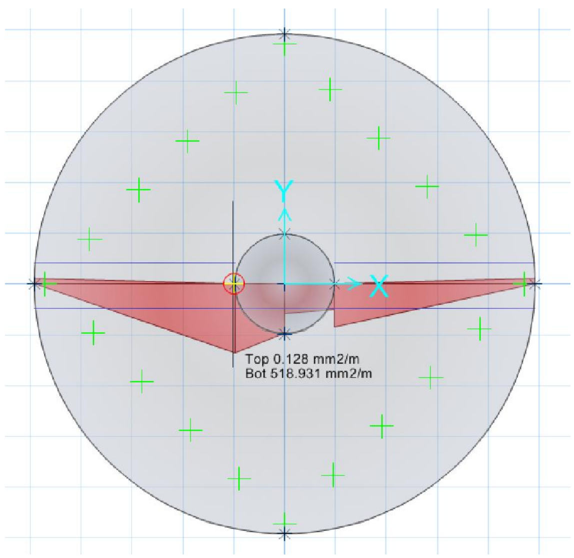


Figure 109

As can be noticed from the footing design diagram in fig. 71, the top reinforcement across strip layer A is around $0mm^2/m$ for top reinforcement and $790mm^2/m$ for bottom reinforcement.

The design outputs can also be displayed in tabular format by clicking on the '**Show Tables…**' menu item from the '**Display**' menu or by just clicking on the equivalent icon  from the tool bar below the menu bar and the following window will pop up.

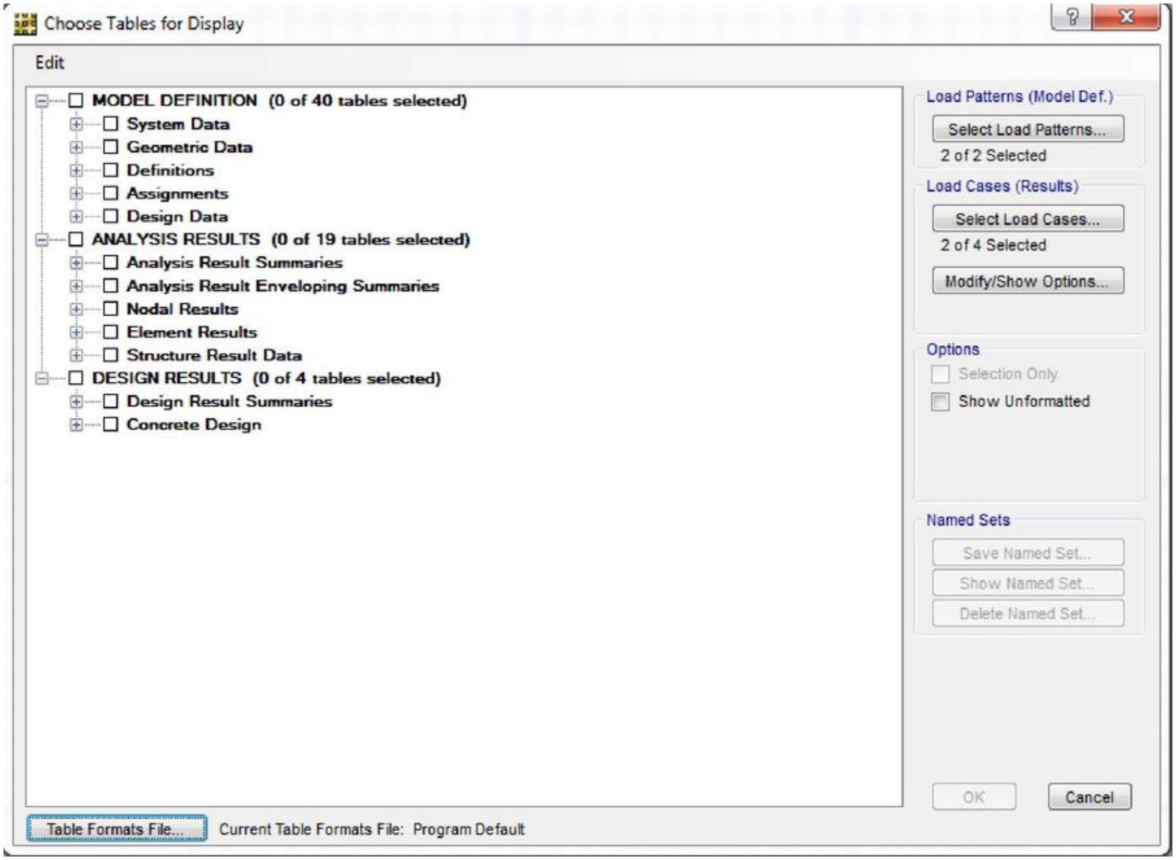


Figure 110

In this window, we can select any of the model definitions or analysis results or design results and press '**OK**' to display the quantity which we want to have a look at. By using the right hand side buttons in the window, the load patterns and the load cases can be selected.

STEP 10: Detailing

After running the analysis and after checking that the results are reasonable, the detailing will be done. However, before running the detailing, the detailing preferences can be set from the '**Detailing**' menu. From the '**Detailing**

Preferences…', the likes of dimensional units and material quantity units can be selected. From '**Slab/Mat Detailing Preferences…**', the likes of rebar curtailment options, the rebar detailing options, rebar selection rules and preferred rebar sizes can be selected. The '**Drawing Sheet Set-up…**' menu allows us to set-up the contents of the drawing sheet. The '**Drawing Format Properties…**' allows us to set some formats in which the output displayed.

To run the detailing, go to '**Run**' menu and click on '**Run Detailing…**' or simultaneously press '**Shift**' and '**F5**' keys or just click on the run detailing icon  from the tool bar just below the menu bar. Then, the '**Run Detailing Options**' window pops up so that we set the detailing options. Set the detailing options which you want and click '**OK**'.

Once the detailing is run, the detailing can be displayed. The detailing display options can be best accessed from the '**Model Explorer**'. When expanded in full, the '**Detailing**' tab of the model explorer looks like:

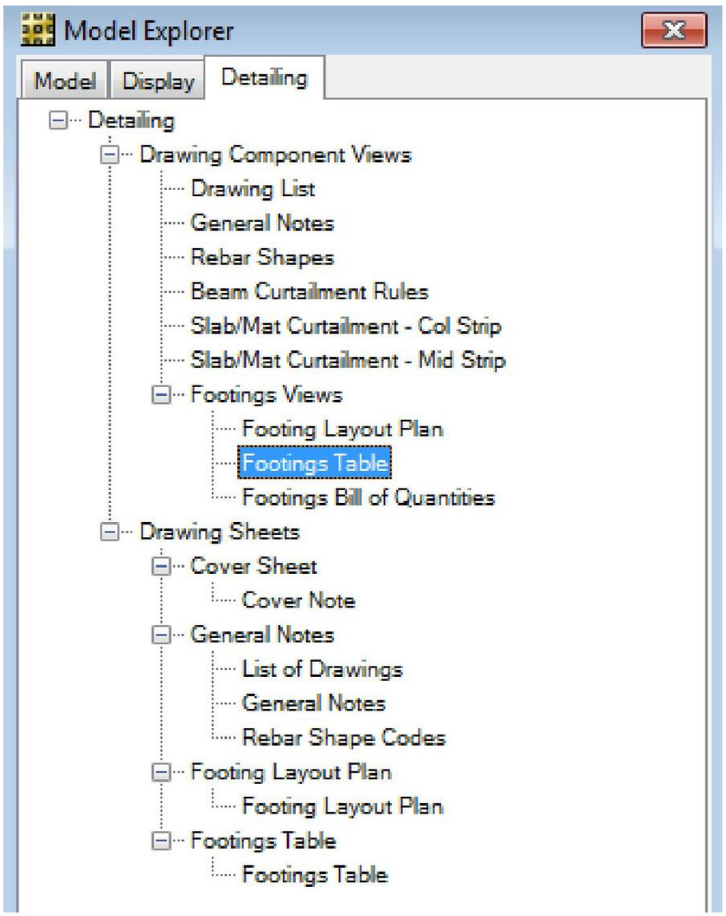


Figure 111

By clicking on any of the options from the detailing tab, a desired detailing can be displayed. For instance, by double clicking on the '**Footing Table**', the following detail of footing can be shown.

FOOTINGS TABLE

SR. NO.	TYPE	NOS	LX	LY	T	REBARS-A	REBARS-B
1	F1	1	3.000 M	3.000 M	0.600 M	7-#3	8-#3

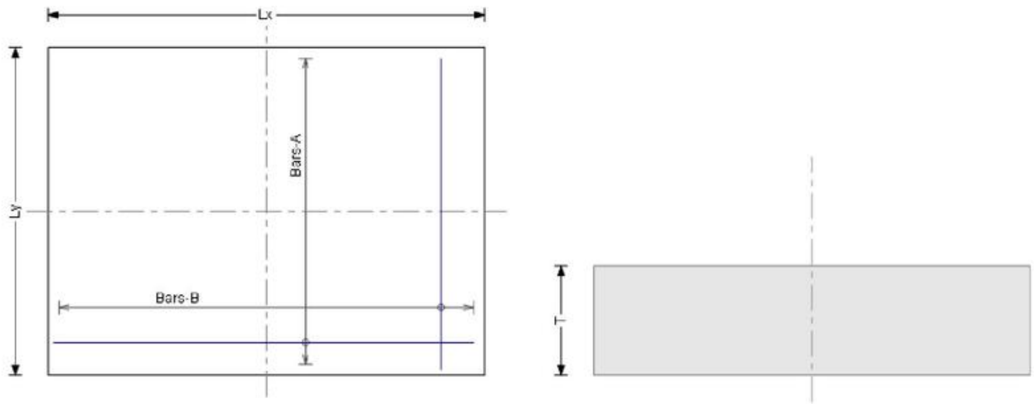


Figure 112

In this detail, the diameter of the reinforcement which is used is 10mm. If you want to change the diameter….

Apart from this detail, other details can also be shown.

STEP 11: Reporting

The last step of foundation design is reporting. Before creating the report, the report preferences should be set up. To do this, go to the 'File' menu and click on 'Report Set-up…' and the following window pops up.

In this '**Report Setup Data**' window, the user preferences regarding the reporting such as the report output type, the report page orientation and the report items can be set along with the load patterns and load combinations. Once the preference is set, the report can be created by clicking on '**Create Report**' command in the '**File**' menu. The '**Advanced Report Writer**' command in the same menu can be used to set some advanced reporting formats.

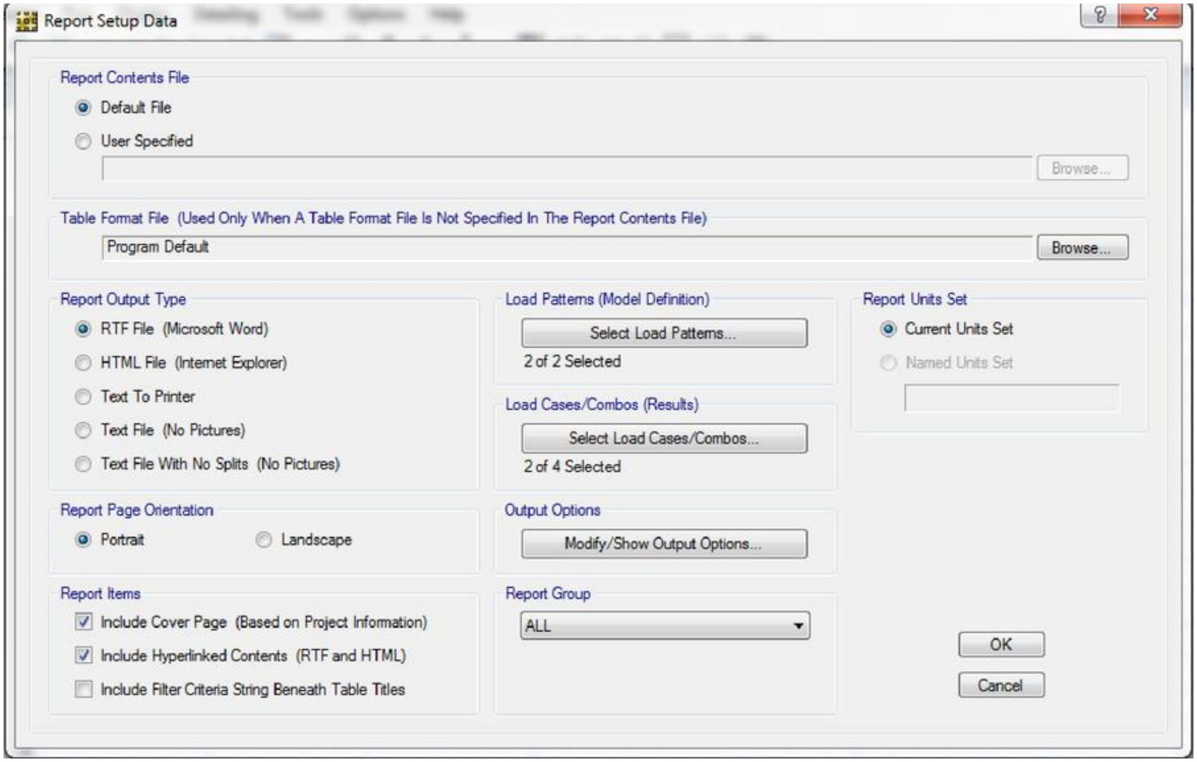


Figure 113

This concludes the tutorial for the design of single footing using a model imported from AutoCAD.

Design Of Combined Footing

Week 5-6

3. Combined Footing

In this part of the tutorial, a combined footing will be designed in three different ways. First, a rectangular combined foundation will be designed using the built-in model of the software itself. Then a trapezoidal isolated foundation will be designed using grids. Lastly, a rectangular combined foundation will be designed by importing the geometry from AutoCAD. In all sections, a design problem will be given first and the procedures which will be followed to carry out the design process in SAFE program will be detailed.

3.1. From the Built-in Model

STEP 1:

Design a rectangular combined footing with the following parameters:

- *Dimension of the footing*: 2.9mX5.8m
- *Dimension of first column*: 0.3mX0.3m
- *Dimension of second column*: 0.35mX0.35m
- *Distance between the columns*: 4.5m
- *Distance between the center of the first column to the left edge*: 0.15m
- *Ultimate bearing capacity of the soil* : 100kN/m^2
- *Maximum allowable settlement of the foundation*: 10mm
- *Concentrated ultimate load on the first column* : 650kN
- *Concentrated ultimate load on the second column*: 1000kN
- *Grade of concrete*: C-25 (25MPa 28-day characteristics cube strength)
- *Grade of reinforcement bar (rebar)*: S-300 (300MPa characteristics yield strength)
- *Overall thickness of the foundation*: 500mm
- *Concrete cover* : 50mm
- *There are no bending moments on the column*

Creating the Model

To create a footing model from the built-in program, there are three ways.

d. Go to **File** menu and click on **New Model …**

e. Click on the icon . This icon is the first among the list of icons just below the menu bar.

f. Simply click on the **Ctrl** and **N** keys simultaneously (**Ctrl+N**).

Performing one of the above three ways results in the popping-up of the following **'New Model Initialization'** window.

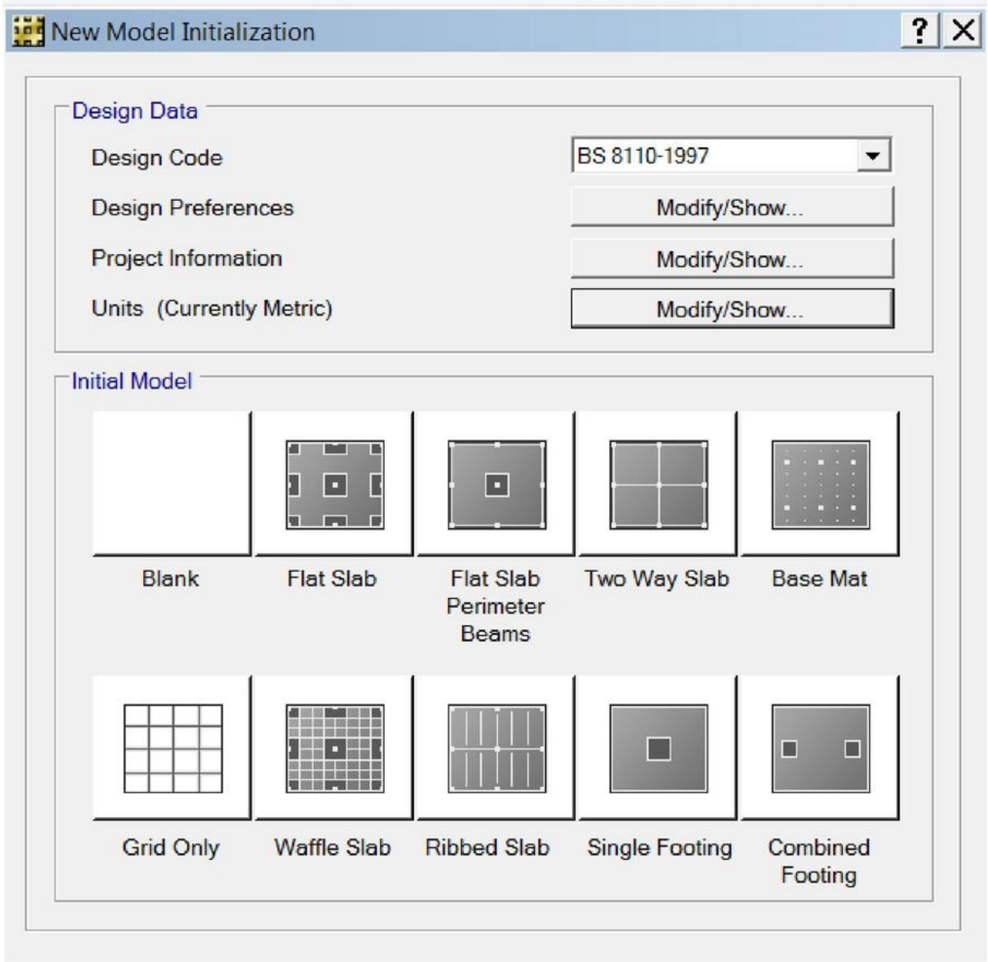


Figure 114

This window consists of two parts: the '**Design Data**' part and the '**Initial Model**' part. In the '**Design Data**', there are '**Design Code**', '**Design Preferences**', '**Project Information**' and '**Units**'.

The '**Design Code**' consists of a list of selected design codes from many countries. Depending on the suitability of the code for your country, you may select one among the list of codes. The design code which follows similar design philosophy to my country code is BS 8110-1997. So, this design code is selected.

After selecting the design code, it is better to first select the units to be used in the design and analysis process. These units can be selected by clicking on the '**Modify/Show**' button in front of '**Units**'. The following window pops up when the button is clicked. To choose metric units, click on '**Metric Defaults**' button.

The click results is metric units for best practices to be selected. It can be observed that, even though all the units are metric, they are not consistent. To select a preferred consistent unit, click on the '**Consistent Units**' button. In this '**Units**' window, the decimal places and minimum number of significant digits for any quantity can also be modified.

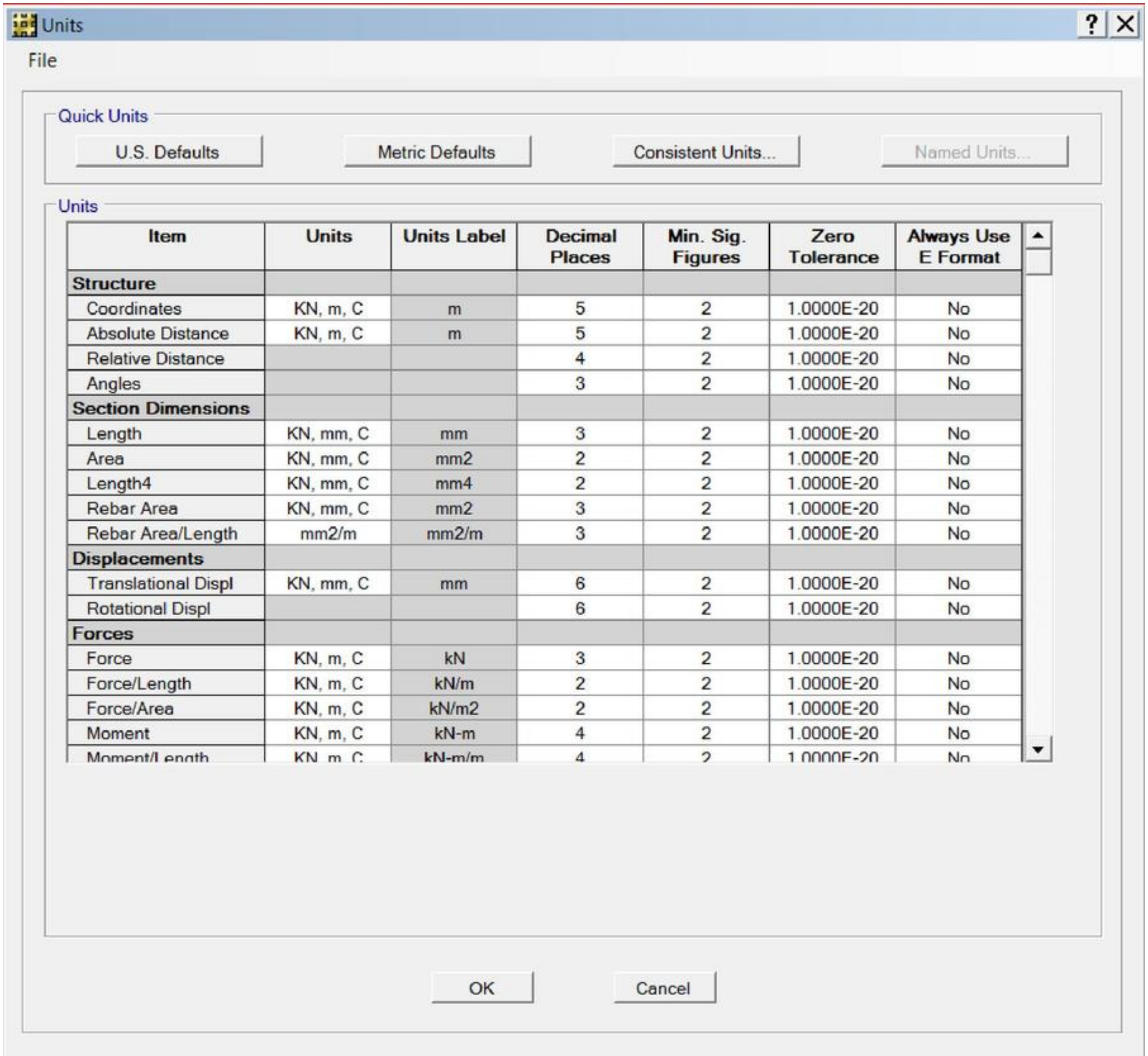


Figure 115

By clicking on the '**Modify/Show**' button in front of the '**Project Information**', information regarding the project, the company and the model can be entered.

By clicking on the '**Modify/Show**' button in front of the '**Design Preferences**', the user preferences regarding the design code, concrete cover for slabs and beams, post-tensioning can be changed. The following **'Design Preferences'** window pops-up when the button is clicked. The '**Min. Cover Slabs**' tab of the window may be of interest for this particular problem as footings will be modelled as slabs. Thus, click on this tab. This is the tab where the concrete cover and preferred rebar size will be entered.

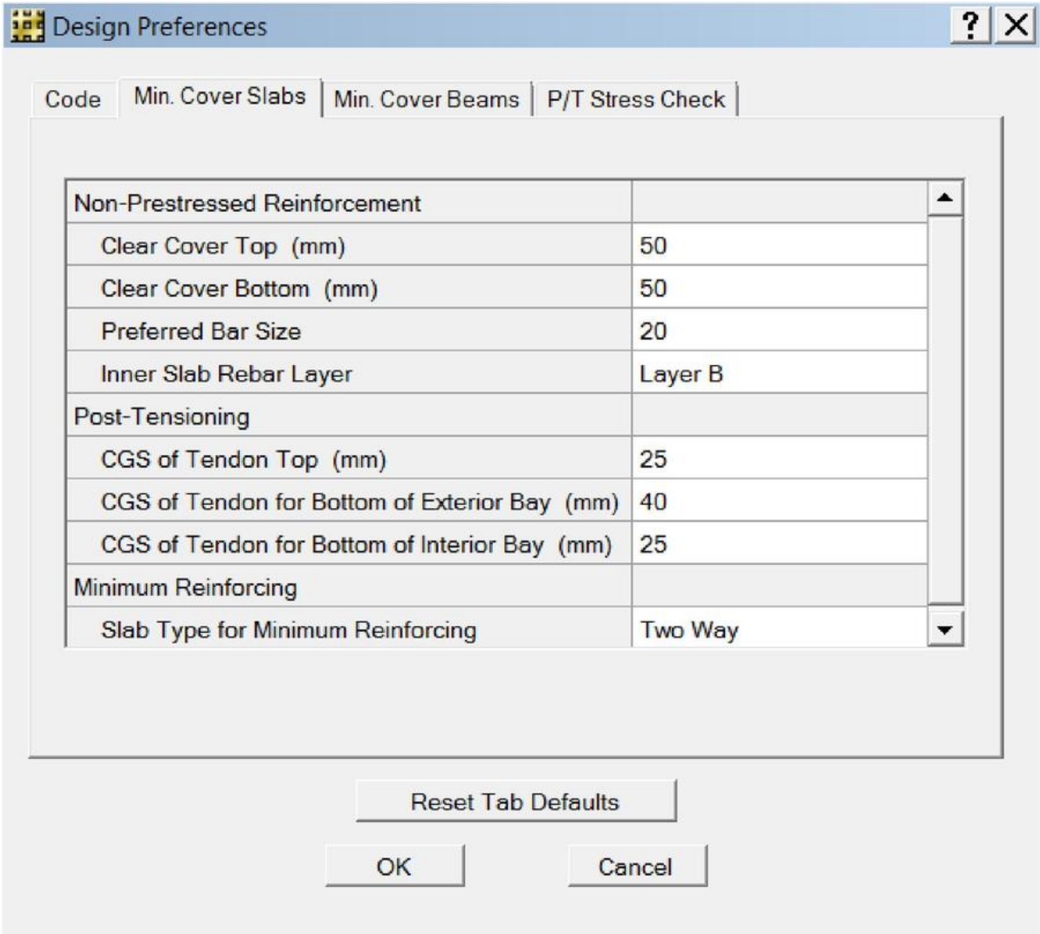


Figure 116.

In the 'Min. Cover Slabs' tab, for '**Non-Prestressed Reinforcement**' , both the '**Clear Cover Top**' and '**Clear Cover Bottom**' should be set to 50mm as the concrete cover in this design problem stated to be 50mm. The '**Preferred Bar Size**' can be set to any reasonable value. Here, the '**Preferred Bar Size**' is set to 20 which is the diameter of the rebar in mm which will be used as the main reinforcement in the foundation. Leave the rest at they are.

Once everything is set correctly in the '**Design Data**' box, go to the '**Initial Model**' box and select the suitable model. Since, an isolated (single) footing will be designed click on the big square button just above '**Combined Footing**'. The button itself shows a plan view of a combined footing. When the button is clicked, the following

'Combined Footing' window will pop up. In this window, there are three boxes: '**Plan Dimensions**', '**Load**' and '**Properties**'.

In the '**Plan Dimensions**' box, the footing and column dimensions will be entered. While entering the footing dimension, it is the distance from the center of the columns to each of the edge which will be entered not the actual dimension of the footing. Since the distance from the left edge to the center of the column is 0.15m, the '**Left Edge Distance**' is set to 0.15m. The distance from the center of the second column to the right edge, if you calculate it from the dimensions of the footing, is 1.15m. Thus, the value of set all the values of '**Right Edge Distance**' is set to 1.15m. Since the column is centrally located in y-direction, the values of '**Top Edge Distance**' and '**Bottom Edge Distance**' are set to be 1.45m.

The radio button below '**Load Spacing and Size**' should be set to '**X Direction**' as the columns will be aligned in x-direction. The value of the '**Spacing**' should be set to 4.5 as the distance between the columns is 4.5m. For the '**Load Size (square)**', the dimensions of any of the columns can be entered and later it can be changed depending on the actual footing dimensions.

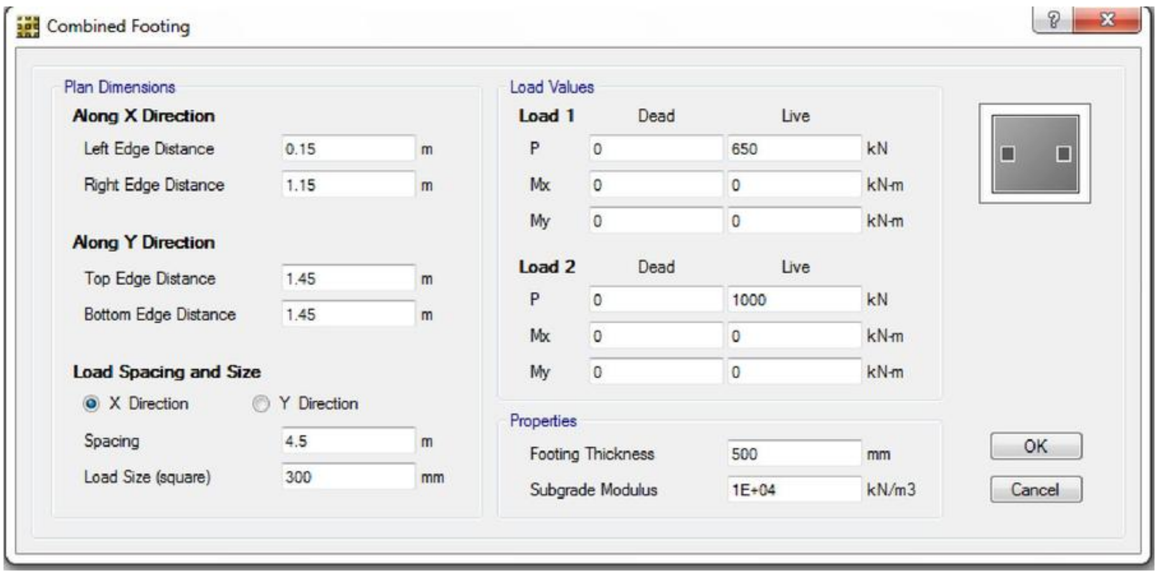


Figure 117

Since in this design problem, the dimension of the first column is 300mmX300mm, enter 300 in the text field as the unit is already in mm. For column shapes other than square, just enter a preliminary column dimension and you can change both the shape and size of columns at a later time.

In the '**Load Values**' box, enter the concentrated loads and moments in the column depending on their type as '**Dead**' or '**Live**'. In this particular problem, the loads on each column are factored or ultimate loads comprised of dead and live loads. But, the self-weight is not included. Since, the self-weight is regarded as a dead load and should be multiplied by a partial safety factor. Thus, it is better if we define the factored loads as live loads with partial safety factor of unity. Thus, set the value for '**P**' of '**Load1**' to be 650 and the value for '**P**' of '**Load2**' to be 1000 in the live load column.

In the '**Properties**' box, the footing thickness and the subgrade modulus of the soil will be entered. The footing thickness of the footing is given to be 500mm. Thus, enter this value as shown in fig. 117. However this value may be revised depending on the punching shear requirements at a later time. The subgrade modulus of a soil is the ratio of the increment of contact pressure to the corresponding change in settlement. In the absence of a supportive test to determine its value, it can also be safely approximated by the ratio of the bearing capacity of the soil to the maximum permissible settlement in the soil. In this particular problem, the subgrade modulus can be approximated by $100kN/m^2/10mm = 10000kN/m^3$. Thus enter this value in the text field in front of the 'Subgrade Modulus'. Then press the '**OK**' button. This exits the '**Combined Footing**' window and a plan view of the foundation will be displayed on the main screen. You can zoom in and zoom out using the wheel of your mouse or using the tool bar under the menu bar.

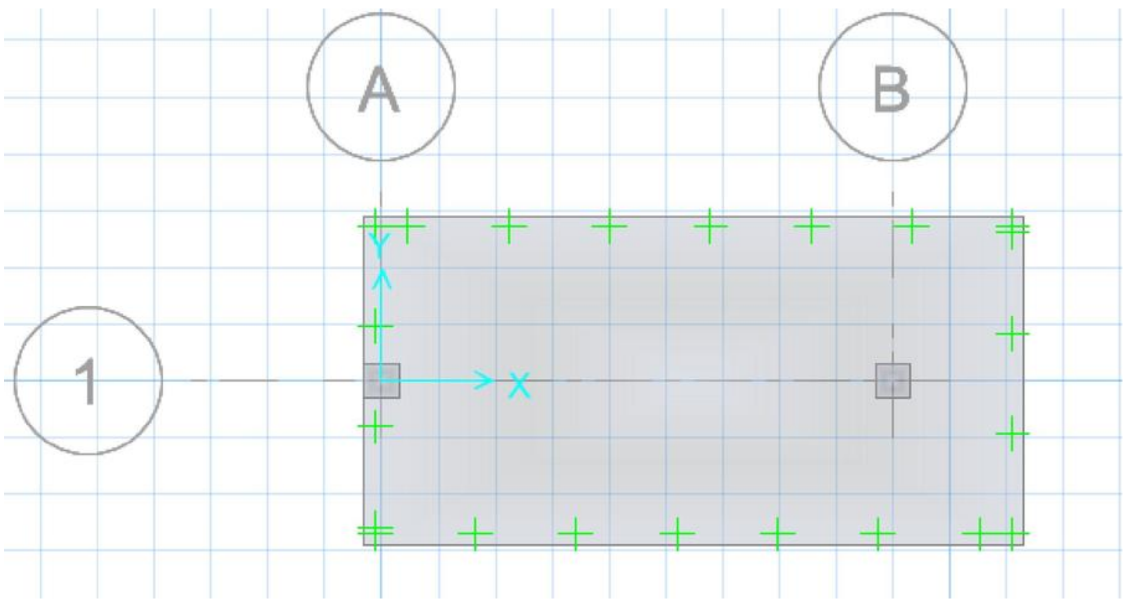


Figure 118

STEP 2: Defining Material Properties

The materials which are involved in the design of the footing should be defined before the analysis. These materials are the concrete, the reinforcement bar (rebar) and the soil which is already defined. Thus, the concrete property and the rebar property will be defined here.

The material definition can be carried out in two ways.

The first one is through the '**Define**' menu in the menu bar. The other one through the model explorer on the left hand side of the home window. To define material properties through the former method, click on '**Define**' menu and again click on '**Materials…**' resulting in the following window depending on prior material definitions.

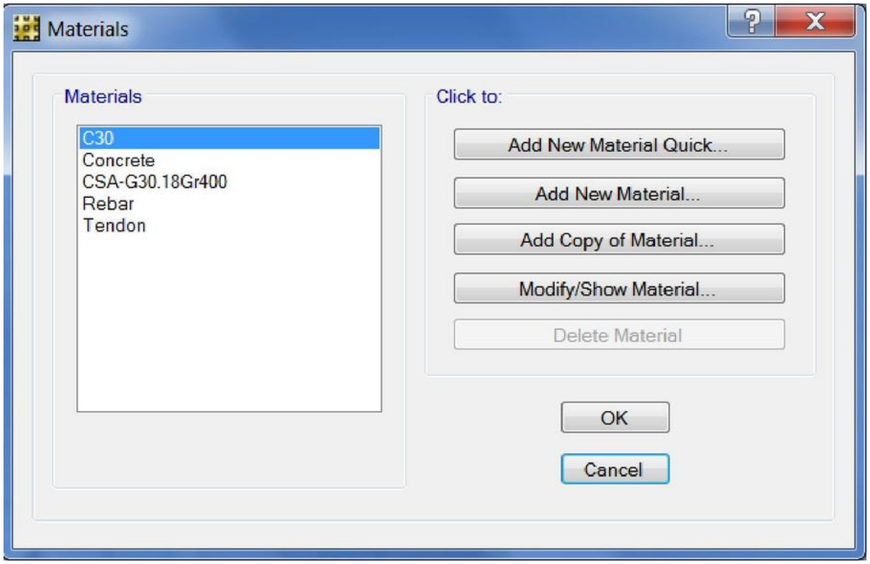


Figure 119

The list in the '**Materials**' box may not be exactly as it appears in your widow. However, that doesn't bring any change in the outcome of the design process as you can customize this list any time.

The '**Add New Material Quick…**' button allows you to define materials quickly from a list of pre-defined materials. The '**Add New Material**' button allows you to define materials by changing their properties. The '**Add Copy of Material**' button allows you to define a material with same property as an already defined material. The '**Modify/Show Material**' button displays the property of an already defined material with the possibility of modification. The '**Delete Material**' button, when active, deletes a defined material property.

For this particular problem, we will define concrete and rebar properties through the '**Add New Material**' button. Thus, click on this button. This can also be achieved by right clicking on 'Materials' on the model explorer. The following window pops up in ether cases.

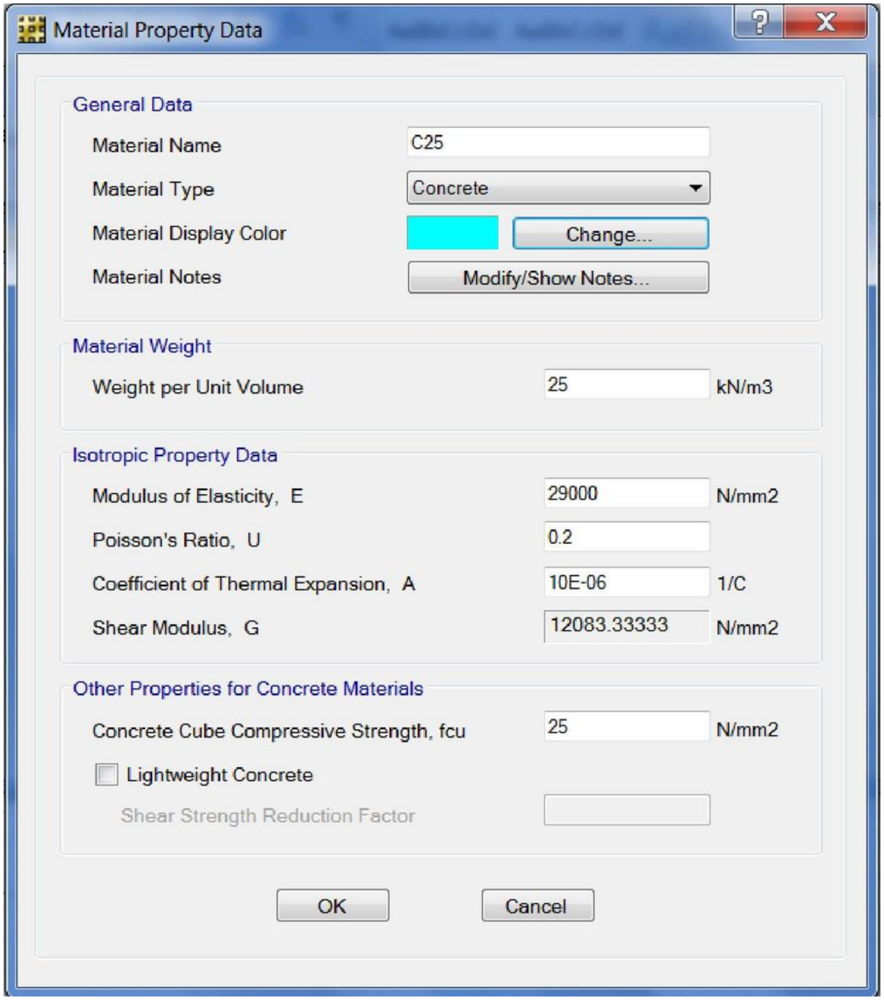


Figure 120

This is the window where material properties are defined. First, let us define the concrete property which will be used in the design. Put any name for the material in the text field in front of '**Material Name**'. But, it is important to make sure that the concrete with the defined material name is assigned for the footing. For this particular problem, let the name of the concrete to be used be '**C25**'. Since we are defining a concrete, the '**Material Type**' should be set to '**Concrete**'.

The unit weight of reinforced concrete may vary depending of the design code of your country. Thus, enter the unit weight of reinforced concrete stipulated in your country code in the text field in front of '**Weight per Unit Volume**'. For this particular example, we will use 25kN/m^3.

For C-25 concrete, the modulus of elasticity according to BS 8110-1197 is around 29GPa. Thus, enter this value in the text field in front of '**Modulus of Elasticity, E**'. If you selected another design code in step 1 while creating the model, you should refer to actual value of this parameter from the code and enter it accordingly. Be aware of the units though.

The values of Poisson's ratio and coefficient of thermal expansion may also be defined in the design code and should be entered accordingly. For this particular problem, a value of 0.2 for '**Poisson's Ratio, U**' and a value of $10 \times 10^{-6}/^{o}C$ for '**Coefficient of Thermal Expansion, A**' will be entered. The '**Shear Modulus, G**' will be automatically calculated in an uneditable text field.

The grade of concrete for this particular problem is C-25 which is a concrete with 28 day characteristics cube compressive strength of 25MPa. The concrete designation may be different for different country codes but the concept is the same. Therefore, enter 25 in the text field in front of '**Concrete Cube Compressive Strength, fcu**'.

If a lightweight concrete is used, check on '**Lightweight Concrete**' and enter the corresponding '**Shear Strength Reduction Factor**' in the space provided.

When you press on '**OK**', a concrete material with the above properties will be added to the list of materials. This material will be assigned for the footing before the analysis.

After defining the concrete property, the program returns to the window shown in Fig. 119. To define a rebar property, we will follow the same procedure as we followed while defining the concrete property. Since a new rebar property will be defined, click on the '**Add New Material…**' button. A '**Material Property Data**' window pops up and when you change the '**Material Type**' to '**Rebar**', the window appears to look like the following.

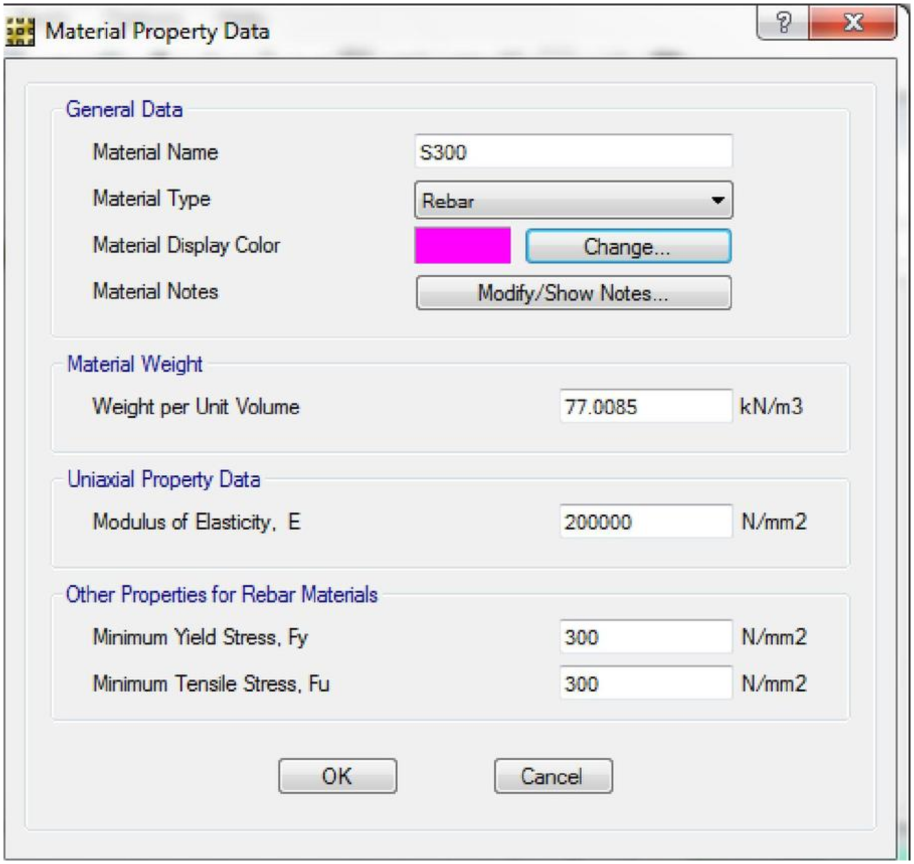


Figure 121

Change the '**Material Name**' to any name you want. Here, we name it '**S300**'. The material type should be '**Rebar**'. The weight per unit volume of steel is stipulated in the design code. For BS 8110-1197, the weight per unit volume is 77.0085kN/m^3. Thus, enter this value in the text field in front of '**Weight per Unit Volume**'. The modulus of elasticity for reinforcement bars according to the same design code is 200GPa. Thus enter this value in the text field in front of '**Modulus of Elasticity, E**' considering the unit.

In the '**Other Properties for Rebar Materials**' box, two quantities are mentioned: minimum yield stress and minimum tensile stress for the reinforcing material. The values of these parameters will be specified in the design code which you defined earlier. If the code assumes that the rebar material exhibits elastic perfectly plastic behavior, the values of these two quantities will be the same. The grade of steel to be used for this particular example is S-300. The yield stress for this type of reinforcement bar is 300MPa. Since the design code of my country assumes that rebars exhibit elastic perfectly plastic behavior, the minimum tensile stress will also be 300MPa. Thus enter 300 in both text fields in front of the '**Minimum yield**

stress, Fy' and '**Minimum Tensile Stress, Fu**'. Then press '**OK**' in both '**Material Property Data**' and '**Materials**' windows concluding the material definition step.

STEP 3: Defining Footing and Column Properties

After defining the material properties, the footing and column properties can be defined. This definition can take place in two ways: from the menu bar and from the model explorer. In SAFE software, footings are modelled as 'footings' and foundation columns are modelled as 'stiff'.

To define footing and column properties from the menu bar, go to '**Define**' menu and click on '**Slab Properties…**'. The following window will pop up.

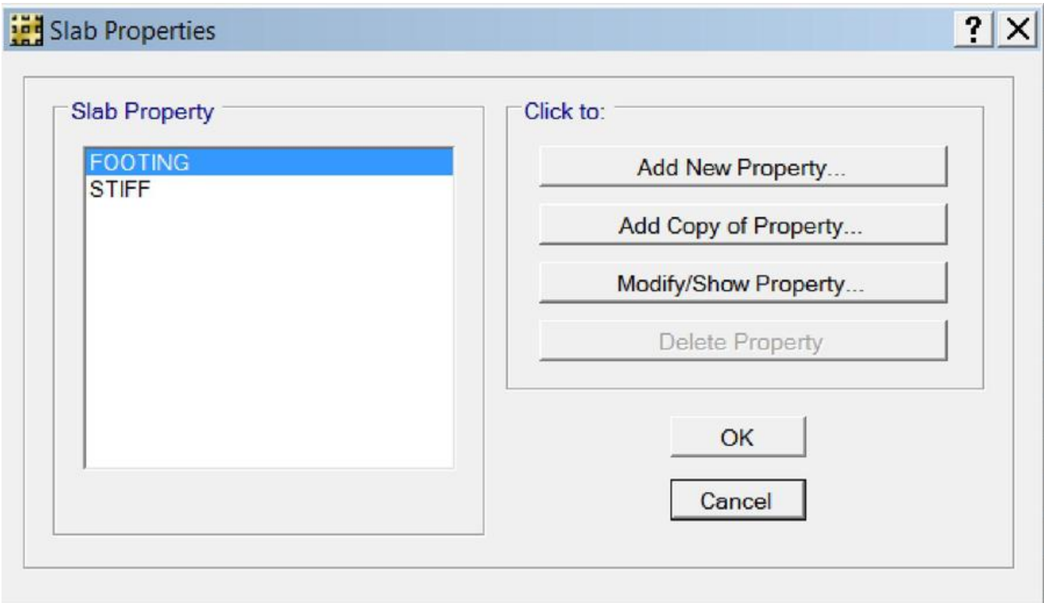


Figure 122

In the '**Slab Property**' box, two foundation components are listed. The names in this box for your particular design may be different from the list of names in the box. The '**FOOTING**' corresponds to the bottom slab portion of the foundation and '**STIFF**' corresponds to the foundation column. The '**Add New Property…**' button prompts the user to enter new properties for the footing and foundation column while the '**Add Copy of Property…**' copies the property of an existing slap. The '**Modify/Show Property…**' allows the user to show the property of an existing component with the possibility of modification. When the '**Delete Property**' button is active, it allows the user to delete an existing slab property.

In this case, we use the '**Modify/Show Property…**' button. However, one can also use the '**Add New Property…**' button. To use '**Modify/Show Property…**'

button, highlight the foundation component to be modified like the '**FOOTING**' is highlighted in Fig. 122 and click on this button. Since '**FOOTING**' is highlighted, we will be modifying the footing property. The following window appears after the click.

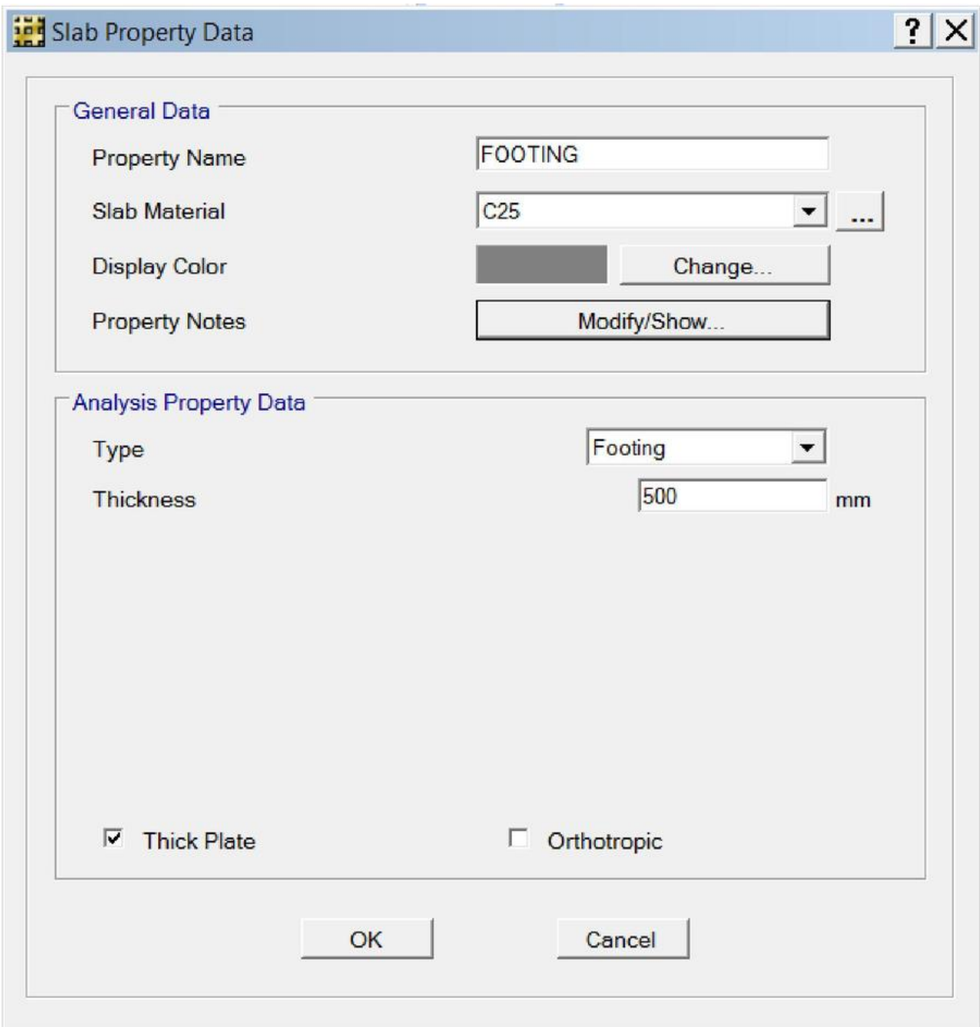


Figure 123

The '**Property Name**' will be automatically assigned to '**FOOTING**'. The '**Slab Material**' should be set to the concrete grade which is defined in step 2. Since the name of the concrete material defined in step 2 for this particular problem is '**C25**', a material with this name should be selected from the list.

In the '**Analysis Property Data**' box, '**Type**' should be set to '**Footing**' as we are modifying the footing property. The '**Thickness**' value should be set to the same value as the one which is used while creating the model. Since the thickness of the footing is 500mm, this value is entered in the text field corresponding to '**Thickness**'. As footings are modelled as thick plates, check the '**Thick Plate**' option. The ‘**Orthotropic**’ check box is selected when a footing with irregular dimension is to be used.

When you press ‘**OK**’, the ‘**Slab Property Data**’ window will be exited and the ‘**Slab Properties**’ window in Fig. 11 gets activated. Now, the property of the foundation column will be modified. To do this, highlight ‘**STIFF**’ in the ‘**Slab Property**’ box and click on the ‘**Modify/Slab Property…**’ button. The following window appears after the click.

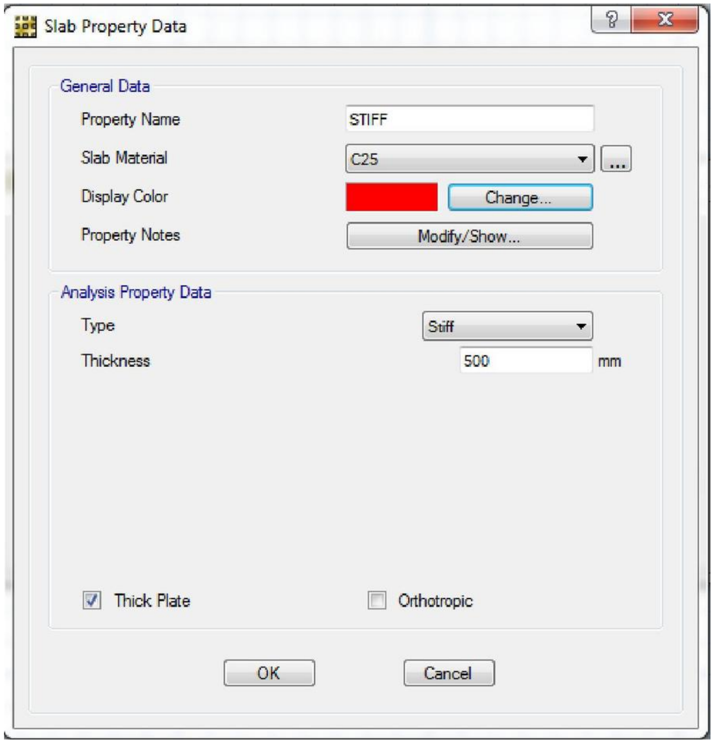


Figure 124

The ‘**Property Name**’ will be automatically assigned to ‘**STIFF**’. The ‘**Slab Material**’ should be set to the concrete grade which is defined in step 2. Since the name of the concrete material defined in step 2 for this particular problem is ‘**C25**’, a material with this name should be selected from the list.

In the ‘**Analysis Property Data**’ box, ‘**Type**’ should be set to ‘**Stiff**’ as we are modifying the footing property. The ‘**Thickness**’ value should be set to the same value as the thickness of the footing which is 500mm, this value is entered in the text field corresponding to ‘**Thickness**’. As foundation columns are modelled as thick plates, check the ‘**Thick Plate**’ option. The ‘**Orthotropic**’ check box is selected when a column with irregular dimension is to be used.

When you press '**OK**', the '**Slab Property Data**' window will be exited and the '**Slab Properties**' window in Fig. 122 gets activated. Again press '**OK**' and exit the window for defining the footing and foundation column.

STEP 4: Defining Load Patterns, Load Cases and Load Combinations

The loads on the foundation should be defined accordingly before the analysis. First, the load pattern should be defined. This can be done from the '**Define**' menu or from the '**Model Explorer**'. This time, we will do it from the model explorer. In the model explorer, expand '**Load Definitions**' and you will see '**Load Patterns**'. When you expand '**Load Patterns**', you will see '**DEAD**' and '**LIVE**'. At the end, the model explorer appears to look like:

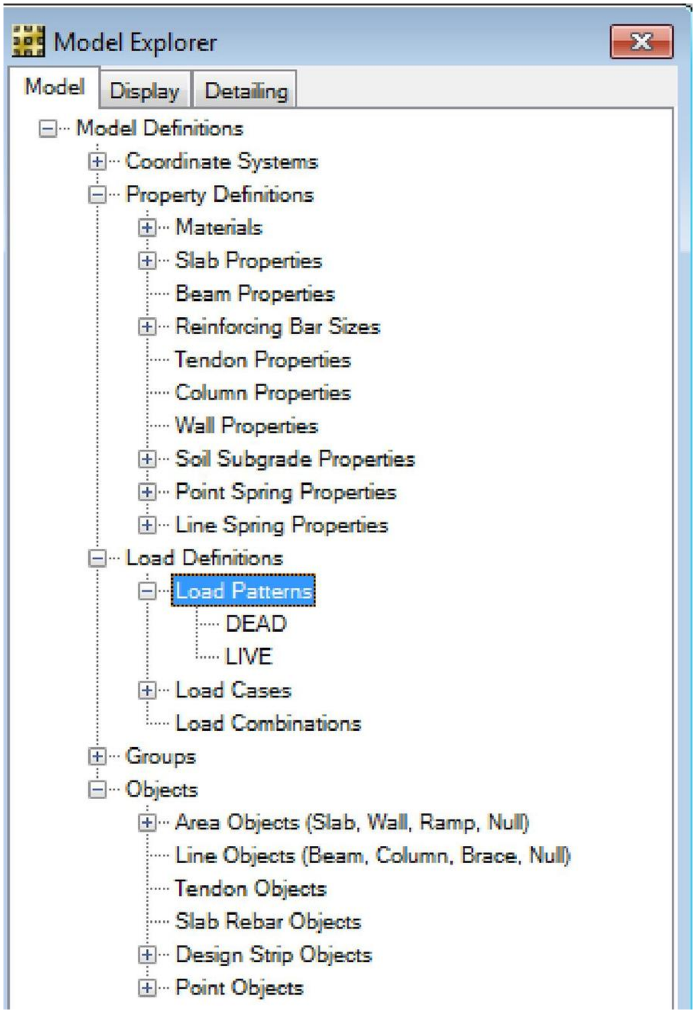


Figure 125

Now, right click on '**Load Patterns**' and click on '**New Load Pattern**' and the following window pops up.

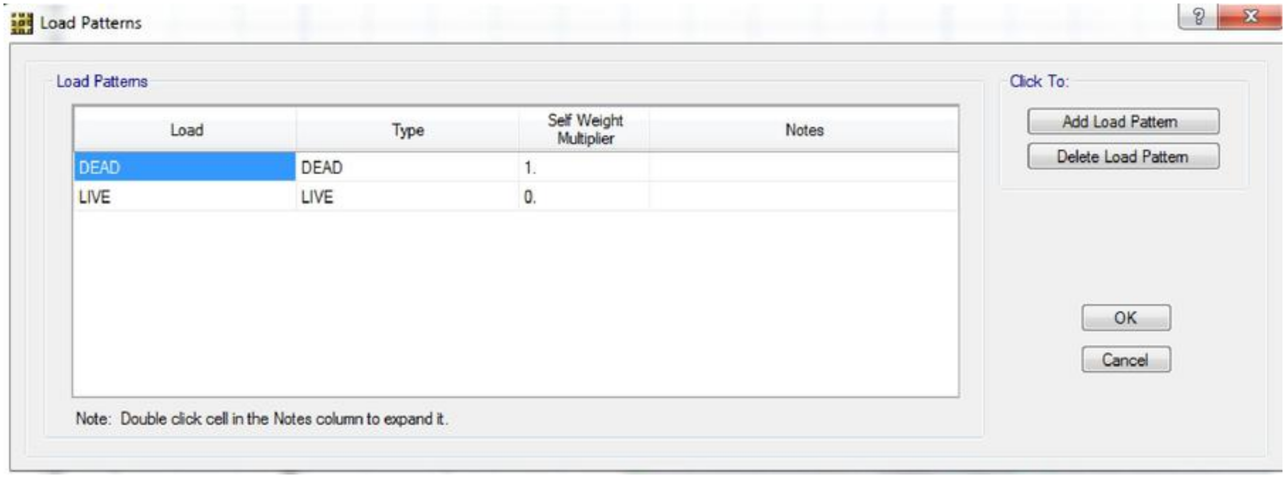


Figure 126

In the '**Load Patterns**' window, two load patterns are already defined: '**DEAD**' and '**LIVE**'. The '**Type**' of the '**Load**' should also be changed accordingly. There are many options for type of loading. The '**Type**' for dead loads should be set to '**DEAD**' and for live loads '**LIVE**'. If there are other types of load patterns on the foundation such as earth quake load, you can add the load pattern with the '**Add Load Pattern**' button. You can also delete any undesirable load pattern using the '**Delete Load Pattern**' button. Since in this example, we have only dead and live loads, we will leave the existing load patterns as they are. The '**Self Weight Multiplier**' value should also be changed accordingly. This value imparts the option whether to include or leave the self-weight of the foundation in addition to external loads. If the self-weight of the foundation is already included as an external dead load or if you want to exclude the effect of self-weight from the analysis, the value under '**Self Weight Multiplier**' should be set to zero. In this example, we will consider the self-weight as an additional load to the external dead load. Thus, the value under '**Self Weight Multiplier**' for the '**DEAD**' load is one. For the '**LIVE**' load, it will be zero. Press '**OK**' and the window will be exited.

After this, the load cases will be defined. Load cases are used to dictate the way the loads are applied (statically or dynamically) or the way the structure responds (linearly or non-linearly) for the defined load patterns. To define a load case, go to '**Define**' menu and click on '**Load Cases…**'. The following window will pop up after the click.

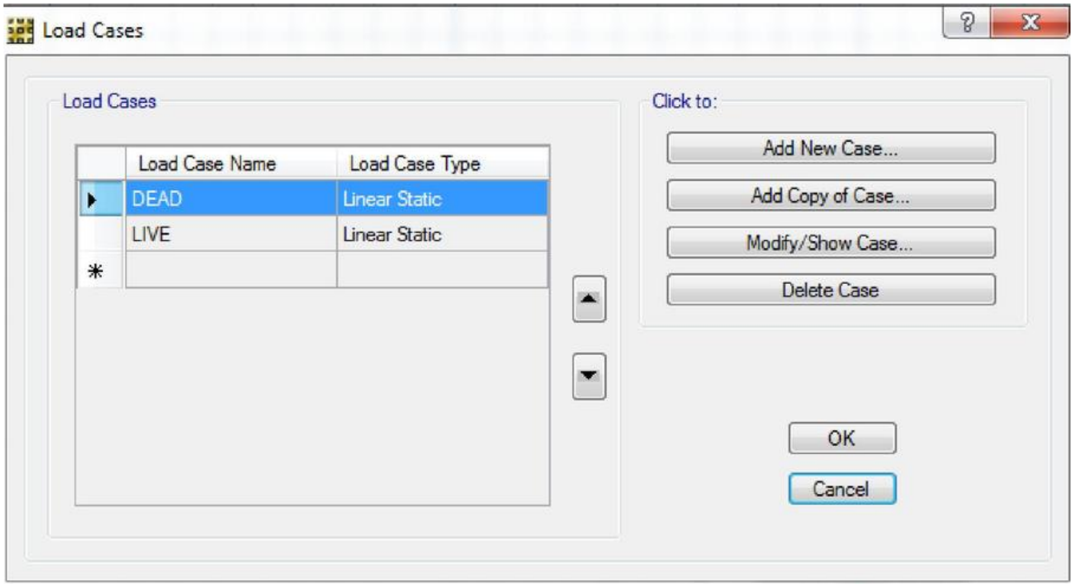


Figure 127

The load patterns which were defined earlier will automatically appear in the list of '**Load Case Name**' of the '**Load Cases**' window. The '**Load Case Type**' column shows the way in which each load pattern will be applied during analysis. If you want to modify this, highlight the load pattern for which you are going to change the load case type and click on the '**Modify/Show Case…**' button. If you click this button, the following window will appear. In this window, you can change the way the load is applied from the '**Load Case Type**' box. The way the structure responds can also be selected from the '**Analysis Type**' box. This problem '**Static**' is for the '**Load Case Type**' and '**Linear**' is selected for the '**Analysis Type**' since the load is static and the foundation responds linearly. The scale factor for the dead load in the '**Loads Applied**' box will be left as one. Press '**OK**' and exit the window.

The load case type for the live load should also be 'Linear Static'. Otherwise, it should be changed by clicking the '**Modify/Show Case…**' button to linear static case. If both the load case types are as desired click '**OK**' and exit the '**Load Cases**' window.

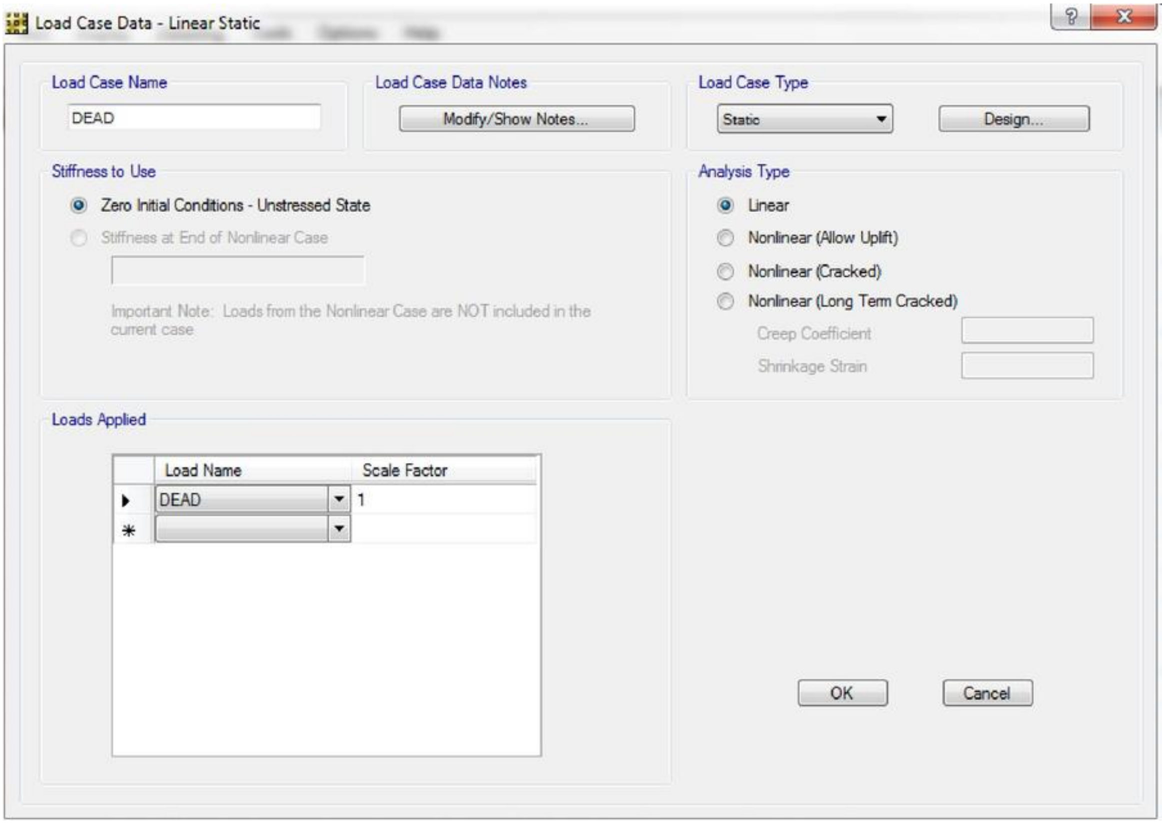


Figure 128

Definition of the load combinations will be the next step. Two load combinations will be considered in this example for ultimate limit state and serviceability limit state. To define load combinations, go to '**Define**' menu and click on '**Load Combinations…**' and the following window pops up.

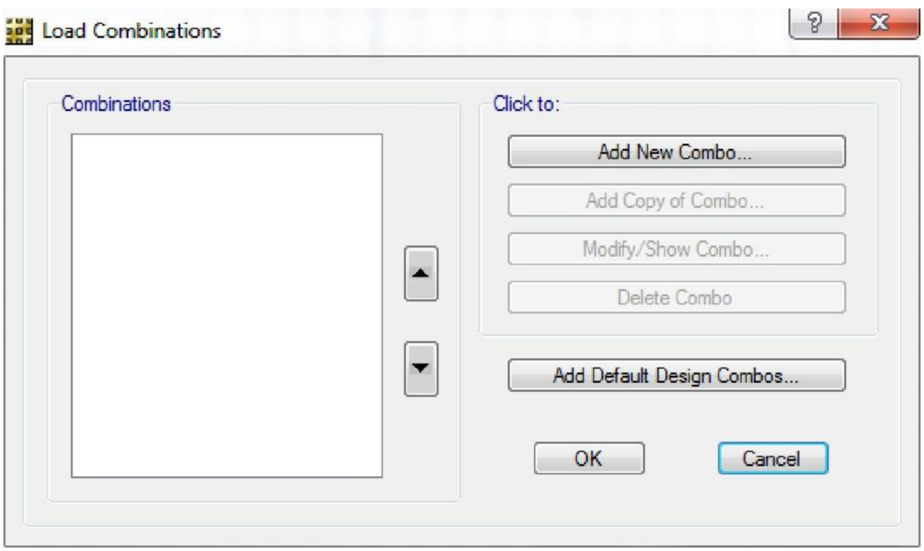


Figure 129

New load combinations will be added through the '**Add New Combo…**' button. Here, two load combinations will be added; one for the ultimate limit state the other for the serviceability limit state. When the '**Add New Combo…**' button is clicked, the following window appears.

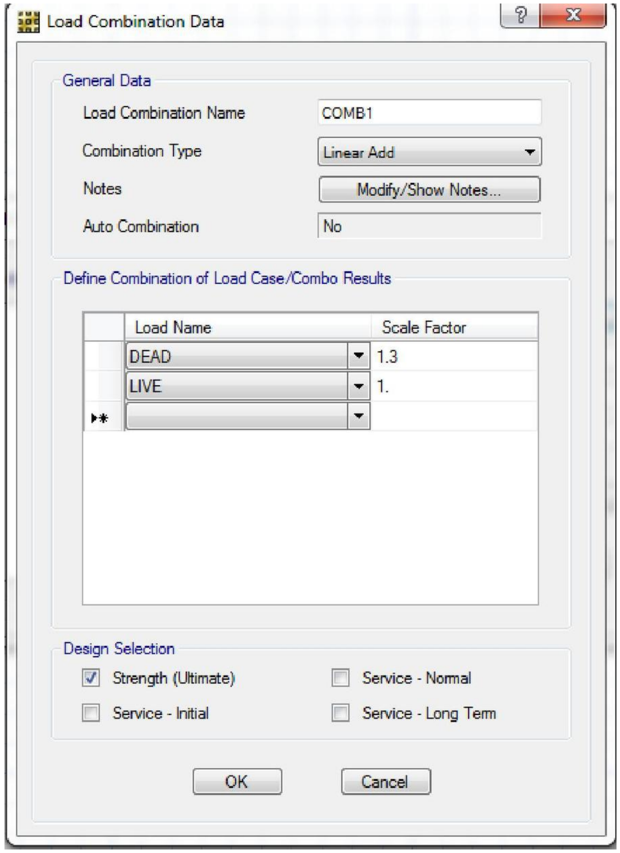


Figure 130

In the '**Load Combination Data**' window, any name can be given for the load combination. The '**Combination Type**' should be set to '**Linear Add**' as the component loads (dead and live) will be added linearly. However, there are also other options from the drop down menu in front of '**Combination Type**'. In the window shown in Fig. 130, the two loads (dead and live) should be activated below the '**Load Name**' column of the '**Define Combination of Load Case/Combo Results**' box. The values in the '**Scale Factor**' column correspond to the partial safety factors for load for the failure mode under consideration. These partial safety factors are specified in the design code of your country. In the design code of my country, the partial safety factor for dead loads for ultimate limit sate case is 1.3 for the cases where there are only dead and live loads. For such load patterns, the partial safety factor for live loads for ultimate limit state case is 1.6. In this particular problem, the loads are already factored and for convenience, they are defined as live loads. The only un-factored load is the self-weight which will be considered as dead load. The self-weight is considered as a dead load while defining the load patterns by making the self-weight multiplier to be equal to one. Thus, enter a value of 1.3 in front of '**DEAD**' and 1 in front of '**LIVE**' in '**Scale Factor**' column. The failure condition which is being under consideration can be defined by selecting and deselecting the check boxes in the '**Design Selection**' box. Since the above scale factors are for the ultimate limit state, check on '**Strength (Ultimate)**'.

STEP 6: Drawing Actual Column Dimensions and Design Strips

The column shape and dimensions can be changed at this stage. The column is shown at the center of the plan view in Fig.118 with somewhat darker color. Since a single column dimension is entered and since, in this problem, there are two types of column sizes, the column dimensions should be revised. The dimension of the second column should be adjusted to 350X350mm.

To change the dimensions of the second column, the existing column should be deleted and another column with the right dimension should be drawn. So, carefully select the second column. At the bottom left corner of the window, it should display, '**1 Areas, 4 Edges selected**'. Otherwise, you should press the '**Esc**' key and select again. Once, the column is selected, delete it using the '**Delete**' key.

To draw a new column of actual dimensions, go to the '**Draw**' menu and click on '**Quick Draw Slabs/Areas Around Points**' or simply click on the equivalent icon  from the left hand side toolbar. After the click, the following window will be displayed.

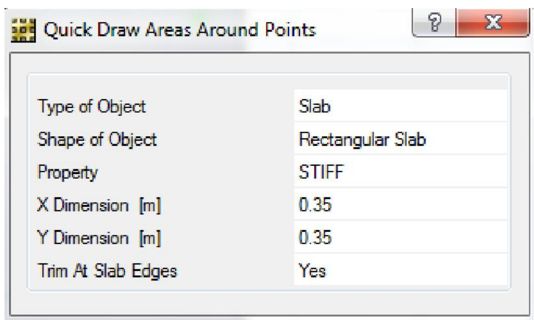


Figure 131

In this window, the '**Shape of Object**' should be '**Rectangular Slab**' and set the '**Property**' to '**STIFF**' as we will be drawing a foundation column. The dimension of the column are 0.35m in both x-direction and y-directions. Thus, set the '**X Dimension [m]**' and '**Y Dimension [m]**' values to 0.35. With great care, move the cursor to the center of the second column and when you see a small red cross, make one left click and close the '**Quick Draw Areas Around Points**' window.

Design strips determine the way in which different quantities related to the reinforcement calculation are calculated. Forces are integrated across the design strips. Thus, the larger the width of coverage of the design strips within the given structure, the higher will be the calculated values of the bending moments and shear forces. Thus, an optimum width of strip is required compromising the safety and economical requirements. The width of the design strip will be specified in the design code. According to the code of my country, the width of column strips for combined foundations should extend up to a distance of 0.5 times the depth from the face of the support in the direction of the line connecting the two columns (x-direction). The area between the column strips in y-direction should be covered with middle strips. In the transverse direction (x-direction), the whole length should be covered with a column strip. These design strips in X and Y direction are usually defined in SAFE software as layer A and layer B. Thus, the column strip at the first column in y-direction will have the following dimensions when the strip is drawn from bottom to top: '**Start Width Left [m]**' = 0.15m, '**Start Width Right [m]**' = 0.15+0.5/2=0.40m, '**End Width Left [m]**' = 0.15m, '**End Width Right [m]**' = 0.15+0.5/2=0.40m.

For combined footings, the existing design strips drawn by the program itself can be used. You can display the design strips by setting the display options by clicking on '**Set Display Options…**' from the '**View**' menu or by simultaneously clicking on '**Ctrl**' and '**W**' keys or by just clicking on the set display options icon  from the tool bar below the menu bar. This results in the following window:

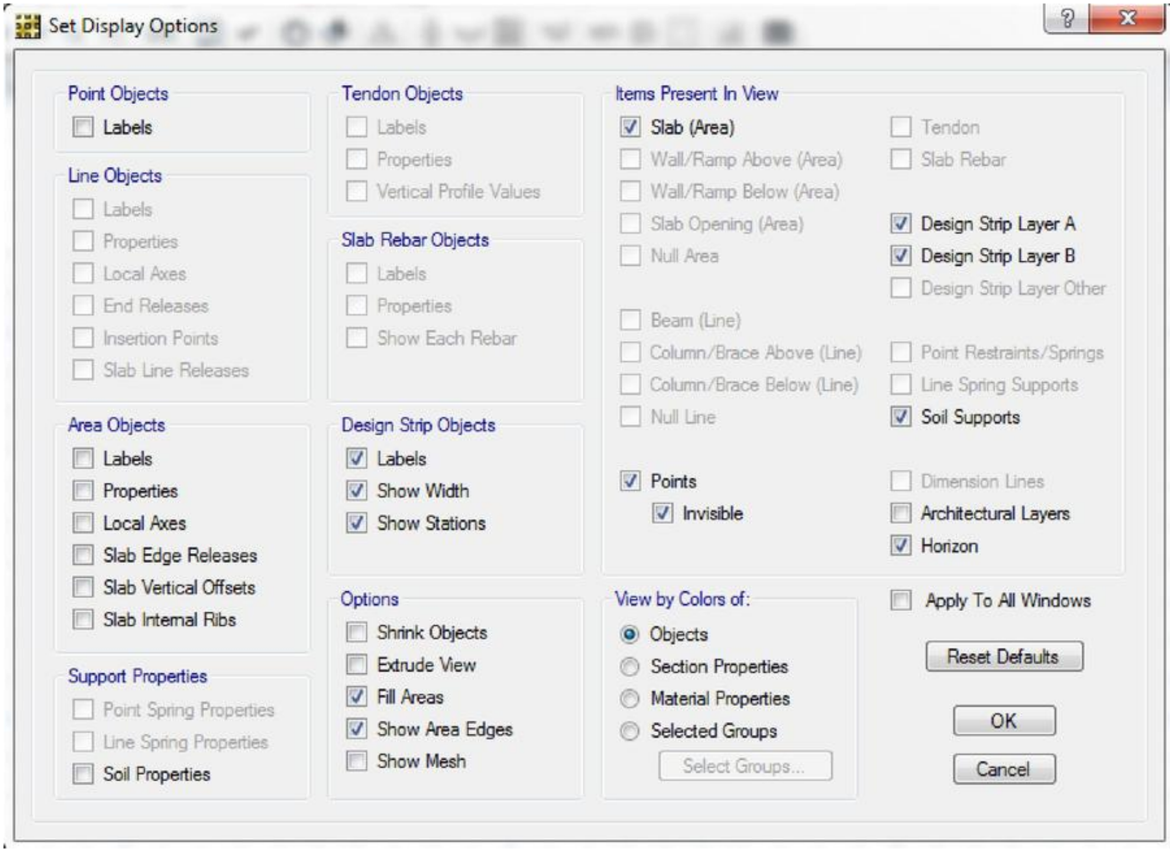


Figure 132

Then, check on ‘**Labels**’, ‘**Show Width**’ and ‘**Show Stations**’ in the ‘**Design Strip Objects**’ box and check both ‘**Design Strip Layer A**’ and ‘**Design Strip Layer B**’ and press ‘**OK**’ and the following window appears displaying the design strips in the two directions.

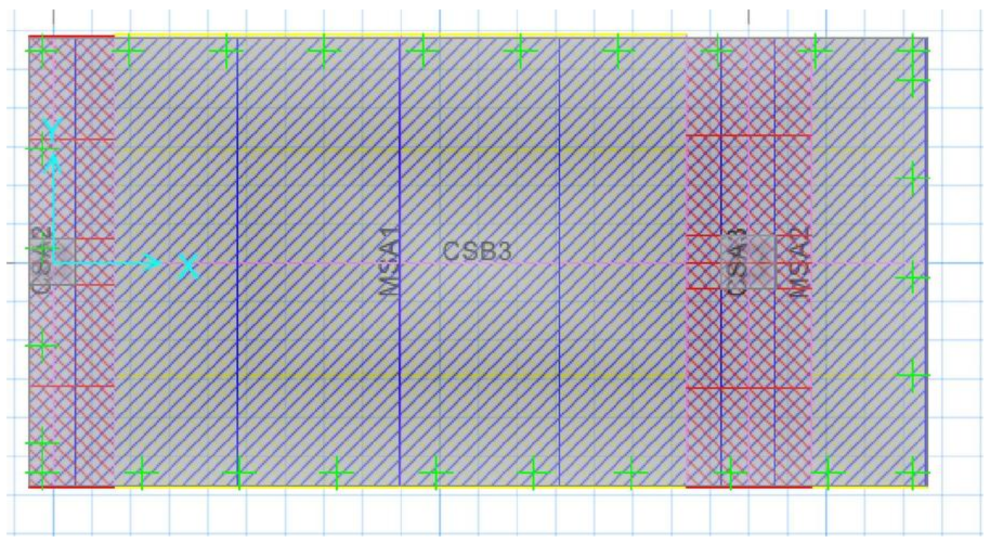


Figure 133

STEP 7: Assigning Slab Data, Support Data and Load Data

The slab data, support data, and load data which are defined in the previous steps should be assigned to the corresponding structural component. The way to do this is first to select the component and next to assign the slab property, support property or loads accordingly. The selection can be done by clicking on the component from the plan view or through the 'Select' menu. The latter option assures that the component is selected exactly as selection by clicking may result in incorrect selection. Thus, all the selections here will be done from the '**Select**' menu. The assignments will be discussed as follows:

f. *Assigning slab data to the footing*
Select the footing through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**'Slab Properties…**'. Then select '**Footing**' and press '**OK**'. Then assign the footing property through the following strings of commands '**Assign**'>'**Slab Data**'>'**Properties…**'. Then select '**FOOTING**' and press '**OK**'.

g. *Assigning slab data to the foundation column*
Select the foundation column through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**'Slab Properties…**'. Then select '**Stiff**' and press '**OK**'. Then assign the footing property through the following strings of commands '**Assign**'>'**Slab Data**'>'**Properties…**'. Then select '**STIFF**' and press '**OK**'.

h. *Assigning support data to the footing*
Select the footing through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**'Slab Properties…**'. Then select '**Footing**' and press '**OK**'. Then assign the footing property through the following strings of commands '**Assign**'>'**Support Data**'>'**Soil Properties…**'. Then select '**SOIL**' and press '**OK**'.

i. *Assigning reinforcement data to the design strips*
Select each design strip through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**Design Strip Layers…**'. Then select '**A**' or '**B**' (one at a time) and press '**OK**'. When you right click on the selected strip layer, the '**Slab-Type Area Object Information**' window pops up. In the '**Design**' tab of this window, set the '**Rebar Material**' to '**S300**' and press '**OK**'. Do this for both strips.

j. *Assigning load on the foundation column*

The loads are already assigned on the foundation column. However the dimension of the second column should be modified. To do this, right click at the center of the second column. In the '**Loads**' tab of the appearing window, click on '**Assign Loads**' button then select '**Force Loads**' then press '**OK**'.

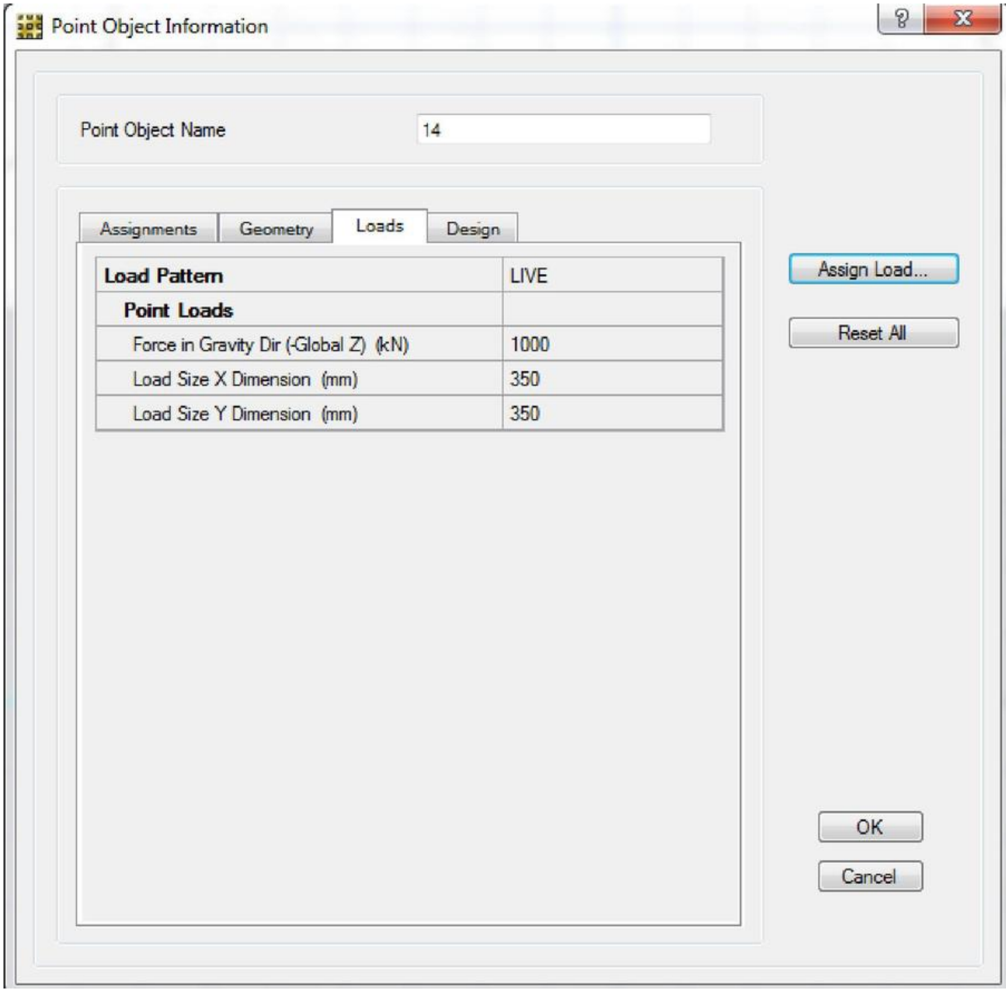


Figure 134

Then select '**LIVE**', enter its value accordingly and in the right direction (axis), then select '**Replace to Existing Loads**' and press '**OK**'. While doing this, the foundation column dimensions should be entered in the '**Size of Load for Punching Shear**' box of '**Point Loads**' window. In this example, since the live load is 1000kN for the second column, this should be entered in the Gravity Direction and the values of '**Load Size in X Dimension (mm)**' and '**Load Size in Y Dimension (mm)**' should both be set to 350.

STEP 8: Running the Analysis

After this, the analysis can be run. But, make sure that the footing and the foundation column are assigned with the correct rebar material. To do this, right click anywhere in the plan view of the footing and the '**Slab-Type Area Object Information**' window will pop-up.

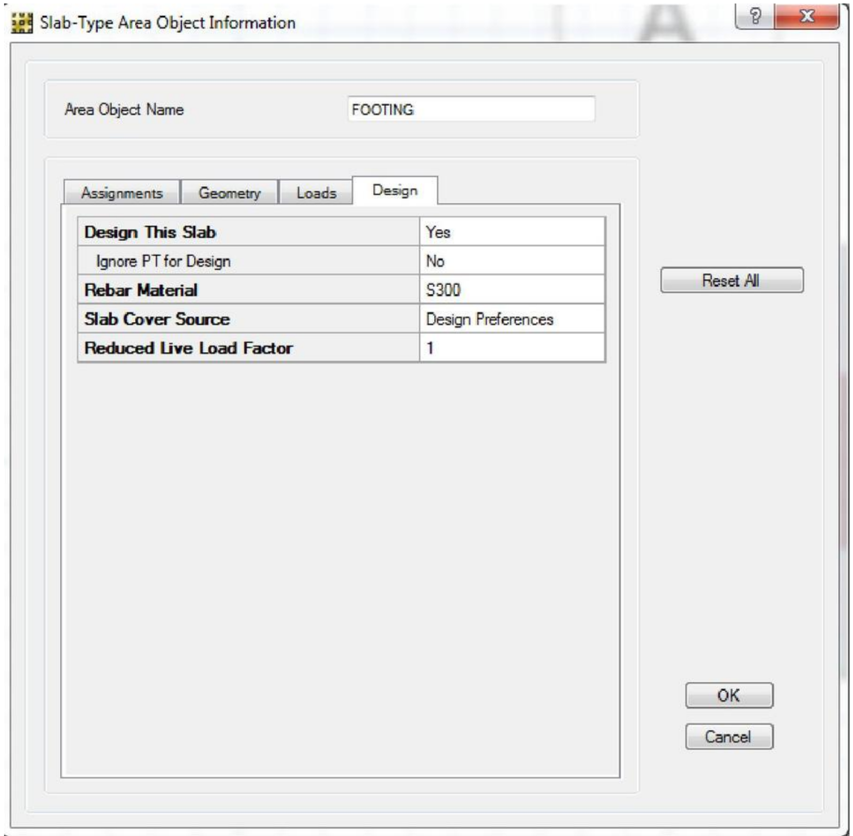


Figure 135

In the '**Design**' tab of this window, set the value of '**Rebar Material**' to '**S300**'. Do this for the column foundation, and both design strips as well. After this, go to the '**Run**' menu and click on the '**Run Analysis & Design**' command or simply click on the '**F5**' key. When you do this, you will be prompted to save the model, if you haven't already don this. When you save the model, the following window showing the displacement of the soil in a banded figure will be displayed.

STEP 9: Displaying the Output

After the analysis is run the following output will be displayed on the plan view window.

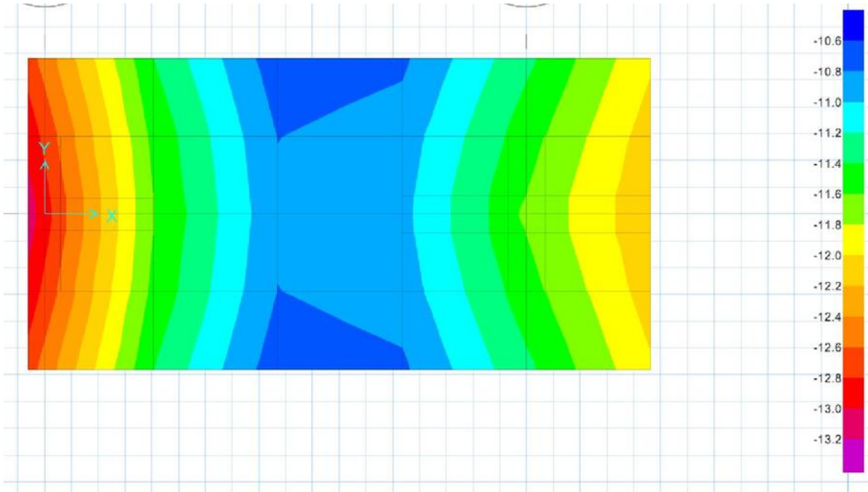


Figure 136

To display the punching shear ratio, go to the '**Display**' menu and click on '**Show Punching Shear Design**'. After this, the punching shear ratio will be displayed in the plan view around the foundation column as in the following figure.

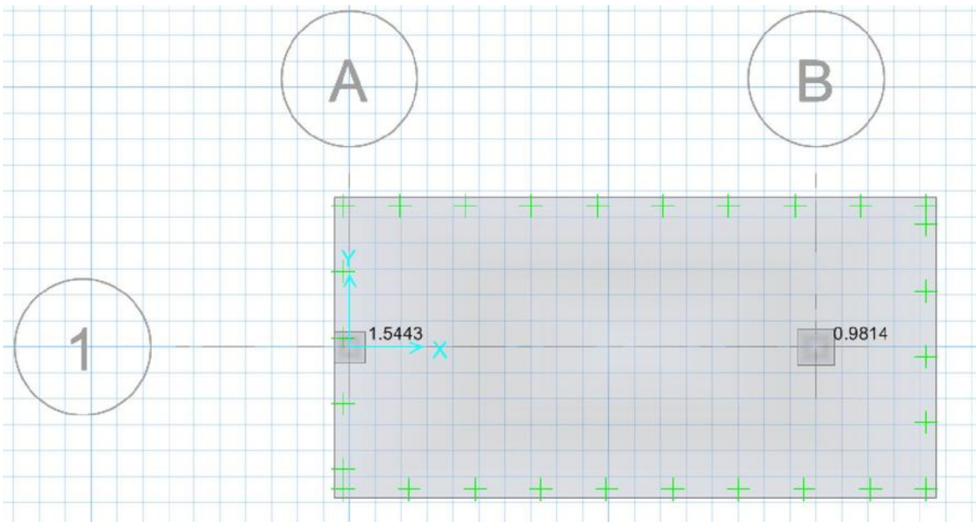


Figure 137

As can be noticed from the figure, the punching shear ratio is 1.5443 for the first column and 0.9814 for the second column. Generally, a punching shear ratio less than one indicates the concrete thickness is adequate to resist punching shear and a value greater than one indicates that the punching shear capacity is exceeded somewhere along the critical section. For economical design, it is recommended to keep the punching shear ratio between 0.95 and 1 as very small values of punching shear ratio means excess concrete thickness is used. However, if the punching shear ratio is greater than one, the thickness of the concrete should be increased and the foundation should be re-designed. In this problem, since the punching shear ratio at column 1 is greater than one, the footing depth should be revised. However, since this is just an illustration of how to design foundations with SAFE, we will proceed as if the depth of the footing was adequate for punching shear. A detailed quantitative description of the foundation design can also be obtained by right clicking on the plan view shown in Fig 137 as shown below. Several trial may be made by zooming in and out to get the quantitative description.

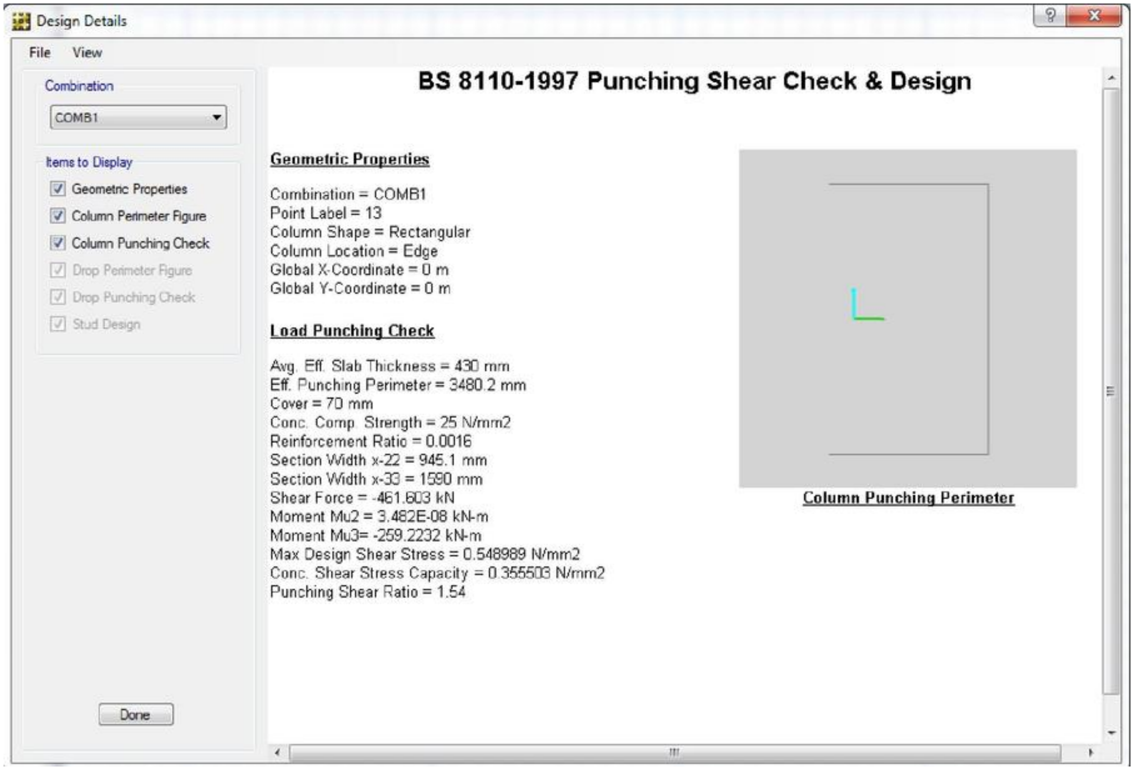


Figure 138

Through the '**Display**' menu, relevant quantities can be displayed on the screen. For instance, the '**Display**'>'**Show Strip Forces**' command or by simply clicking the '**F8**' key, the following window will be displayed.

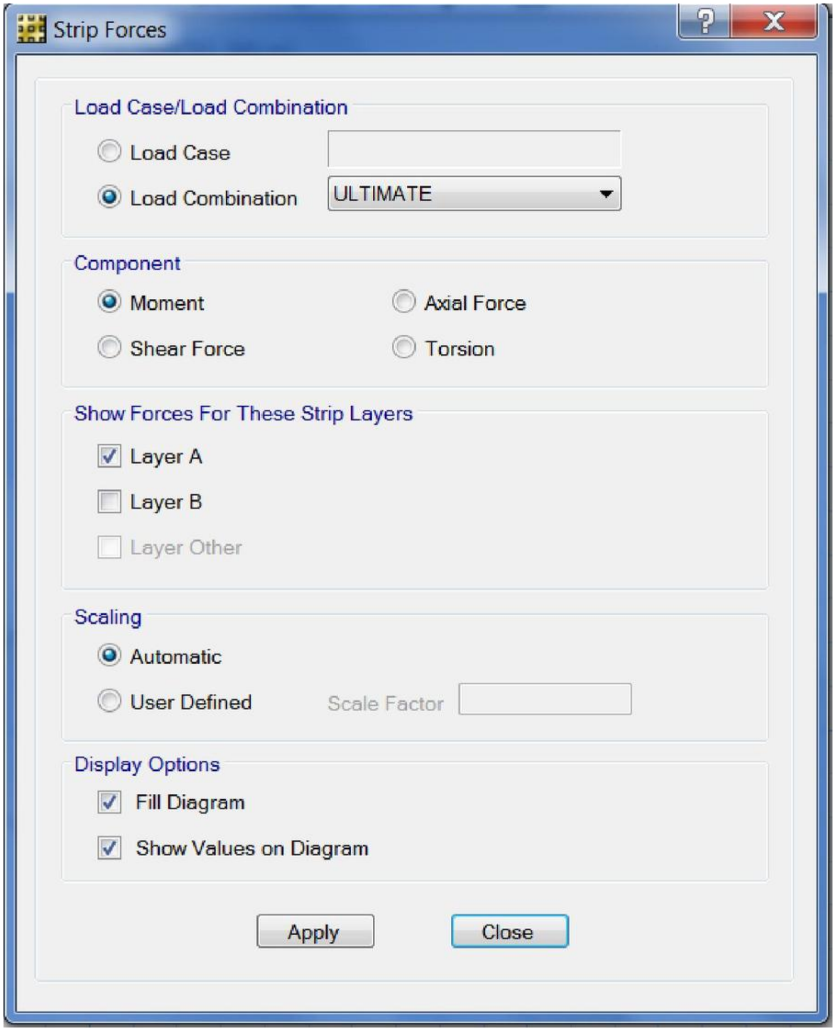


Figure 139

In this window, the '**Load Case/Load Combination**' box provides with radio buttons to select for which particular load case or load combination that we want to display the output. The '**Component**' box contains four radio buttons to select which quantity to display. The '**Show Forces For These Strip Layers**' box allows us to select the strip layer for which the quantity is displayed. Both strip layers can be selected at the same time. From the '**Scaling**' box, we can select whether automatic scaling or user defined scaling is used while displaying the diagram. The '**Display Options**' box allows us to fill or not to fill the diagram and to display or not to display the values on the diagram. For the preferences shown in Fig. 30, the following diagram will be displayed.

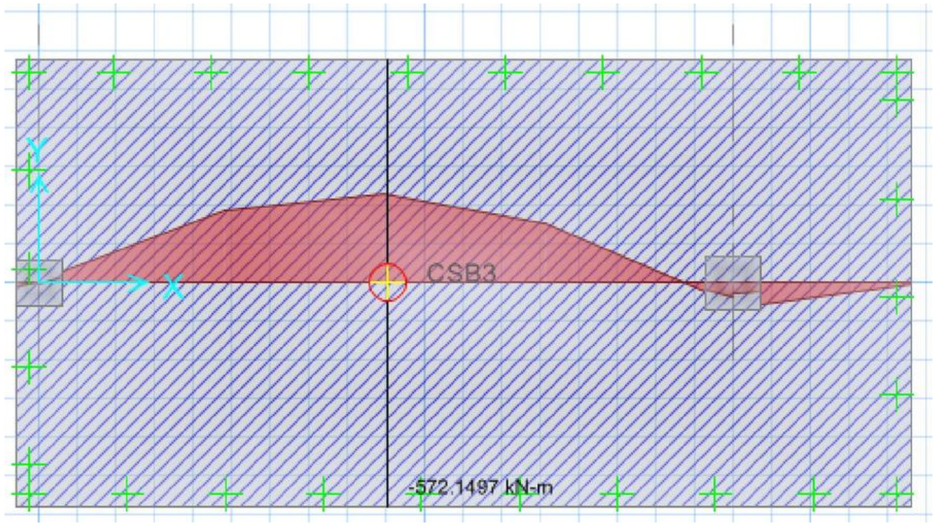


Figure 140

The '**Display**'>**'Show Slab Design…**' command results in the window shown in Fig 141. In this window, several options can be set in order to display the footing design in the way we wanted to. The '**Choose Display Type**' box allows us to select the '**Design Basis**' between '**Strip Based**' and '**Finite Element Based**'. Unless some differences in the way the design is displayed, there is no difference in the amount of reinforcement between these selections. Through the '**Display Type**' in the same box, it can be selected whether to display flexural reinforcement or shear reinforcement. This box also allows us to impose or not to impose minimum reinforcement during the design. The '**Rebar Location Shown**' box allows us to select which reinforcement, top or bottom or both, to be displayed. The '**Reinforcing Display Type**' box allows us to set the manner in which the amount of reinforcement is displayed. The option whether to show the reinforcing envelop diagram and the reinforcing extent can be set by the check boxes in the '**Reinforcing Diagram**' window. The strip layer direction for which the amount of reinforcement is displayed can be chosen from the '**Choose Strip Direction**' box. The '**Display Options**' box allows us whether to display output in filled diagram or not and whether the values at controlling stations will be displayed or not. If we want to display the amount of reinforcement above some specified reinforcement bar area or spacing, we can use the options in the '**Show Rebar Above Specified Value**' box. When the '**Typical Uniform Reinforcing Specified Below**' radio button is selected, the **'Typical Uniform Reinforcing'** box get activated. In this box, we can set a specific value above which the reinforcement amount will be displayed. The reinforcement diagram output, for the options set in Fig. 141, will be shown below in fig. 142.

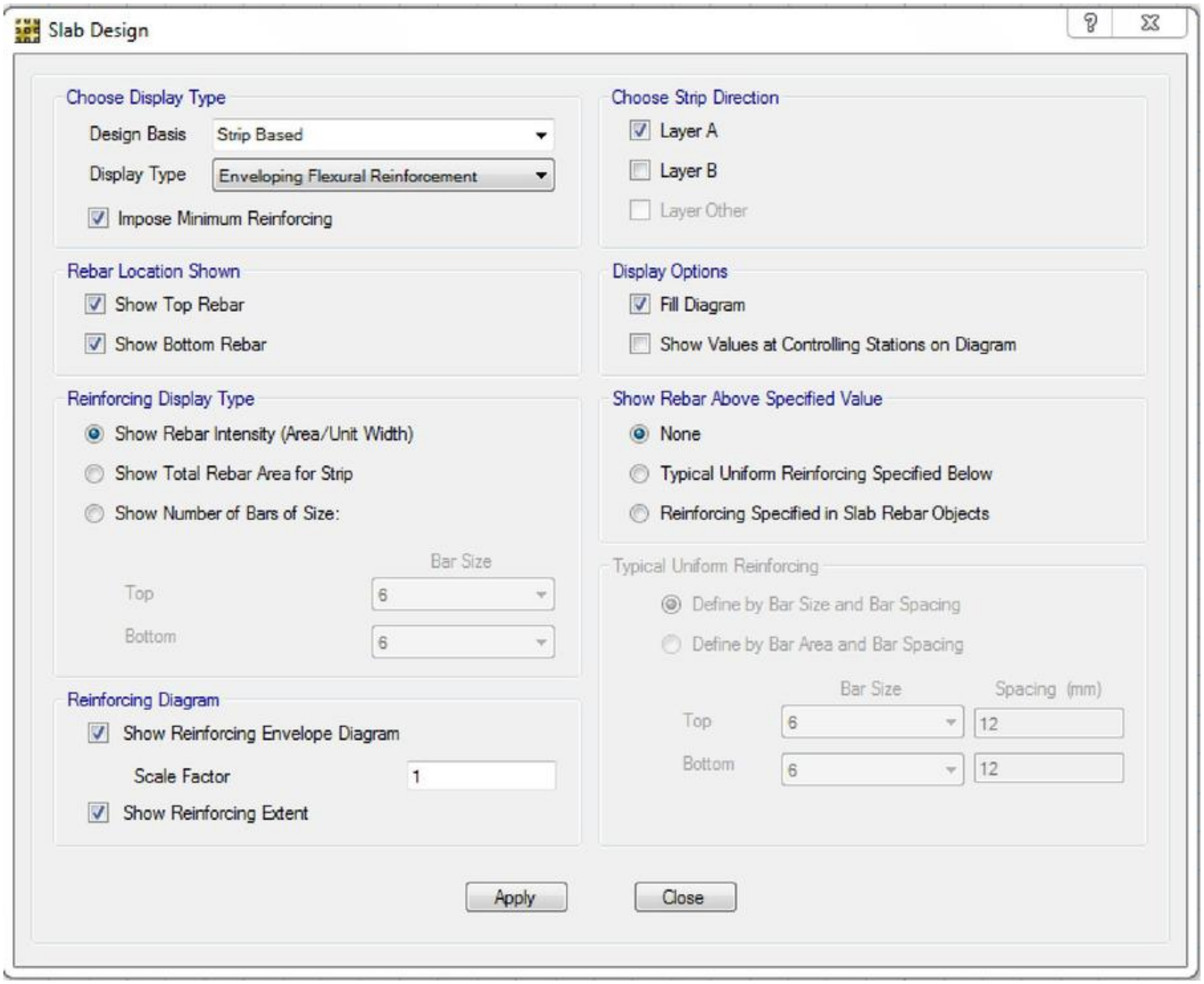


Figure 141

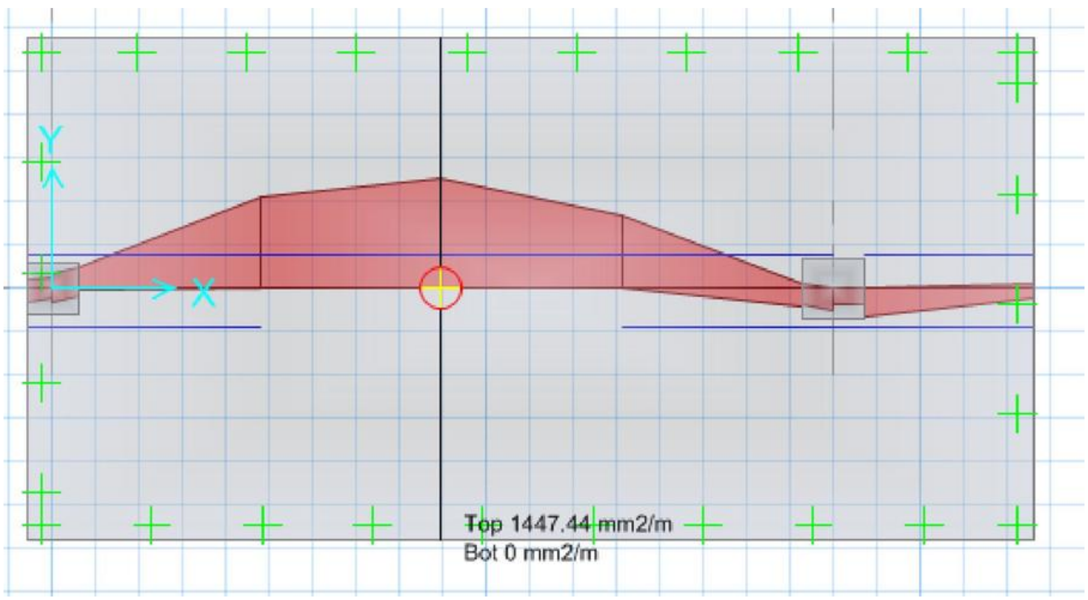


Figure 142

As can be noticed from the footing design diagram in fig. 142, the top reinforcement across strip layer A is 1834mm^2/m and 0mm^2/m for bottom reinforcement at the location indicated by the vertical line.

The design outputs can also be displayed in tabular format by clicking on the '**Show Tables…**' menu item from the '**Display**' menu or by just clicking on the equivalent icon  from the tool bar below the menu bar and the following window will pop up.

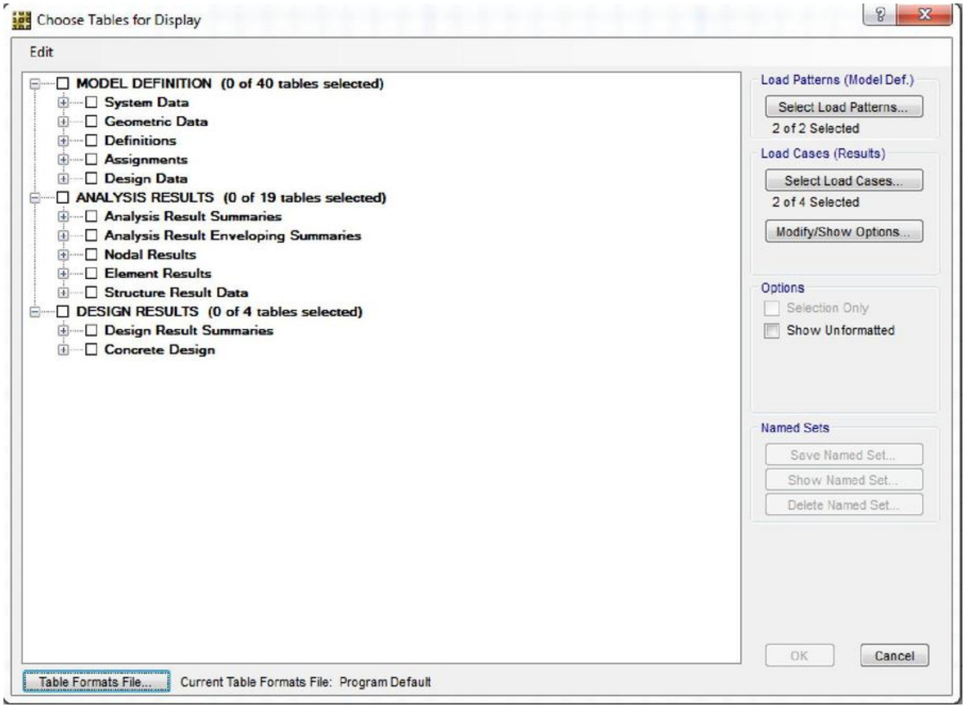


Figure 143

In this window, we can select any of the model definitions or analysis results or design results and press '**OK**' to display the quantity which we want to have a look at. By using the right hand side buttons in the window, the load patterns and the load cases can be selected.

STEP 10: Detailing

After running the analysis and after checking that the results are reasonable, the detailing will be done. However, before running the detailing, the detailing preferences can be set from the '**Detailing**' menu. From the '**Detailing Preferences…**', the likes of dimensional units and material quantity units can be selected. From '**Slab/Mat Detailing Preferences…**', the likes of rebar curtailment options, the rebar detailing options, rebar selection rules and preferred

rebar sizes can be selected. The '**Drawing Sheet Set-up…**' menu allows us to set-up the contents of the drawing sheet. The '**Drawing Format Properties…**' allows us to set some formats in which the output displayed.

To run the detailing, go to '**Run**' menu and click on '**Run Detailing…**' or simultaneously press '**Shift**' and '**F5**' keys or just click on the run detailing icon  from the tool bar just below the menu bar. Then, the '**Run Detailing Options**' window pops up so that we set the detailing options. Set the detailing options which you want and click '**OK**'.

Once the detailing is run, the detailing can be displayed. The detailing display options can be best accessed from the '**Model Explorer**'. When expanded in full, the '**Detailing**' tab of the model explorer looks like:

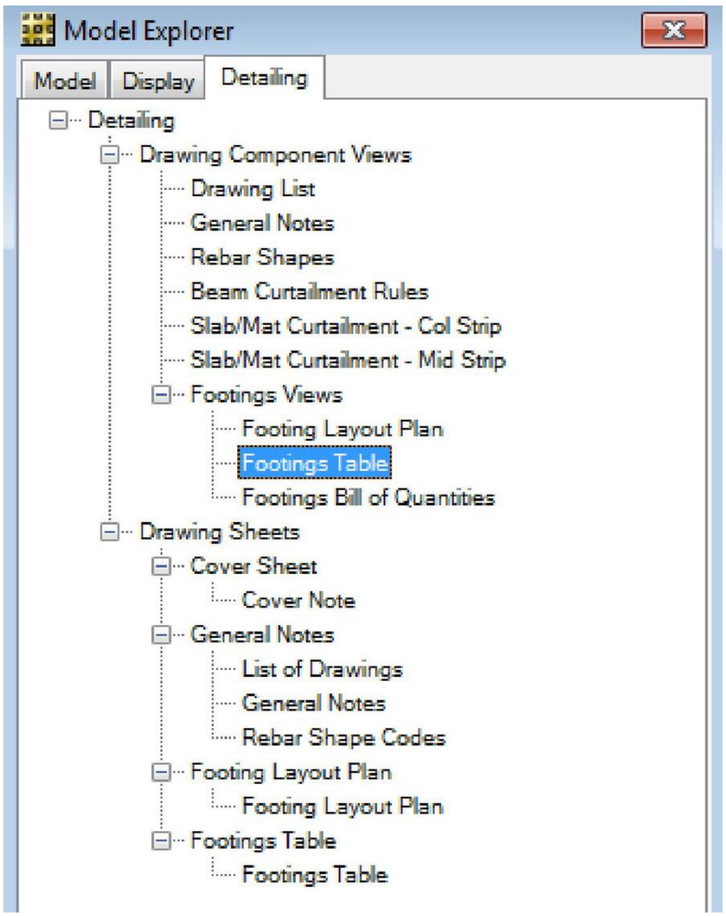


Figure 144

By clicking on any of the options from the detailing tab, a desired detailing can be displayed. For instance, by double clicking on the '**Footing Table**', the following detail of footing can be shown.

FOOTINGS TABLE

SR. NO.	TYPE	NOS	LX	LY	T	REBARS-A	REBARS-B
1	F1	1	5.800 M	2.900 M	0.500 M	15-10	13-10

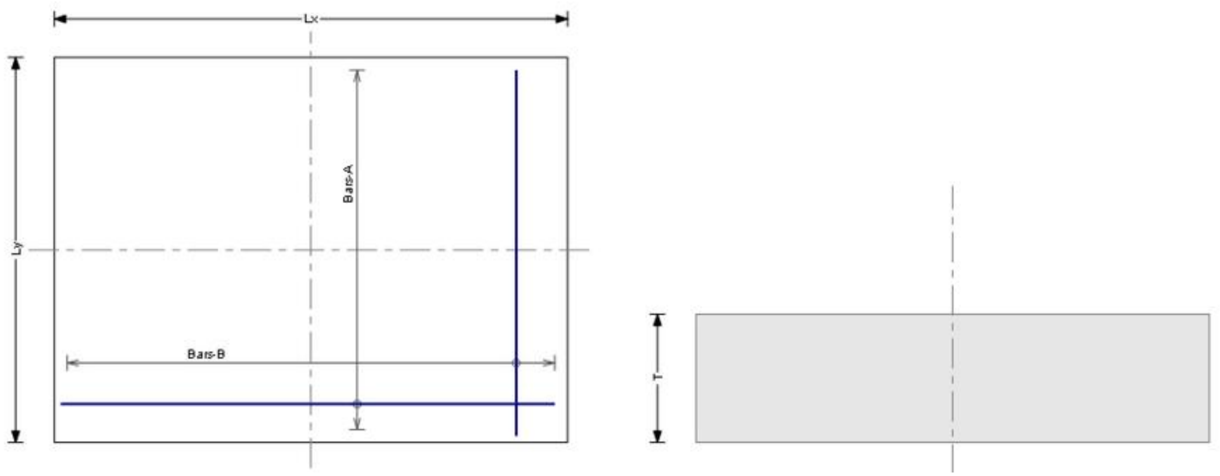


Figure 145

In this detail, the diameter of the reinforcement which is used is 10mm. If you want to change the diameter….

Apart from this detail, other details can also be shown.

STEP 11: Reporting

The last step of foundation design is reporting. Before creating the report, the report preferences should be set up. To do this, go to the 'File' menu and click on 'Report Set-up…' and the following window pops up.

In this '**Report Setup Data**' window, the user preferences regarding the reporting such as the report output type, the report page orientation and the report items can be set along with the load patterns and load combinations. Once the preference is set, the report can be created by clicking on '**Create Report**' command in the '**File**' menu. The '**Advanced Report Writer**' command in the same menu can be used to set some advanced reporting formats.

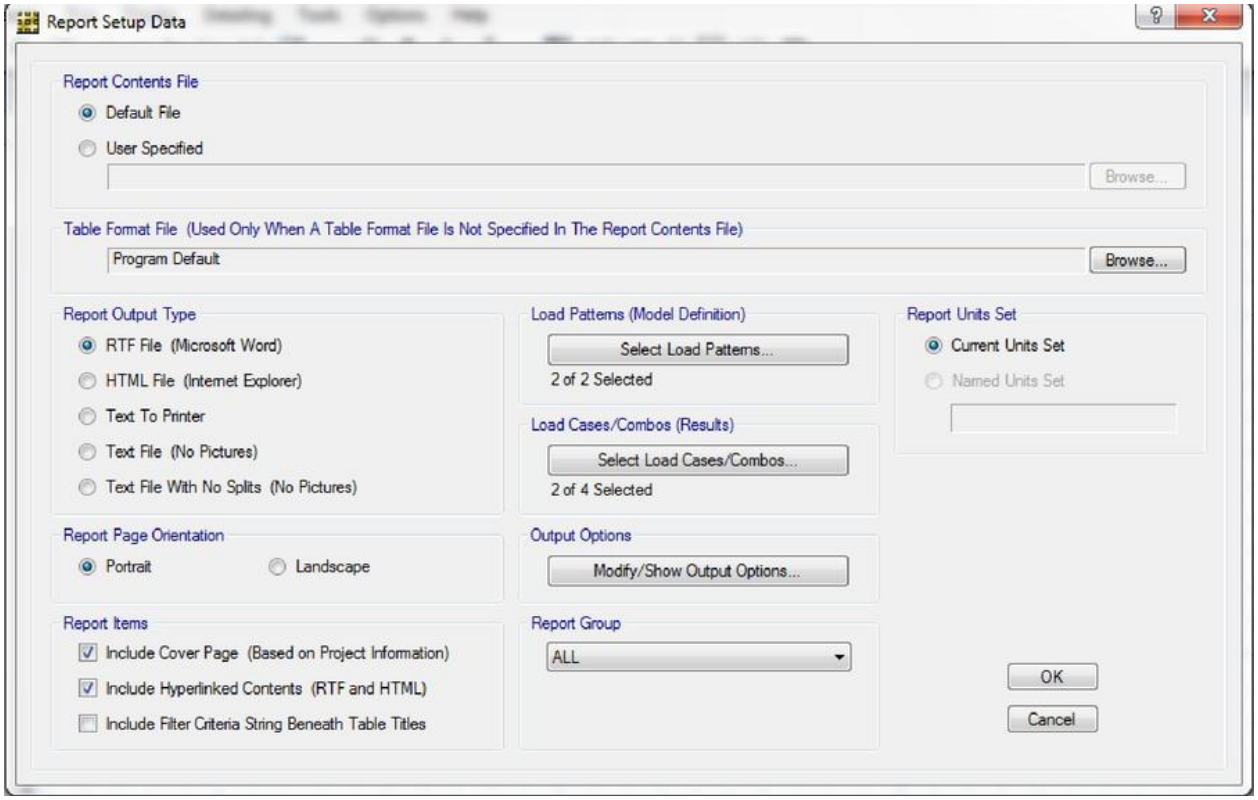
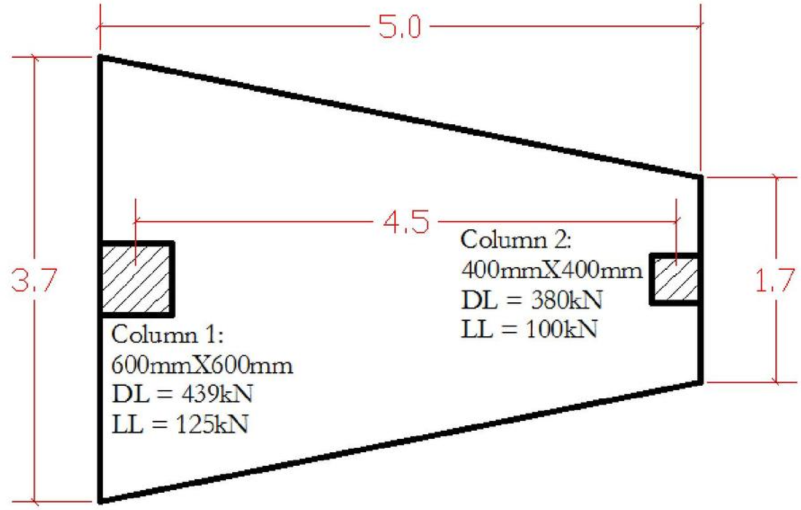


Figure 146

This concludes the tutorial for the design of single footing using the built-in model.

3.2. Using Grids

Design a trapezoidal combined footing shown in the figure below



- *Ultimate bearing capacity of the soil* : 100kPa
- *Allowable settlement of the soil* : 10mm
- *Grade of concrete*: C-30 (30MPa 28-day characteristics cube strength)
- *Grade of reinforcement bar (rebar)*: S-400 (400MPa characteristics yield strength)
- *Overall thickness of the foundation*: 600mm
- *Concrete cover* : 50mm

STEP 1: Creating the Model

To create a footing model by using a grid, there are three ways. Go to '**File'** menu and click on '**New Model …'** or just click on the icon  . This icon is the first among the list of icons just below the menu bar or simply click on the '**Ctrl'** and '**N'** keys simultaneously (**Ctrl+N**).

Performing one of the above three ways results in the popping-up of the following **'New Model Initialization'** window.

This window consists of two parts: the '**Design Data**' part and the '**Initial Model**' part. In the '**Design Data**', there are '**Design Code**', '**Design Preferences**', '**Project Information**' and '**Units**'.

The '**Design Code**' consists of a list of selected design codes from many countries. Depending on the suitability of the code for your country, you may select one among the list of codes. The design code which follows similar design philosophy to my country code is BS 8110-1997. So, this design code is selected.

After selecting the design code, it is better to first select the units to be used in the design and analysis process. These units can be selected by clicking on the '**Modify/Show**' button in front of '**Units**'. The following window pops up when the button is clicked. To choose metric units, click on '**Metric Defaults**' button. The click results in metric units for best practices to be selected. It can be observed that, even though all the units are metric, they are not consistent. To select a preferred consistent unit, click on the '**Consistent Units**' button. In this '**Units**' window, the decimal places and minimum number of significant digits for any quantity can also be modified.

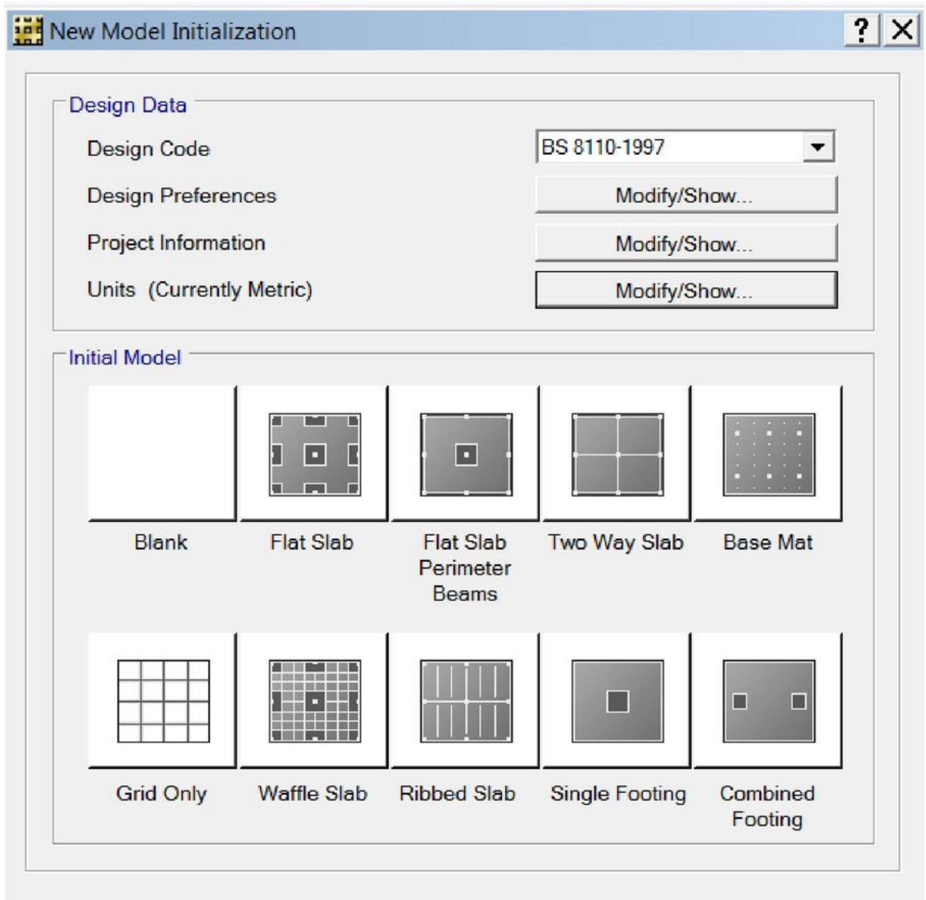


Figure 147

By clicking on the '**Modify/Show**' button in front of the '**Project Information**', information regarding the project, the company and the model can be entered.

By clicking on the '**Modify/Show**' button in front of the '**Design Preferences**', the user preferences regarding the design code, concrete cover for slabs and beams, post-tensioning can be changed. The following **'Design Preferences'** window pops-up when the button is clicked. The '**Min. Cover Slabs**' tab of the window may be of interest for this particular problem as footings will be modelled as slabs. Thus, click on this tab. This is the tab where the concrete cover and preferred rebar size will be entered.

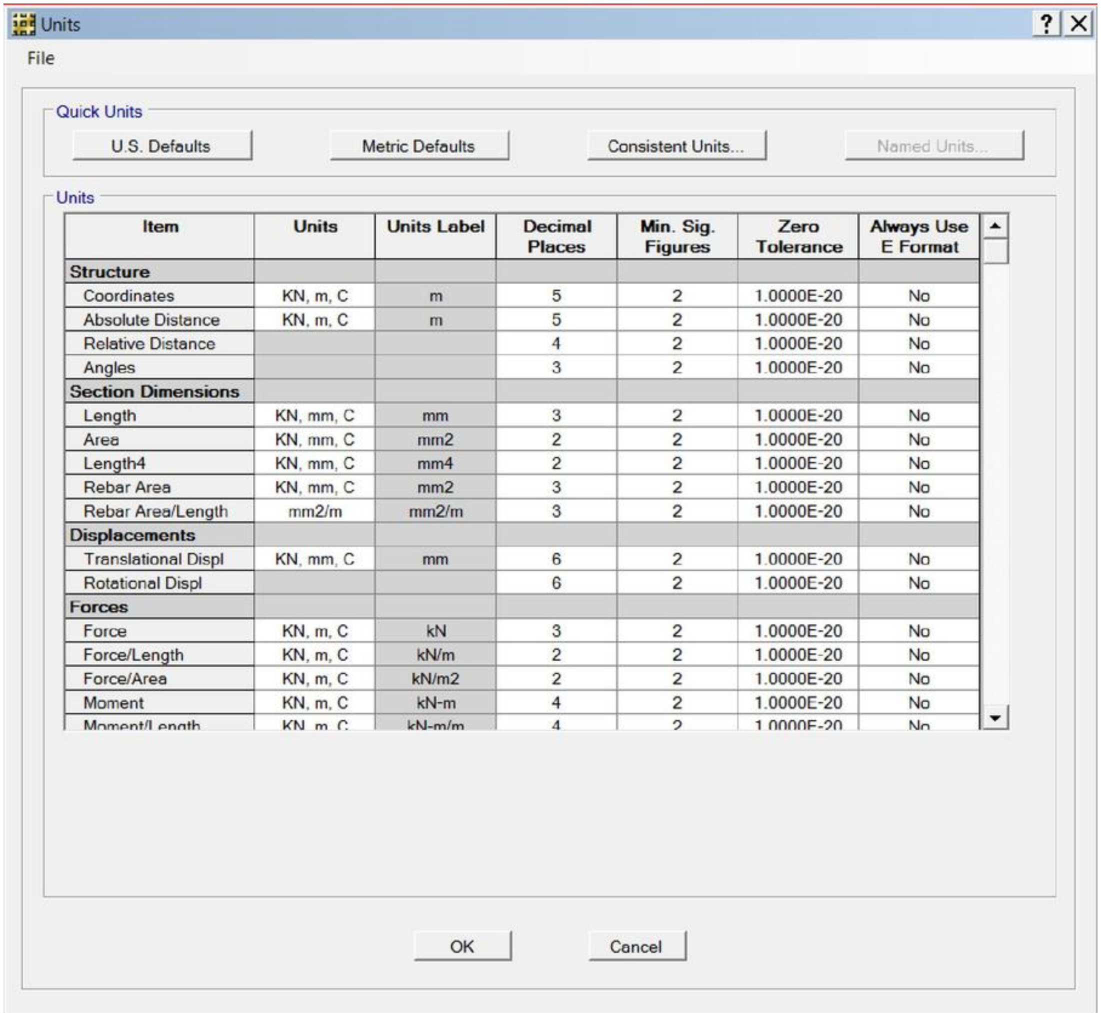


Figure 148

In the '**Min. Cover Slabs**' tab, for '**Non-Prestressed Reinforcement**' , both the '**Clear Cover Top**' and '**Clear Cover Bottom**' should be set to 50mm as the concrete cover in this design problem stated to be 50mm. The '**Preferred Bar Size**' can be set to any reasonable value. Here, the '**Preferred Bar Size**' is set to 20 which is the diameter of the rebar in mm which will be used as the main reinforcement in the foundation. Leave the rest at they are.

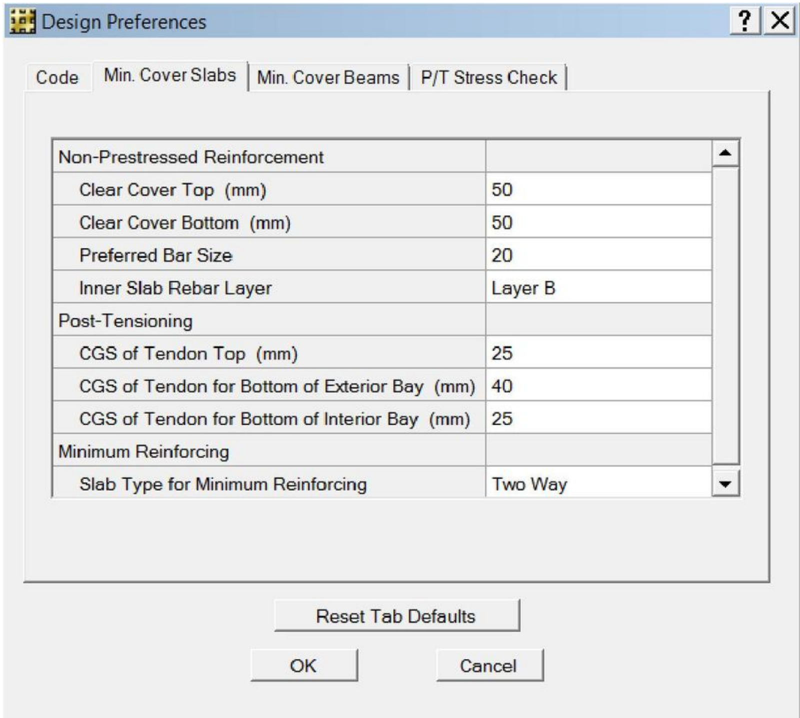


Figure 149

Once everything is set correctly in the '**Design Data**' box, go to the '**Initial Model**' box and select the suitable model. Since an isolated (single) footing will be designed from grids click on the big square button just above '**Grid Only**'. The button itself shows crossing grids. When the button is clicked, the following **'Coordinate System Definition'** window, shown in fig. 150, will pop up.

The footing in this example is trapezoidal with two columns. The grids which will be created will be boundaries of the outer edges of the footing and the outer edges of the foundation column. As a result, there is uneven spacing between the grids and there will be slanting grids. To create such slanting grids with uneven spacing, it is better to create it by clicking the '**Edit Grid**' button. When you click this button, a '**Coordinate System**' window will pop up. In the '**Display Grid Data as**' box, two radio buttons are there to choose between '**Ordinate**' and '**Spacing**'. These are the options how the location of different grids is defined. If the '**Ordinate**' option is selected, the X and Y-coordinates of the grids will be entered. If the '**Spacing**' option is selected, the spacing between the grids will be selected. If the grids are irregular in shape with non-vertical and non-horizontal grids, they can be defined in the '**General Grid Data**' box by entering the XY coordinates of the starting and ending points of the grids. Regular grids can also be defined by this option too. For this particular example, we will use the '**General Grid Data**' box. To use this '**General Grid Data**' box, first delete the existing coordinates from '**X**

Grid Data' and '**Y Grid Data**' by clicking on the left hand side arrow and pressing the '**Delete**' key.

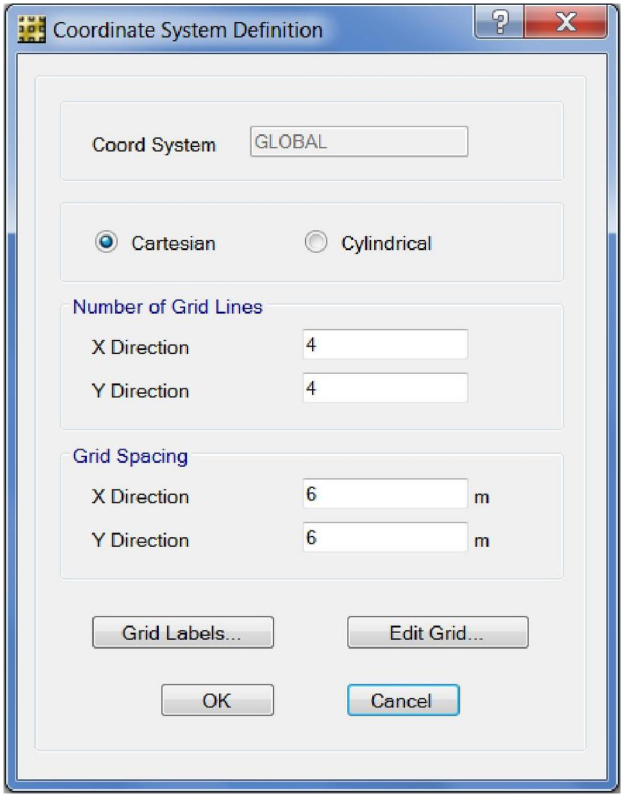


Figure 150

In this example, the values which will be entered are shown in the coordinates of the starting point (X1, Y1) and the coordinates of the ending point (X2, Y2) for all the grid lines in the following table. Here, the bottom left corner of the footing will be considered as the origin of the coordinate system.

Grid ID.	X1	Y1	X2	Y2	Description
1	0	0	5	1	Bottom edge
2	0	1.55	5	1.55	Bottom grid for column1
3	0	2.15	5	2.15	Top grid for column1
4	0	3.7	5	2.7	Top edge
5	0	1.65	5	1.65	Bottom grid for column2
6	0	2.05	5	2.05	Top grid for column 2
A	0	0	0	3.7	Left edge
B	0.6	0	0.6	3.7	Right grid for column1
C	4.6	0	4.6	3.7	Left grid for column2
D	5	0	5	3.7	Right edge

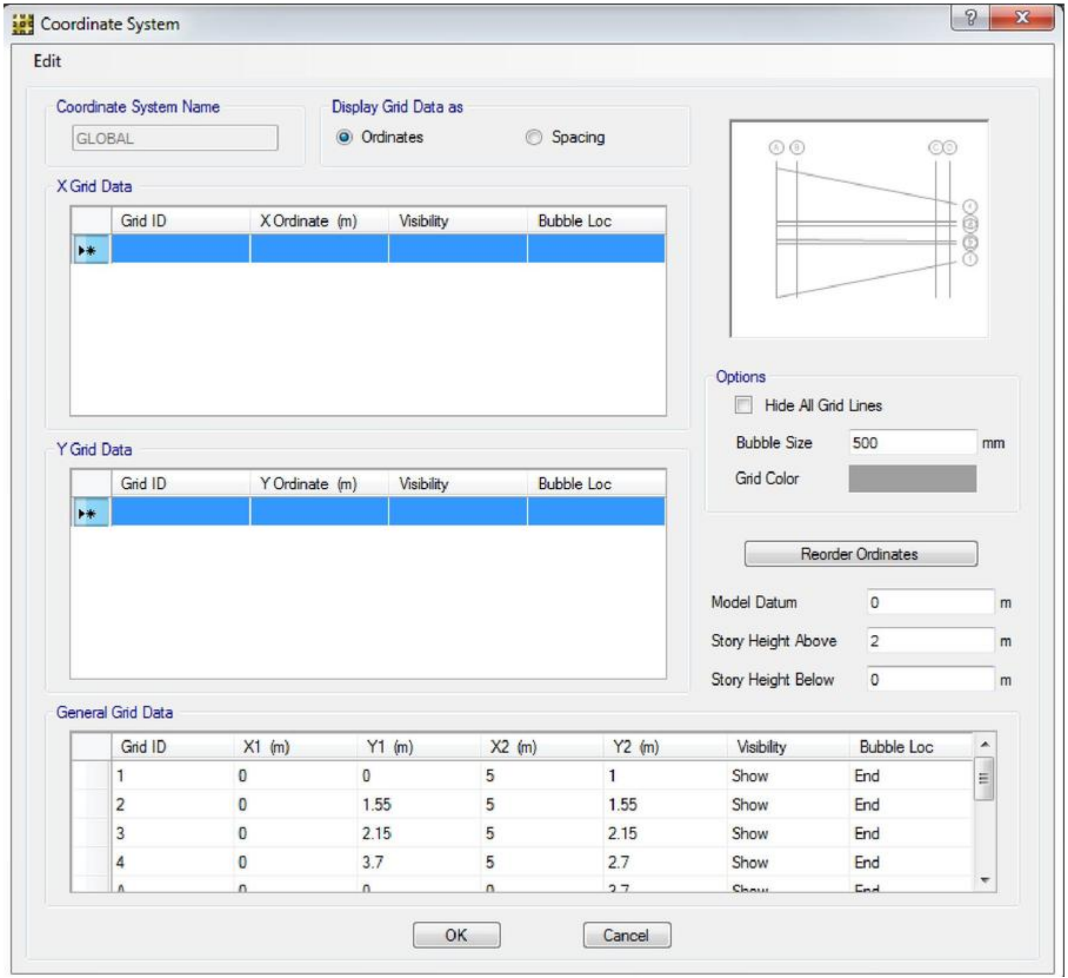


Figure 151

The '**Model Datum**' is the level of the model in z-direction. Any value can be set to this. Set the '**Story Height Above**' value to some reasonable non zero value, say 2, and set the '**Story Height Below**' to 0 as there will be no story below the footing. Then press '**OK**' and the grids will be displayed on the '**Plan View**' window as below:

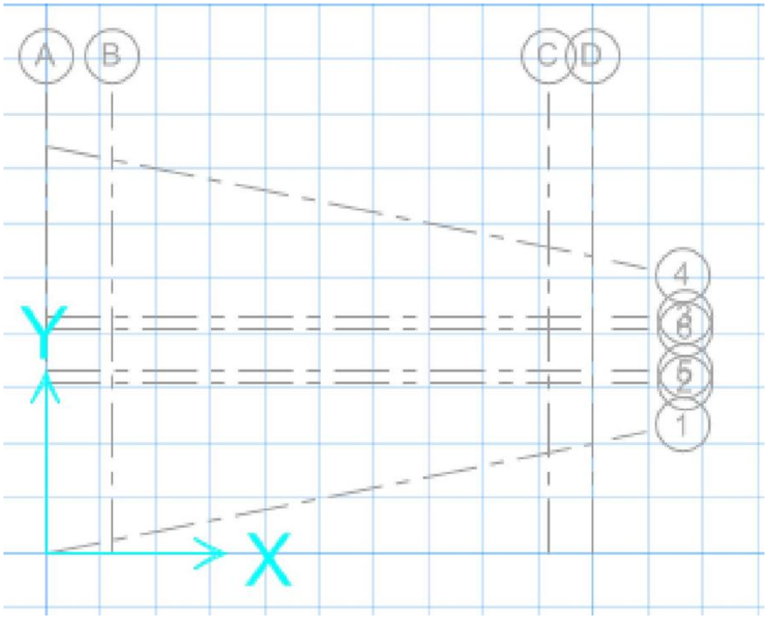


Figure 152

STEP 2: Defining Material Properties

The materials which are involved in the design of the footing should be defined before the analysis. These materials are the concrete, the reinforcement bar (rebar) and the soil support. Thus, these properties will be defined here.

The material definition can be carried out in two ways.

The first one is through the '**Define**' menu in the menu bar. The other one through the model explorer on the left hand side of the home window. To define material properties through the former method, click on '**Define**' menu and again click on '**Materials…**' resulting in the following window depending on prior material definitions.

The list in the '**Materials**' box may not be exactly as it appears in your widow. However, that doesn't bring any change in the outcome of the design process as you can customize this list any time.

The '**Add New Material Quick…**' button allows you to define materials quickly from a list of pre-defined materials. The '**Add New Material**' button allows you to define materials by changing their properties. The '**Add Copy of Material**' button allows you to define a material with same property as an already defined material. The '**Modify/Show Material**' button displays the property of an already defined material with the possibility of modification. The '**Delete Material**' button, when active, deletes a defined material property.

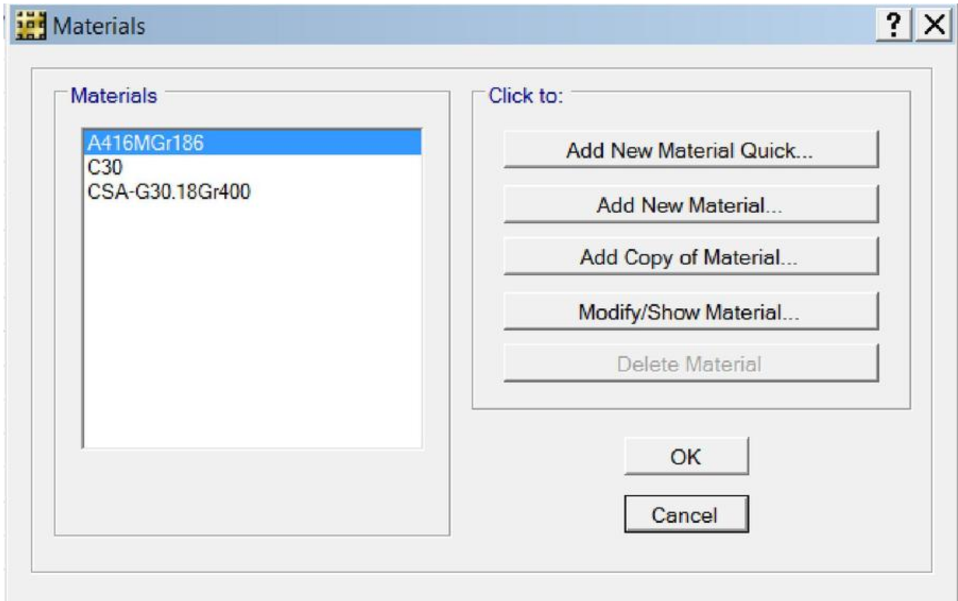


Figure 153

For this particular problem, we will modify the existing '**C30**' concrete using the '**Modify/Show Material**' button and define a new rebar property through the '**Add New Material**' button.

Highlight 'C30' in the 'Materials' box and click on the '**Modify/Show Material**' button. This can also be achieved by right clicking on '**Materials**' on the model explorer. The following window will pop up.

This is the window where concrete properties will be modified. Put any name for the material in the text field in front of '**Material Name**'. But, it is important to make sure that the concrete with the defined material name is assigned for the footing. For this particular problem, let us keep the name of the concrete as '**C30**'. Since we are defining a concrete, the '**Material Type**' should be set to '**Concrete**'.

The unit weight of reinforced concrete may vary depending of the design code of your country. Thus, enter the unit weight of reinforced concrete stipulated in your country code in the text field in front of '**Weight per Unit Volume**'. For this particular example, we will use $25kN/m^3$.

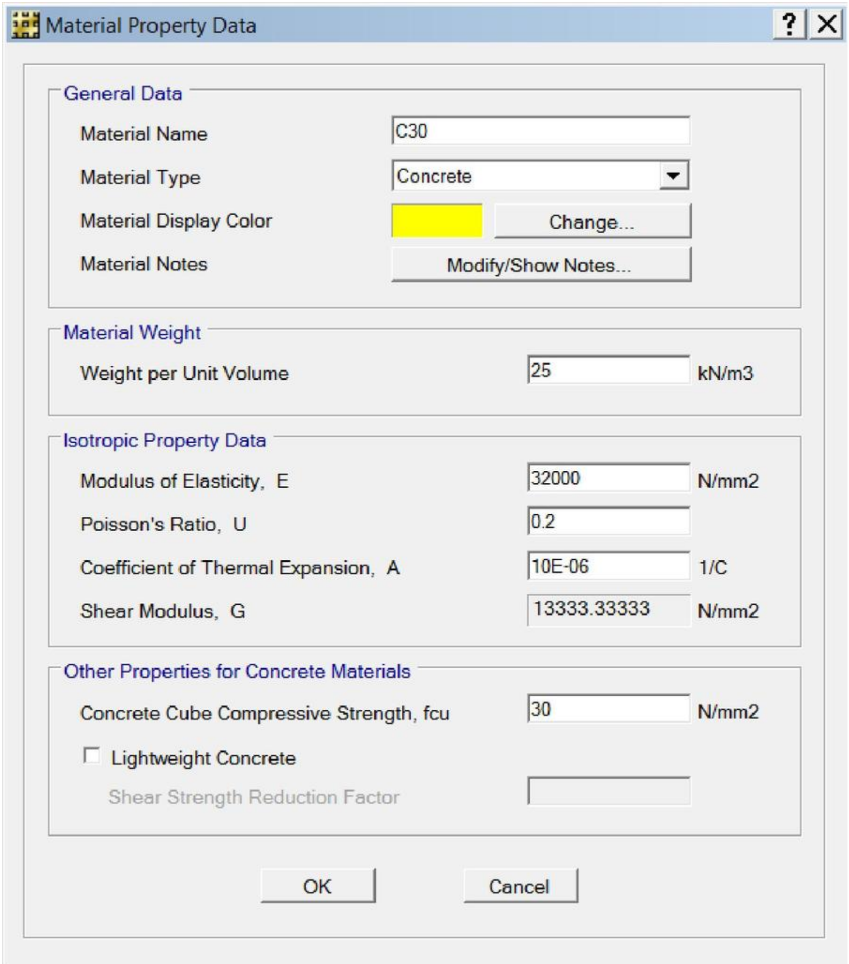


Figure 154

For C-30 concrete, the modulus of elasticity according to BS 8110-1197 is around 32GPa. Thus, enter this value in the text field in front of '**Modulus of Elasticity, E**'. If you selected another design code in step 1 while creating the model, you should refer to actual value of this parameter from the code and enter it accordingly. Be aware of the units though.

The values of Poisson's ratio and coefficient of thermal expansion may also be defined in the design code and should be entered accordingly. For this particular problem, a value of 0.2 for '**Poisson's Ratio, U**' and a value of $10x10^{-6}/^{o}C$ for '**Coefficient of Thermal Expansion, A**' will be entered. The '**Shear Modulus, G**' will be automatically calculated in an un-editable text field.

The grade of concrete for this particular problem is C-30. C-30 is a concrete with 28 day characteristics cube compressive strength of 30MPa. The concrete designation may be different for different country codes but the concept is the

same. Therefore, enter 30 in the text field in front of '**Concrete Cube Compressive Strength, fcu**'.

If a lightweight concrete is used, check on '**Lightweight Concrete**' and enter the corresponding '**Shear Strength Reduction Factor**' in the space provided.

When you press on '**OK**', a concrete material with the above properties will be added to the list of materials. This material will be assigned for the footing before the analysis.

After modifying the concrete property, the program returns to the window shown in Fig. 44. To define a rebar property, we will follow the same procedure as we followed while defining the concrete property. Since a new rebar property will be defined, click on the '**Add New Material…**' button. A '**Material Property Data**' window pops up and when you change the '**Material Type**' to '**Rebar**', the window appears to look like the following.

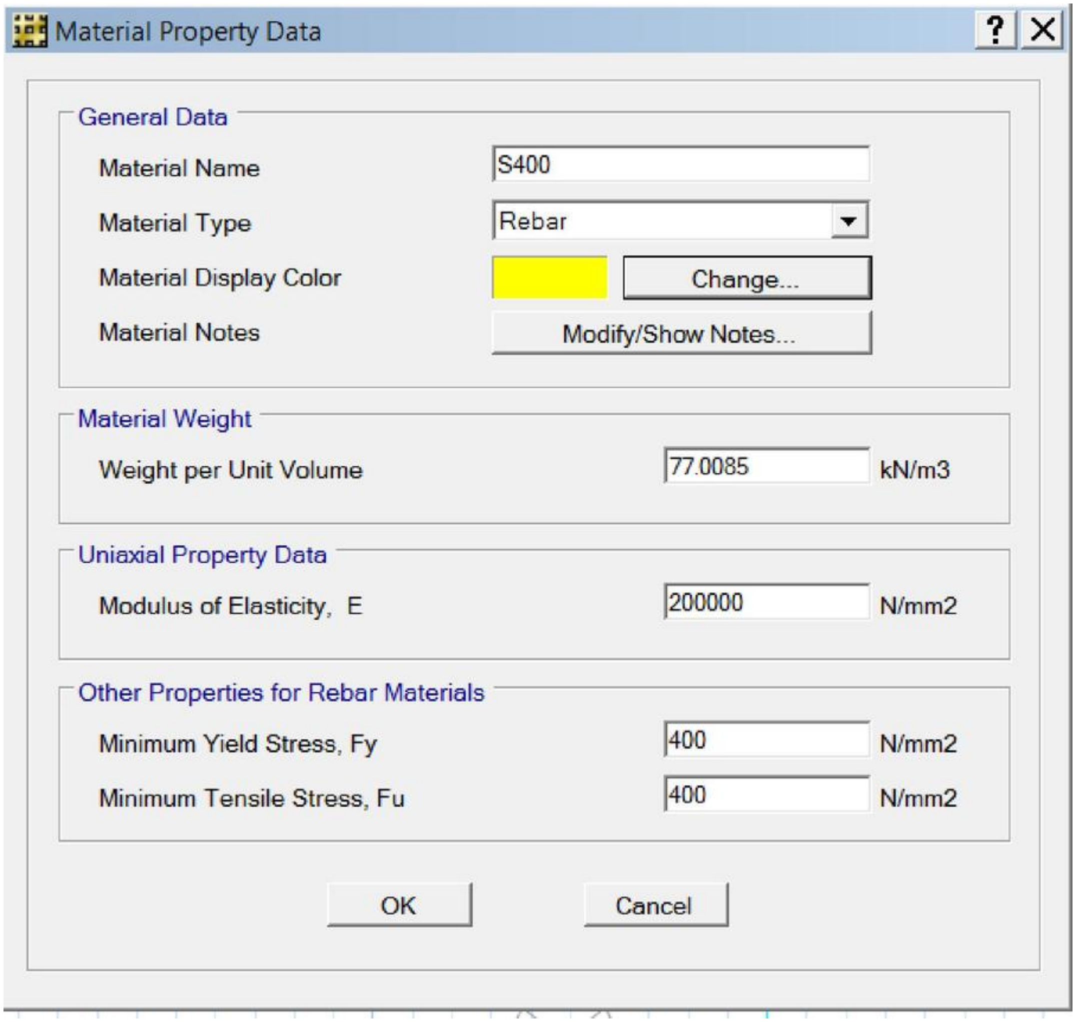


Figure 155

Change the '**Material Name**' to any name you want. Here, we name it '**S400**'. The material type should be '**Rebar**'. The weight per unit volume of steel is stipulated in the design code. For BS 8110-1197, the weight per unit volume is 77.0085kN/m^3. Thus, enter this value in the text field in front of '**Weight per Unit Volume**'. The modulus of elasticity for reinforcement bars according to the same design code is 200GPa. Thus enter this value in the text field in front of '**Modulus of Elasticity, E**' considering the unit.

In the '**Other Properties for Rebar Materials**' box, two quantities are mentioned: minimum yield stress and minimum tensile stress for the reinforcing material. The values of these parameters will be specified in the design code which you defined earlier. If the code assumes that the rebar material exhibits elastic perfectly plastic behavior, the values of these two quantities will be the same. The grade of steel to be used for this particular example is S-400. The yield stress for this type of reinforcement bar is 400MPa. Since the design code of my country assumes that rebars exhibit elastic perfectly plastic behavior, the minimum tensile stress will also be 400MPa. Thus enter 400 in both text fields in front of the '**Minimum yield stress, Fy**' and '**Minimum Tensile Stress, Fu**'. Then press '**OK**' in both '**Material Property Data**' and '**Materials**' windows concluding the material definition step.

The other property which should be defined is the soil support. To define the soil properties, go to the '**Define**' menu and click on '**Soil Subgrade Properties**' menu item and the following window appears.

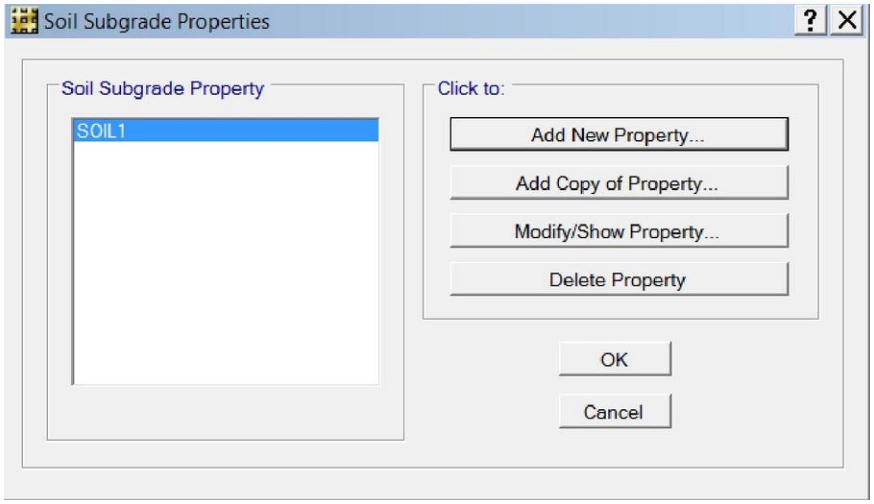


Figure 156

By using the buttons in this window, a new soil property or copy of soil property can be added. An existing soil property can also be modified or deleted. For this problem, let us add a new soil property by using the '**Add New Property…**' button. Thus, click on this button and the following window pops up.

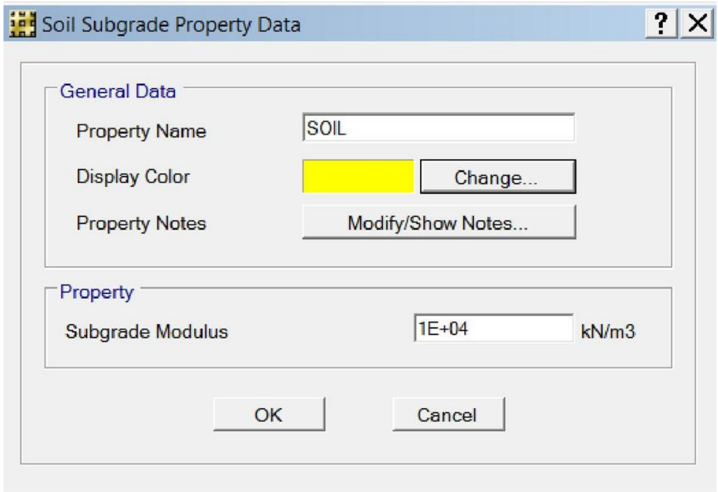


Figure 157

In this window, set the property name to '**SOIL**' and change the subgrade modulus to 10,000kN/m^3. Because, the subgrade modulus which can be assume by the ratio of the bearing capacity by the allowable settlement for this particular example is so. Then press '**OK**' in both the '**Soil Subgrade Property Data**' and '**Soil Subgrade Properties**' windows.

STEP 3: Defining Footing and Column Properties

After defining the material properties, the footing and column properties can be defined. This definition can take place in two ways: from the menu bar and from the model explorer. In SAFE software, footings are modelled as 'footings' and foundation columns are modelled as 'stiff'.

To define footing and column properties from the menu bar, go to '**Define**' menu and click on '**Slab Properties…**'. The following window will pop up.

The '**Add New Property…**' button prompts the user to enter new properties for the footing and foundation column while the '**Add Copy of Property…**' copies the property of an existing slap. The '**Modify/Show Property…**' allows the user to show the property of an existing component with the possibility of modification. When the '**Delete Property**' button is active, it allows the user to delete an existing slab property.

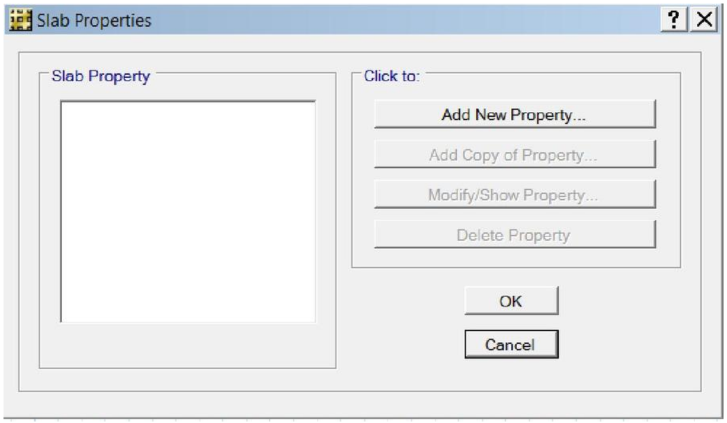


Figure 158

In this case, we use the '**Add New Property…**' button to add new slab properties for the footing and foundation column. Click the button and the following window will pop up.

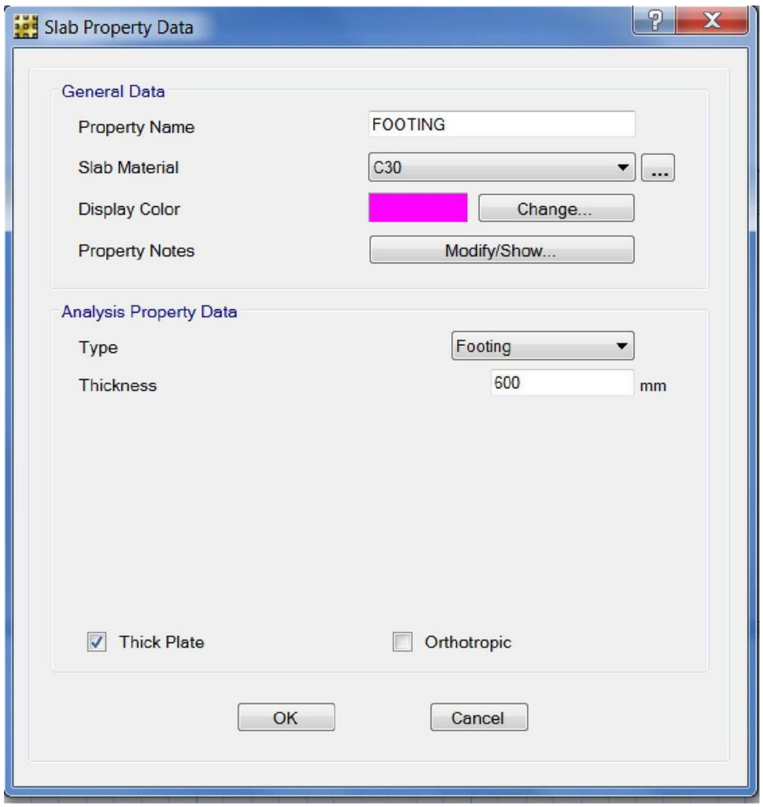


Figure 159

The '**Property Name**' can be assigned with any name; but, in this case, the property name which will be used is '**FOOTING**'. The '**Slab Material**' should be set to the concrete grade which is defined in step 2. Since the name of the concrete material defined in step 2 for this particular problem is '**C30**', a material with this name should be selected from the list.

In the '**Analysis Property Data**' box, '**Type**' should be set to '**Footing**'. The '**Thickness**' value should be set to the thickness of the footing defined in the example. Since the thickness of the footing is 600mm, this value is entered in the text field corresponding to '**Thickness**'. As footings are modelled as thick plates, check the '**Thick Plate**' option. The '**Orthotropic**' check box is selected when a footing with irregular dimension is to be used.

When you press '**OK**', the '**Slab Property Data**' window will be exited and the '**Slab Properties**' window gets activated. Now, the property of the foundation column will be added. To do this, again click on the '**Add New Property…**' button. The following window appears after the click.

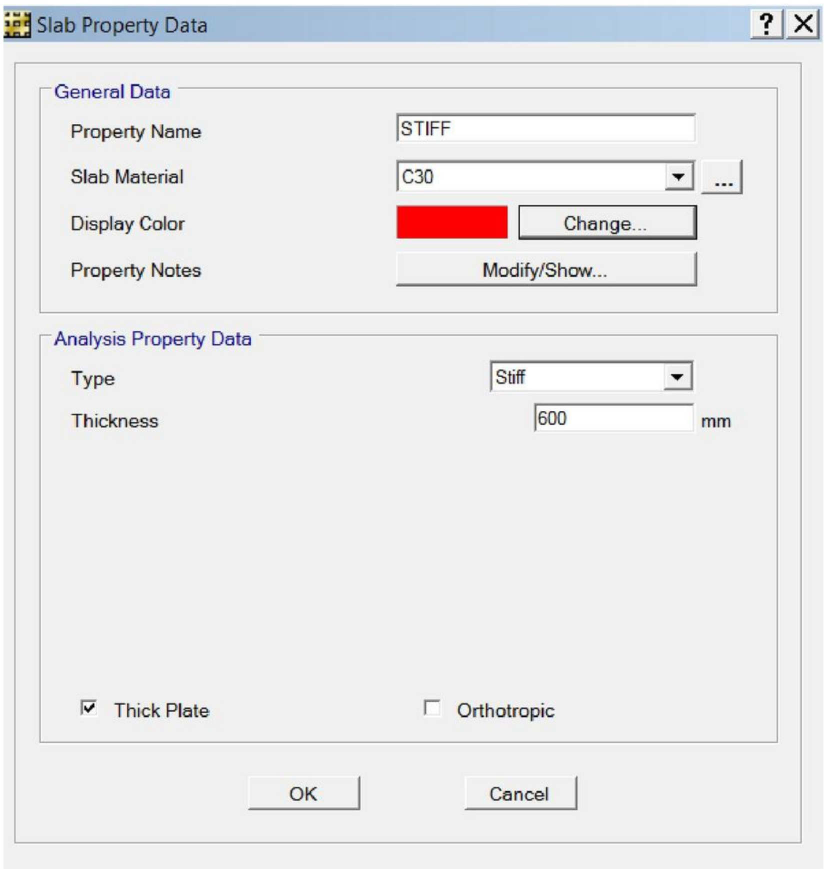


Figure 160

The '**Property Name**' can be any name but we use '**STIFF**'. The '**Slab Material**' should be set to the concrete grade which is defined in step 2. Since the name of the concrete material defined in step 2 for this particular problem is '**C30**', a material with this name should be selected from the list.

In the '**Analysis Property Data**' box, '**Type**' should be set to '**Stiff**' as we are defining the property of column. The '**Thickness**' value should be equal to the footing thickness which is 600mm. This value should be entered in the text field corresponding to '**Thickness**'. As foundation columns are modelled as thick plates, check the '**Thick Plate**' option. The '**Orthotropic**' check box is selected when a column with irregular dimension is to be used.

When you press '**OK**', the '**Slab Property Data**' window will be exited and the '**Slab Properties**' window gets activated. Again press '**OK**' and exit the window for defining the footing and foundation column.

STEP 4: Defining Load Patterns, Load Cases and Load Combinations

The loads on the foundation should be defined accordingly before the analysis. First, the load pattern should be defined. This can be done from the '**Define**' menu or from the '**Model Explorer**'. This time, we will do it from the model explorer. In the model explorer, expand '**Load Definitions**' and you will see '**Load Patterns**'. When you expand '**Load Patterns**', you will see '**DEAD**' and '**LIVE**'. At the end, the model explorer appears to look like:

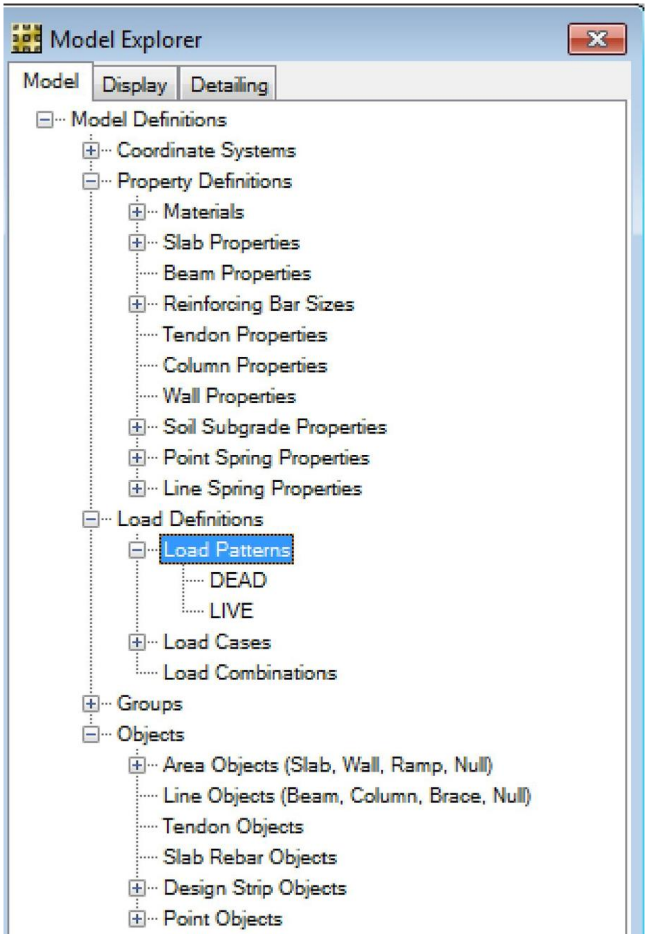


Figure 161

Now, right click on ‘**Load Patterns**’ and click on ‘**New Load Pattern**’ and the following window pops up.

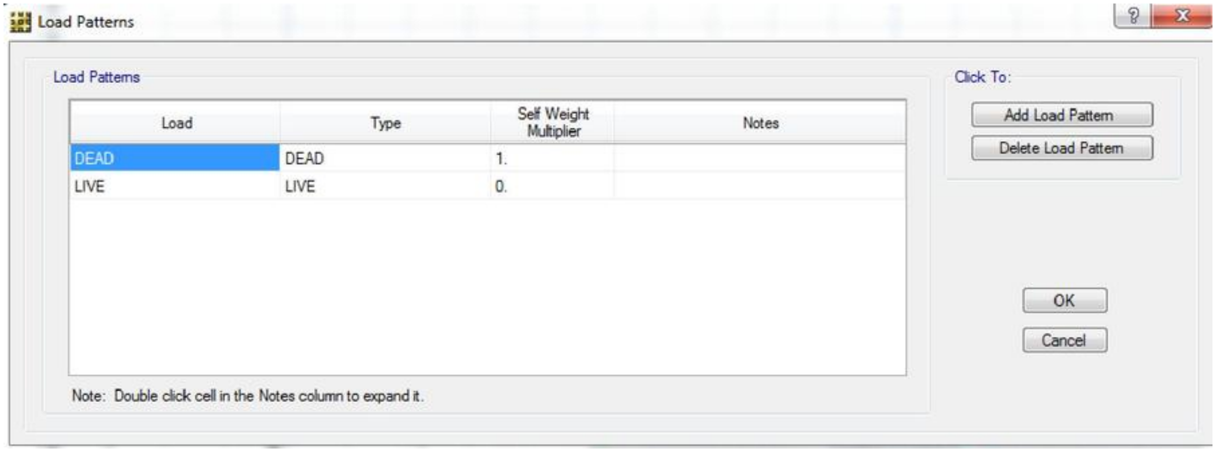


Figure 162

In the ‘**Load Patterns**’ window, two load patterns are already defined: ‘**DEAD**’ and ‘**LIVE**’. The ‘**Type**’ of the ‘**Load**’ should also be changed accordingly. There are many options for type of loading. The ‘**Type**’ for dead loads should be set to ‘**DEAD**’ and for live loads ‘**LIVE**’. If there are other types of load patterns on the foundation such as earth quake load, you can add the load pattern with the ‘**Add Load Pattern**’ button. You can also delete any undesirable load pattern using the ‘**Delete Load Pattern**’ button. Since in this example, we have only dead and live loads, we will leave the existing load patterns as they are. The ‘**Self Weight Multiplier**’ value should also be changed accordingly. This value imparts the option whether to consider or ignore the self-weight of the foundation in addition to external loads. If the self-weight of the foundation is already included as an external dead load or if you want to exclude the effect of self-weight from the analysis, the value under ‘**Self Weight Multiplier**’ should be set to zero. In this example, we will consider the self-weight as an additional load to the external dead load. Thus, the value under ‘**Self Weight Multiplier**’ for the ‘**DEAD**’ load is one. For the ‘**LIVE**’ load, it will be zero. Press ‘**OK**’ and the window will be exited.

After this, the load cases will be defined. Load cases are used to dictate the way the loads are applied (statically or dynamically) or the way the structure responds (linearly or non-linearly) for the defined load patterns. To define a load case, go to ‘**Define**’ menu and click on ‘**Load Cases…**’. The following window will pop up after the click.

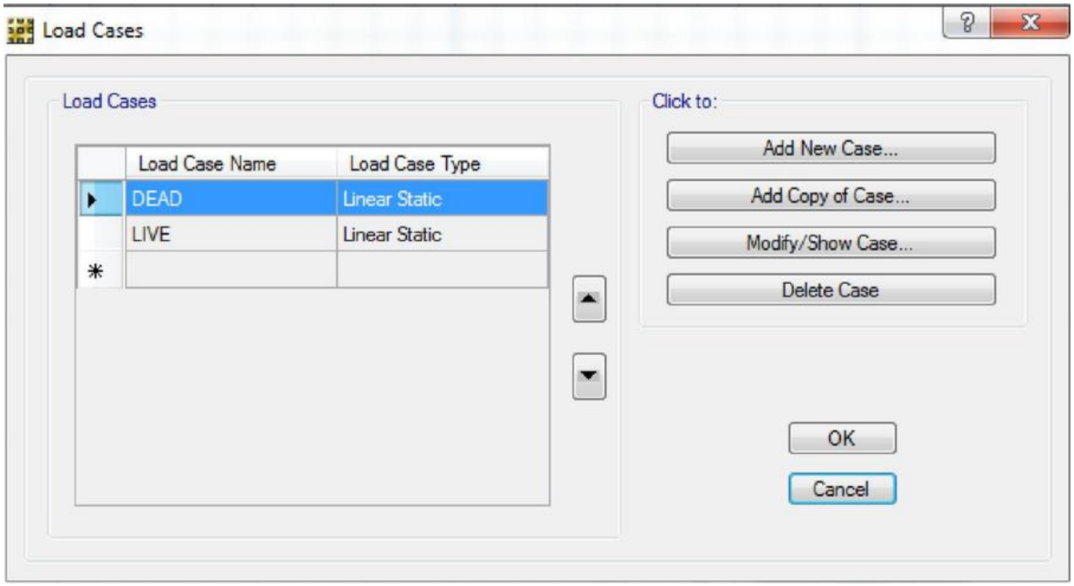


Figure 163

The load patterns which were defined earlier will automatically appear in the list of '**Load Case Name**' of the '**Load Cases**' window. The '**Load Case Type**' column shows the way in which each load pattern will be applied during analysis. If you want to modify this, highlight the load pattern for which you are going to change the load case type and click on the '**Modify/Show Case…**' button. If you click this button, the following window will appear. In this window, you can change the way the load is applied from the '**Load Case Type**' box. The way the structure responds can also be selected from the '**Analysis Type**' box. This problem '**Static**' is for the '**Load Case Type**' and '**Linear**' is selected for the '**Analysis Type**' since the load is static and the foundation responds linearly. The scale factor for the dead load in the '**Loads Applied**' box will be left as one. Press '**OK**' and exit the window.

The load case type for the live load should also be 'Linear Static'. Otherwise, it should be changed by clicking the '**Modify/Show Case…**' button to linear static case. If both the load case types are as desired click '**OK**' and exit the '**Load Cases**' window.

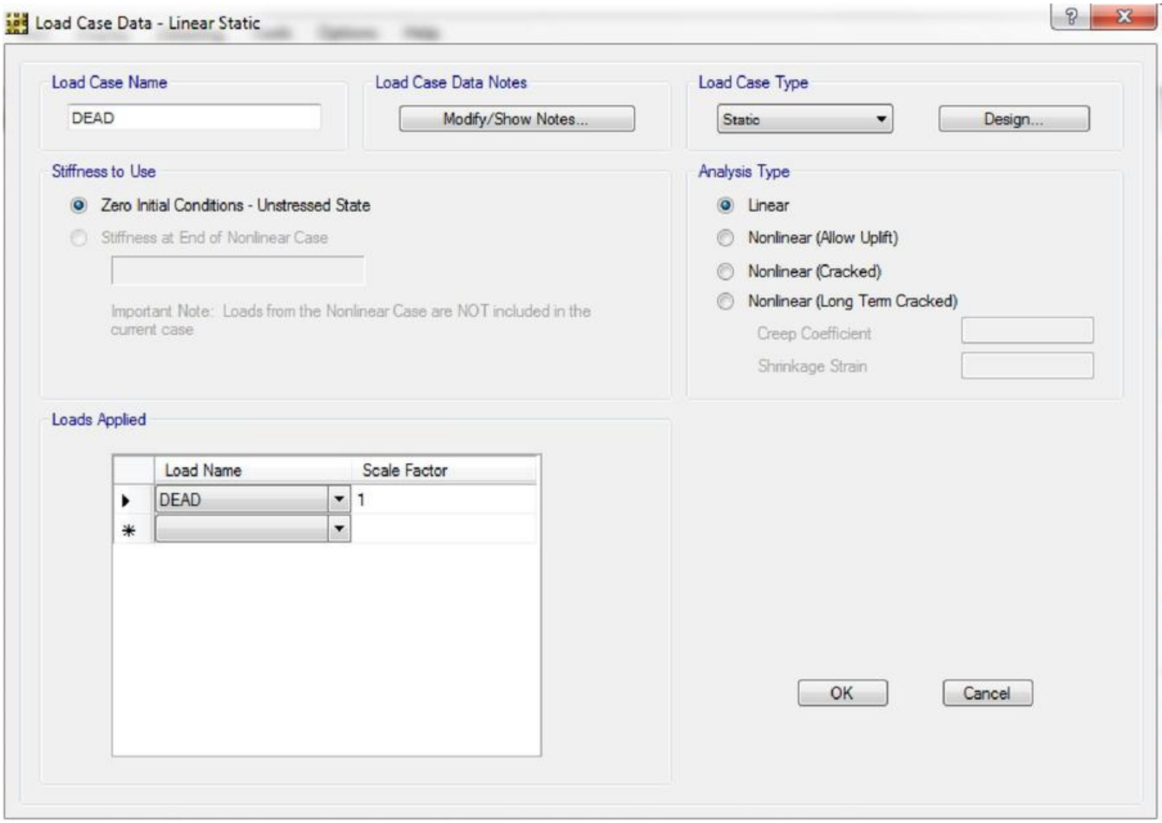


Figure 164

Definition of the load combinations will be the next step. Two load combinations will be considered in this example for ultimate limit state and serviceability limit state. To define load combinations, go to '**Define**' menu and click on '**Load Combinations…**' and the following window pops up.

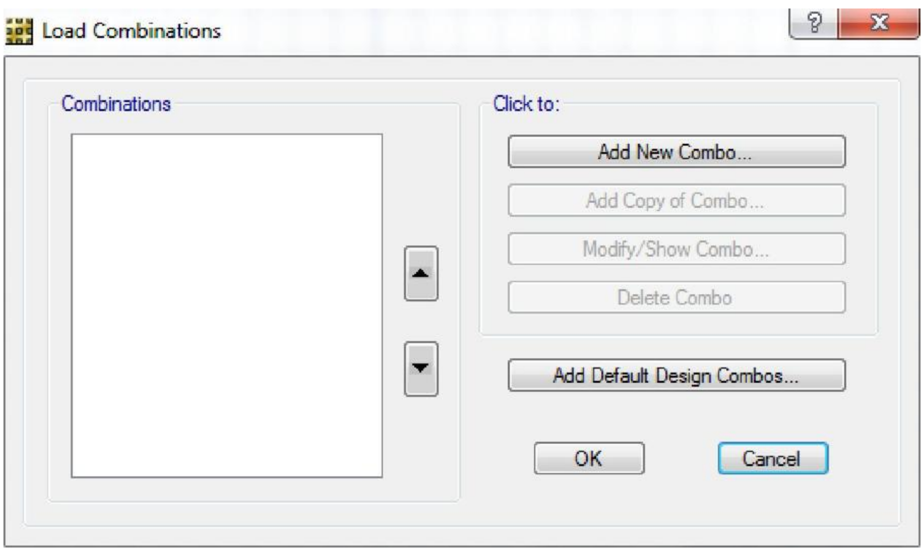


Figure 165

New load combinations will be added through the '**Add New Combo…**' button. Here, two load combinations will be added; one for the ultimate limit state the other for the serviceability limit state. When the '**Add New Combo…**' button is clicked, the following window appears.

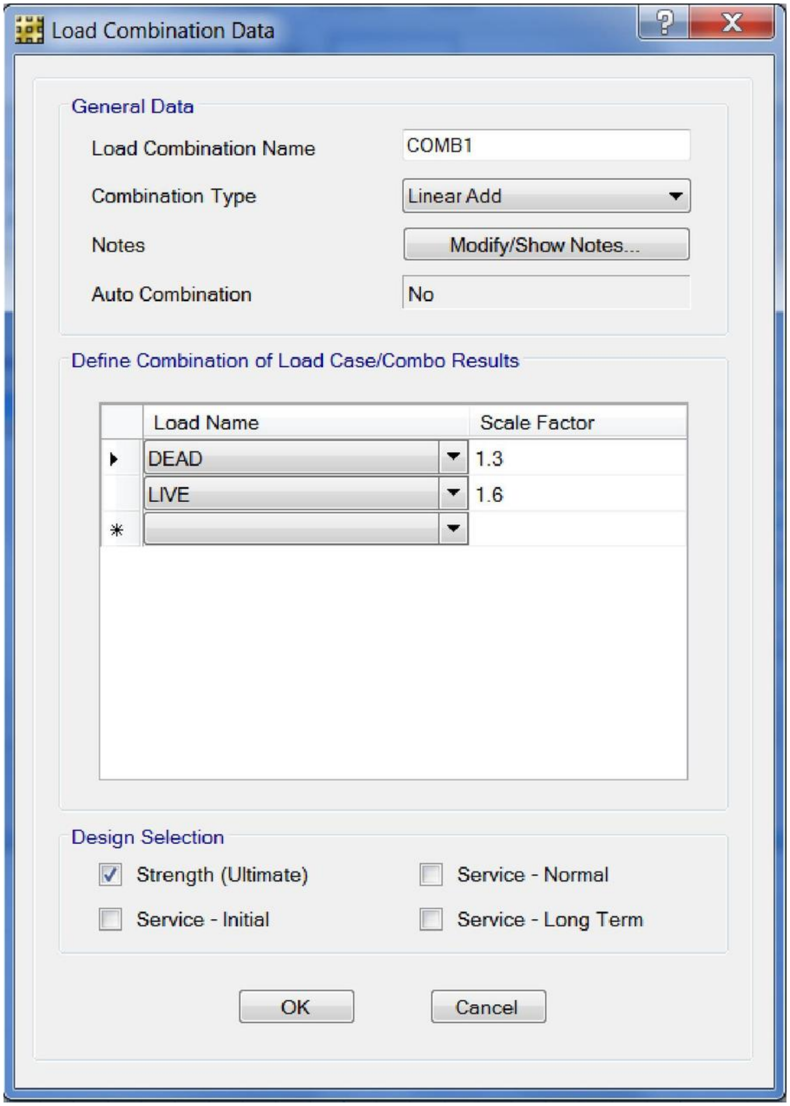


Figure 166

In the '**Load Combination Data**' window, any name can be given for the load combination. The '**Combination Type**' should be set to '**Linear Add**' as the component loads (dead and live) will be added linearly. However, there are also other options from the drop down menu in front of '**Combination Type**'. In the window shown in Fig. 166, the two loads (dead and live) should be activated below the '**Load Name**' column of the '**Define Combination of Load Case/Combo Results**' box. The values in the '**Scale Factor**' column correspond to the partial

safety factors for load for the failure mode under consideration. These partial safety factors are specified in the design code of your country. In the design code of my country, the partial safety factor for dead loads for ultimate limit sate case is 1.3 and for serviceability limit state is 1 for the cases where there are only dead and live loads. For such load patterns, the partial safety factor for live loads for ultimate limit sate case is 1.6 and for serviceability limit state is 1. Thus, enter a value of 1.3 in front of '**DEAD**' and 1.6 in front of '**LIVE**' in '**Scale Factor**' column. The failure condition which is being under consideration can be defined by selecting and deselecting the check boxes in the '**Design Selection**' box. Since the above scale factors are for the ultimate limit state, check on '**Strength (Ultimate)**'. The '**Load Combination Data**' window for the serviceability limit state looks like the following window.

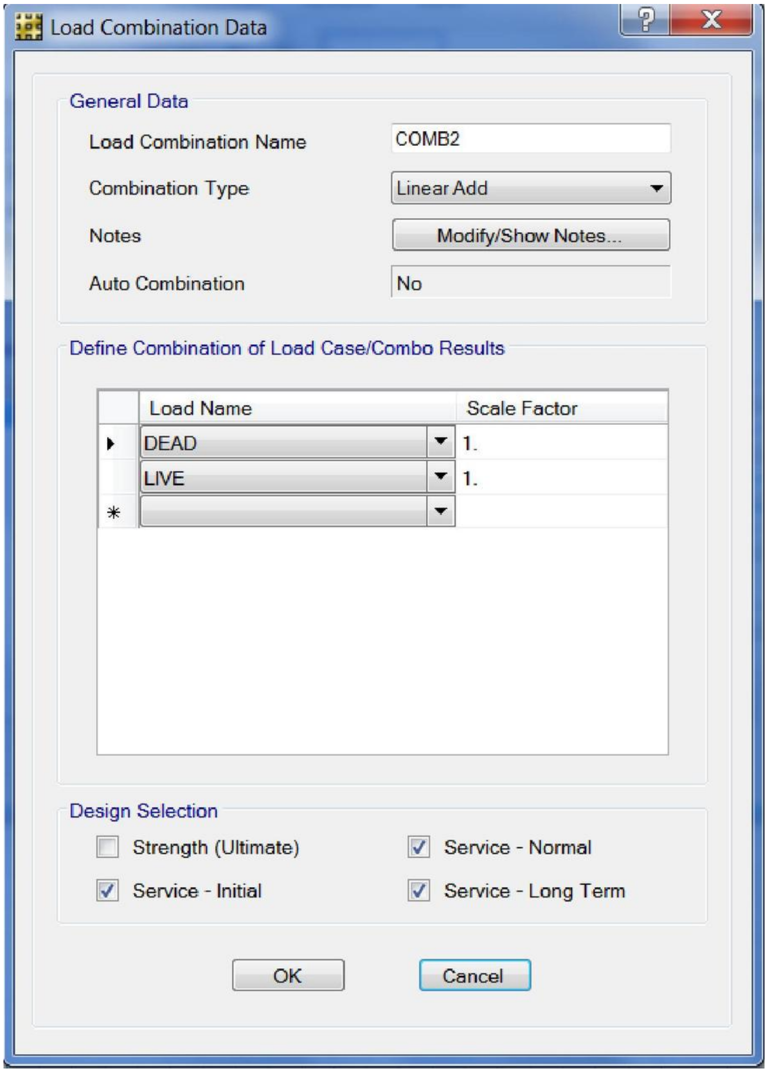


Figure 167

STEP 6: Drawing the Footing Components and Design Strips

The footing, the foundation column and a point where the loads will be applied on the foundation column should now be drawn on the grid.

iv. *Drawing the footing*

Since the footing will be drawn around points and since the footing is rectangular, go to the '**Draw**' menu and click on '**Draw Slabs/Areas**' or simply click on the equivalent icon  on the left hand side tool bar. The following window will appear on the screen after the click.

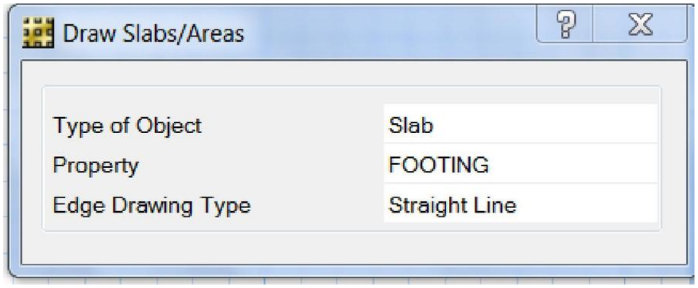


Figure 168

Making sure that the '**Property**' is set to '**FOOTING**', and the '**Edge Drawing Type**' to '**Straight Line**', click on the grid where the four corners of the footing will be located. Repeating the click at the first click point will draw a trapezoidal foundation.

v. *Drawing the foundation column*

While the window in fig 168 is active, change the '**Property**' to '**STIFF**' and click on the grid where the four corners of the foundation columns will be located. Repeating the click at the first click for both foundation columns will draw the foundation columns with their respective sizes. Note that, the first column is 0.6mX0.6m while the second column is 0.4mX0.4m.

vi. *Drawing the point on the foundation column where the load will be applied*

Go to '**Draw**' menu and click on '**Draw Points**', then click on the mid-point of the foundation columns and the points will be created. If the cursor could not snap to the midpoint, you can adjust the '**Snap Options**' from the '**Draw**' menu.

After this, the design strips will be drawn. Design strips determine the way in which different quantities related to the reinforcement calculation are calculated. Forces are integrated across the design strips. Thus, the larger the width of coverage of the design strips within the given structure, the higher will be the calculated values of the bending moments and shear forces. Thus, an optimum width of strip is required compromising the safety and economical requirements. The width of the design strip will be specified in the design code. According to the code of my country, the width of design strips for isolated foundations is 1m. Thus, a one meter design strip will be drawn in both X and Y directions on the foundation. These design strips in X and Y direction are usually defined in SAFE software as layer A and layer B.

To draw the design strip, go to the '**Draw**' menu and click on '**Design Strips**' or simply click on the equivalent icon  from the left hand sided tool bar and the following window pops up.

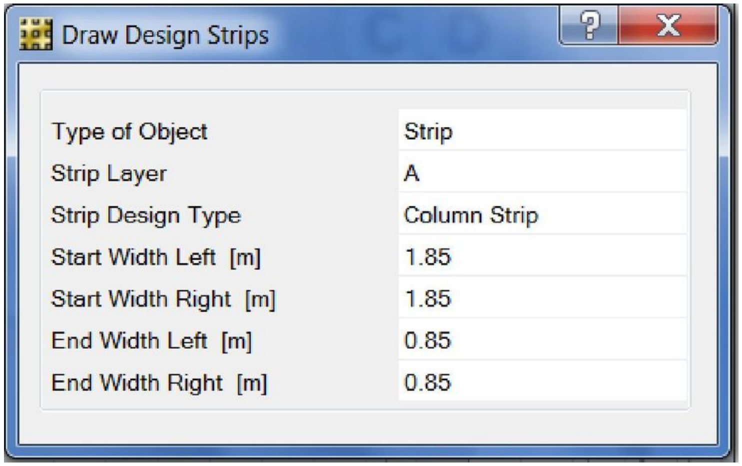
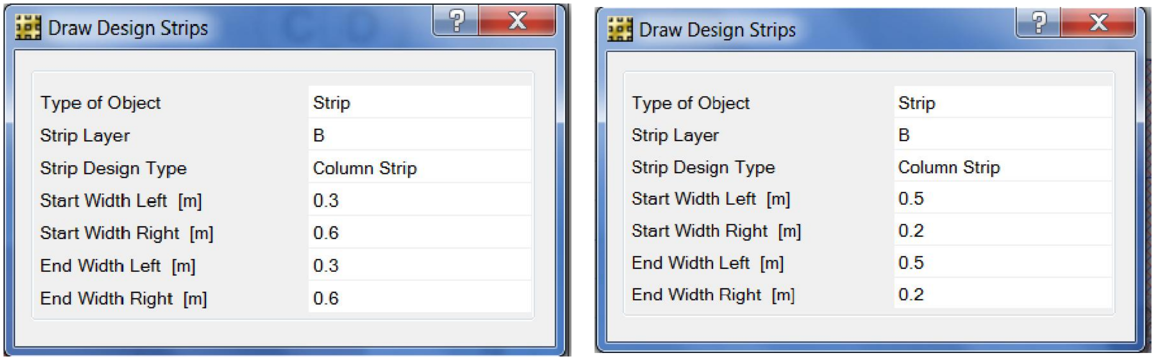


Figure 169

In this window, the '**Strip Layer**' should be selected to be either '**A**' or '**B**'. But if '**A**' is for design strip in X direction '**B**' should be for Y direction and vice versa. Since we are drawing a strip around the column to consider maximum moment and shear forces, the '**Strip Design Type**' should be set to '**Column Strip**'. Since the whole footing area should be covered with design strip in x-direction, the width of the strip in x-direction will be 3.7m at the start and 1.7m at the end. Since, the footing is symmetrical about the centerline in x-direction, the 'Start Width Left [m]' and 'Start Width Right [m]' values will be 1.85 while the 'End Width Left [m]' and 'End Width Right [m]' values will be 0.85. To draw the design strip in the X direction, without closing the window, left click at the center of left side of the footing on the plan view parallel to the Y axis and again left click at the center of the parallel side and right click. This creates a design strip in X direction. In doing so, if you can't snap to the center of the side of the footing, you can modify the snap options by clicking on the '**Snap Options…**' command from the '**Draw**' menu and adjusting the options which you want to snap to.

The design strip in Y direction can also be drawn in a similar procedure after changing the '**Strip Layer**' to '**B**'. The design strips in y direction extend for a distance of half of the depth from the face of the support for each column. Thus, the widths of the strips for each column 1 and column 2 are shown in Fig. 170(a) and 170(b) respectively. When you make the click to draw the strips make sure that you click at the bottom first and then at the top for both design strips.



(a) (b)

Figure 170

Middle strips should also be drawn between the column strips in y-direction (transverse direction). To draw middle strips, change the '**Strip Design Type**' in the '**Draw Design Strips**' window to '**Middle Strip**' and enter the right values of the widths of the middle strips to the right and to the left of the point from which you will start drawing the strips. For this example, if we start drawing the middle strip from x = 2.5m, the values of '**Start Width Left [m]**', '**Start Width Right [m]**', '**End Width Left [m]**' and '**End Width Right [m]**' are 1.6,1.8,1.6 and 1.8 respectively. So, enter these values and draw the middle strip between the two column strips.

You can display the design strips by setting the display options by clicking on '**Set Display Options…**' from the '**View**' menu or by simultaneously clicking on '**Ctrl**' and '**W**' keys or by just clicking on the set display options icon from the tool bar below the menu bar. This results in the following window:

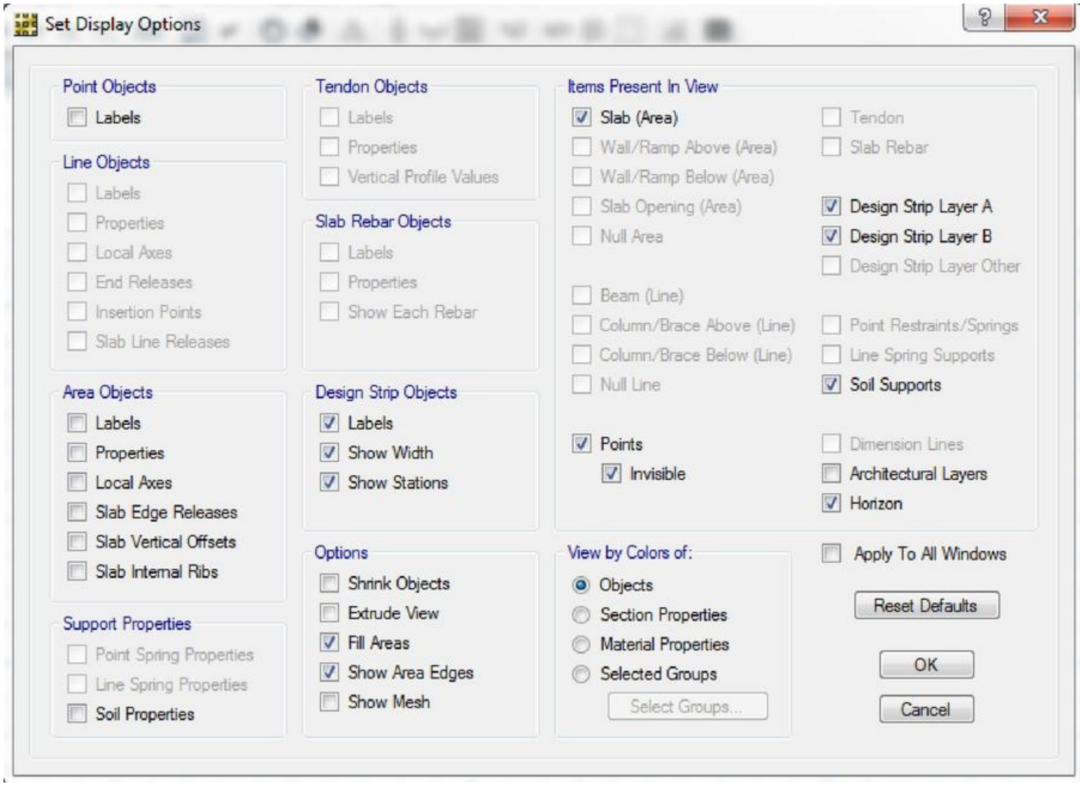


Figure 171

Then, check on ‘**Labels**’, ‘**Show Width**’ and ‘**Show Stations**’ in the ‘**Design Strip Objects**’ box and press ‘**OK**’ and after drawing dimension lines by using the command ‘**Draw**’>’**Draw Dimension Lines**’, the following window appears displaying the design strips in the two directions.

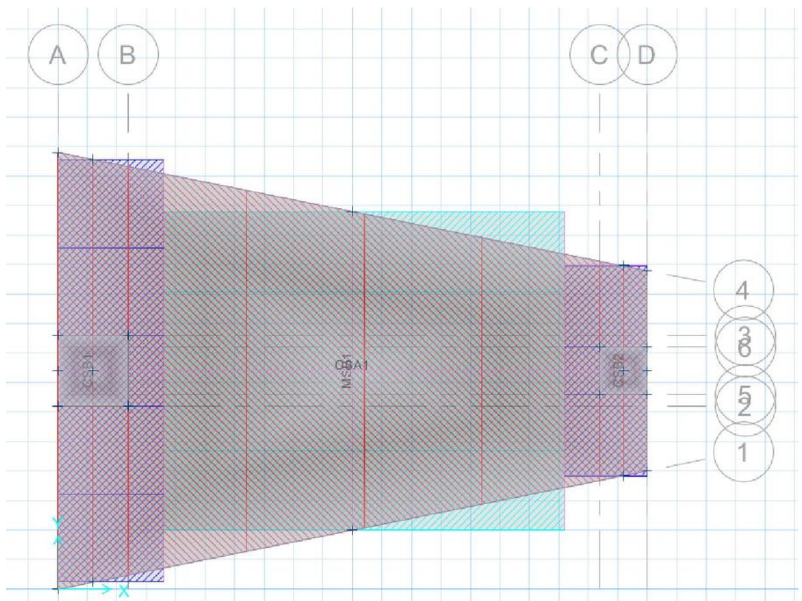


Figure 172

STEP 7: Assigning Slab Data, Support Data and Load Data

The slab data, support data, and load data which are defined in the previous steps should be assigned to the corresponding structural component. The way to do this is first to select the component and next to assign the slab property, support property or loads accordingly. The selection can be done by clicking on the component from the plan view or through the '**Select**' menu. The latter option assures that the component is selected exactly as selection by clicking may result in incorrect selection. Thus, all the selections here will be done from the '**Select**' menu. The assignments will be discussed as follows:

d. *Assigning support data to the footing*
Select the footing through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**'Slab Properties…**'. Then select '**FOOTING**' and press '**OK**'. Then assign the footing property through the following strings of commands '**Assign**'>'**Support Data**'>'**Soil Properties…**'. Then select '**SOIL**' and press '**OK**'.

e. *Assigning reinforcement data to the design strips*
Select each design strip through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**Design Strip Layers…**'. Then select '**A**' or '**B**' (one at a time) and press '**OK**'. When you right click on the selected strip layer, the '**Slab-Type Area Object Information**' window pops up. In the '**Design**' tab of this window, set the '**Rebar Material**' to '**S400**' and press '**OK**'. Do this for both strips.

f. *Assigning load on the foundation columns*
To assign load on the foundation columns, right click on the point at the center of each foundation column (one at a time) and a '**Point Object Information**' window pops up. In this window, click on the '**Loads**' tab.

The dead load and the live load can be assigned through the '**Assign Load…**' button. The procedure is: click on '**Assign Load…**' button, then select '**Force Loads**' then press '**OK**' then select either '**DEAD**' or '**LIVE**' depending on which loads you want to enter their values then enter their values accordingly (both the concentrated load and the bending moment) and in the right direction (axis), then select '**Add to Existing Loads**' and press '**OK**'. While doing this, the foundation column dimensions should be entered in the '**Size of Load for Punching Shear**' box of '**Point Loads**' window.

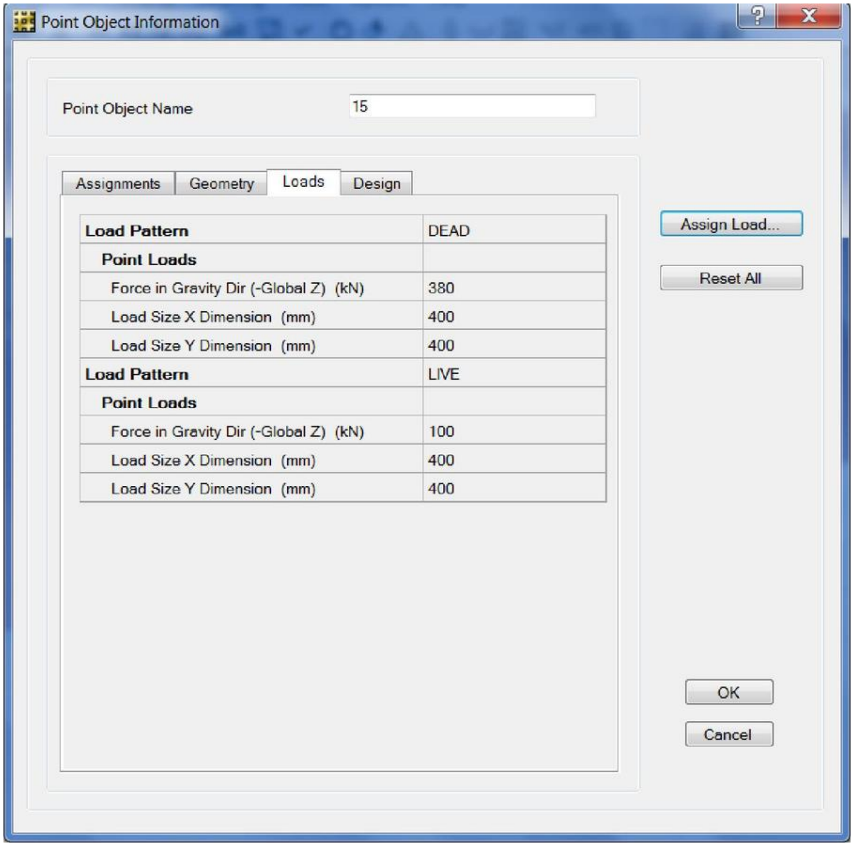


Figure 173

You can also delete any forces which are wrongly entered through '**Delete Existing Loads**' radio button option. In this example, the dead concentrated load is 439kN and the live load is 125kN for the column1. For the second column, the dead concentrated load is 380kN and the live concentrated load is 100kN for the second column. These values are entered accordingly and the final values for the second column are shown in Fig 173.

STEP 8: Running the Analysis

After this, the analysis can be run. But, make sure that the footing and the foundation column are assigned with the correct rebar material. To do this, right click anywhere in the plan view of the footing and the '**Slab-Type Area Object Information**' window will pop-up.

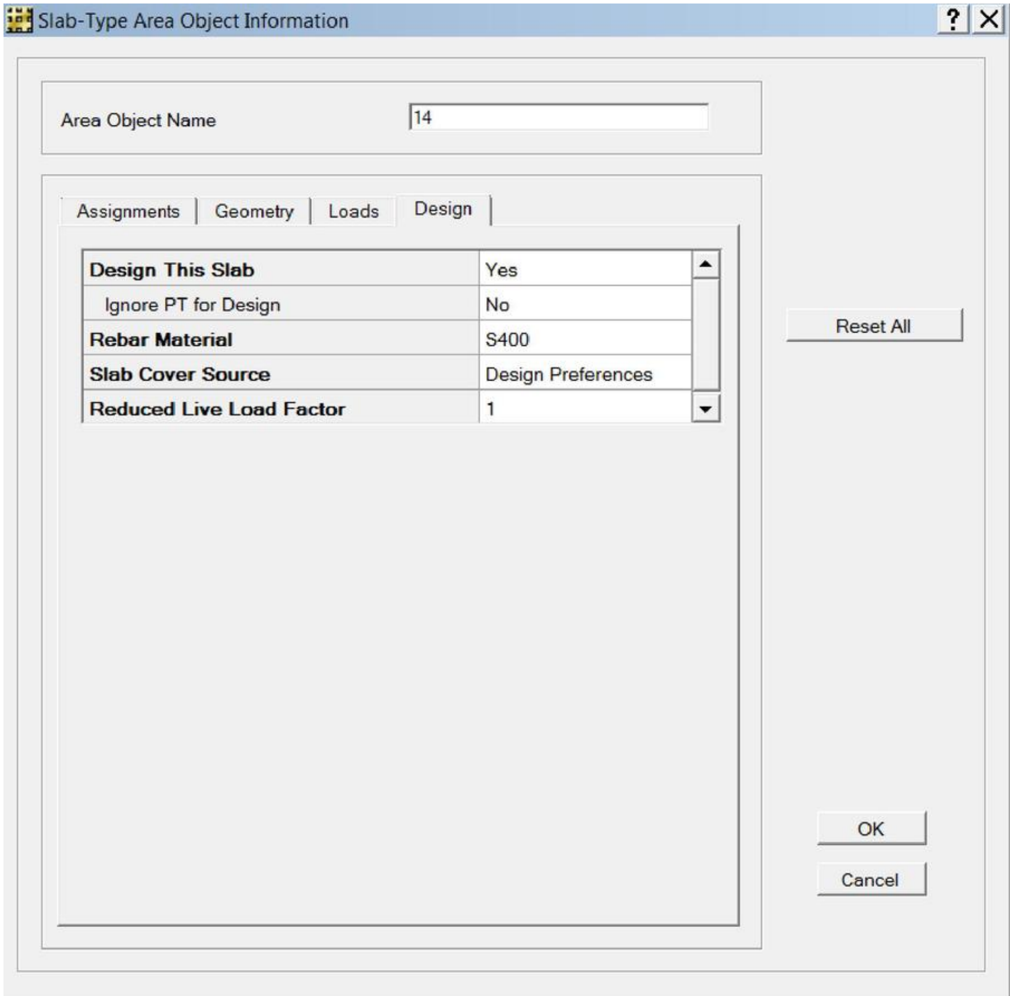


Figure 174

In the '**Design**' tab of this window, set the value of '**Rebar Material**' to '**S400**'. Do this for the column foundation, and both design strips as well. After this, go to the '**Run**' menu and click on '**Run Analysis & Design**' command or simply click on the '**F5**' key. When you do this, you will be prompted to save the model, if you haven't already done this.

STEP 9: Displaying the Output

After running the analysis the following output diagram will be displayed on the plan view window.

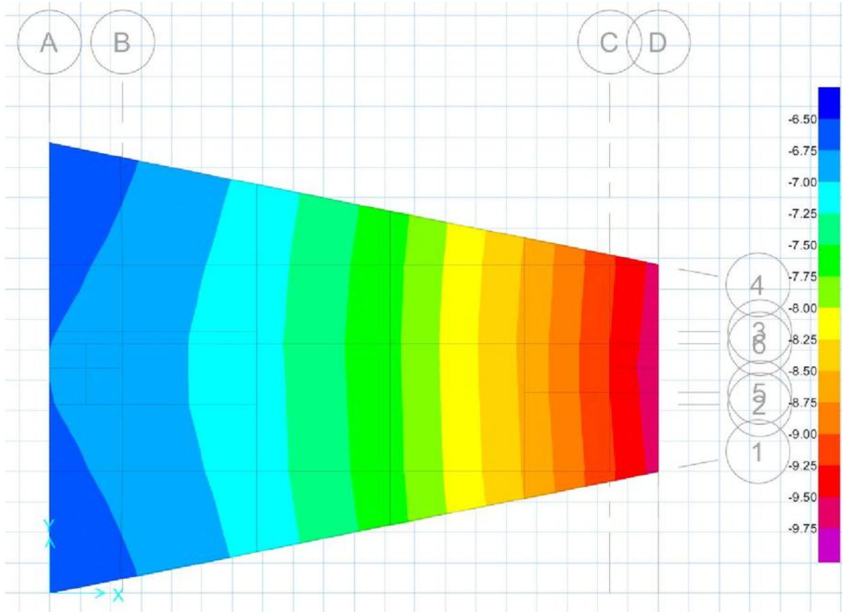


Figure 175

To display the punching shear ratio, go to the '**Display**' menu and click on '**Show Punching Shear Design**'. After this, the punching shear ratio will be displayed in the plan view around the foundation column as in the following figure.

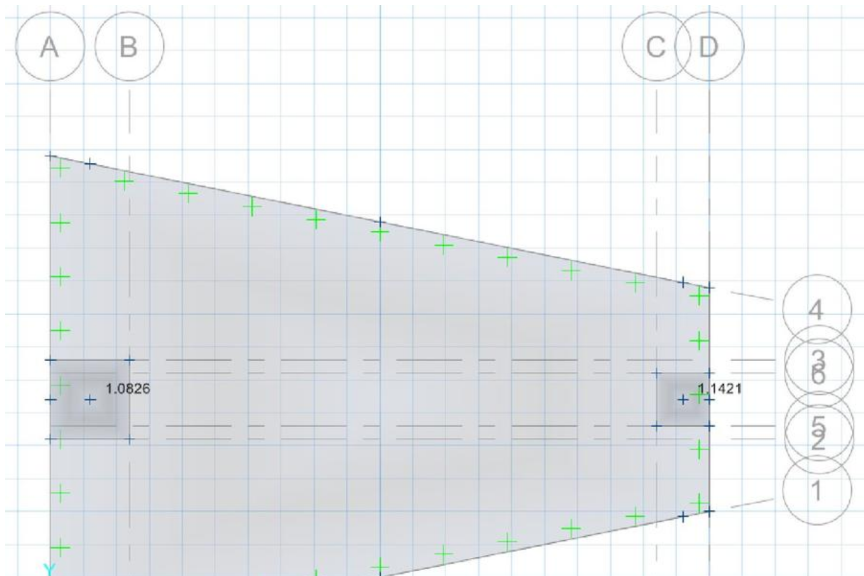


Figure 176

As can be noticed from the figure, the punching shear ratio is for the first column is 1.0826 and for the second column is 1.1421. Generally, a punching shear ratio less than one indicates the concrete thickness is adequate to resist punching shear and a value greater than one indicates that the punching shear capacity is exceeded somewhere along the critical section. For economical design, it is recommended to keep the punching shear ratio between 0.95 and 1 as very small values of punching shear ratio means excess concrete thickness is used. However, if the punching shear ratio is greater than one, the thickness of the concrete should be increased and the foundation should be re-designed. Since the objective of this tutorial is to show you how to design foundations with SAFE, we will proceed with the design process as if the depth was adequate for punching shear requirement. A detailed quantitative description of the foundation design for each column can also be obtained by right clicking on the plan view. The detailed description for the first column is shown in Fig 177. Several trial may be made by zooming in and out to get the quantitative description.

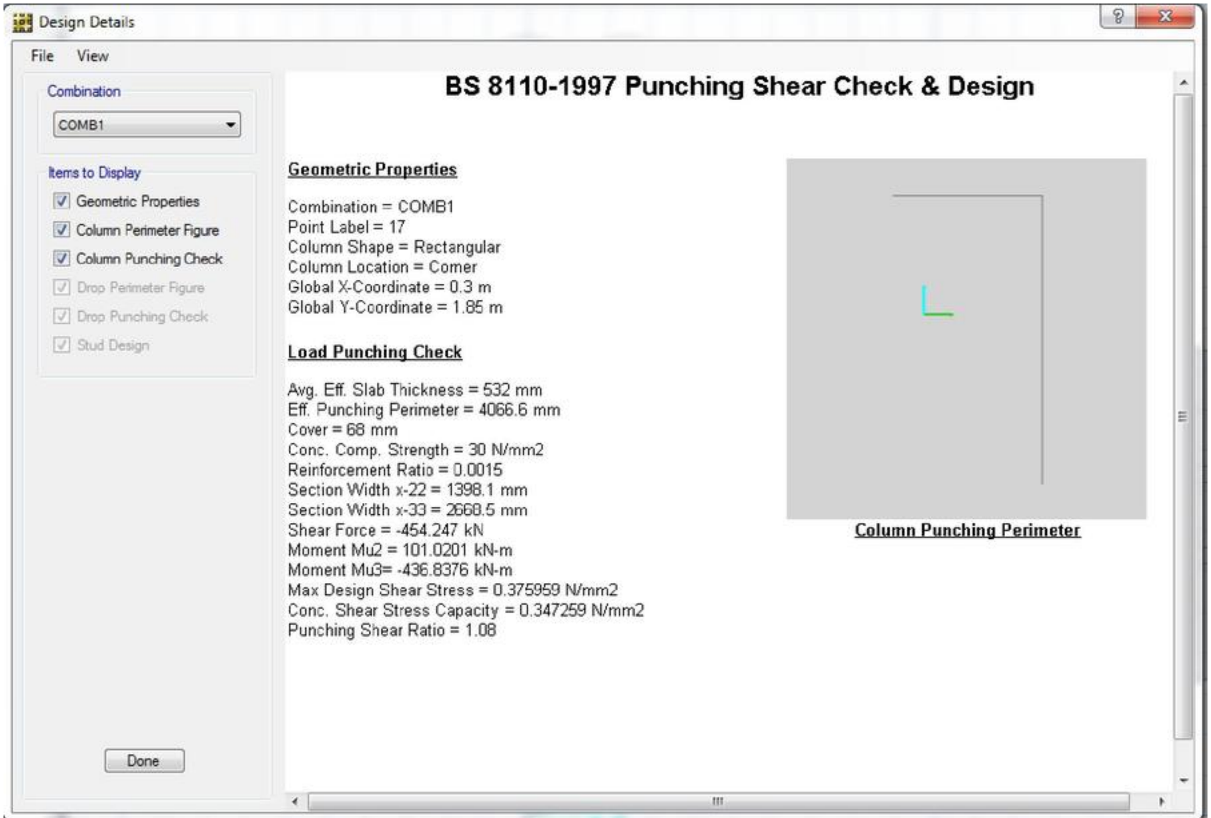


Figure 177

Through the '**Display**' menu, relevant quantities can be displayed on the screen. For instance, the '**Display**'>'**Show Strip Forces**' command or by simply clicking the '**F8**' key, the following window will be displayed.

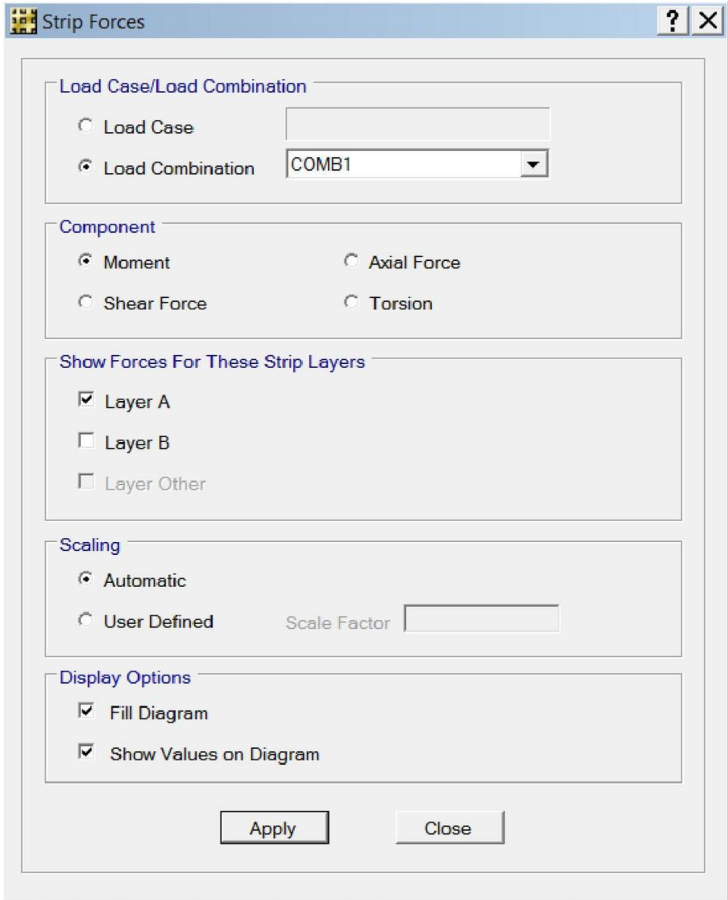


Figure 178

In this window, the '**Load Case/Load Combination**' box provides with radio buttons to select for which particular load case or load combination that we want to display the output. The '**Component**' box contains four radio buttons to select which quantity to display. The '**Show Forces For These Strip Layers**' box allows us to select the strip layer for which the quantity is displayed. Both strip layers can be selected at the same time. From the '**Scaling**' box, we can select whether automatic scaling or user defined scaling is used while displaying the diagram. The '**Display Options**' box allows us to fill or not to fill the diagram and to display or not to display the values on the diagram. For the preferences shown in Fig. 68, the following diagram will be displayed.

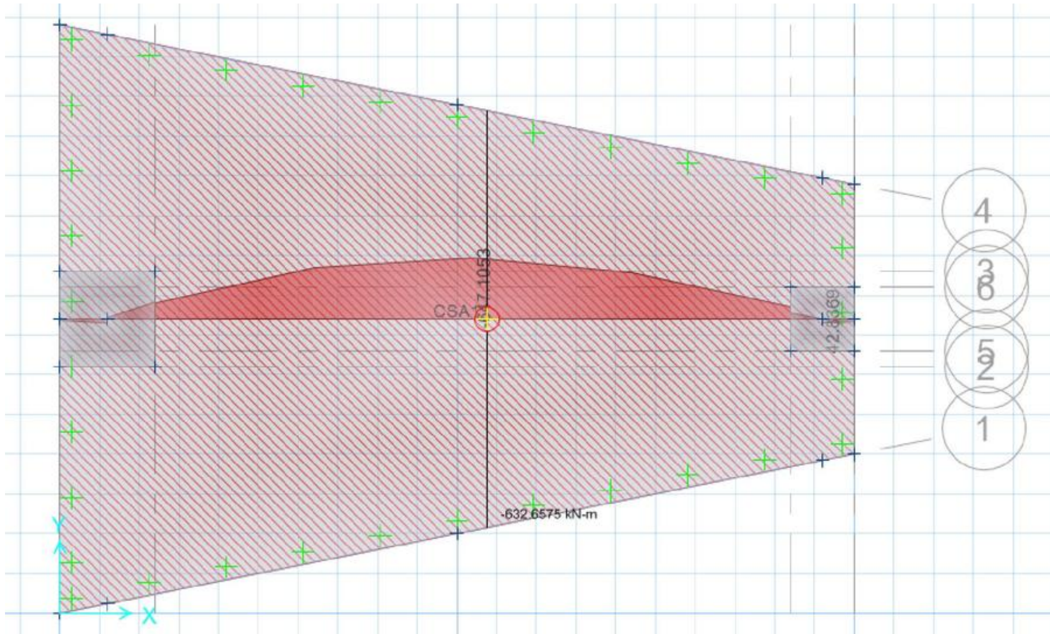


Figure 179

The '**Display**'>'**Show Slab Design…**' command results in the window shown in Fig 180. In this window, several options can be set in order to display the footing design in the way we wanted to. The '**Choose Display Type**' box allows us to select the '**Design Basis**' between '**Strip Based**' and '**Finite Element Based**'. Unless some differences in the way the design is displayed, there is no difference in the amount of reinforcement between these selections. Through the '**Display Type**' in the same box, it can be selected whether to display flexural reinforcement or shear reinforcement. This box also allows us to impose or not to impose minimum reinforcement during the design. The '**Rebar Location Shown**' box allows us to select which reinforcement, top or bottom or both, to be displayed. The '**Reinforcing Display Type**' box allows us to set the manner in which the amount of reinforcement is displayed. The option whether to show the reinforcing envelop diagram and the reinforcing extent can be set by the check boxes in the '**Reinforcing Diagram**' window. The strip layer direction for which the amount of reinforcement is displayed can be chosen from the '**Choose Strip Direction**' box. The '**Display Options**' box allows us whether to display output in filled diagram or not and whether the values at controlling stations will be displayed or not. If we want to display the amount of reinforcement above some specified reinforcement bar area or spacing, we can use the options in the '**Show Rebar Above Specified Value**' box. When the '**Typical Uniform Reinforcing Specified Below**' radio button is selected, the '**Typical Uniform Reinforcing'** box get activated. In this box, we can set a specific value above which the

reinforcement amount will be displayed. The reinforcement diagram output, for the options set in Fig. 180, will be shown below in fig. 181.

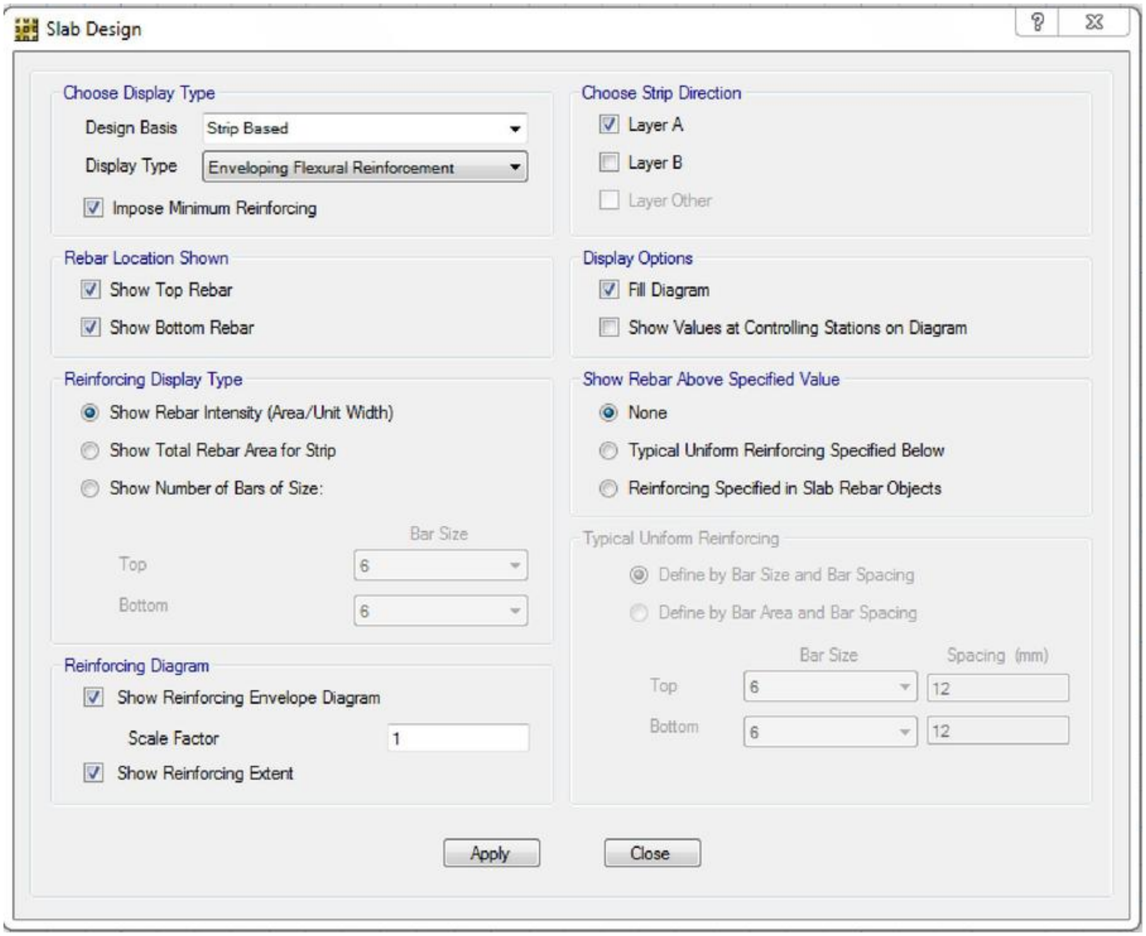


Figure 180

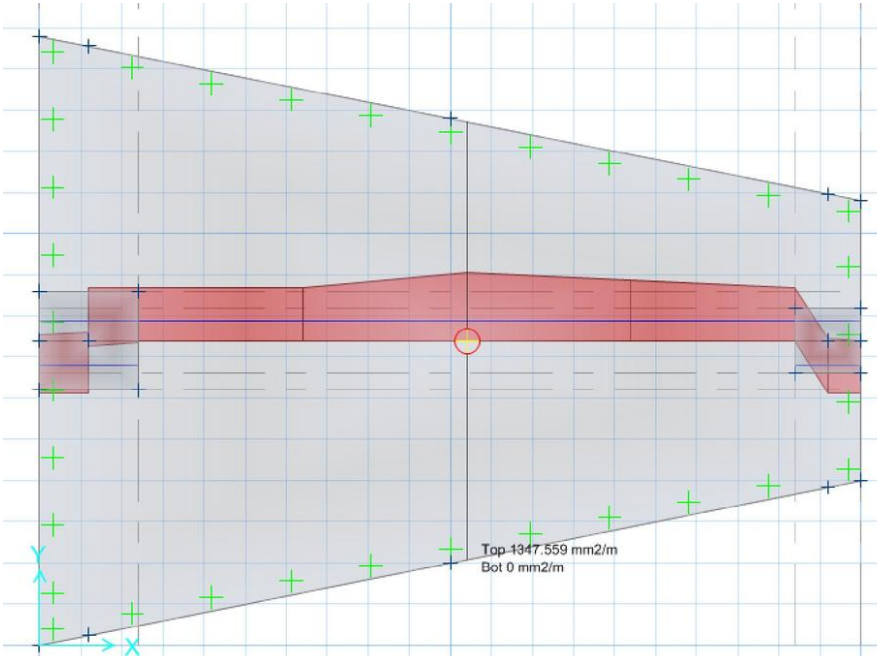


Figure 181

As can be noticed from the footing design diagram in fig. 71, the top reinforcement across strip layer A is around 0mm^2/m for bottom reinforcement and 1348mm^2/m for top reinforcement.

The design outputs can also be displayed in tabular format by clicking on the '**Show Tables…**' menu item from the '**Display**' menu or by just clicking on the equivalent icon  from the tool bar below the menu bar and the following window will pop up.

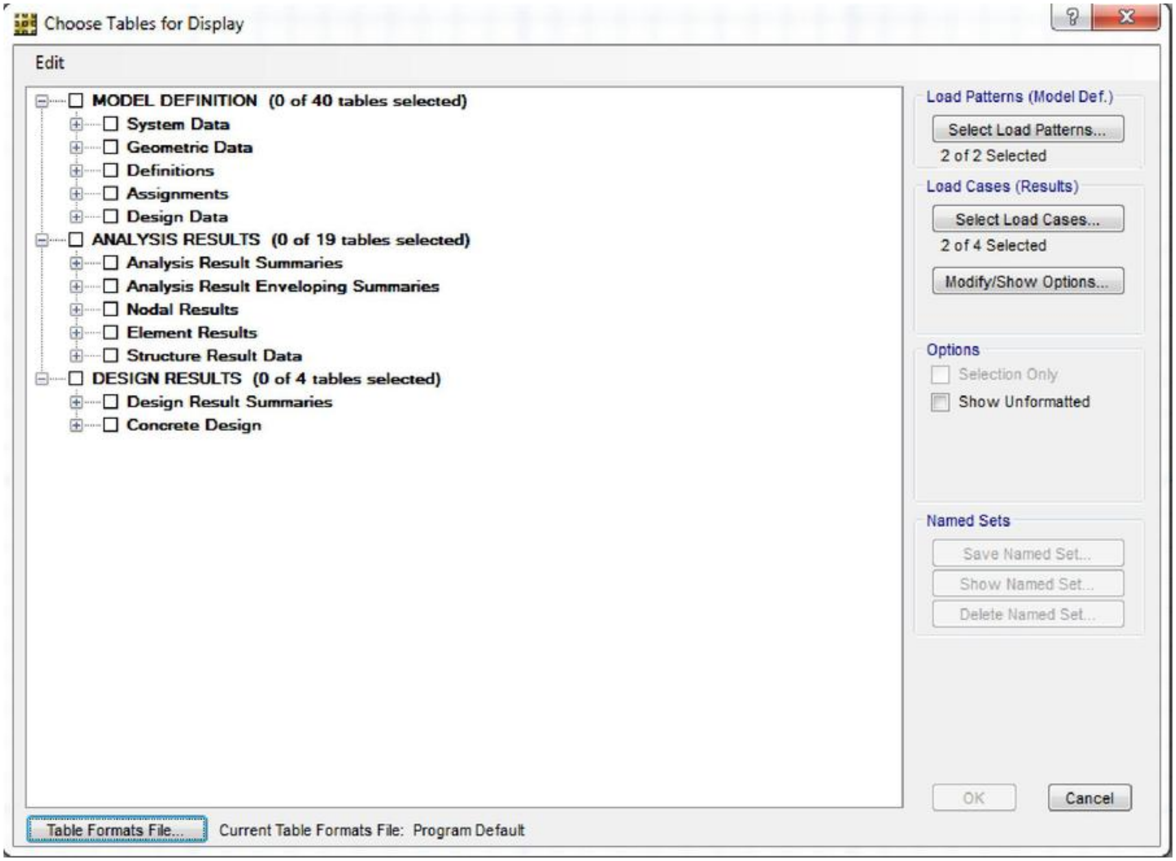


Figure 182

In this window, we can select any of the model definitions or analysis results or design results and press '**OK**' to display the quantity which we want to have a look at. By using the right hand side buttons in the window, the load patterns and the load cases can be selected.

STEP 10: Detailing

After running the analysis and after checking that the results are reasonable, the detailing will be done. However, before running the detailing, the detailing preferences can be set from the '**Detailing**' menu. From the '**Detailing Preferences…**', the likes of dimensional units and material quantity units can be selected. From '**Slab/Mat Detailing Preferences…**', the likes of rebar curtailment options, the rebar detailing options, rebar selection rules and preferred rebar sizes can be selected. The '**Drawing Sheet Set-up…**' menu allows us to set-up the contents of the drawing sheet. The '**Drawing Format Properties…**' allows us to set some formats in which the output displayed.

To run the detailing, go to '**Run**' menu and click on '**Run Detailing…**' or simultaneously press '**Shift**' and '**F5**' keys or just click on the run detailing icon  from the tool bar just below the menu bar. Then, the '**Run Detailing Options**' window pops up so that we set the detailing options. Set the detailing options which you want and click '**OK**'.

Once the detailing is run, the detailing can be displayed. The detailing display options can be best accessed from the '**Model Explorer**'. When expanded in full, the '**Detailing**' tab of the model explorer looks like:

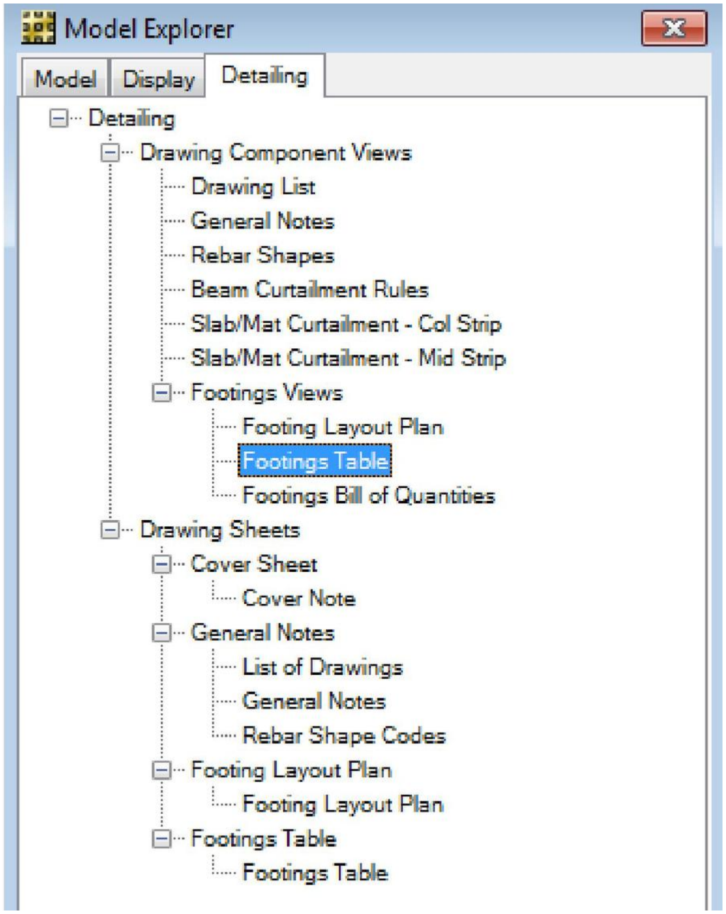


Figure 183

By clicking on any of the options from the detailing tab, a desired detailing can be displayed. For instance, by double clicking on the '**Footing Table**', the following detail of footing can be shown. In the detail, the foundation is displayed as a rectangular footing, which is not in accordance with the actual geometry. Thus, good judgment is required while utilizing the design results.

FOOTINGS TABLE

SR. NO.	TYPE	NOS	LX	LY	T	REBARS-A	REBARS-B
1	F1	1	5.000 M	3.700 M	0.600 M	10-8	11-10

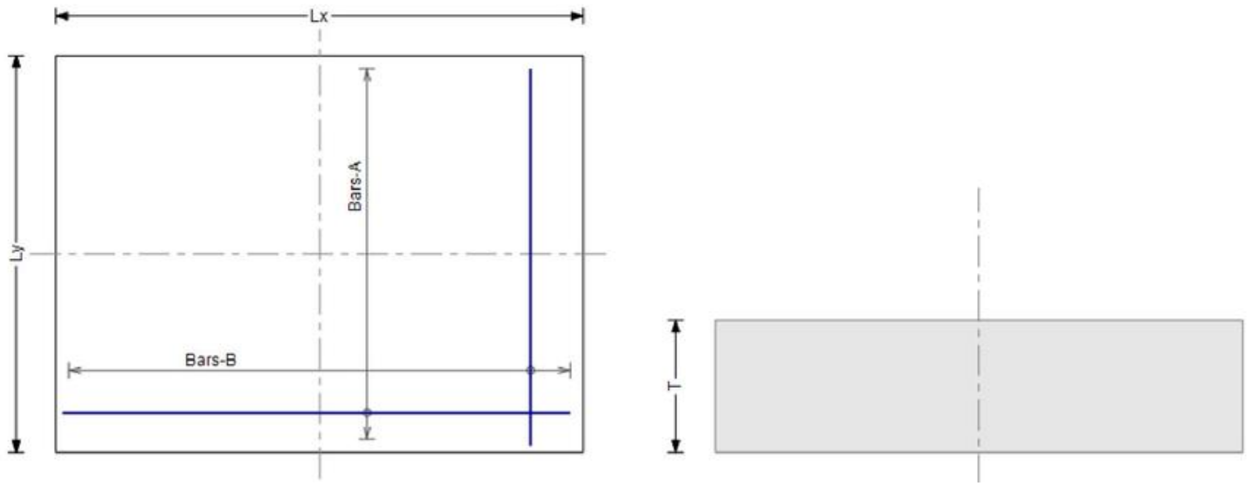


Figure 184

In this detail, the diameter of the reinforcement which is used is 10mm. If you want to change the diameter….

Apart from this detail, other details can also be shown.

STEP 11: Reporting

The last step of foundation design is reporting. Before creating the report, the report preferences should be set up. To do this, go to the 'File' menu and click on 'Report Set-up…' and the following window pops up.

In this '**Report Setup Data**' window, the user preferences regarding the reporting such as the report output type, the report page orientation and the report items can be set along with the load patterns and load combinations. Once the preference is set, the report can be created by clicking on '**Create Report**' command in the '**File**' menu. The '**Advanced Report Writer**' command in the same menu can be used to set some advanced reporting formats.

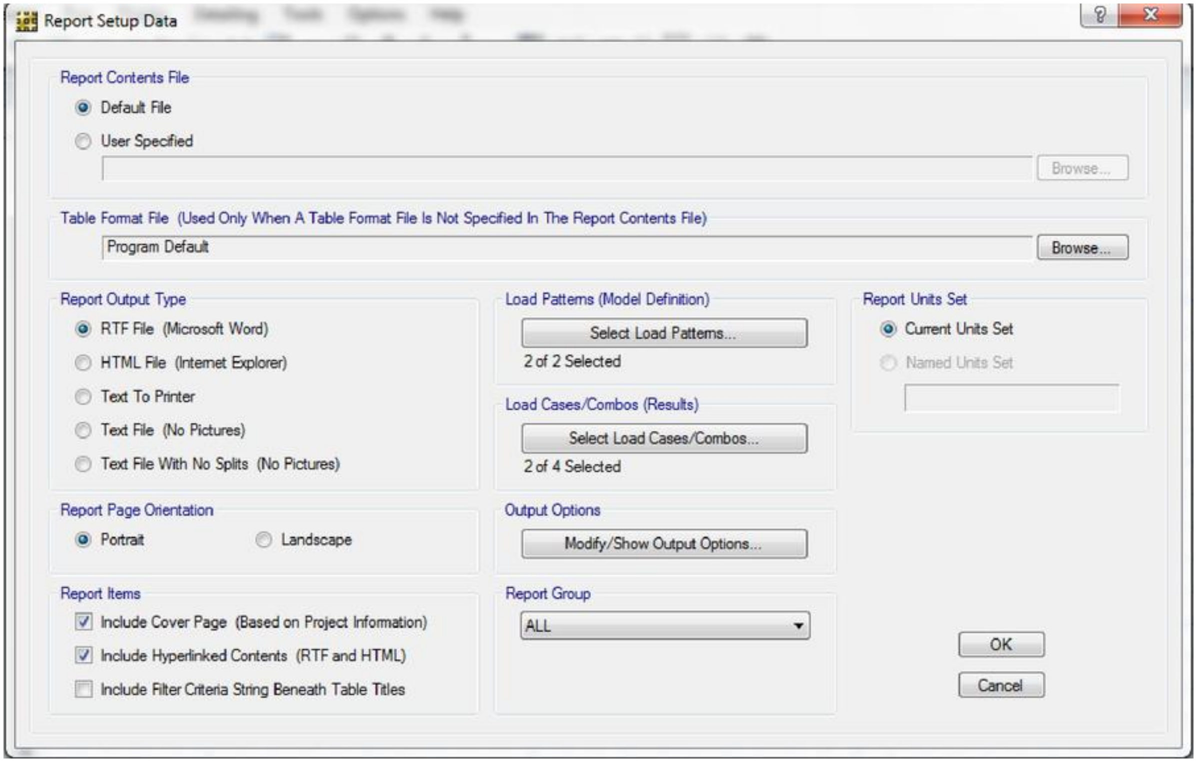


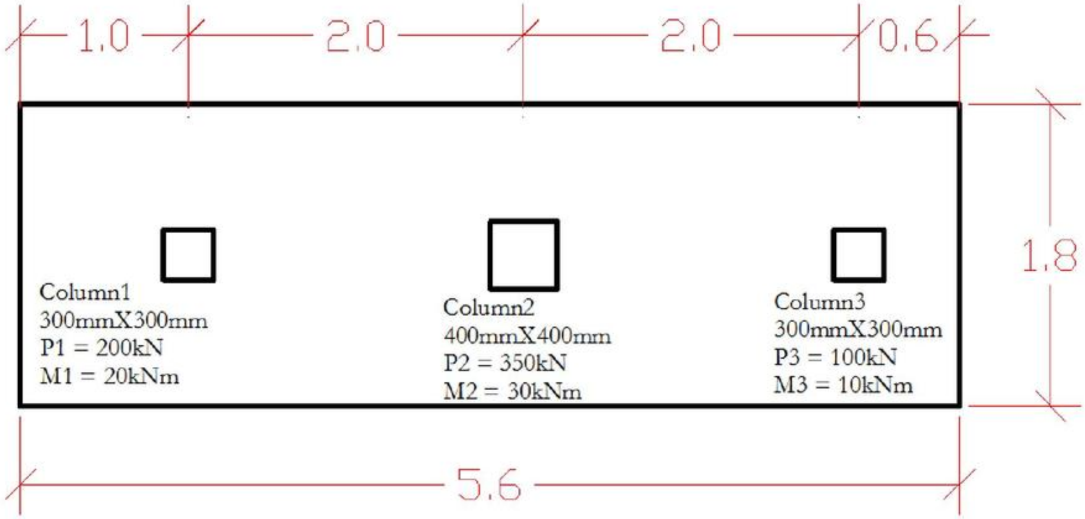
Figure 185

This concludes the tutorial for the design of single footing using the grids.

3.3. From AutoCAD

STEP 1:

Design the combined foundation shown below with the given parameters:



- *Ultimate bearing capacity of the soil* : 100kN/m^2
- *Maximum allowable settlement of the foundation*: 10mm
- *Grade of concrete*: C-20 (20MPa 28-day characteristics cube strength)
- *Grade of reinforcement bar (rebar)*: S-460 (460MPa characteristics yield strength)
- *Overall thickness of the foundation*: 600mm
- *Concrete cover* : 50mm
- *All the given loads are factored (ultimate) loads.*

Creating the Model

To create a footing model from an AutoCAD file, first draw the plan view of the foundation on AutoCAD. In this case, it will be a rectangle with dimensions of 5.6mX1.8m (footing) and three squares (foundation columns). When you draw the plan view on AutoCAD, make sure that you are drawing it on a new layer. If you want the origin of the SAFE mode to coincide with the center of the circular foundation, make the center of the circle coincide with the global origin of the AutoCAD file when you draw it. Then save the AutoCAD file in DXF format. To save an AutoCAD file in DXF format, go to '**File**' menu in the AutoCAD file and click on '**Save As'**. At the bottom of the upcoming window, just below the text field where you will enter the file name, you will see '**Files of type**' with a drop-down menu list. From the drop down menu list, select the option which has '**(.dxf)**' at the end. Then enter the file name and save it at any location in your computer where you can easily remember close it. This saves the file in DXF format.

Once you saved the AutoCAD file in DXF format, open the SAFE software and when you use the command '**File**'>'**Import**'>'**.DXF/.DWG Architectural Plan**' or when you simultaneously click on '**Ctrl**'+'**Shift**'+'**I**' keys, and you will be prompted to open a file. Open the DXF file from the location where you saved it and the following window will pop up.

In this window, just change the '**CAD Drawing Units**' to '**m**' since in the AutoCAD file is the model is drawn in meter units. Then press '**OK**'.

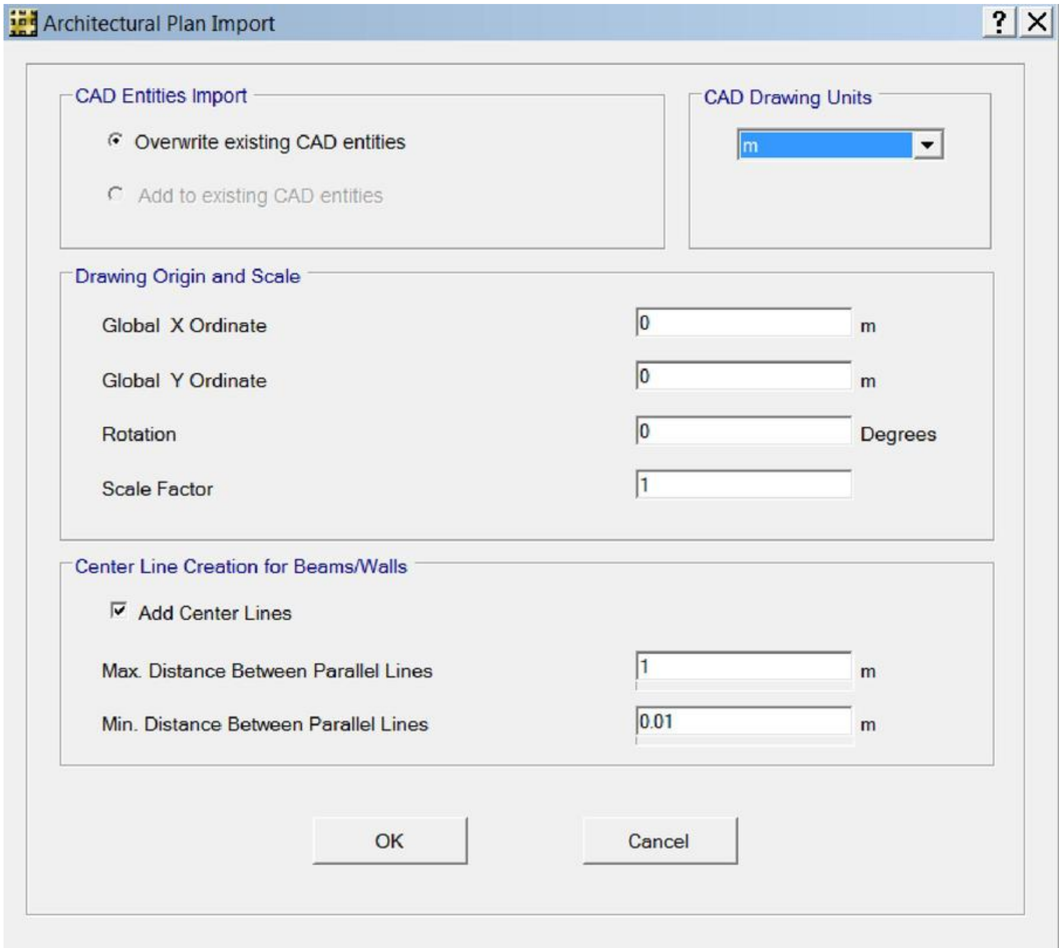


Figure 186

Then, the following model appears in the plan view. As can be noted, the left bottom corner of the footing coincides with the origin of the coordinate system. To achieve this, make sure that that this corner coincides with the origin of the Global coordinate system while drawing the plan view on the AutoCAD.

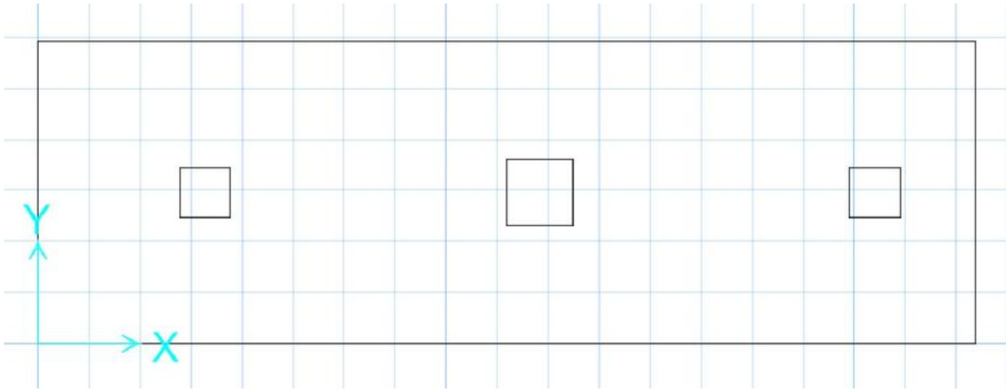
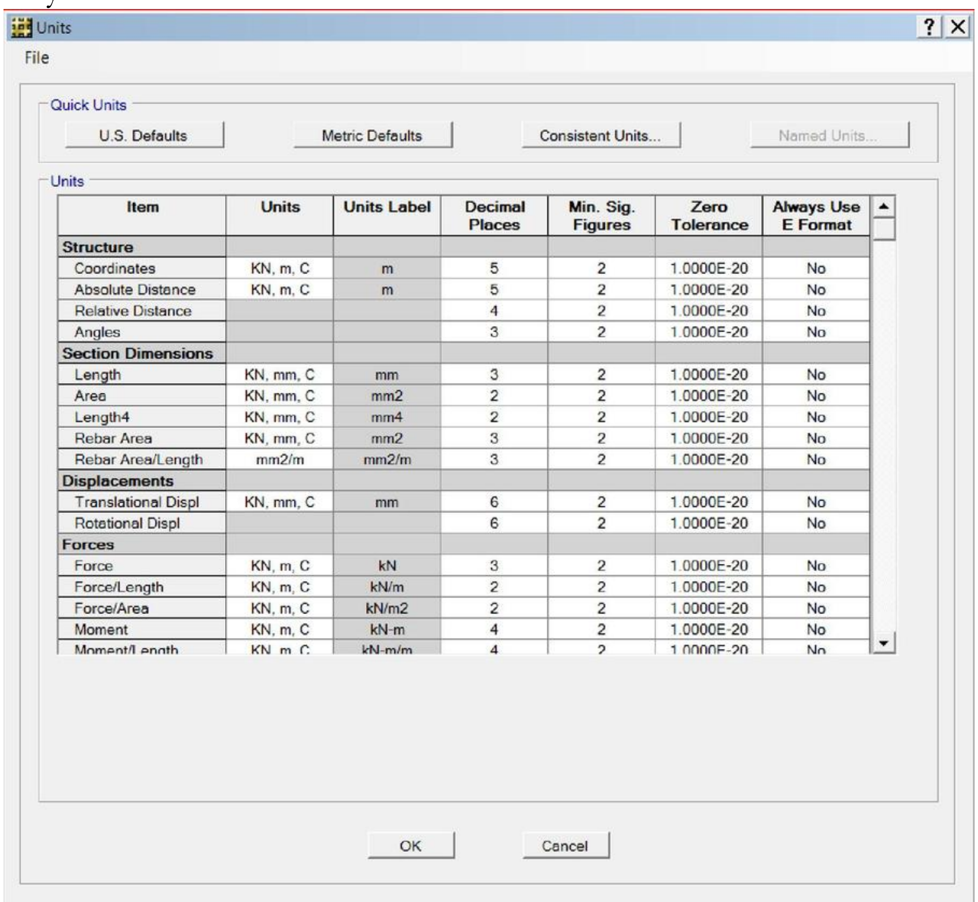


Figure 187

After the model is created, the preferences regarding the units, the design codes and concrete cover should be entered. To change the units, click on '**Units…**' button which is found in the right bottom corner of the main window and the following widow will pop up. In this window, click on '**Metric Defaults**' and press '**OK**' as we will be using metric units. If you want to use U.S. units click on '**U.S. Defaults**' and if you want to use a particular metric or U.S. unit consistently click on '**Consistent Units…**' and select that particular unit which you want to use consistently.



Item	Units	Units Label	Decimal Places	Min. Sig. Figures	Zero Tolerance	Always Use E Format
Structure						
Coordinates	KN, m, C	m	5	2	1.0000E-20	No
Absolute Distance	KN, m, C	m	5	2	1.0000E-20	No
Relative Distance			4	2	1.0000E-20	No
Angles			3	2	1.0000E-20	No
Section Dimensions						
Length	KN, mm, C	mm	3	2	1.0000E-20	No
Area	KN, mm, C	mm2	2	2	1.0000E-20	No
Length4	KN, mm, C	mm4	2	2	1.0000E-20	No
Rebar Area	KN, mm, C	mm2	3	2	1.0000E-20	No
Rebar Area/Length	mm2/m	mm2/m	3	2	1.0000E-20	No
Displacements						
Translational Displ	KN, mm, C	mm	6	2	1.0000E-20	No
Rotational Displ			6	2	1.0000E-20	No
Forces						
Force	KN, m, C	kN	3	2	1.0000E-20	No
Force/Length	KN, m, C	kN/m	2	2	1.0000E-20	No
Force/Area	KN, m, C	kN/m2	2	2	1.0000E-20	No
Moment	KN, m, C	kN-m	4	2	1.0000E-20	No
Moment/Length	KN, m, C	kN-m/m	4	2	1.0000E-20	No

Figure 188

To select the design code and other design preferences, go to '**Design**' menu and click on '**Design Preferences…**' and the following window will pop up. In the '**Code**' tab of this window, change the '**Design Code**' to '**BS 8110-1997**' or any other code which you want to design the circular foundation with. In the '**Min. Cover Slabs**' tab, for '**Non-Prestressed Reinforcement**' , both the '**Clear Cover Top**' and '**Clear Cover Bottom**' should be set to 50mm as the concrete cover in this design problem stated to be 50mm. The '**Preferred Bar Size**' can be set to any reasonable value. Here, the '**Preferred Bar Size**' is set to #14 which is the bar number which will be used as the main reinforcement in the foundation. Leave the rest as they are and press '**OK**'.

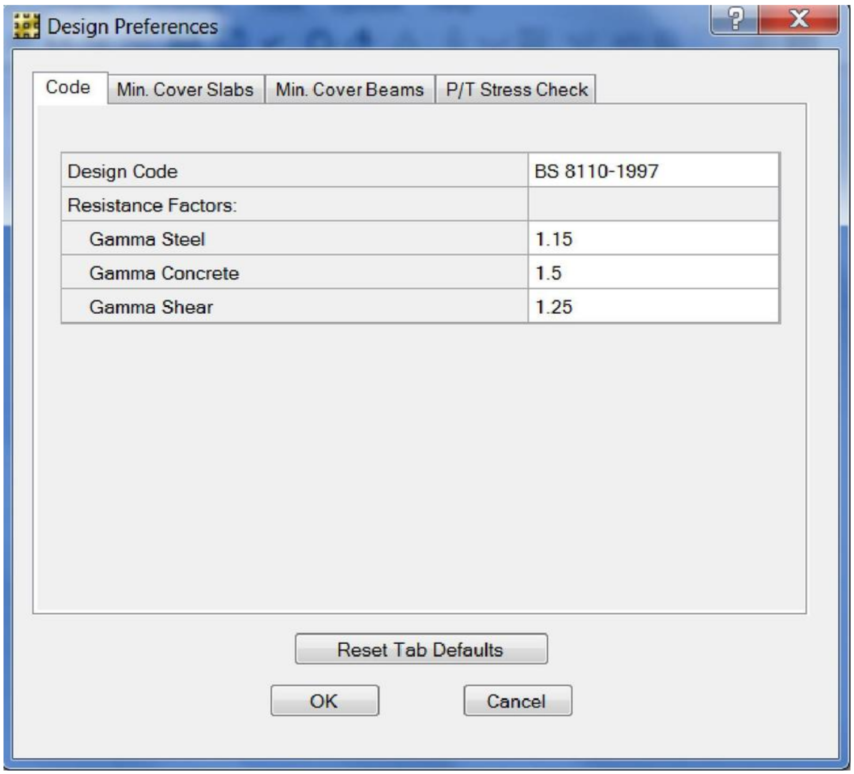


Figure 189

STEP 2: Defining Material Properties

The materials which are involved in the design of the footing should be defined before the analysis. These materials are the concrete, the reinforcement bar (rebar) and the soil support. Thus, these properties will be defined here.

The material definition can be carried out in two ways.

The first one is through the '**Define**' menu in the menu bar. The other one through the model explorer on the left hand side of the home window. To define material properties through the former method, click on '**Define**' menu and again click on '**Materials…**' resulting in the following window depending on prior material definitions.

The list in the '**Materials**' box may not be exactly as it appears in your widow. However, that doesn't bring any change in the outcome of the design process as you can customize this list any time.

The '**Add New Material Quick…**' button allows you to define materials quickly from a list of pre-defined materials. The '**Add New Material**' button allows you to define materials by changing their properties. The '**Add Copy of Material**' button allows you to define a material with same property as an already defined material. The '**Modify/Show Material**' button displays the property of an already defined material with the possibility of modification. The '**Delete Material**' button, when active, deletes a defined material property.

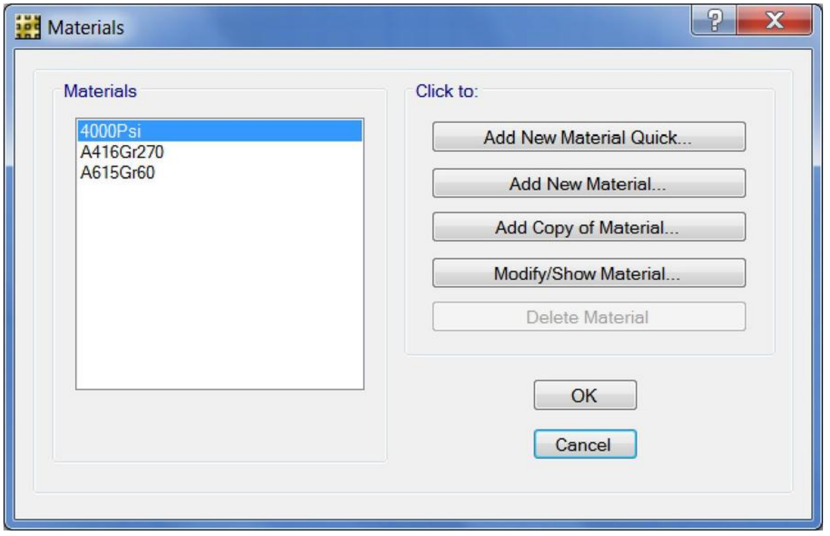


Figure 190

For this particular problem, we will add new concrete and a rebar materials using the '**Add New Material…**' button. Thus click on this button and the following window will pop up.

This is the window where concrete properties will be modified. Put any name for the material in the text field in front of '**Material Name**'. But, it is important to make sure that the concrete with the defined material name is assigned for the footing. For this particular problem, let us use the name of the concrete as '**C20**'. Since we are defining a concrete, the '**Material Type**' should be set to '**Concrete**'.

The unit weight of reinforced concrete may vary depending of the design code of your country. Thus, enter the unit weight of reinforced concrete stipulated in your country code in the text field in front of '**Weight per Unit Volume**'. For this particular example, we will use 25kN/m^3.

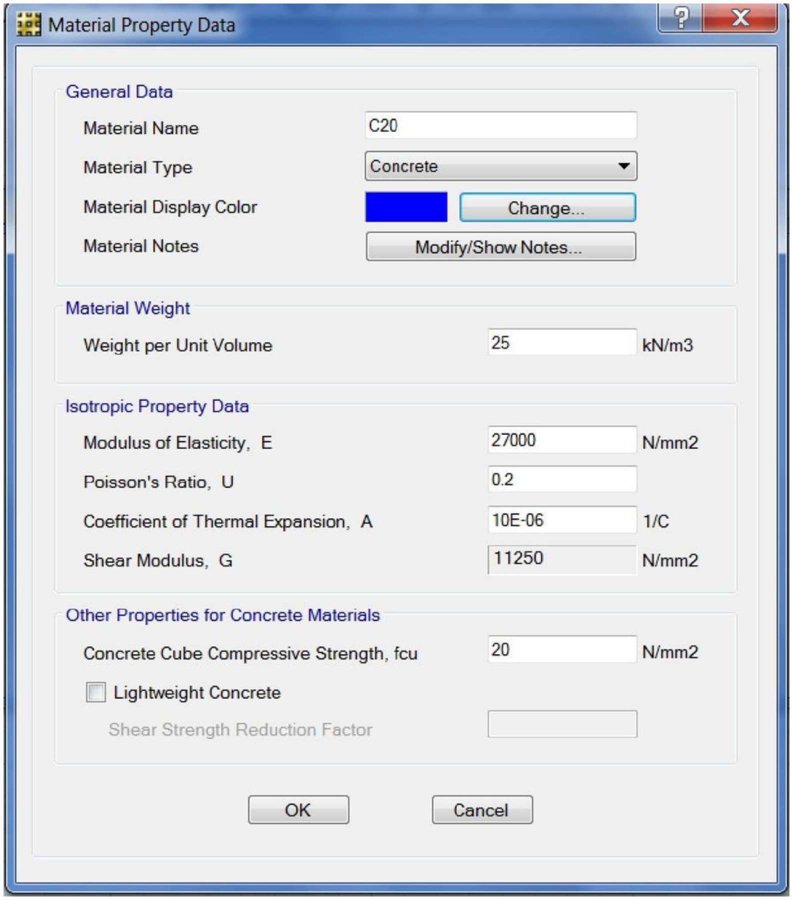


Figure 191

For C-20 concrete, the modulus of elasticity according to BS 8110-1197 is around 27GPa. Thus, enter this value in the text field in front of '**Modulus of Elasticity, E**'. If you selected another design code in step 1 while creating the model, you should refer to actual value of this parameter from the code and enter it accordingly. Be aware of the units though.

The values of Poisson's ratio and coefficient of thermal expansion may also be defined in the design code and should be entered accordingly. For this particular problem, a value of 0.2 for '**Poisson's Ratio, U**' and a value of 10x10^{-6}/^{o}C for '**Coefficient of Thermal Expansion, A**' will be entered. The '**Shear Modulus, G**' will be automatically calculated in an un-editable text field.

The grade of concrete for this particular problem is C-20. C-20 is a concrete with 28 day characteristics cube compressive strength of 20MPa. The concrete designation may be different for different country codes but the concept is the

same. Therefore, enter 20 in the text field in front of '**Concrete Cube Compressive Strength, fcu**'.

If a lightweight concrete is used, check on '**Lightweight Concrete**' and enter the corresponding '**Shear Strength Reduction Factor**' in the space provided.

When you press on '**OK**', a concrete material with the above properties will be added to the list of materials. This material will be assigned for the footing before the analysis.

After adding a new concrete property, the program returns to the window shown in Fig. 190. To define a rebar property, we will follow the same procedure as we followed while defining the concrete property. Since a new rebar property will be defined, click on the '**Add New Material…**' button. A '**Material Property Data**' window pops up and when you change the '**Material Type**' to '**Rebar**', the window appears to look like the following.

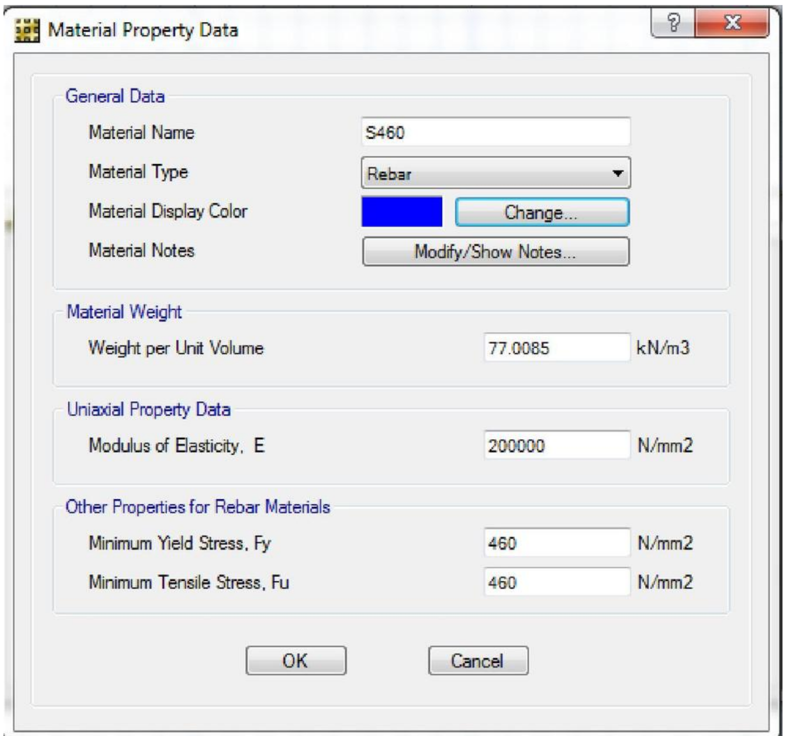


Figure 192

Change the '**Material Name**' to any name you want. Here, we name it '**S460**'. The material type should be '**Rebar**'. The weight per unit volume of steel is stipulated in the design code. For BS 8110-1197, the weight per unit volume is 77.0085kN/m^3. Thus, enter this value in the text field in front of '**Weight per Unit Volume**'. The modulus of elasticity for reinforcement bars according to the same design code is

200GPa. Thus enter this value in the text field in front of '**Modulus of Elasticity, E**' considering the unit.

In the '**Other Properties for Rebar Materials**' box, two quantities are mentioned: minimum yield stress and minimum tensile stress for the reinforcing material. The values of these parameters will be specified in the design code which you defined earlier. If the code assumes that the rebar material exhibits elastic perfectly plastic behavior, the values of these two quantities will be the same. The grade of steel to be used for this particular example is S-460. The yield stress for this type of reinforcement bar is 460MPa. Since the design code of my country assumes that rebars exhibit elastic perfectly plastic behavior, the minimum tensile stress will also be 460MPa. Thus enter 460 in both text fields in front of the '**Minimum yield stress, Fy**' and '**Minimum Tensile Stress, Fu**'. Then press '**OK**' in both '**Material Property Data**' and '**Materials**' windows concluding the material definition step.

The other property which should be defined is the soil support. To define the soil properties, go to the '**Define**' menu and click on '**Soil Subgrade Properties**' menu item and the following window appears.

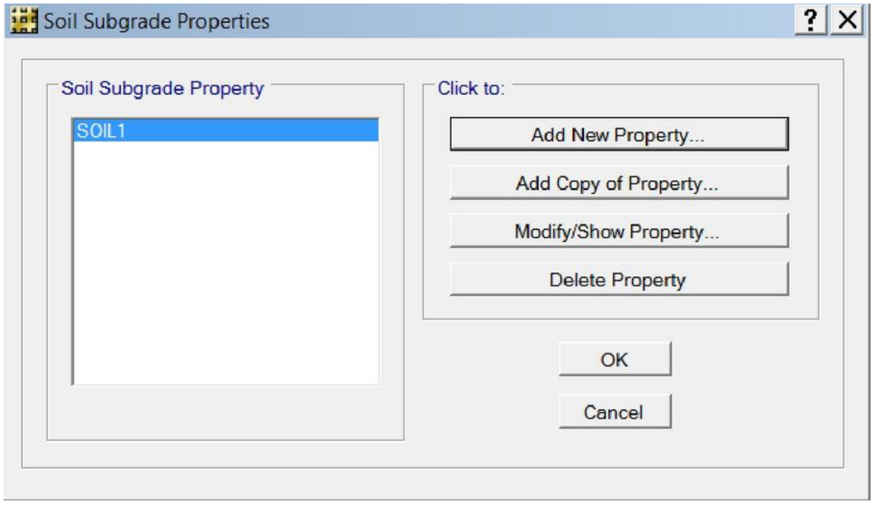


Figure 193

By using the buttons in this window, a new soil property or copy of soil property can be added. An existing soil property can also be modified or deleted. For this problem, let us add a new soil property by using the '**Add New Property…**' button. Thus, click on this button and the following window pops up.

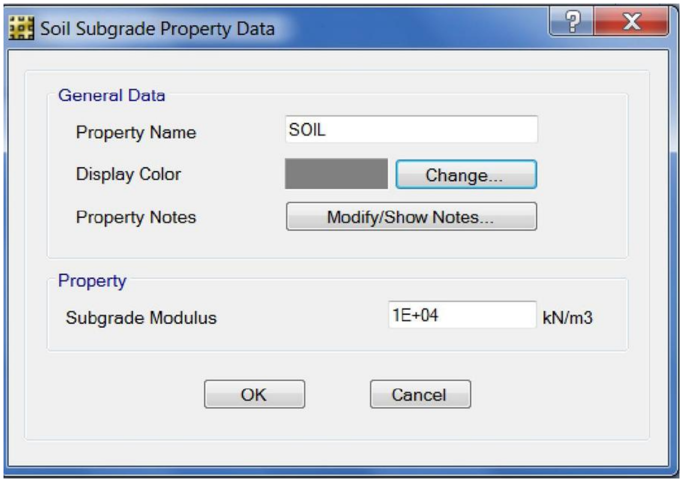


Figure 194

In this window, set the property name to '**SOIL**' and change the subgrade modulus to 10,000kN/m^3. Because, the subgrade modulus which can be assume by the ratio of the bearing capacity by the allowable settlement for this particular example is so. Then press '**OK**' in both the '**Soil Subgrade Property Data**' and '**Soil Subgrade Properties**' windows.

STEP 3: Defining Footing and Column Properties

After defining the material properties, the footing and column properties can be defined. This definition can take place in two ways: from the menu bar and from the model explorer. In SAFE software, footings are modelled as 'footings' and foundation columns are modelled as 'stiff'.

To define footing and column properties from the menu bar, go to '**Define**' menu and click on '**Slab Properties…**'. The following window will pop up.

The '**Add New Property…**' button prompts the user to enter new properties for the footing and foundation column while the '**Add Copy of Property…**' copies the property of an existing slap. The '**Modify/Show Property…**' allows the user to show the property of an existing component with the possibility of modification. When the '**Delete Property**' button is active, it allows the user to delete an existing slab property.

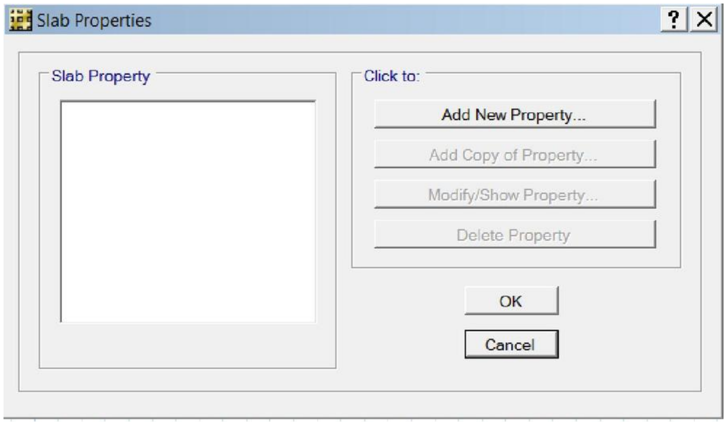


Figure 195

In this case, we use the '**Add New Property…**' button to add new slab properties for the footing and foundation column. Click the button and the following window will pop up.

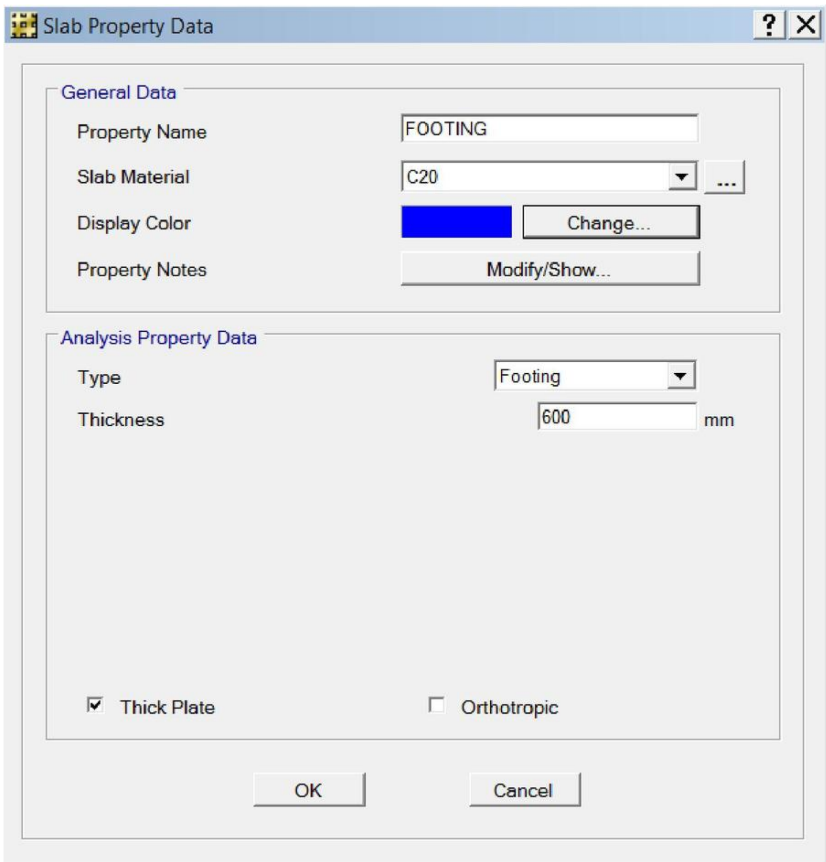


Figure 196

The '**Property Name**' can be assigned with any name; but, in this case, the property name which will be used is '**FOOTING**'. The '**Slab Material**' should be set to the concrete grade which is defined in step 2. Since the name of the concrete material defined in step 2 for this particular problem is '**C20**', a material with this name should be selected from the list.

In the '**Analysis Property Data**' box, '**Type**' should be set to '**Footing**'. The '**Thickness**' value should be set to the thickness of the footing defined in the example. Since the thickness of the footing is 600mm, this value is entered in the text field corresponding to '**Thickness**'. As footings are modelled as thick plates, check the '**Thick Plate**' option. The '**Orthotropic**' check box is selected when a footing with irregular dimension is to be used.

When you press '**OK**', the '**Slab Property Data**' window will be exited and the '**Slab Properties**' window gets activated. Now, the property of the foundation column will be added. To do this, again click on the '**Add New Property…**' button. The following window appears after the click.

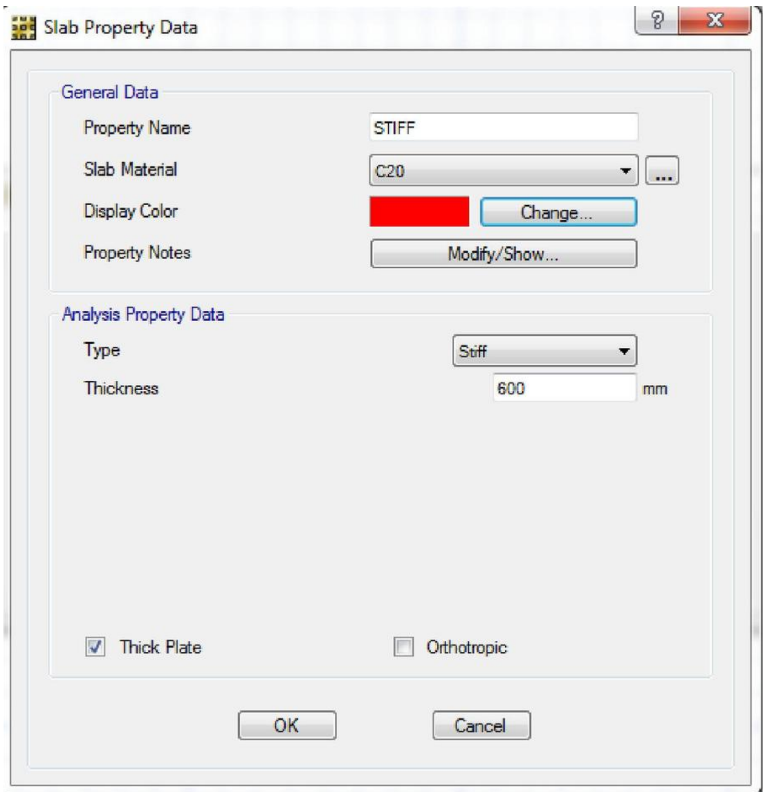


Figure 197

The '**Property Name**' can be any name but we use '**STIFF**'. The '**Slab Material**' should be set to the concrete grade which is defined in step 2. Since the name of the concrete material defined in step 2 for this particular problem is '**C20**', a material with this name should be selected from the list.

In the '**Analysis Property Data**' box, '**Type**' should be set to '**Stiff**' as we are defining the property of column. The '**Thickness**' value should be thickness of the footing which is 600mm. This value should be entered in the text field corresponding to '**Thickness**'. As foundation columns are modelled as thick plates, check the '**Thick Plate**' option. The '**Orthotropic**' check box is selected when a column with irregular dimension is to be used.

When you press '**OK**', the '**Slab Property Data**' window will be exited and the '**Slab Properties**' window gets activated. Again press '**OK**' and exit the window for defining the footing and foundation column.

STEP 4: Defining Load Patterns, Load Cases and Load Combinations

The loads on the foundation should be defined accordingly before the analysis. First, the load pattern should be defined. This can be done from the '**Define**' menu or from the '**Model Explorer**'. This time, we will do it from the model explorer. In the model explorer, expand '**Load Definitions**' and you will see '**Load Patterns**'. When you expand '**Load Patterns**', you will see '**DEAD**' and '**LIVE**'. At the end, the model explorer appears to look like:

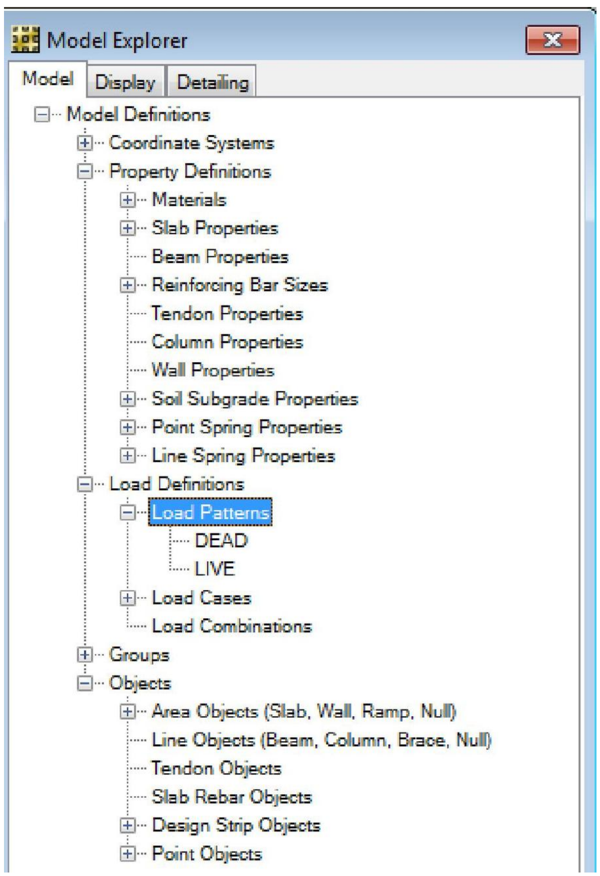


Figure 198

Now, right click on '**Load Patterns**' and click on '**New Load Pattern**' and the following window pops up.

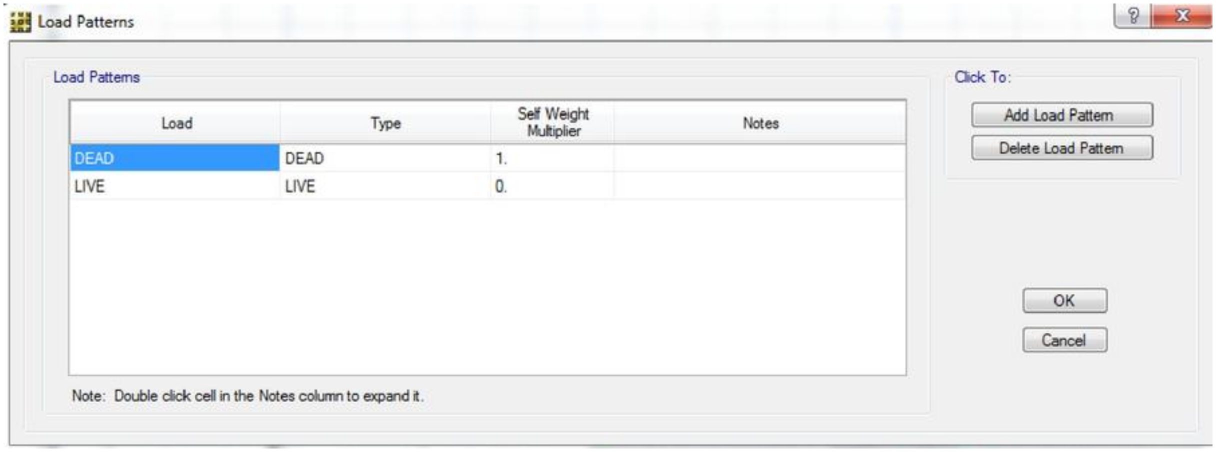


Figure 199

In the '**Load Patterns**' window, two load patterns are already defined: '**DEAD**' and '**LIVE**'. The '**Type**' of the '**Load**' should also be changed accordingly. There are many options for type of loading. The '**Type**' for dead loads should be set to '**DEAD**' and for live loads '**LIVE**'. If there are other types of load patterns on the foundation such as earth quake load, you can add the load pattern with the '**Add Load Pattern**' button. You can also delete any undesirable load pattern using the '**Delete Load Pattern**' button. Since in this example, we have only dead and live loads, we will leave the existing load patterns as they are. The '**Self Weight Multiplier**' value should also be changed accordingly. This value imparts the option whether to consider or ignore the self-weight of the foundation in addition to external loads. If the self-weight of the foundation is already included as an external dead load or if you want to exclude the effect of self-weight from the analysis, the value under '**Self Weight Multiplier**' should be set to zero. In this example, we will consider the self-weight as an additional load to the external dead load. Thus, the value under '**Self Weight Multiplier**' for the '**DEAD**' load is one. For the '**LIVE**' load, it will be zero. Press '**OK**' and the window will be exited.

After this, the load cases will be defined. Load cases are used to dictate the way the loads are applied (statically or dynamically) or the way the structure responds (linearly or non-linearly) for the defined load patterns. To define a load case, go to '**Define**' menu and click on '**Load Cases…**'. The following window will pop up after the click.

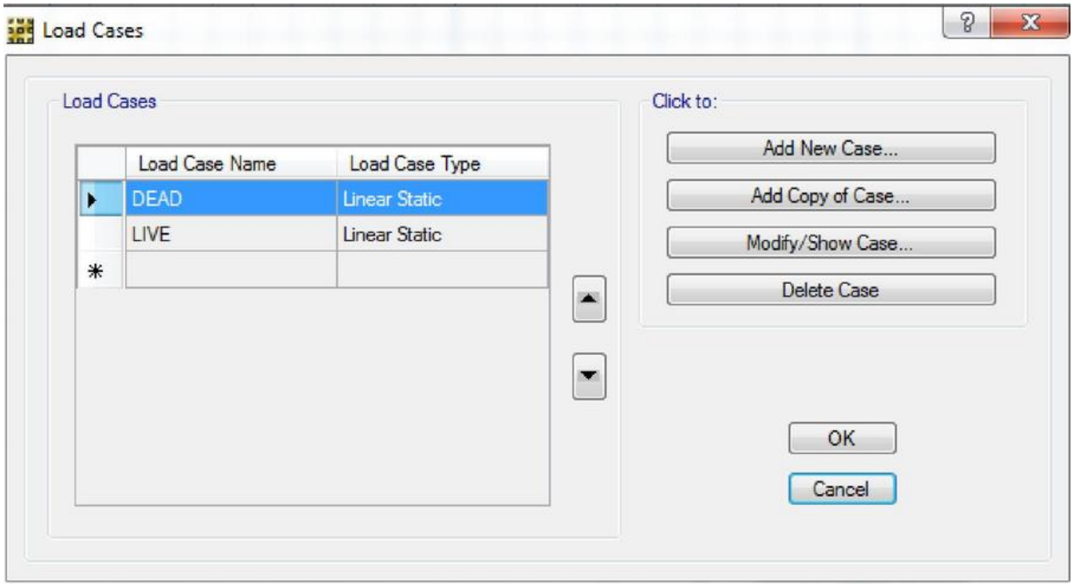


Figure 200

The load patterns which were defined earlier will automatically appear in the list of '**Load Case Name**' of the '**Load Cases**' window. The '**Load Case Type**' column shows the way in which each load pattern will be applied during analysis. If you want to modify this, highlight the load pattern for which you are going to change the load case type and click on the '**Modify/Show Case…**' button. If you click this button, the following window will appear. In this window, you can change the way the load is applied from the '**Load Case Type**' box. The way the structure responds can also be selected from the '**Analysis Type**' box. This problem '**Static**' is for the '**Load Case Type**' and '**Linear**' is selected for the '**Analysis Type**' since the load is static and the foundation responds linearly. The scale factor for the dead load in the '**Loads Applied**' box will be left as one. Press '**OK**' and exit the window.

The load case type for the live load should also be 'Linear Static'. Otherwise, it should be changed by clicking the '**Modify/Show Case…**' button to linear static case. If both the load case types are as desired click '**OK**' and exit the '**Load Cases**' window.

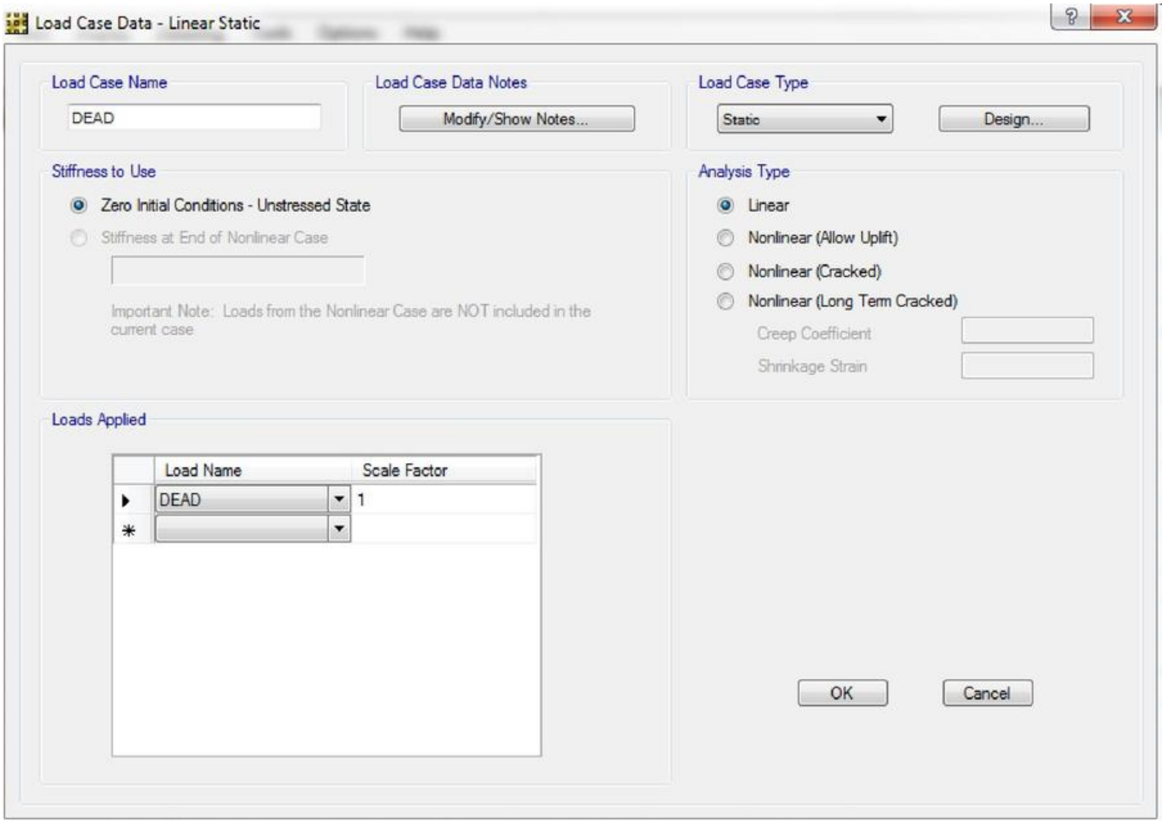


Figure 201

Definition of the load combinations will be the next step. Since the loads are factored, only one load combination will be considered. To define load combination, go to '**Define**' menu and click on '**Load Combinations…**' and the following window pops up.

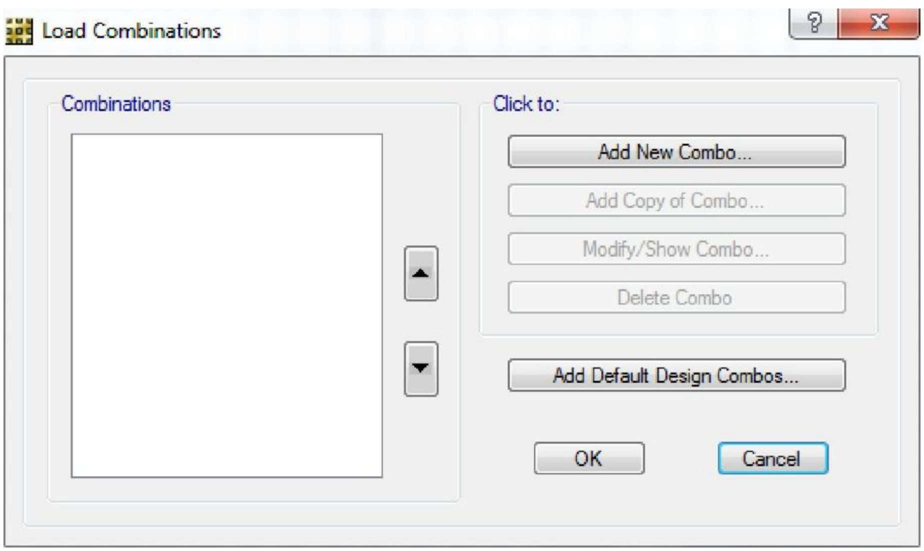


Figure 202

New load combinations will be added through the '**Add New Combo…**' button. The load combination which will be considered is factoring only the self-weight which is considered as a dead load and the other factored loads will be considered as live loads for convenience and the scaling factor for the live loads will be one. When the '**Add New Combo…**' button is clicked, the following window appears.

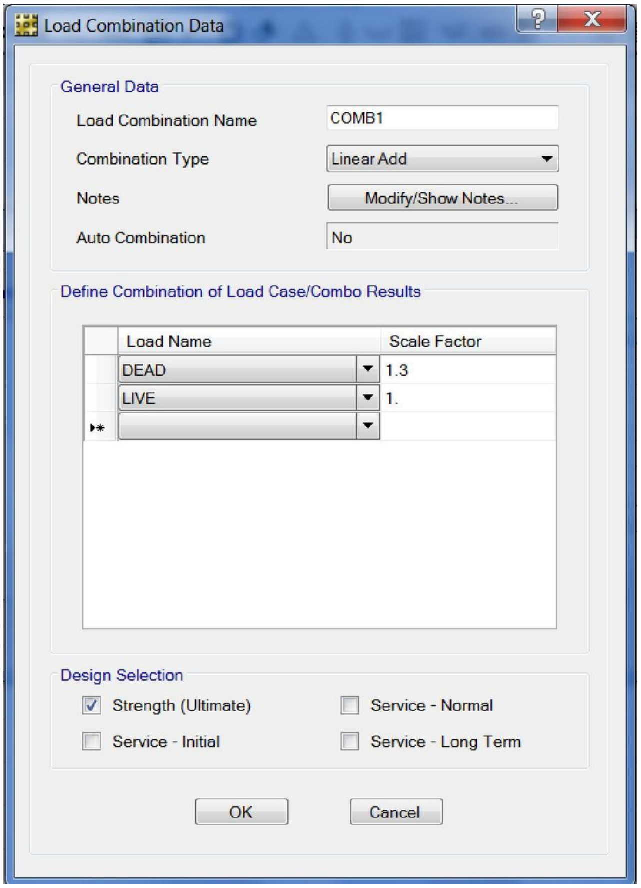


Figure 203

In the '**Load Combination Data**' window, any name can be given for the load combination. The '**Combination Type**' should be set to '**Linear Add**' as the component loads (dead and live) will be added linearly. However, there are also other options from the drop down menu in front of '**Combination Type**'. In the window shown in Fig. 203, the two loads (dead and live) should be activated below the '**Load Name**' column of the '**Define Combination of Load Case/Combo Results**' box. The values in the '**Scale Factor**' column correspond to the partial safety factors for load for the failure mode under consideration. These partial safety factors are specified in the design code of your country. In the design code of my country, the partial safety factor for dead loads for ultimate limit sate case is 1.3 for the case where there are only dead and live loads. For such load patterns, the partial safety factor for live loads for ultimate limit sate case would have been 1.6. But, in this case, since the loads are already factored and for convenience reason since the factored loads are considered as live loads, a scaling factor of unity will be used. Thus, enter a value of 1.3 in front of '**DEAD**' and 1 in front of '**LIVE**' in '**Scale Factor**' column. The failure condition which is being under consideration can be defined by selecting and deselecting the check boxes in the '**Design Selection**' box. The serviceability limit state will not be considered for this case.

STEP 6: Drawing the Footing Components and Design Strips

The footing, the foundation column and a point where the loads will be applied on the foundation column should now be drawn on the grid.

j. Drawing the footing

The footing will be drawn as a rectangular slab. Thus, go to the '**Draw**' menu and click on '**Draw Rectangular Slabs/Areas…**' or click on the equivalent icon  from the left hand side tool bar and the following window will pop up.

Figure 204

Make sure that the '**Property**' is set to '**FOOTING**' and click at one corner of the footing. Then, without releasing the click, drag into the opposite corner and release the click. This draws a rectangular footing.

iv. Drawing the foundation column

While the window in fig 204 is active, change the '**Property**' to '**STIFF**' and draw the three columns by following a similar procedure which you used while drawing the footing.

v. Drawing the points on the foundation columns where the load will be applied

Go to '**Draw**' menu and click on '**Draw Points**', then click on the mid-point of each of the footing and the points will be created. If the cursor could not snap to the midpoint, you can adjust the '**Snap Options**' from the '**Draw**' menu.

After this, the design strips will be drawn. Design strips determine the way in which different quantities related to the reinforcement calculation are calculated. Forces are integrated across the design strips. Thus, the larger the width of coverage of the design strips within the given structure, the higher will be the calculated values of the bending moments and shear forces. Thus, an optimum width of strip is required compromising the safety and economical requirements. The width of the design strip will be specified in the design code. According to the code of my country, the width of design strips for combined foundations in the longitudinal direction should cover the whole area of the footing. In this particular problem, the width of the strips in x-direction is 1.8m. The design strips in the transverse direction will be defined for each column and should extend to a distance of half the depth of the footing from the face of the column on each side. These design strips in X and Y direction are usually defined in SAFE software as layer A and layer B.

To draw the design strip, go to the '**Draw**' menu and click on '**Design Strips**' or simply click on the equivalent icon  from the left hand sided tool bar and the following window pops up.

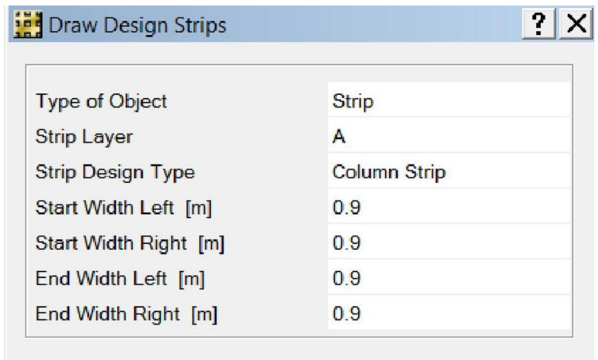
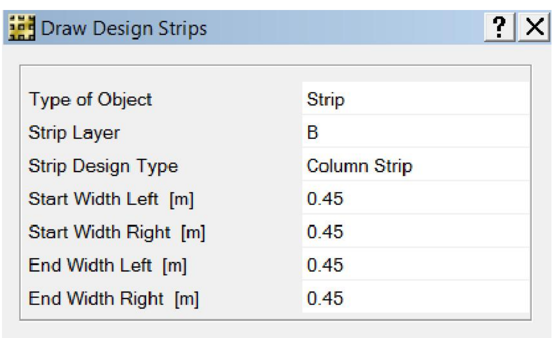
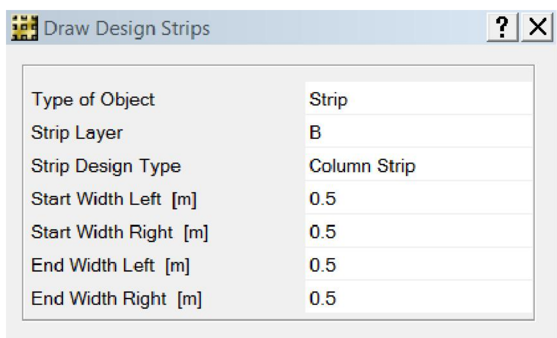


Figure 205

In this window, the '**Strip Layer**' should be selected to be either '**A**' or '**B**'. But if '**A**' is for design strip in X direction '**B**' should be for Y direction and vice versa. The window in fug 205 is for the design strip in X direction. The '**Draw Design Strips**' windows for column1 and column3 in the transverse direction are shown in fig. 206(a) and for column2 in the same direction is shown in fig. 206(b).



(a)



(b)

Figure 206

Since we are drawing a strip around the column to consider maximum moment and shear forces, the '**Strip Design Type**' should be set to '**Column Strip**'. While drawing the design strips, if you can't snap to the intended points on the footing, you can modify the snap options by clicking on the '**Snap Options…**' command from the '**Draw**' menu and adjusting the options which you want to snap to. Or you can use other ways to draw the design strips. The design strip in Y direction can also be drawn in a similar procedure after changing the '**Strip Layer**' to '**B**'.

You can display the design strips by setting the display options by clicking on '**Set Display Options…**' from the '**View**' menu or by simultaneously clicking on '**Ctrl**' and '**W**' keys or by just clicking on the set display options icon  from the tool bar below the menu bar. This results in the following window:

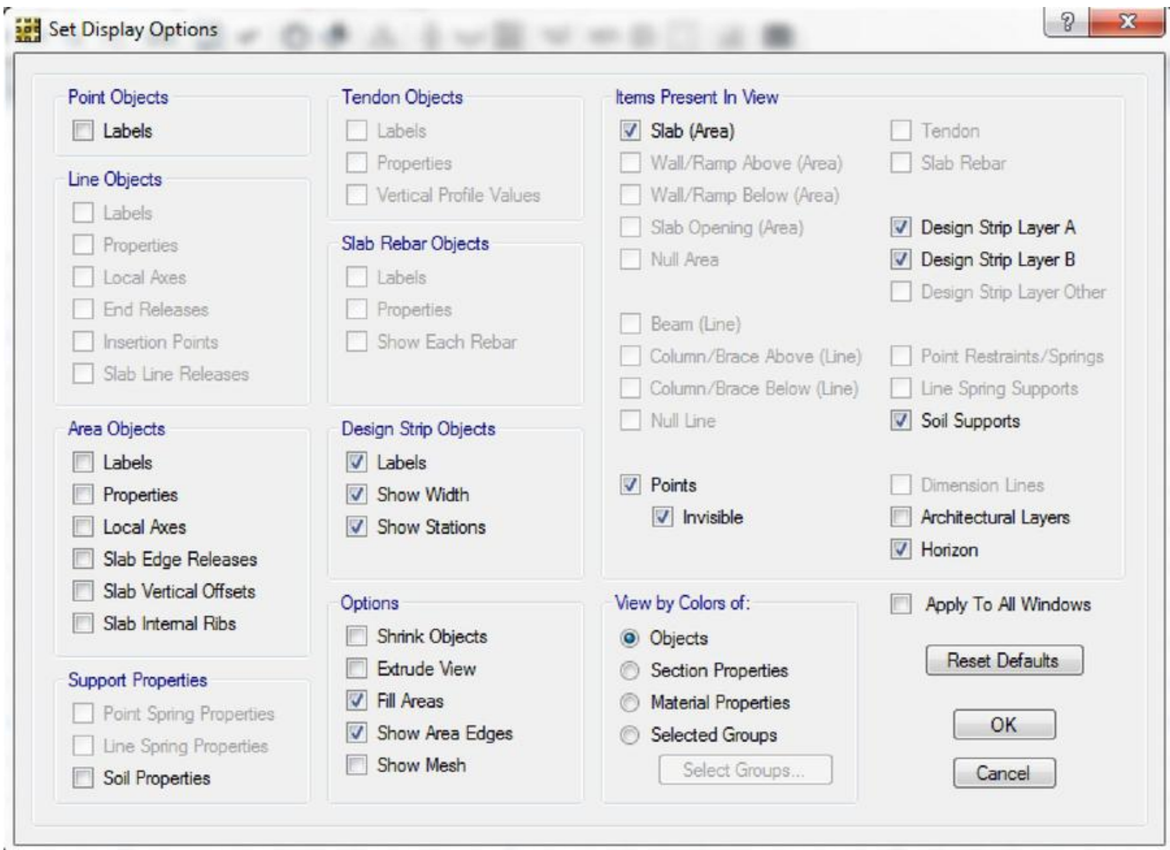


Figure 207

Then, check on '**Labels**', '**Show Width**' and '**Show Stations**' in the '**Design Strip Objects**' box and press '**OK**' and after drawing dimension lines by using the command '**Draw**'>'**Draw Dimension Lines**', the following window appears displaying the design strips in the two directions.

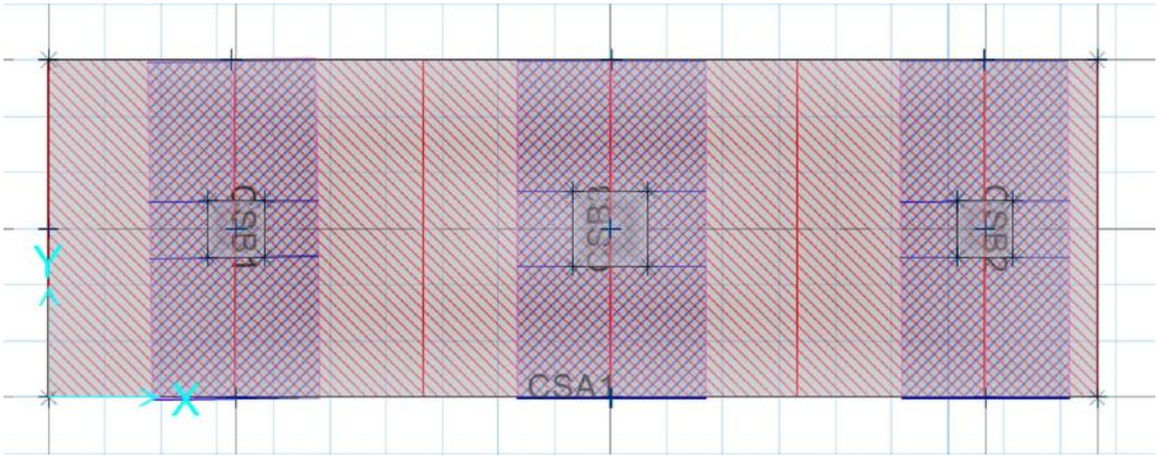


Figure 208

STEP 7: Assigning Slab Data, Support Data and Load Data

The slab data, support data, and load data which are defined in the previous steps should be assigned to the corresponding structural component. The way to do this is first to select the component and next to assign the slab property, support property or loads accordingly. The selection can be done by clicking on the component from the plan view or through the '**Select**' menu. The latter option assures that the component is selected exactly as selection by clicking may result in incorrect selection. Thus, all the selections here will be done from the '**Select**' menu. The assignments will be discussed as follows:

d. *Assigning support data to the footing*
Select the footing through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**'Slab Properties…**'. Then select '**FOOTING**' and press '**OK**'. Then assign the footing property through the following strings of commands '**Assign**'>'**Support Data**'>'**Soil Properties…**'. Then select '**SOIL**' and press '**OK**'.

e. *Assigning reinforcement data to the design strips*
Select each design strip through the following strings of commands '**Select**'>'**Select**'>'**Properties**'>**Design Strip Layers…**'. Then select '**A**' or '**B**' (one at a time) and press '**OK**'. When you right click on the selected strip layer, the '**Slab-Type Area Object Information**' window pops up. In the '**Design**' tab of this window, set the '**Rebar Material**' to '**S460**' and press '**OK**'. Do this for both strips.

f. *Assigning load on the foundation column*

To assign load on the foundation column, right click on the point at the center of the foundation column and a '**Point Object Information**' window pops up. In this window, click on the '**Loads**' tab.

The dead load and the live load can be assigned through the '**Assign Load…**' button. The procedure is: click on '**Assign Load…**' button, then select '**Force Loads**' then press '**OK**' then select either '**DEAD**' or '**LIVE**' depending on which loads you want to enter their values then enter their values accordingly (both the concentrated load and the bending moment) and in the right direction (axis), then select '**Add to Existing Loads**' and press '**OK**'. While doing this, the foundation column dimensions should be entered in the '**Size of Load for Punching Shear**' box of '**Point Loads**' window.

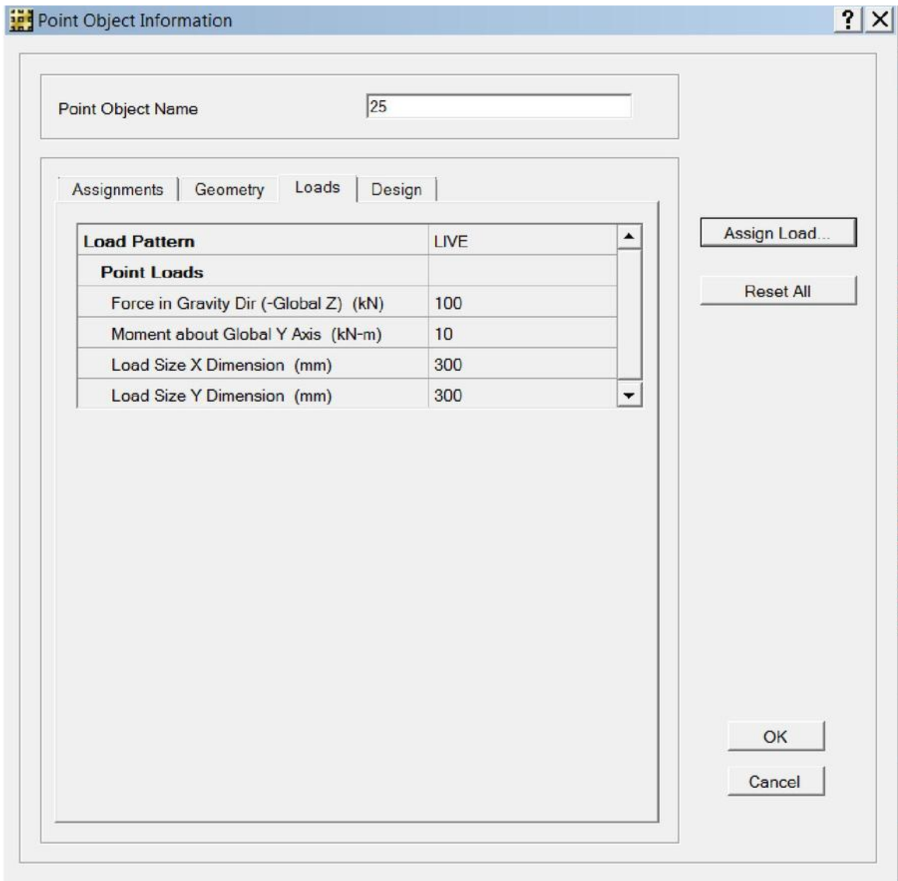


Figure 209

You can also delete any forces which are wrongly entered through '**Delete Existing Loads**' radio button option. In this example, since the factored concentrated load is 100kN and since the bending moment about y-axis is 10kNm, these values are entered in the Gravity Direction for the concentrated load and along y-axis for the bending moment and the final values are shown in the above figure.

STEP 8: Running the Analysis

After this, the analysis can be run. But, make sure that the footing and the foundation column are assigned with the correct rebar material. To do this, right click anywhere in the plan view of the footing and the '**Slab-Type Area Object Information**' window will pop-up.

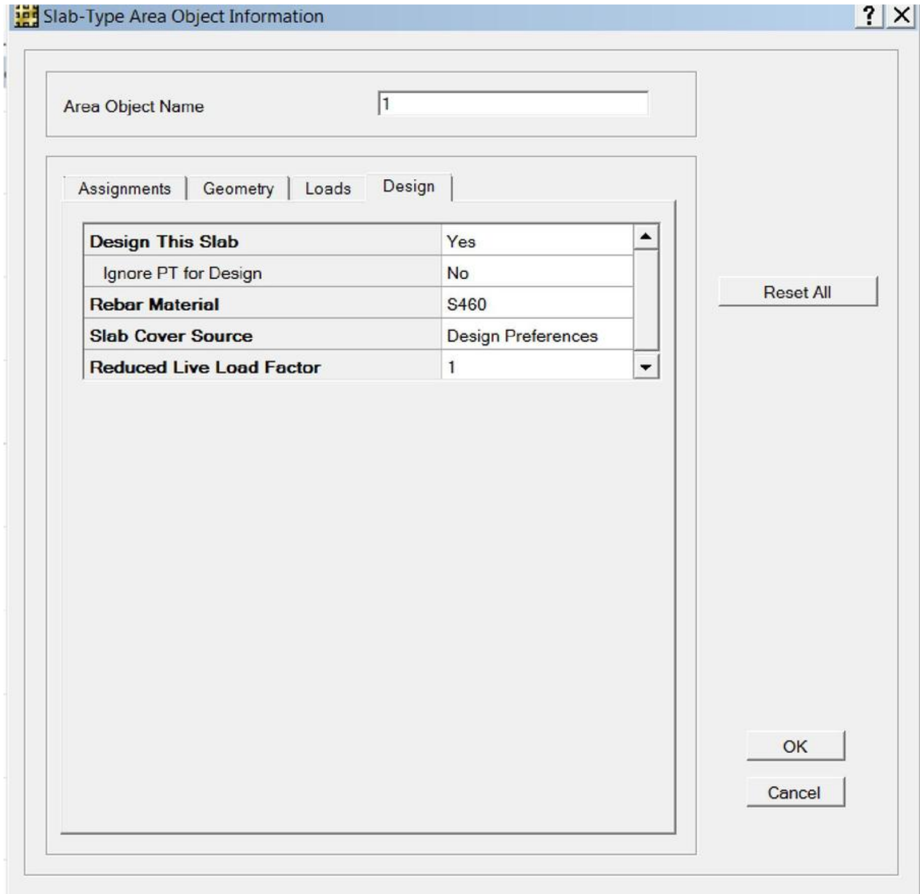


Figure 210

In the '**Design**' tab of this window, set the value of '**Rebar Material**' to '**S460**'. Do this for the column foundation, and both design strips as well. After this, go to the '**Run**' menu and click on the '**Run Analysis**' command.

STEP 9: Displaying the Output

Once the analysis is run, the output will be displayed. Particularly, the punching shear design is of great importance as the footing cannot be designed without the punching shear requirement being adequately satisfied. To do this, go to the '**Run**' menu and click on '**Run Analysis & Design**' command or simply click on the '**F5**' key. When you do this, you will be prompted to save the model, if you haven't already don this. When you save the model, the following window showing the displacement of the soil in a banded figure will be displayed.

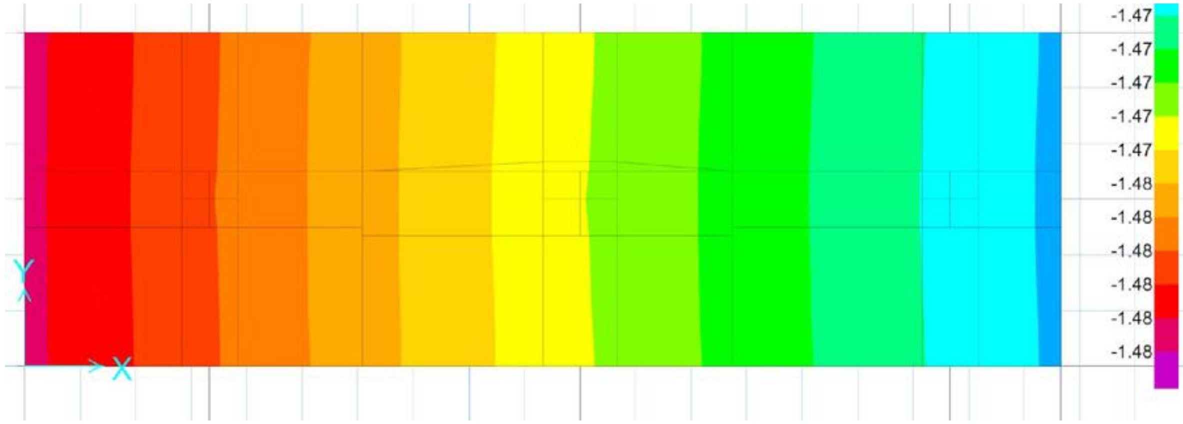


Figure 211

To display the punching shear ratio, go to the '**Display**' menu and click on '**Show Punching Shear Design**'. After this, the punching shear ratio will be displayed in the plan view around the foundation column as in the following figure.

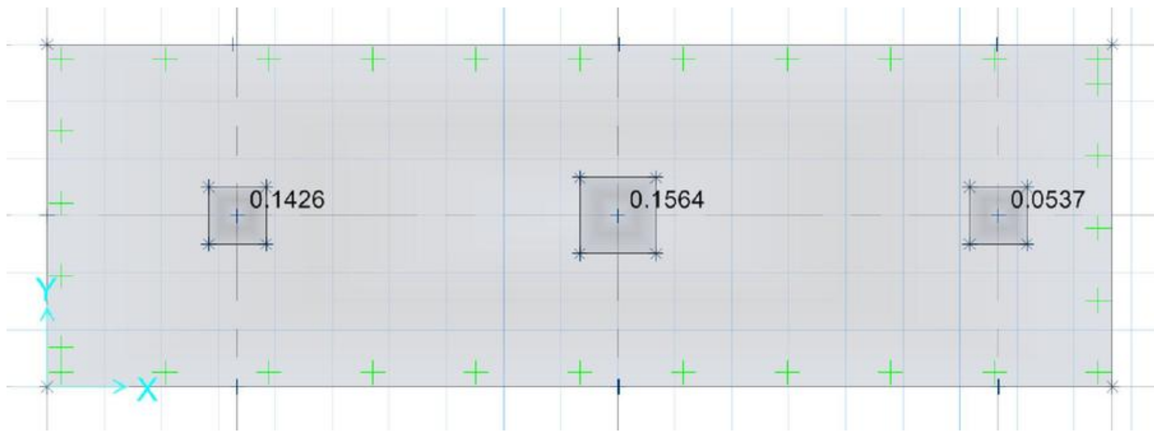


Figure 212

As can be noticed from the figure, the punching shear ratio is 0.1426 for the first column, 0.1564 for the second column and 0.0537 for the third column. Generally, a punching shear ratio less than one indicates the concrete thickness is adequate to resist punching shear and a value greater than one indicates that the punching shear capacity is exceeded somewhere along the critical section. For economical design, it is recommended to keep the punching shear ratio between 0.95 and 1 as very small values of punching shear ratio means excess concrete thickness is used. However, if the punching shear ratio is greater than one, like in this example, the thickness of the concrete should be increased or the grade of the concrete should be increased and the foundation should be re-designed. A detailed quantitative description of the foundation design can also be obtained by right clicking any of the columns shown in Fig 213 as shown below. Several trial may be made by zooming in and out to get the quantitative description.

Figure 213

Through the ‘**Display**’ menu, relevant quantities can be displayed on the screen. For instance, the ‘**Display**’>’**Show Strip Forces**’ command or by simply clicking the ‘**F8**’ key, the following window will be displayed.

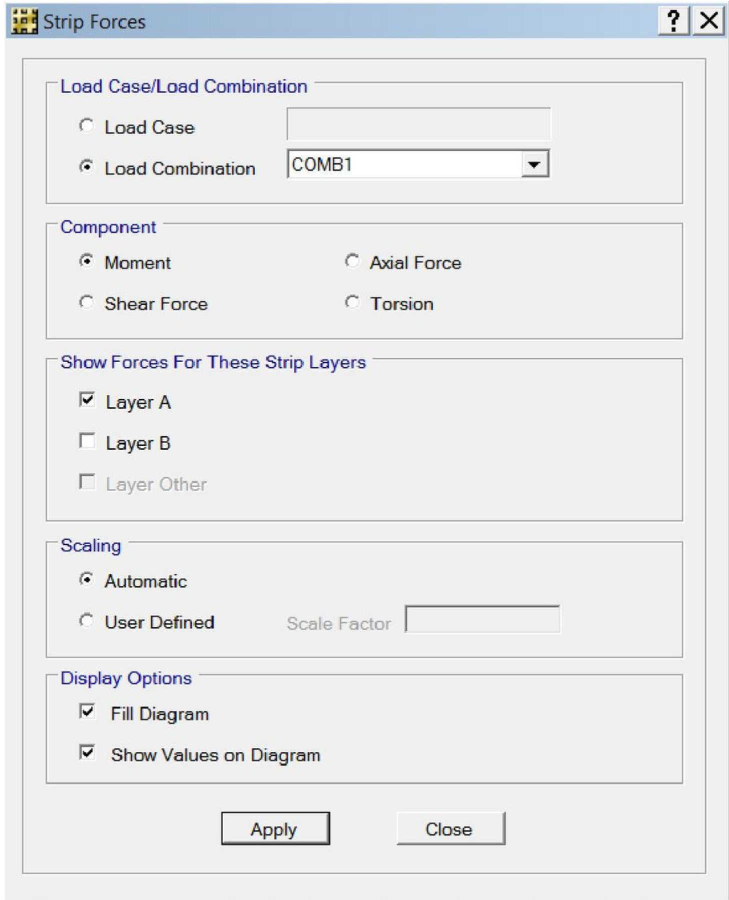


Figure 214

In this window, the '**Load Case/Load Combination**' box provides with radio buttons to select for which particular load case or load combination that we want to display the output. The '**Component**' box contains four radio buttons to select which quantity to display. The '**Show Forces For These Strip Layers**' box allows us to select the strip layer for which the quantity is displayed. Both strip layers can be selected at the same time. From the '**Scaling**' box, we can select whether automatic scaling or user defined scaling is used while displaying the diagram. The '**Display Options**' box allows us to fill or not to fill the diagram and to display or not to display the values on the diagram. For the preferences shown in Fig. 68, the following diagram will be displayed.

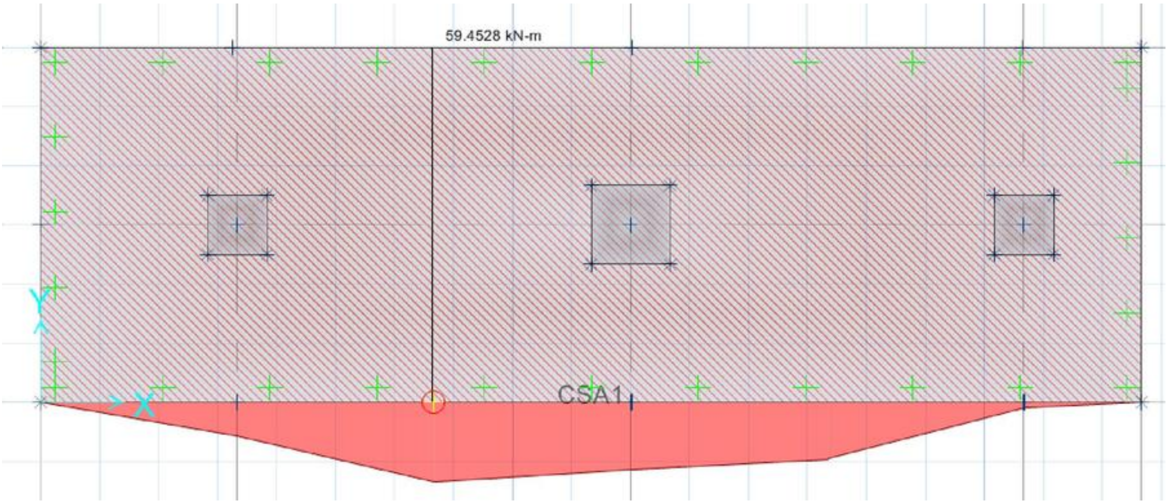


Figure 215

The ‘**Display**’>’**Show Slab Design…**’ command results in the window shown in Fig 216. In this window, several options can be set in order to display the footing design in the way we wanted to. The ‘**Choose Display Type**’ box allows us to select the ‘**Design Basis**’ between ‘**Strip Based**’ and ‘**Finite Element Based**’. Unless some differences in the way the design is displayed, there is no difference in the amount of reinforcement between these selections. Through the ‘**Display Type**’ in the same box, it can be selected whether to display flexural reinforcement or shear reinforcement. This box also allows us to impose or not to impose minimum reinforcement during the design. The ‘**Rebar Location Shown**’ box allows us to select which reinforcement, top or bottom or both, to be displayed. The ‘**Reinforcing Display Type**’ box allows us to set the manner in which the amount of reinforcement is displayed. The option whether to show the reinforcing envelop diagram and the reinforcing extent can be set by the check boxes in the ‘**Reinforcing Diagram**’ window. The strip layer direction for which the amount of reinforcement is displayed can be chosen from the ‘**Choose Strip Direction**’ box. The ‘**Display Options**’ box allows us whether to display output in filled diagram or not and whether the values at controlling stations will be displayed or not. If we want to display the amount of reinforcement above some specified reinforcement bar area or spacing, we can use the options in the ‘**Show Rebar Above Specified Value**’ box. When the ‘**Typical Uniform Reinforcing Specified Below**’ radio button is selected, the **‘Typical Uniform Reinforcing’** box get activated. In this box, we can set a specific value above which the reinforcement amount will be displayed. The reinforcement diagram output, for the options set in Fig. 108, will be shown below in fig. 109.

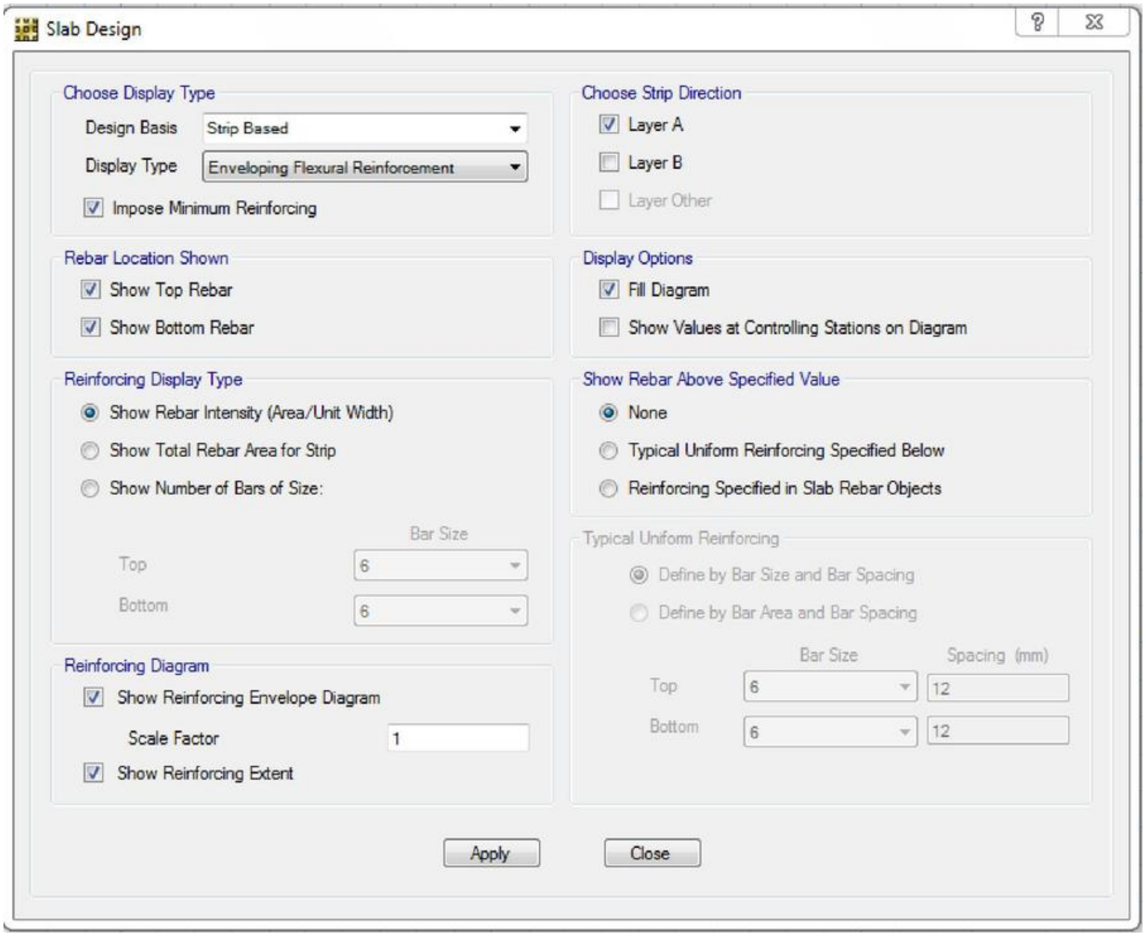


Figure 216

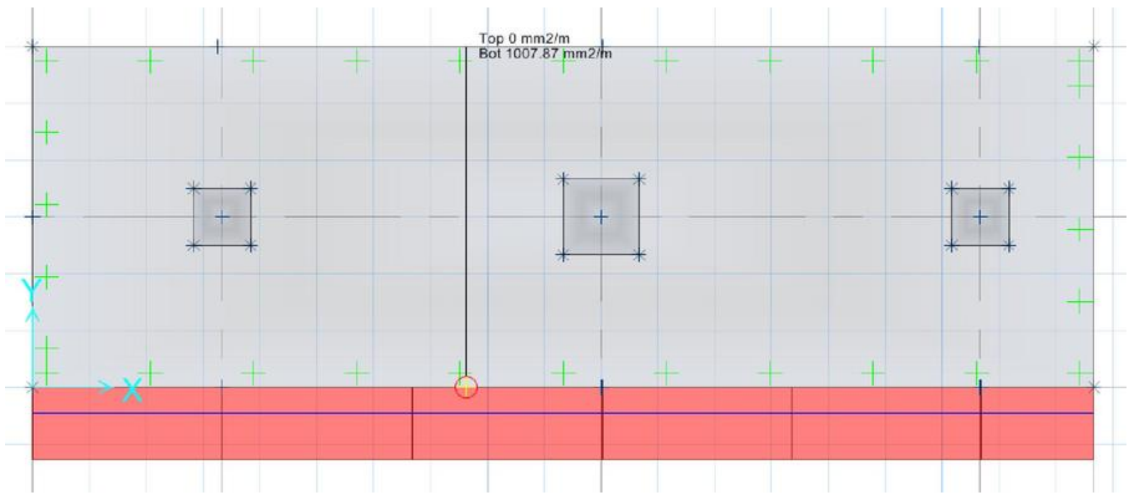


Figure 217

As can be noticed from the footing design diagram in fig. 71, the top reinforcement across strip layer A is around 0mm^2/m for top reinforcement and 1008smm^2/m for bottom reinforcement.

The design outputs can also be displayed in tabular format by clicking on the '**Show Tables…**' menu item from the '**Display**' menu or by just clicking on the equivalent icon  from the tool bar below the menu bar and the following window will pop up.

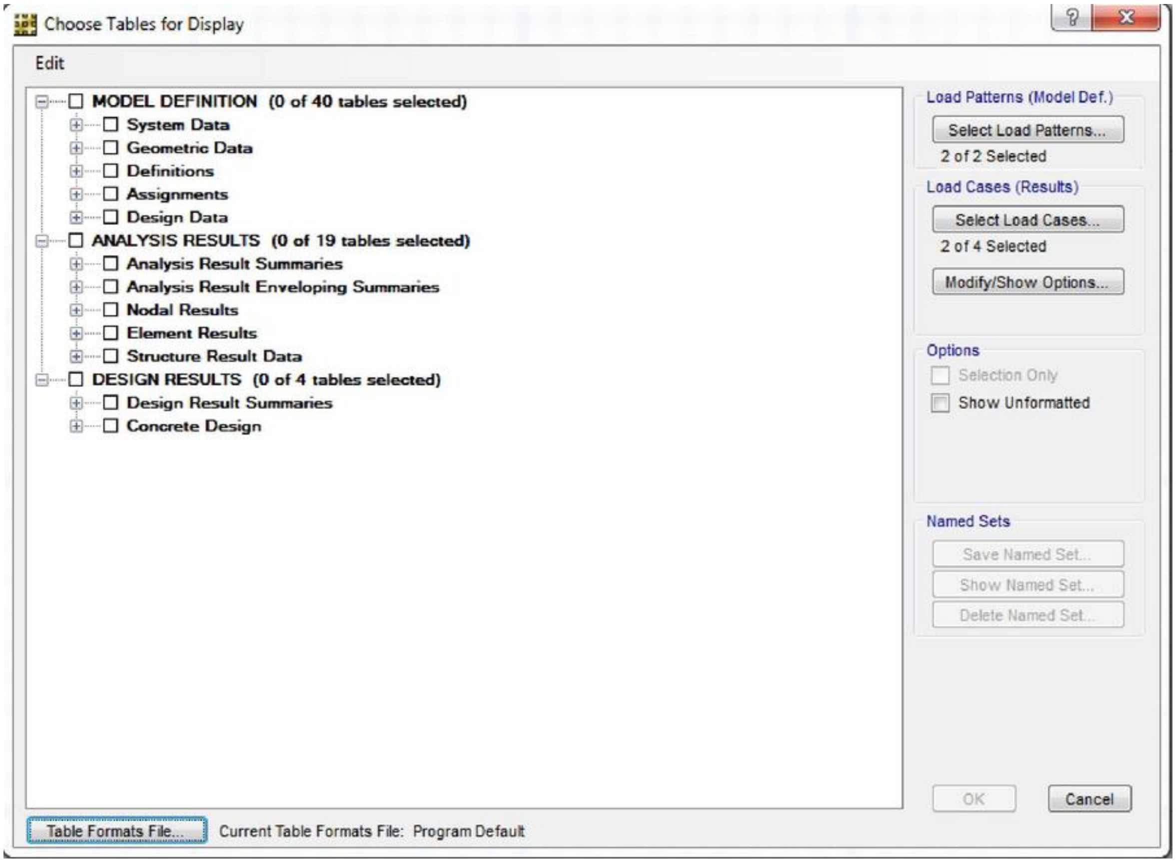


Figure 218

In this window, we can select any of the model definitions or analysis results or design results and press '**OK**' to display the quantity which we want to have a look at. By using the right hand side buttons in the window, the load patterns and the load cases can be selected.

STEP 10: Detailing

After running the analysis and after checking that the results are reasonable, the detailing will be done. However, before running the detailing, the detailing preferences can be set from the '**Detailing**' menu. From the '**Detailing Preferences…**', the likes of dimensional units and material quantity units can be selected. From '**Slab/Mat Detailing Preferences…**', the likes of rebar curtailment options, the rebar detailing options, rebar selection rules and preferred

rebar sizes can be selected. The '**Drawing Sheet Set-up…**' menu allows us to set-up the contents of the drawing sheet. The '**Drawing Format Properties…**' allows us to set some formats in which the output displayed.

To run the detailing, go to '**Run**' menu and click on '**Run Detailing…**' or simultaneously press '**Shift**' and '**F5**' keys or just click on the run detailing icon  from the tool bar just below the menu bar. Then, the '**Run Detailing Options**' window pops up so that we set the detailing options. Set the detailing options which you want and click '**OK**'.

Once the detailing is run, the detailing can be displayed. The detailing display options can be best accessed from the '**Model Explorer**'. When expanded in full, the '**Detailing**' tab of the model explorer looks like:

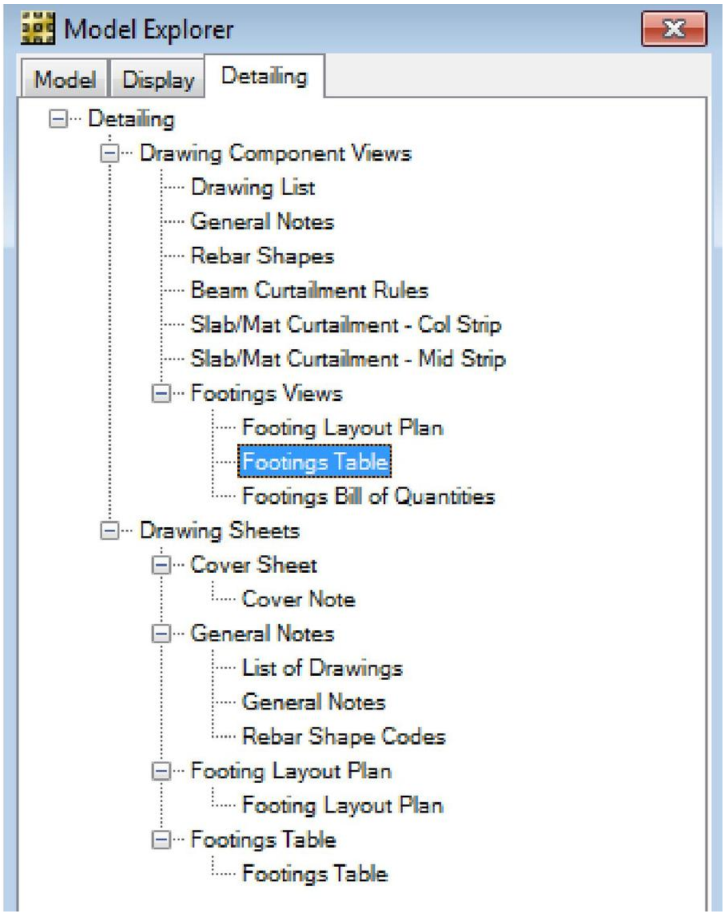


Figure 219

By clicking on any of the options from the detailing tab, a desired detailing can be displayed. For instance, by double clicking on the '**Footing Table**', the following detail of footing can be shown.

FOOTINGS TABLE

SR. NO.	TYPE	NOS	LX	LY	T	REBARS-A	REBARS-B
1	F1	1	5.600 M	1.800 M	0.600 M	8-#3	8-#3

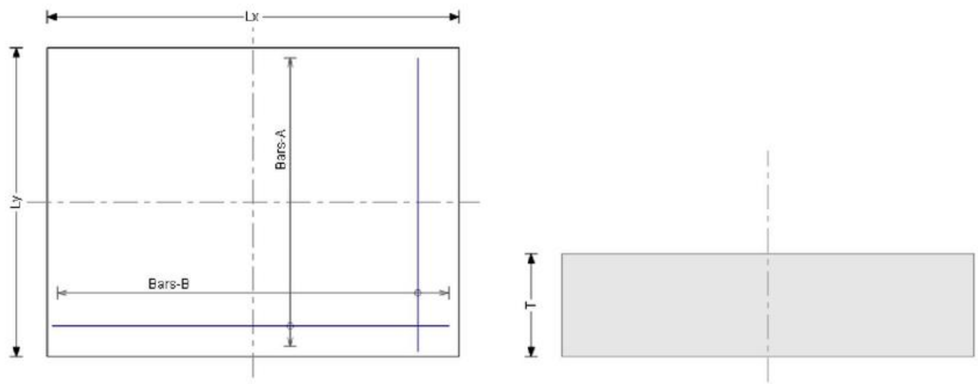


Figure 220

In this detail, the diameter of the reinforcement which is used is 10mm. If you want to change the diameter….

Apart from this detail, other details can also be shown.

STEP 11: Reporting

The last step of foundation design is reporting. Before creating the report, the report preferences should be set up. To do this, go to the 'File' menu and click on 'Report Set-up…' and the following window pops up.

In this '**Report Setup Data**' window, the user preferences regarding the reporting such as the report output type, the report page orientation and the report items can be set along with the load patterns and load combinations. Once the preference is set, the report can be created by clicking on '**Create Report**' command in the '**File**' menu. The '**Advanced Report Writer**' command in the same menu can be used to set some advanced reporting formats.

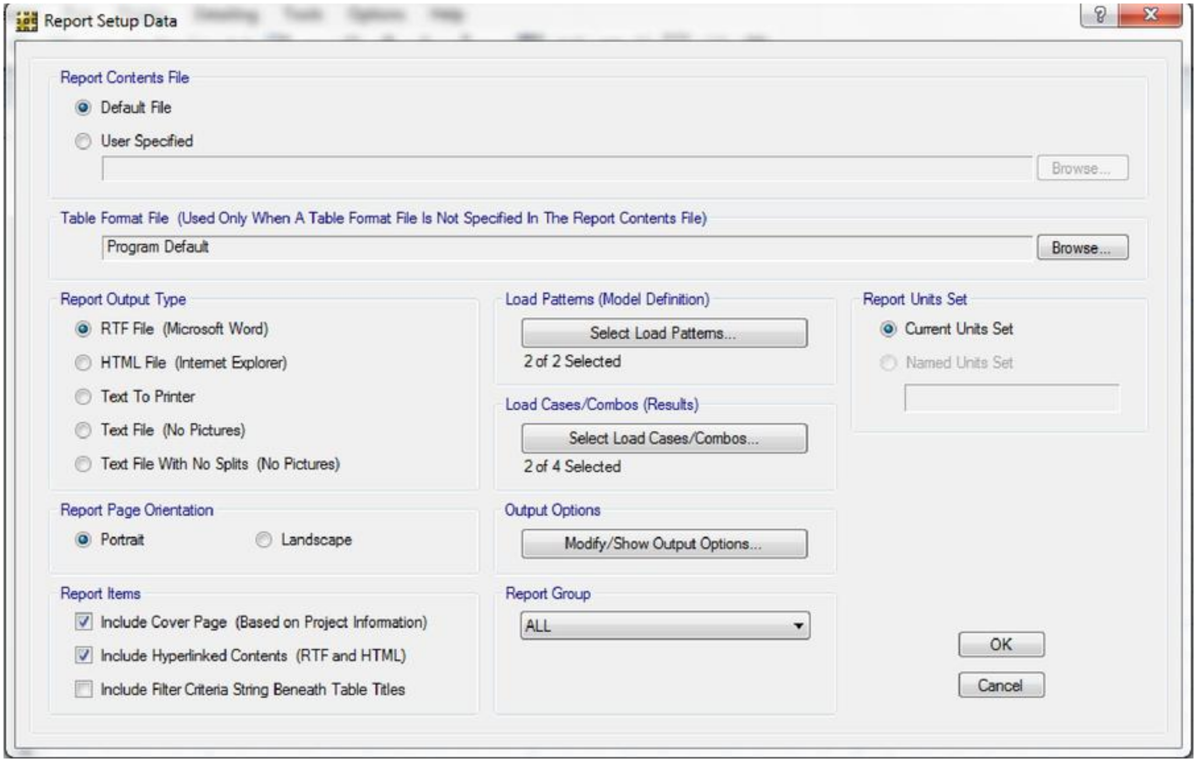


Figure 221

This concludes the tutorial for the design of single footing using a model imported from AutoCAD.

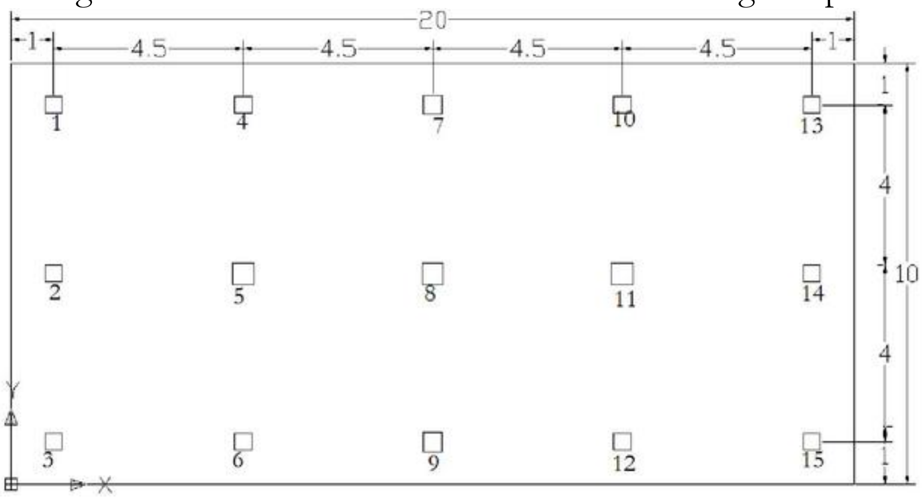
Design of Mat Foundation

Week 7–9

4. Mat Foundation

In this part of the tutorial, a mat foundation will be designed by using an imported model from AutoCAD. This is an advantageous procedure because geometrical asymmetries, different sizes of columns and different loadings can be accommodated by drawing a single AutoCAD drawing. Designing mat foundations from the built-in model is beneficial when the column dimensions and loadings are identical which may not often be the case. Thus, this procedure of designing foundations will not be considered in this tutorial. Of course, there is a possibility for modification of the inbuilt model once it is created. And creating mat foundation models by using grids may involve creating several grid lines especially when the geometry is asymmetrical and column dimensions are not identical. Thus, this way of creation of models will not be discussed. However, if these ways of creating the model are mandatory, the procedure of creating is no different from that of combined foundations which are discussed earlier.

Design the mat foundation shown below with the given parameters:



Columns	Dimension (m)	Axial Load (-z direction) (kN)	Mx (kNm)	My(kNm)
1	0.40x0.40	500	70	100
2	0.40x0.40	700	-10	75
3	0.40x0.40	500	-40	65
4	0.40x0.40	700	50	60
5	0.50x0.50	1000	-5	20
6	0.40x0.40	700	-45	65
7	0.45x0.45	900	55	-5
8	0.50x0.50	1200	15	20
9	0.45x0.45	900	-45	-10
10	0.40x0.40	700	50	-55
11	0.50x0.50	1000	-10	-25
12	0.40x0.40	700	-50	-60
13	0.40x0.40	500	50	-75
14	0.40x0.40	700	-10	-60
15	0.40x0.40	500	-80	-90

- *Ultimate bearing capacity of the soil* : 100kN/m^2
- *Maximum allowable settlement of the foundation*: 10mm
- *Grade of concrete*: C-25 (25MPa 28-day characteristics cube strength)
- *Grade of reinforcement bar (rebar)*: S-400 (400MPa characteristics yield strength)
- *Overall thickness of the foundation*: 1000mm
- *Concrete cover* : 50mm
- *All the given loads are factored (ultimate) loads.*

STEP 1: Creating the Model

To create a footing model from an AutoCAD file, first draw the plan view of the foundation on AutoCAD. In this case, a rectangle with dimension of 10mX20m and a number of squares representing the foundation columns. When you draw the plan view on AutoCAD, make sure that you are drawing it on a new layer. If you want the origin of the SAFE mode to coincide with the center of the circular foundation, make the center of the circle coincide with the global origin of the AutoCAD file when you draw it. Then save the AutoCAD file in DXF format. To save an AutoCAD file in DXF format, go to '**File**' menu in the AutoCAD file and click on '**Save As'**. At the bottom of the upcoming window, just below the text field where you will enter the file name, you will see '**Files of type**' with a drop-down menu list. From the drop down menu list, select the option which has '**(.dxf)**' at the end. Then enter the file name and save it at any location in your computer where you can easily remember close it. This saves the file in DXF format.

Once you saved the AutoCAD file in DXF format, open the SAFE software and when you use the command '**File**'>'**Import**'>'.**DXF/.DWG Architectural Plan**' or when you simultaneously click on '**Ctrl**'+'**Shift**'+'**I**' keys, and you will be prompted to open a file. Open the DXF file from the location where you saved it and the following window will pop up.

In this window, just change the '**CAD Drawing Units**' to '**m**' since in the AutoCAD file is the model is drawn in meter units. Then press '**OK**'.

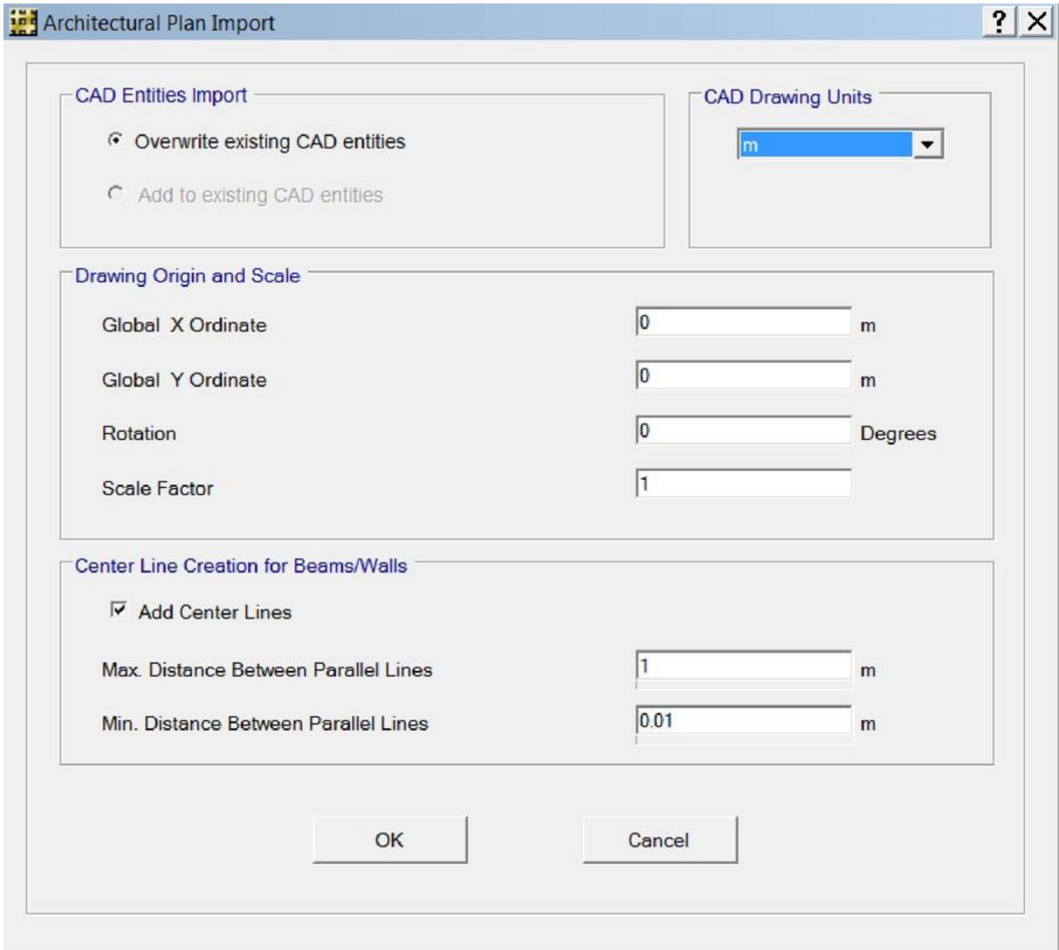


Figure 222

Then, the following model appears in the plan view.

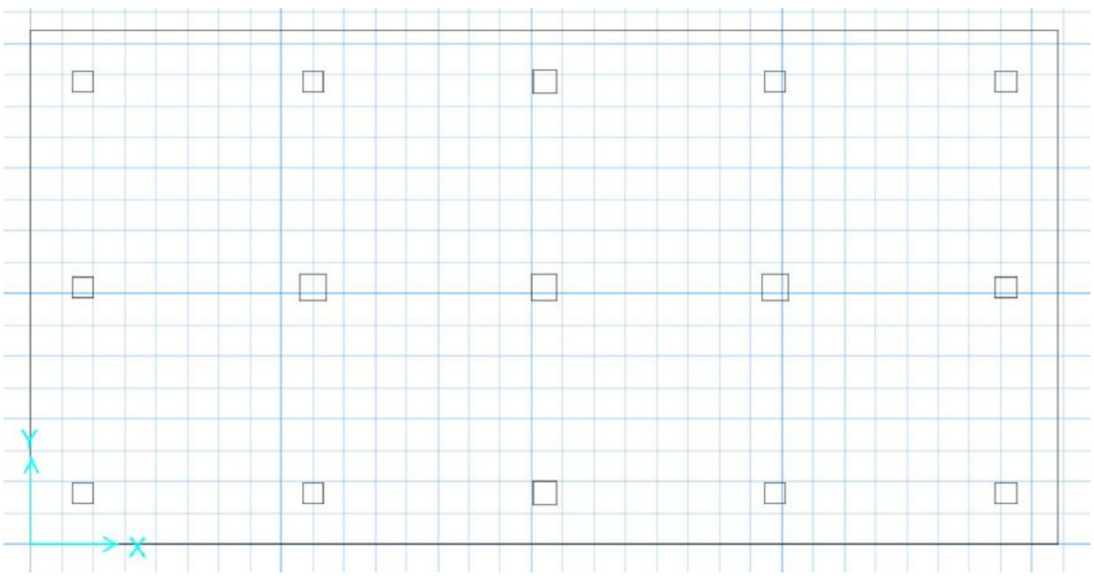
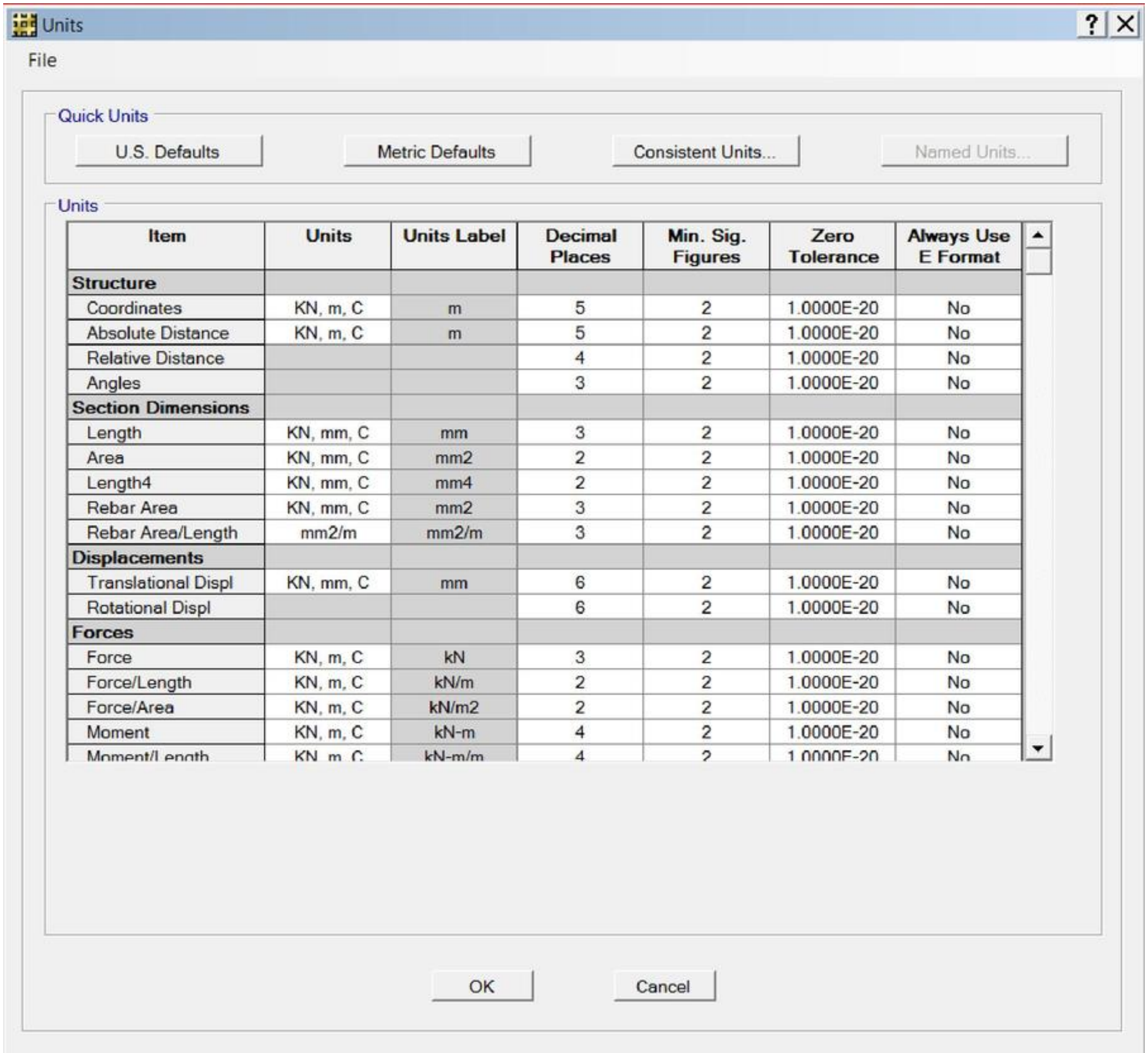


Figure 223

After the model is created, the preferences regarding the units, the design codes and concrete cover should be entered. To change the units, click on '**Units…**' button which is found in the right bottom corner of the main window and the following widow will pop up. In this window, click on '**Metric Defaults**' and press '**OK**' as we will be using metric units. If you want to use U.S. units click on '**U.S. Defaults**' and if you want to use a particular metric or U.S. unit consistently click on '**Consistent Units…**' and select that particular unit which you want to use consistently.



Item	Units	Units Label	Decimal Places	Min. Sig. Figures	Zero Tolerance	Always Use E Format
Structure						
Coordinates	KN, m, C	m	5	2	1.0000E-20	No
Absolute Distance	KN, m, C	m	5	2	1.0000E-20	No
Relative Distance			4	2	1.0000E-20	No
Angles			3	2	1.0000E-20	No
Section Dimensions						
Length	KN, mm, C	mm	3	2	1.0000E-20	No
Area	KN, mm, C	mm2	2	2	1.0000E-20	No
Length4	KN, mm, C	mm4	2	2	1.0000E-20	No
Rebar Area	KN, mm, C	mm2	3	2	1.0000E-20	No
Rebar Area/Length	mm2/m	mm2/m	3	2	1.0000E-20	No
Displacements						
Translational Displ	KN, mm, C	mm	6	2	1.0000E-20	No
Rotational Displ			6	2	1.0000E-20	No
Forces						
Force	KN, m, C	kN	3	2	1.0000E-20	No
Force/Length	KN, m, C	kN/m	2	2	1.0000E-20	No
Force/Area	KN, m, C	kN/m2	2	2	1.0000E-20	No
Moment	KN, m, C	kN-m	4	2	1.0000E-20	No
Moment/Length	KN, m, C	kN-m/m	4	2	1.0000E-20	No

Figure 224

To select the design code and other design preferences, go to '**Design**' menu and click on '**Design Preferences…**' and the following window will pop up. In the

'**Code**' tab of this window, change the '**Design Code**' to '**BS 8110-1997**' or any other code which you want to design the circular foundation with. In the '**Min. Cover Slabs**' tab, for '**Non-Prestressed Reinforcement**' , both the '**Clear Cover Top**' and '**Clear Cover Bottom**' should be set to 50mm as the concrete cover in this design problem stated to be 50mm. The '**Preferred Bar Size**' can be set to any reasonable value. Here, the '**Preferred Bar Size**' is set to #14 which is the bar number which will be used as the main reinforcement in the foundation. Leave the rest as they are and press '**OK**'.

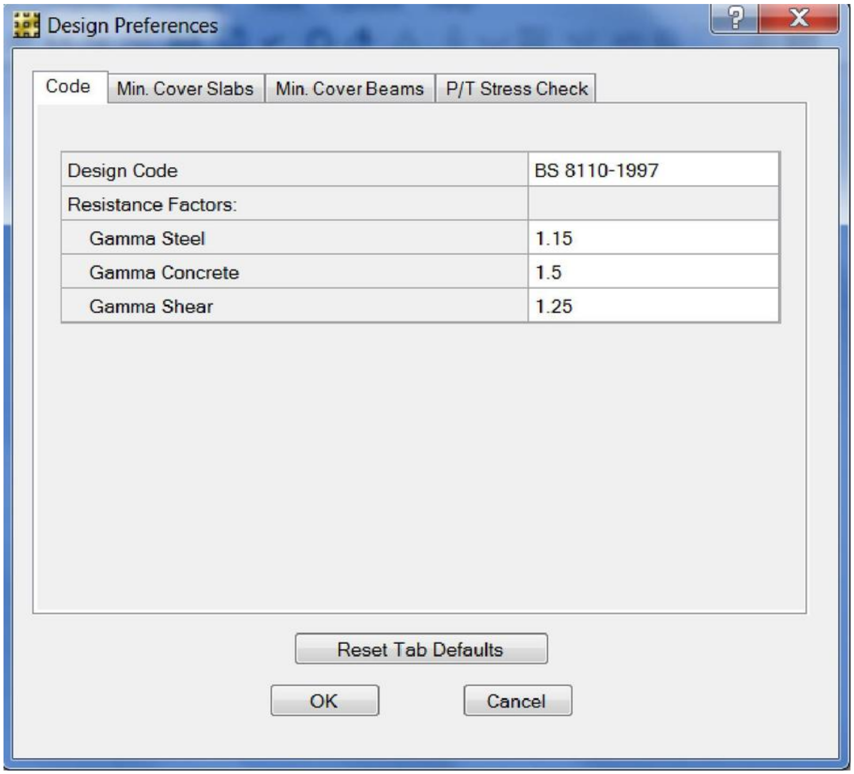


Figure 225

STEP 2: Defining Material Properties

The materials which are involved in the design of the footing should be defined before the analysis. These materials are the concrete, the reinforcement bar (rebar) and the soil support. Thus, these properties will be defined here.

The material definition can be carried out in two ways.

The first one is through the '**Define**' menu in the menu bar. The other one through the model explorer on the left hand side of the home window. To define material properties through the former method, click on '**Define**' menu and again click on

‘**Materials…**’ resulting in the following window depending on prior material definitions.

The list in the ‘**Materials**’ box may not be exactly as it appears in your widow. However, that doesn’t bring any change in the outcome of the design process as you can customize this list any time.

The ‘**Add New Material Quick…**’ button allows you to define materials quickly from a list of pre-defined materials. The ‘**Add New Material**’ button allows you to define materials by changing their properties. The ‘**Add Copy of Material**’ button allows you to define a material with same property as an already defined material. The ‘**Modify/Show Material**’ button displays the property of an already defined material with the possibility of modification. The ‘**Delete Material**’ button, when active, deletes a defined material property.

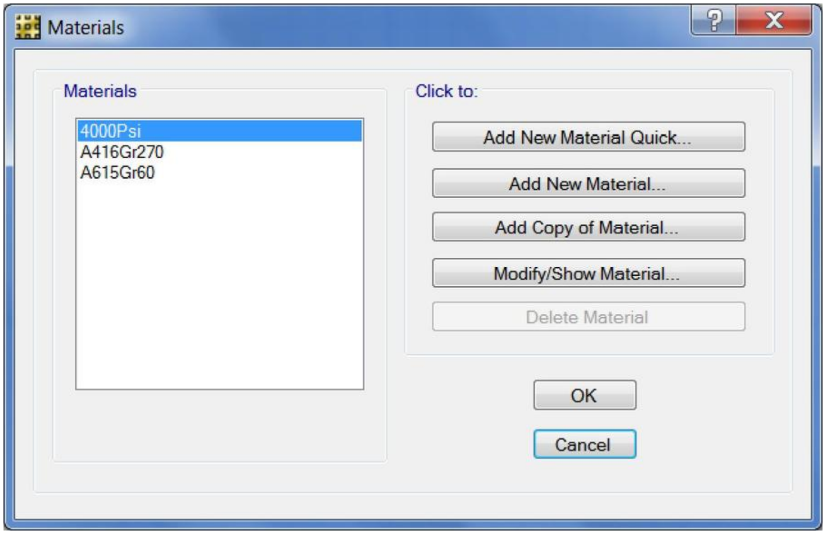


Figure 226

For this particular problem, we will add new concrete and a rebar materials using the ‘**Add New Material…**’ button. Thus click on this button and the following window will pop up.

This is the window where concrete properties will be modified. Put any name for the material in the text field in front of ‘**Material Name**’. But, it is important to make sure that the concrete with the defined material name is assigned for the footing. For this particular problem, let us use the name of the concrete as ‘**C25**’. Since we are defining a concrete, the ‘**Material Type**’ should be set to ‘**Concrete**’.

The unit weight of reinforced concrete may vary depending of the design code of your country. Thus, enter the unit weight of reinforced concrete stipulated in your country code in the text field in front of '**Weight per Unit Volume**'. For this particular example, we will use 25kN/m^3.

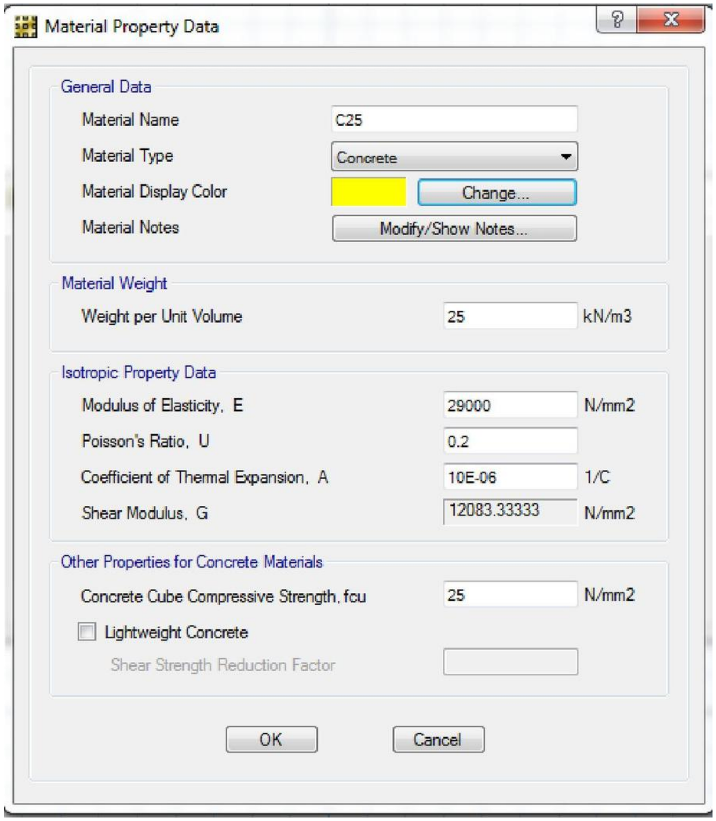


Figure 227

For C-20 concrete, the modulus of elasticity according to BS 8110-1197 is around 29GPa. Thus, enter this value in the text field in front of '**Modulus of Elasticity, E**'. If you selected another design code in step 1 while creating the model, you should refer to actual value of this parameter from the code and enter it accordingly. Be aware of the units though.

The values of Poisson's ratio and coefficient of thermal expansion may also be defined in the design code and should be entered accordingly. For this particular problem, a value of 0.2 for '**Poisson's Ratio, U**' and a value of $10x10^{-6}/^{o}C$ for '**Coefficient of Thermal Expansion, A**' will be entered. The '**Shear Modulus, G**' will be automatically calculated in an un-editable text field.

The grade of concrete for this particular problem is C-25 which is a concrete with 28 day characteristics cube compressive strength of 20MPa. The concrete designation may be different for different country codes but the concept is the

same. Therefore, enter 25 in the text field in front of '**Concrete Cube Compressive Strength, fcu**'.

If a lightweight concrete is used, check on '**Lightweight Concrete**' and enter the corresponding '**Shear Strength Reduction Factor**' in the space provided.

When you press on '**OK**', a concrete material with the above properties will be added to the list of materials. This material will be assigned for the footing before the analysis.

After adding a new concrete property, the program returns to the window shown in Fig. 226 To define a rebar property, we will follow the same procedure as we followed while defining the concrete property. Since a new rebar property will be defined, click on the '**Add New Material…**' button. A '**Material Property Data**' window pops up and when you change the '**Material Type**' to '**Rebar**', the window appears to look like the following.

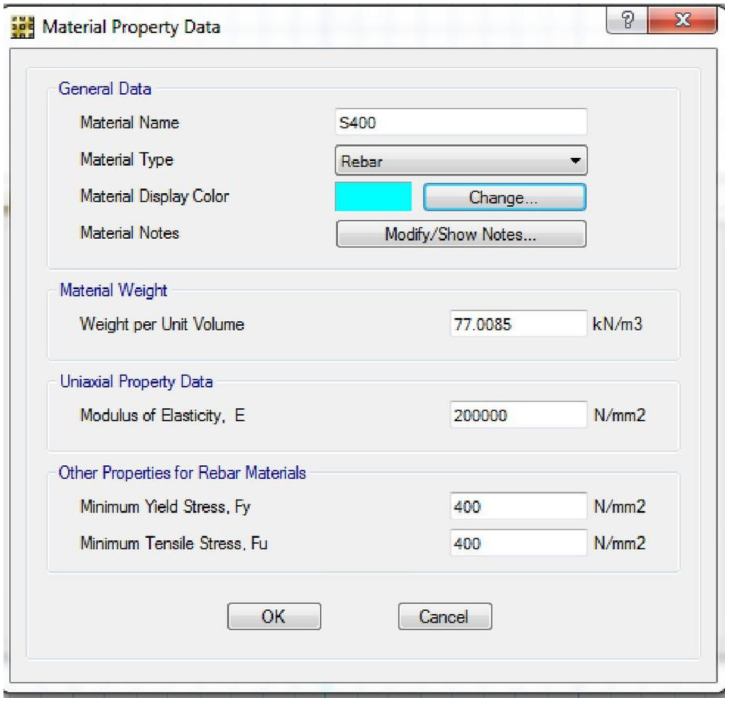


Figure 228

Change the '**Material Name**' to any name you want. Here, we name it '**S400**'. The material type should be '**Rebar**'. The weight per unit volume of steel is stipulated in the design code. For BS 8110-1197, the weight per unit volume is 77.0085kN/m^3. Thus, enter this value in the text field in front of '**Weight per Unit Volume**'. The modulus of elasticity for reinforcement bars according to the same design code is 200GPa. Thus enter this value in the text field in front of '**Modulus of Elasticity, E**' considering the unit.

In the '**Other Properties for Rebar Materials**' box, two quantities are mentioned: minimum yield stress and minimum tensile stress for the reinforcing material. The values of these parameters will be specified in the design code which you defined earlier. If the code assumes that the rebar material exhibits elastic perfectly plastic behavior, the values of these two quantities will be the same. The grade of steel to be used for this particular example is S-400. The yield stress for this type of reinforcement bar is 400MPa. Since the design code of my country assumes that rebars exhibit elastic perfectly plastic behavior, the minimum tensile stress will also be 400MPa. Thus enter 400 in both text fields in front of the '**Minimum yield stress, Fy**' and '**Minimum Tensile Stress, Fu**'. Then press '**OK**' in both '**Material Property Data**' and '**Materials**' windows concluding the material definition step.

The other property which should be defined is the soil support. To define the soil properties, go to the '**Define**' menu and click on '**Soil Subgrade Properties**' menu item and the following window appears.

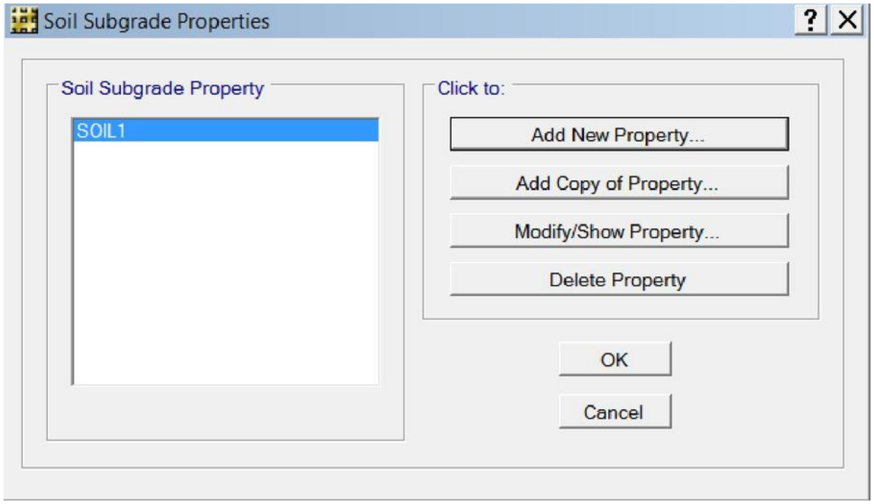


Figure 229

By using the buttons in this window, a new soil property or copy of soil property can be added. An existing soil property can also be modified or deleted. For this problem, let us add a new soil property by using the '**Add New Property…**' button. Thus, click on this button and the following window pops up.

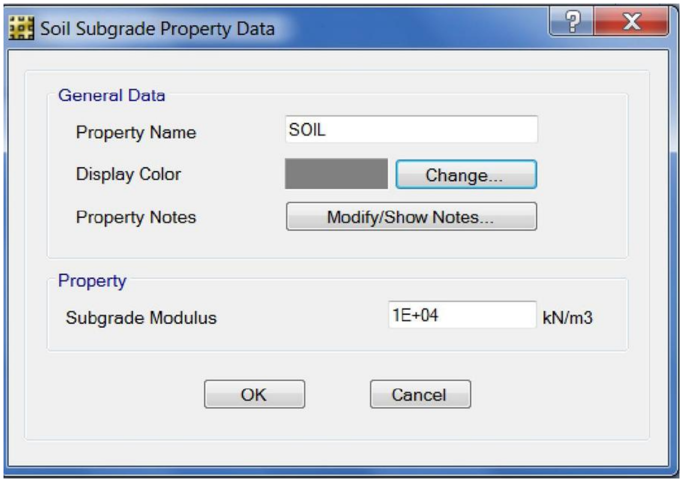


Figure 230

In this window, set the property name to '**SOIL**' and change the subgrade modulus to 10,000kN/m^3. Because, the subgrade modulus which can be assume by the ratio of the bearing capacity by the allowable settlement for this particular example is so. Then press '**OK**' in both the '**Soil Subgrade Property Data**' and '**Soil Subgrade Properties**' windows.

STEP 3: Defining Footing and Column Properties

After defining the material properties, the footing and column properties can be defined. This definition can take place in two ways: from the menu bar and from the model explorer. In SAFE software, footings are modelled as 'footings' and foundation columns are modelled as 'stiff'.

To define footing and column properties from the menu bar, go to '**Define**' menu and click on '**Slab Properties…**'. The following window will pop up.

The '**Add New Property…**' button prompts the user to enter new properties for the footing and foundation column while the '**Add Copy of Property…**' copies the property of an existing slap. The '**Modify/Show Property…**' allows the user to show the property of an existing component with the possibility of modification. When the '**Delete Property**' button is active, it allows the user to delete an existing slab property.

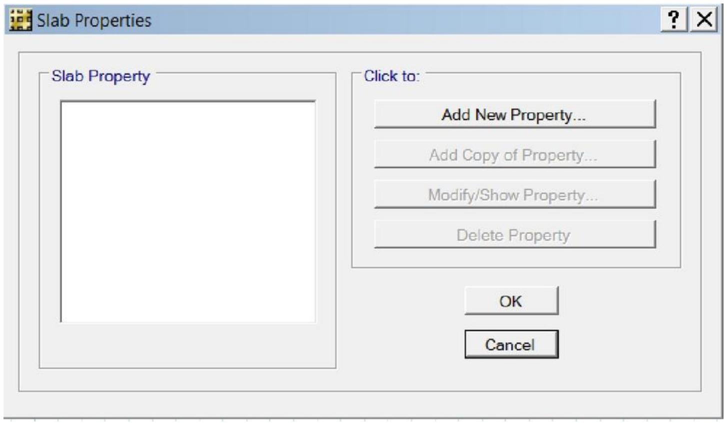


Figure 231

In this case, we use the '**Add New Property…**' button to add new slab properties for the footing and foundation column. Click the button and the following window will pop up.

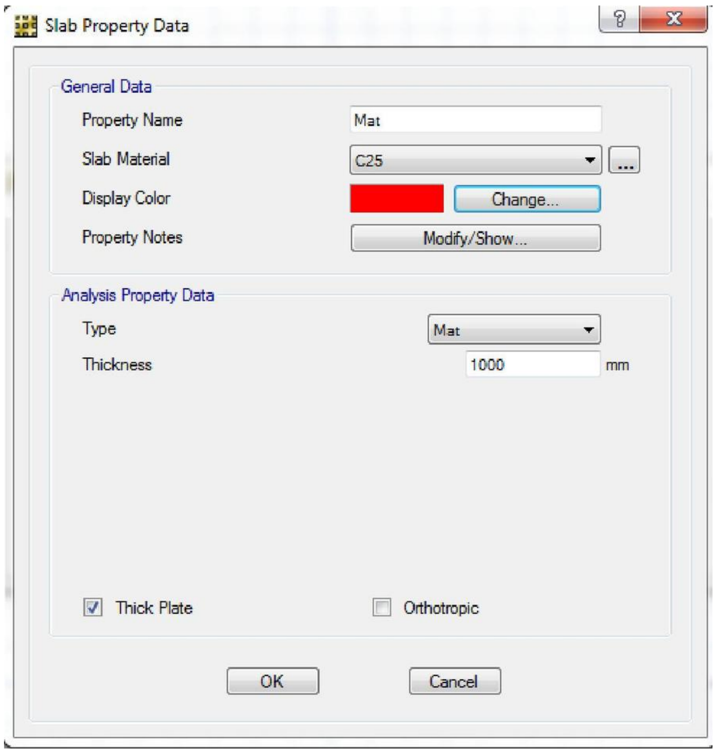


Figure 232

The '**Property Name**' can be assigned with any name; but, in this case, the property name which will be used is '**Mat**'. The '**Slab Material**' should be set to the concrete grade which is defined in step 2. Since the name of the concrete material defined in step 2 for this particular problem is '**C25**', a material with this name should be selected from the list.

In the ‘**Analysis Property Data**’ box, ‘**Type**’ should be set to ‘**Mat**’. The ‘**Thickness**’ value should be set to the thickness of the footing defined in the example. Since the thickness of the footing is 1000mm, this value is entered in the text field corresponding to ‘**Thickness**’. As footings are modelled as thick plates, check the ‘**Thick Plate**’ option. The ‘**Orthotropic**’ check box is selected when a footing with irregular dimension is to be used.

When you press ‘**OK**’, the ‘**Slab Property Data**’ window will be exited and the ‘**Slab Properties**’ window gets activated. Now, the property of the foundation column will be added. To do this, again click on the ‘**Add New Property…**’ button. The following window appears after the click.

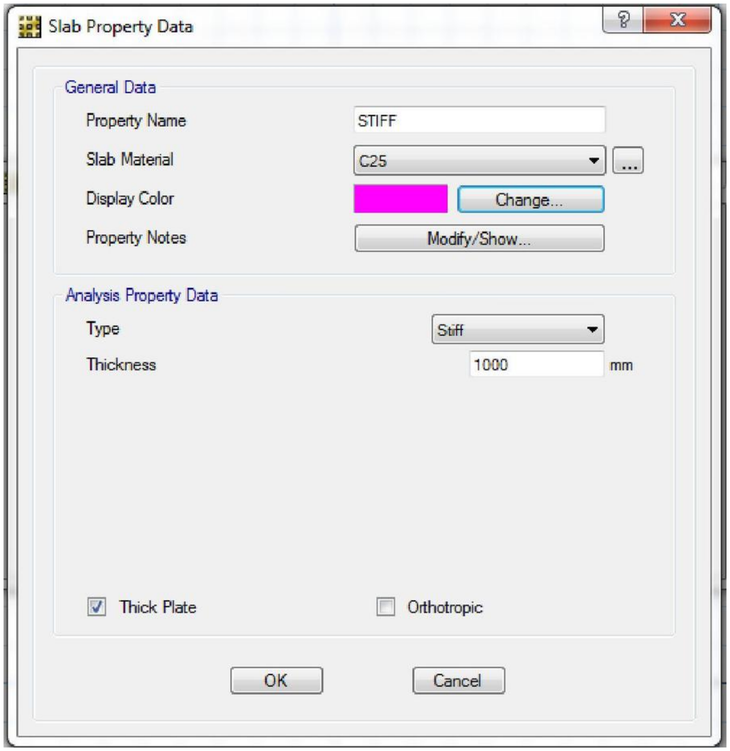


Figure 233

The ‘**Property Name**’ can be any name but we use ‘**STIFF**’. The ‘**Slab Material**’ should be set to the concrete grade which is defined in step 2. Since the name of the concrete material defined in step 2 for this particular problem is ‘**C25**’, a material with this name should be selected from the list.

In the ‘**Analysis Property Data**’ box, ‘**Type**’ should be set to ‘**Stiff**’ as we are defining the property of column. The ‘**Thickness**’ value should equal to the thickness of the mat. As foundation columns are modelled as thick plates, check the ‘**Thick Plate**’ option. The ‘**Orthotropic**’ check box is selected when a column with irregular dimension is to be used.

When you press ‘**OK**’, the ‘**Slab Property Data**’ window will be exited and the ‘**Slab Properties**’ window gets activated. Again press ‘**OK**’ and exit the window for defining the footing and foundation column.

STEP 4: Defining Load Patterns, Load Cases and Load Combinations

The loads on the foundation should be defined accordingly before the analysis. First, the load pattern should be defined. This can be done from the ‘**Define**’ menu or from the ‘**Model Explorer**’. This time, we will do it from the model explorer. In the model explorer, expand ‘**Load Definitions**’ and you will see ‘**Load Patterns**’. When you expand ‘**Load Patterns**’, you will see ‘**DEAD**’ and ‘**LIVE**’. At the end, the model explorer appears to look like:

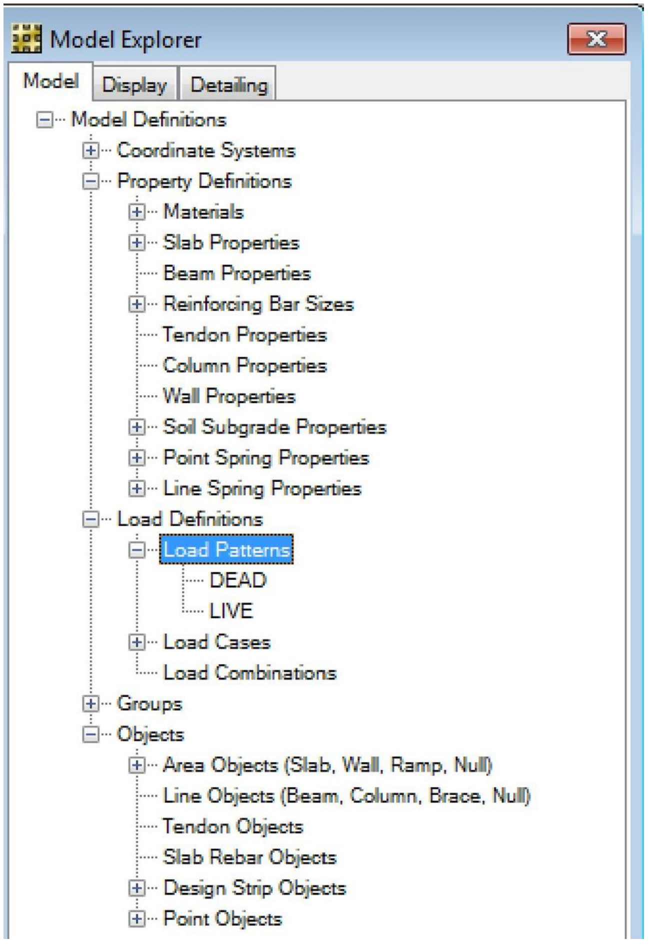


Figure 234

Now, right click on ‘**Load Patterns**’ and click on ‘**New Load Pattern**’ and the following window pops up.

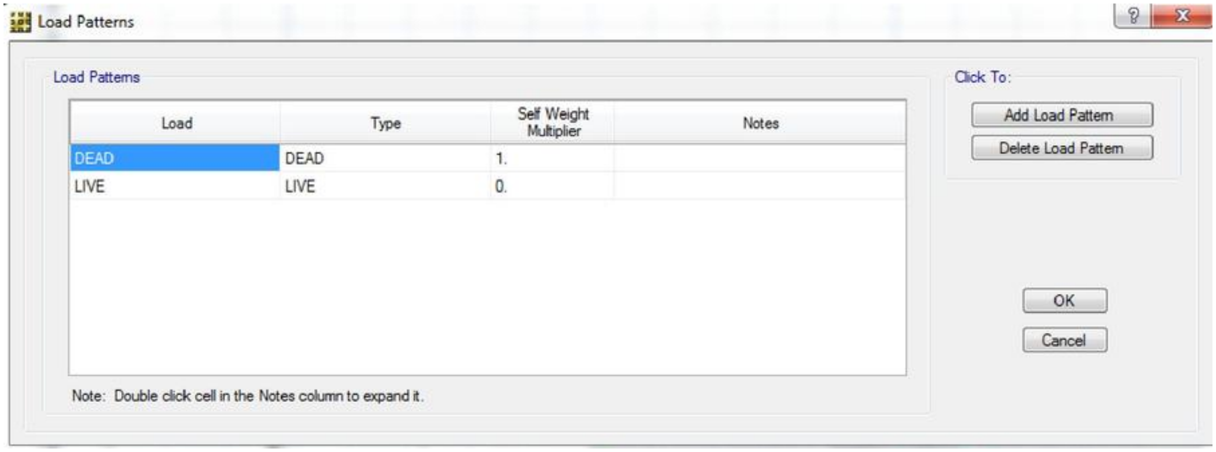


Figure 235

In the '**Load Patterns**' window, two load patterns are already defined: '**DEAD**' and '**LIVE**'. The '**Type**' of the '**Load**' should also be changed accordingly. There are many options for type of loading. The '**Type**' for dead loads should be set to '**DEAD**' and for live loads '**LIVE**'. If there are other types of load patterns on the foundation such as earth quake load, you can add the load pattern with the '**Add Load Pattern**' button. You can also delete any undesirable load pattern using the '**Delete Load Pattern**' button. Since in this example, we have only dead and live loads, we will leave the existing load patterns as they are. The '**Self Weight Multiplier**' value should also be changed accordingly. This value imparts the option whether to consider or ignore the self-weight of the foundation in addition to external loads. If the self-weight of the foundation is already included as an external dead load or if you want to exclude the effect of self-weight from the analysis, the value under '**Self Weight Multiplier**' should be set to zero. In this example, we will consider the self-weight as an additional load to the external dead load. Thus, the value under '**Self Weight Multiplier**' for the '**DEAD**' load is one. For the '**LIVE**' load, it will be zero. Press '**OK**' and the window will be exited.

After this, the load cases will be defined. Load cases are used to dictate the way the loads are applied (statically or dynamically) or the way the structure responds (linearly or non-linearly) for the defined load patterns. To define a load case, go to '**Define**' menu and click on '**Load Cases…**'. The following window will pop up after the click.

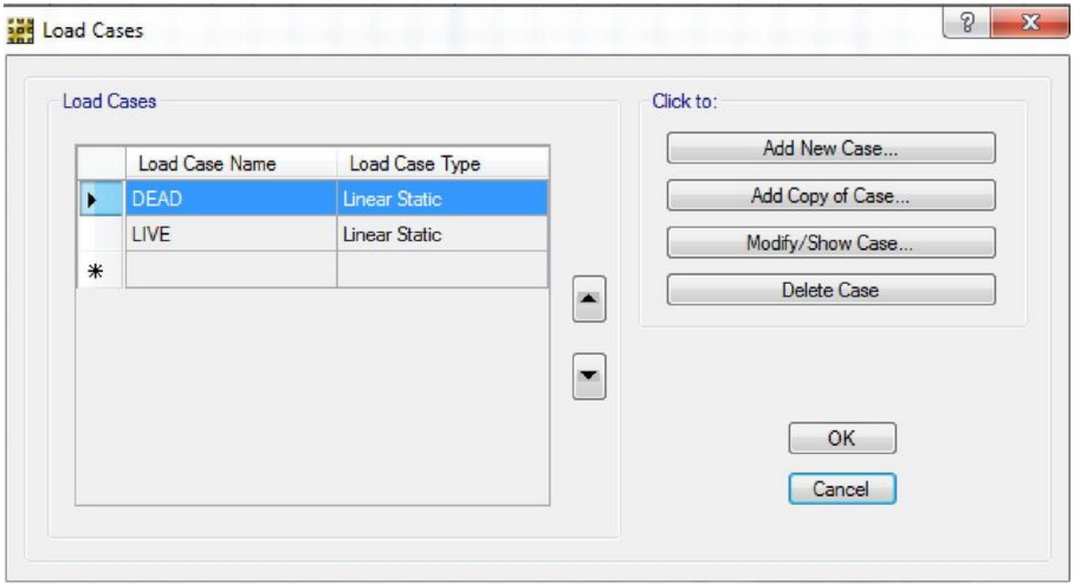


Figure 236

The load patterns which were defined earlier will automatically appear in the list of '**Load Case Name**' of the '**Load Cases**' window. The '**Load Case Type**' column shows the way in which each load pattern will be applied during analysis. If you want to modify this, highlight the load pattern for which you are going to change the load case type and click on the '**Modify/Show Case…**' button. If you click this button, the following window will appear. In this window, you can change the way the load is applied from the '**Load Case Type**' box. The way the structure responds can also be selected from the '**Analysis Type**' box. This problem '**Static**' is for the '**Load Case Type**' and '**Linear**' is selected for the '**Analysis Type**' since the load is static and the foundation responds linearly. The scale factor for the dead load in the '**Loads Applied**' box will be left as one. Press '**OK**' and exit the window.

The load case type for the live load should also be 'Linear Static'. Otherwise, it should be changed by clicking the '**Modify/Show Case…**' button to linear static case. If both the load case types are as desired click '**OK**' and exit the '**Load Cases**' window.

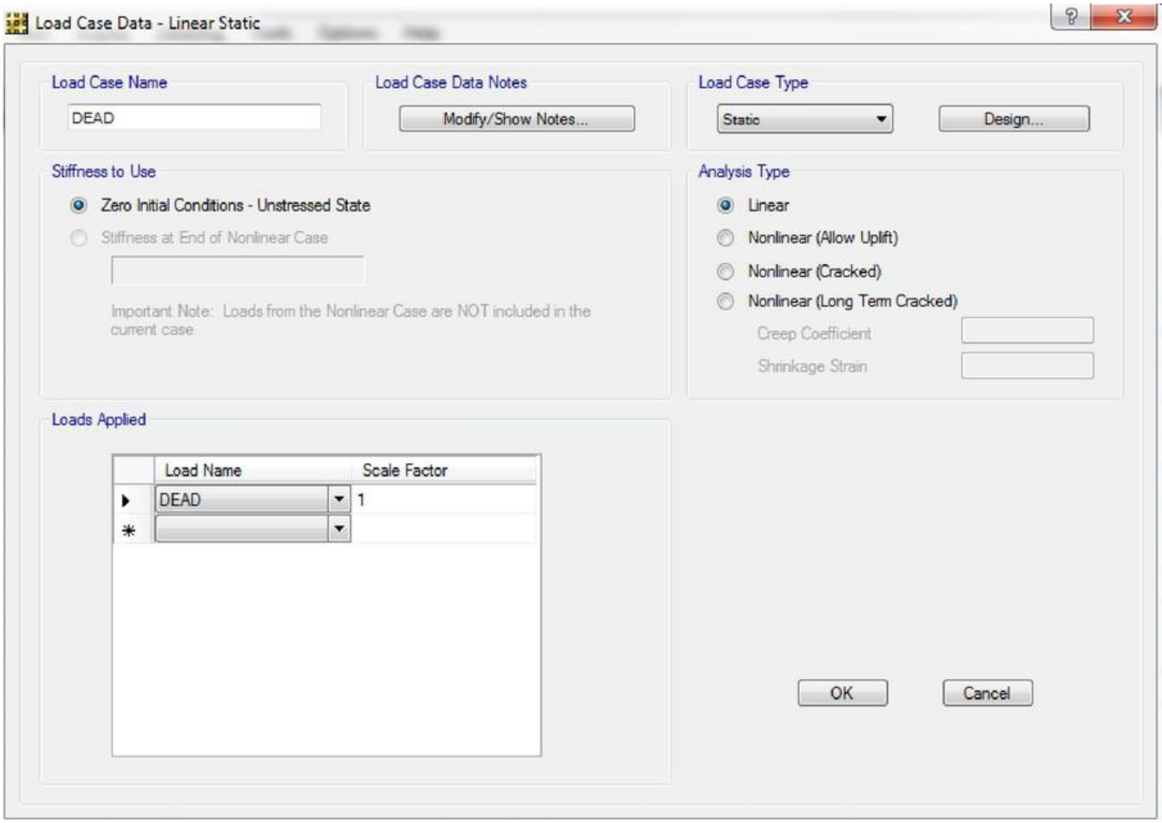


Figure 237

Definition of the load combinations will be the next step. Since the loads are factored, only one load combination will be considered. To define load combination, go to '**Define**' menu and click on '**Load Combinations…**' and the following window pops up.

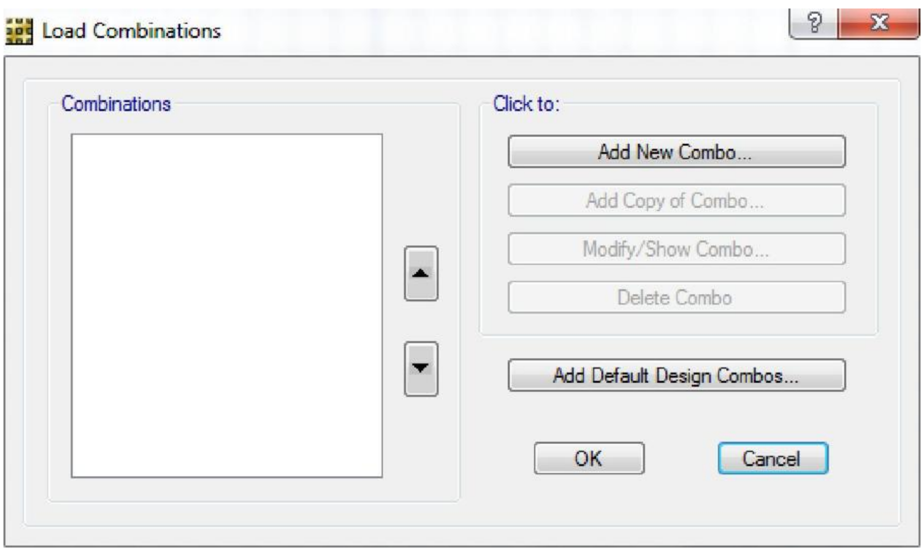


Figure 238

New load combinations will be added through the '**Add New Combo…**' button. The load combination which will be considered is factoring only the self-weight which is considered as a dead load and the other factored loads will be considered as live loads for convenience and the scaling factor for the live loads will be one. When the '**Add New Combo…**' button is clicked, the following window appears.

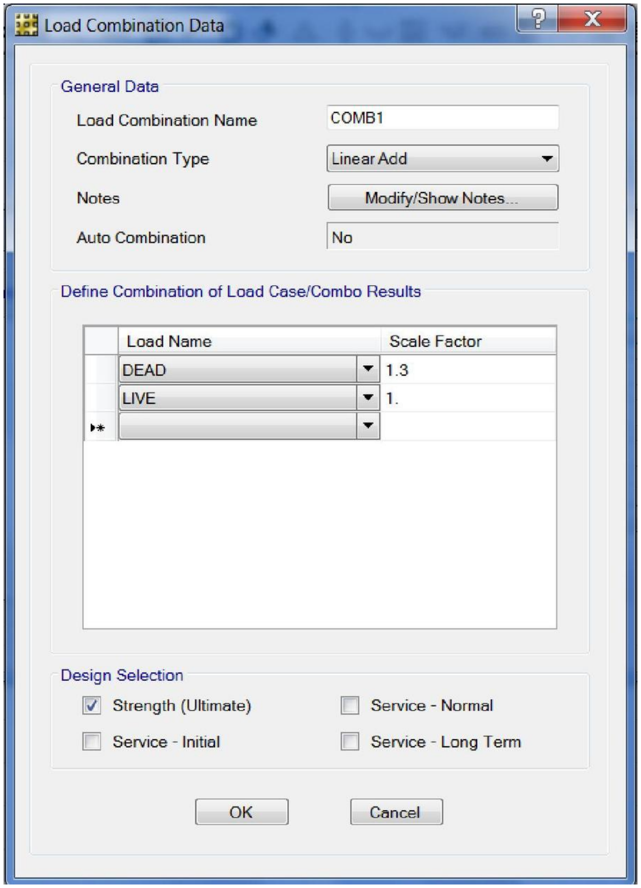


Figure 239

In the '**Load Combination Data**' window, any name can be given for the load combination. The '**Combination Type**' should be set to '**Linear Add**' as the component loads (dead and live) will be added linearly. However, there are also other options from the drop down menu in front of '**Combination Type**'. In the window shown in Fig. 239, the two loads (dead and live) should be activated below the '**Load Name**' column of the '**Define Combination of Load Case/Combo Results**' box. The values in the '**Scale Factor**' column correspond to the partial safety factors for load for the failure mode under consideration. These partial safety factors are specified in the design code of your country. In the design code of my country, the partial safety factor for dead loads for ultimate limit sate case is 1.3 for the case where there are only dead and live loads. For such load patterns, the partial safety factor for live loads for ultimate limit sate case would have been 1.6. But, in this case, since the loads are already factored and for convenience reason since the factored loads are considered as live loads, a scaling factor of unity will be used. Thus, enter a value of 1.3 in front of '**DEAD**' and 1 in front of '**LIVE**' in '**Scale Factor**' column. The failure condition which is being under consideration can be defined by selecting and deselecting the check boxes in the '**Design Selection**' box. The serviceability limit state will not be considered for this case.

STEP 6: Drawing the Footing Components and Design Strips

The footing, the foundation column and a point where the loads will be applied on the foundation column should now be drawn on the grid.

k. *Drawing the mat*

The footing will be drawn as a rectangular slab. Thus, go to the '**Draw**' menu and click on '**Draw Rectangular Slabs/Areas…**' or click on the equivalent icon  from the left hand side tool bar and the following window will pop up.

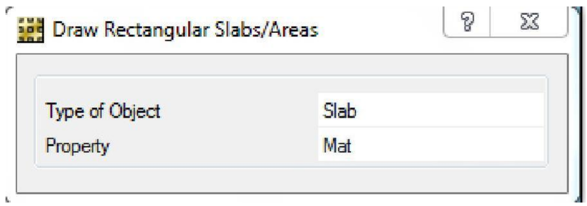


Figure 240

Make sure that the '**Property**' is set to '**Mat**' and click at one corner of the footing. Then, without releasing the click, drag into the opposite corner and release the click. This draws a rectangular footing.

vi. *Drawing the foundation column*

While the window in fig 240 is active, change the '**Property**' to '**STIFF**' and draw the three columns by following a similar procedure which you used while drawing the footing.

vii. *Drawing the points on the foundation columns where the load will be applied*

Go to '**Draw**' menu and click on '**Draw Points**', then click on the mid-point of each of the footing and the pointss will be created. If the cursor could not snap to the midpoint, you can adjust the '**Snap Options**' from the '**Draw**' menu.

After this, the design strips will be drawn. Design strips determine the way in which different quantities related to the reinforcement calculation are calculated. Forces are integrated across the design strips. Thus, the larger the width of coverage of the design strips within the given structure, the higher will be the calculated values of the bending moments and shear forces. Thus, an optimum width of strip is required compromising the safety and economical requirements. The width of the design strip will be specified in the design code. According to the code of my country, the width of design strips for combined foundations in the longitudinal direction should cover the whole area of the footing. In this particular problem, the width of the strips in x-direction is 1.8m. The design strips in the transverse direction will be defined for each column and should extend to a distance of half the depth of the footing from the face of the column on each side. These design strips in X and Y direction are usually defined in SAFE software as layer A and layer B.

To draw the design strip, go to the '**Draw**' menu and click on '**Design Strips**' or simply click on the equivalent icon  from the left hand sided tool bar and the following window pops up. Column strips of width 2m around each column will be drawn in both directions. The remaining spaces between the columns strips will be filled with middle strips.

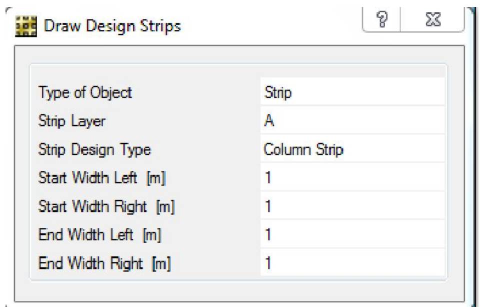
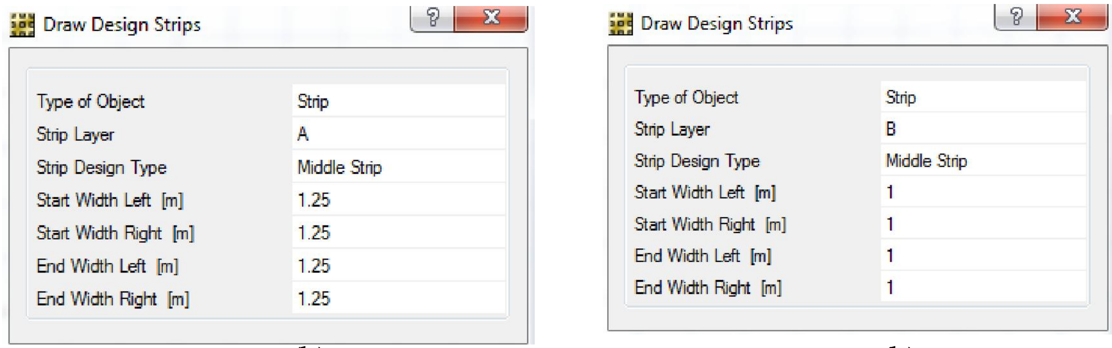


Figure 241

In this window, the '**Strip Layer**' should be selected to be either '**A**' or '**B**'. But if '**A**' is for design strip in X direction '**B**' should be for Y direction and vice versa. The window in fug 241 is for the column strip in Y direction. The column strip in X direction can be drawn by just changing the '**Strip Layer**' to '**B**'. The '**Draw Design Strips**' windows for middle strips in X and Y directions are shown in fig. 242(a) and fig. 2242(b) respectively.



(b) (b)

Figure 242

You can display the design strips by setting the display options by clicking on ‘**Set Display Options…**’ from the ‘**View**’ menu or by simultaneously clicking on ‘**Ctrl**’ and ‘**W**’ keys or by just clicking on the set display options icon from the tool bar below the menu bar. This results in the following window:

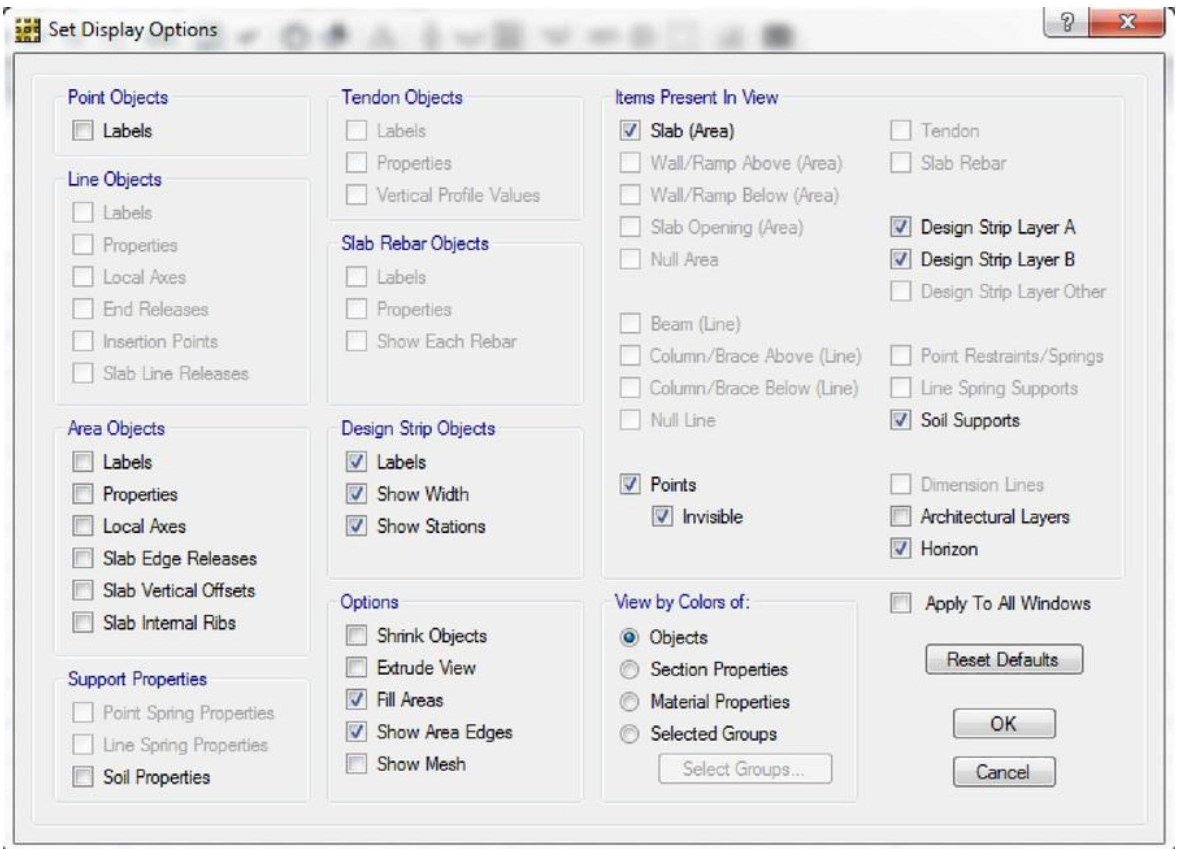


Figure 243

Then, check on ‘**Labels**’, ‘**Show Width**’ and ‘**Show Stations**’ in the ‘**Design Strip Objects**’ box and press ‘**OK**’ and after drawing dimension lines by using the command ‘**Draw**’>’**Draw Dimension Lines**’, the following window appears displaying the design strips in the two directions.

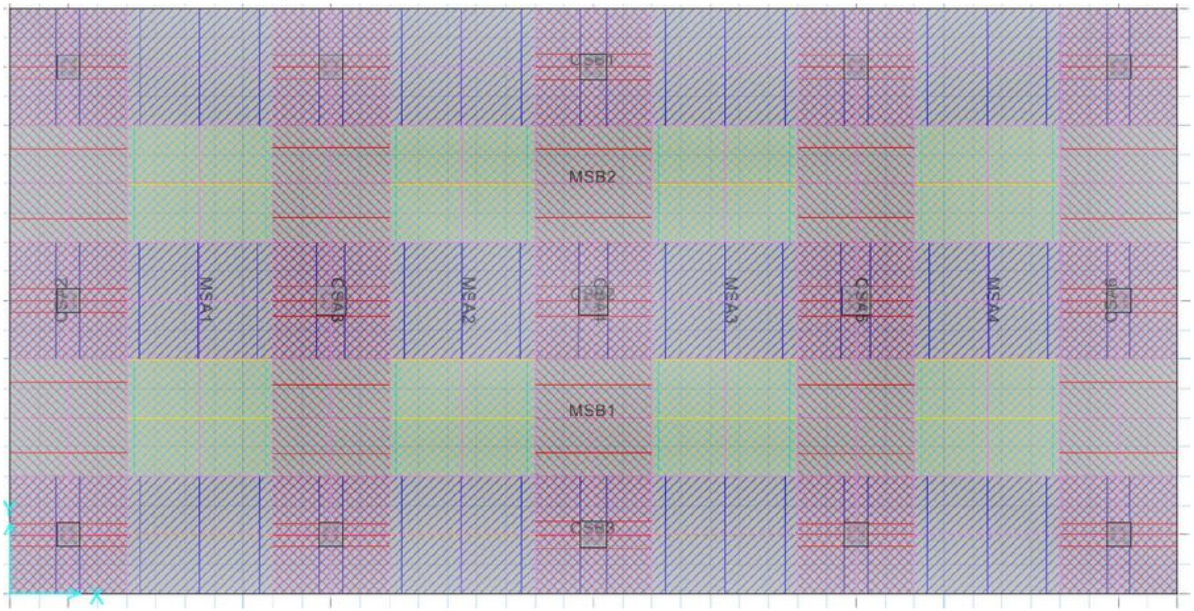


Figure 244

STEP 7: Assigning Slab Data, Support Data and Load Data

The slab data, support data, and load data which are defined in the previous steps should be assigned to the corresponding structural component. The way to do this is first to select the component and next to assign the slab property, support property or loads accordingly. The selection can be done by clicking on the component from the plan view or through the ‘**Select**’ menu. The latter option assures that the component is selected exactly as selection by clicking may result in incorrect selection. Thus, all the selections here will be done from the ‘**Select**’ menu. The assignments will be discussed as follows:

g. *Assigning support data to the footing*
Select the footing through the following strings of commands ‘**Select**’>’**Select**’>’**Properties**’>**’Slab Properties…**’. Then select ‘**FOOTING**’ and press ‘**OK**’. Then assign the footing property through the following strings of commands ‘**Assign**’>’**Support Data**’>’**Soil Properties…**’. Then select ‘**SOIL**’ and press ‘**OK**’.

h. *Assigning reinforcement data to the design strips*
Select each design strip through the following strings of commands ‘**Select**’>’**Select**’>’**Properties**’>**Design Strip Layers…**’. Then select ‘**A**’ or ‘**B**’

(one at a time) and press '**OK**'. When you right click on the selected strip layer, the '**Slab-Type Area Object Information**' window pops up. In the '**Design**' tab of this window, set the '**Rebar Material**' to '**S400**' and press '**OK**'. Do this for all strips.

i. *Assigning load on the foundation column*

To assign load on the foundation column, right click on the point at the center of the foundation column and a '**Point Object Information**' window pops up. In this window, click on the '**Loads**' tab.

The dead load and the live load can be assigned through the '**Assign Load…**' button. The procedure is: click on '**Assign Load…**' button, then select '**Force Loads**' then press '**OK**' then select either '**DEAD**' or '**LIVE**' depending on which loads you want to enter their values then enter their values accordingly (both the concentrated load and the bending moment) and in the right direction (axis), then select '**Add to Existing Loads**' and press '**OK**'. While doing this, the foundation column dimensions should be entered in the '**Size of Load for Punching Shear**' box of '**Point Loads**' window.

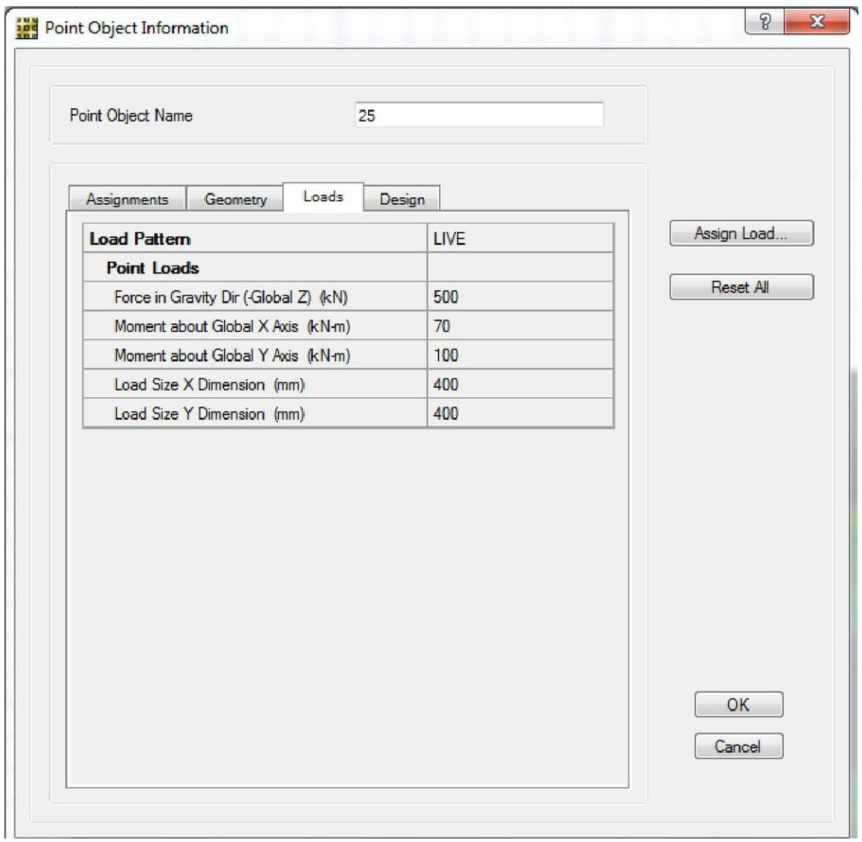


Figure 245

You can also delete any forces which are wrongly entered through '**Delete Existing Loads**' radio button option. In this example, for column 1, since the factored concentrated load is 500kN and since the bending moments about x-axis and y-axis respectively are 70kNm and 100kNm, these values are entered in the Gravity Direction for the concentrated load and along the respective axes for the bending moment and the final values are shown in the above figure. The loading for the rest columns will be entered in a similar way.

STEP 8: Running the Analysis

After this, the analysis can be run. But, make sure that the footing and the foundation column are assigned with the correct rebar material. To do this, right click anywhere in the plan view of the footing and the '**Slab-Type Area Object Information**' window will pop-up.

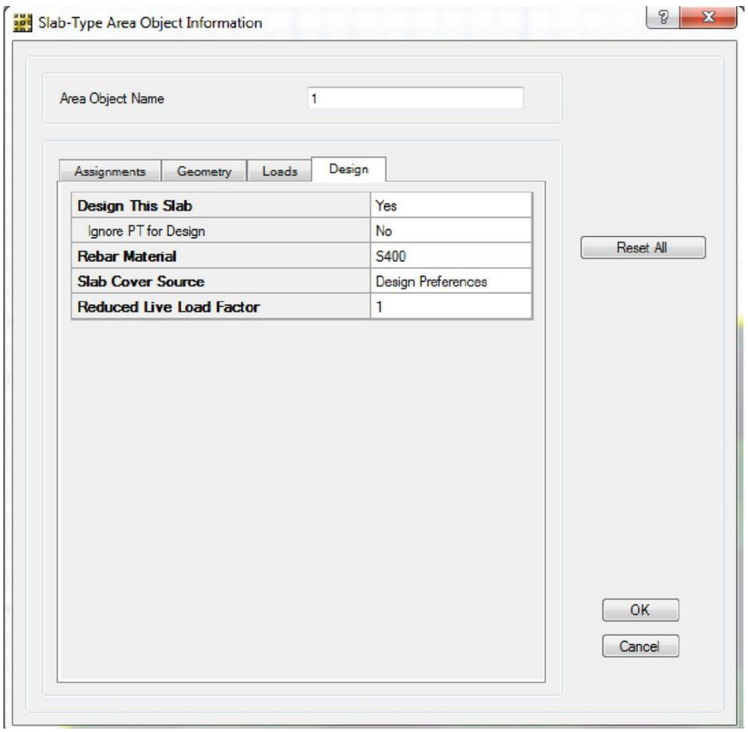


Figure 246

In the '**Design**' tab of this window, set the value of '**Rebar Material**' to '**S400**'. Do this for all column foundations, and all design strips as well. After this, go to the '**Run**' menu and click on the '**Run Analysis**' command.

STEP 9: Displaying the Output

Once the analysis is run, the output will be displayed. Particularly, the punching shear design is of great importance as the footing cannot be designed without the punching shear requirement being adequately satisfied. To do this, go to the '**Run**' menu and click on '**Run Analysis & Design**' command or simply click on the '**F5**' key. When you do this, you will be prompted to save the model, if you haven't already don this. When you save the model, the following window showing the displacement of the soil in a banded figure will be displayed.

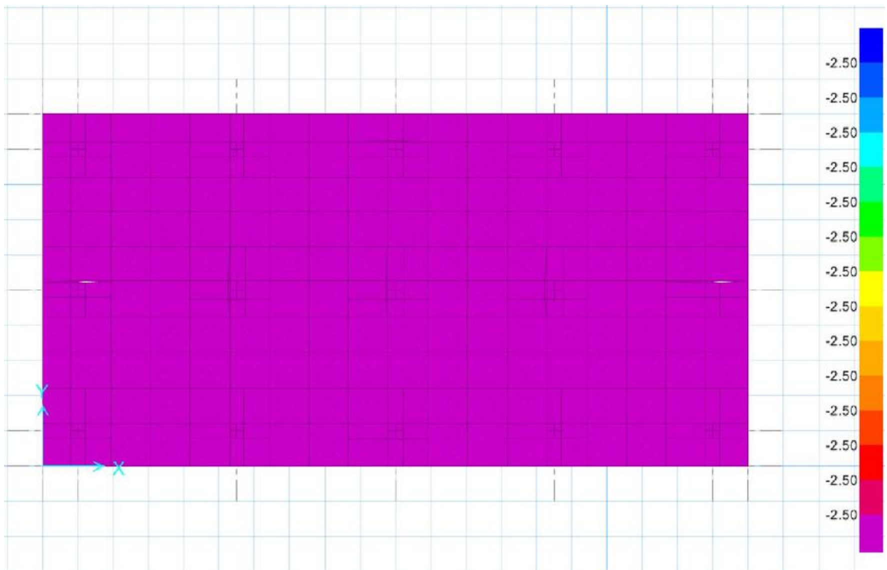


Figure 247

To display the punching shear ratio, go to the '**Display**' menu and click on '**Show Punching Shear Design**'. After this, the punching shear ratio will be displayed in the plan view around the foundation column as in the following figure.

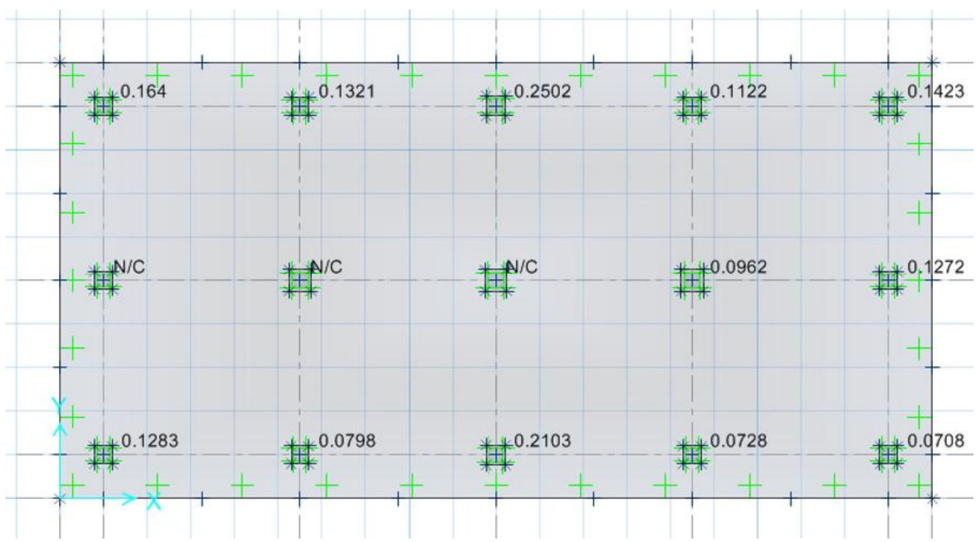


Figure 248

As can be noticed from the figure, the punching shear ratio is 0.164 for the first column, '**N/C**' for the second, fifth and eighth columns. This usually happens when the column dimension is not entered when the loads are assigned or when the mat thickness is inappropriate. In this example, all the column dimensions are entered correctly and the reason for the appearance of '**N/C**' on some of the columns is due to inappropriate mat thickness. For the rest columns, the punching shear ratio is well below unity. Generally, a punching shear ratio less than one indicates the concrete thickness is adequate to resist punching shear and a value greater than one indicates that the punching shear capacity is exceeded somewhere along the critical section. For economical design, it is recommended to keep the punching shear ratio between 0.95 and 1 as very small values of punching shear ratio means excess concrete thickness is used.

Thus, to get an acceptable punching shear ratio, change the thickness of the slab to 600mm. To do this, first unlock the model by pressing button  from the list of buttons below the menu bar. Then, go to the '**Define**' menu and click on '**Slab Properties**'. After this, highlight '**Mat**' and '**STIFF**' one at a time and click on '**Show/Modify Properties**' button and change the value for '**Thickness**' to '**600**'.

After this, if you click on '**Run**' > '**Run Analysis & Design'**, the following window will appear.

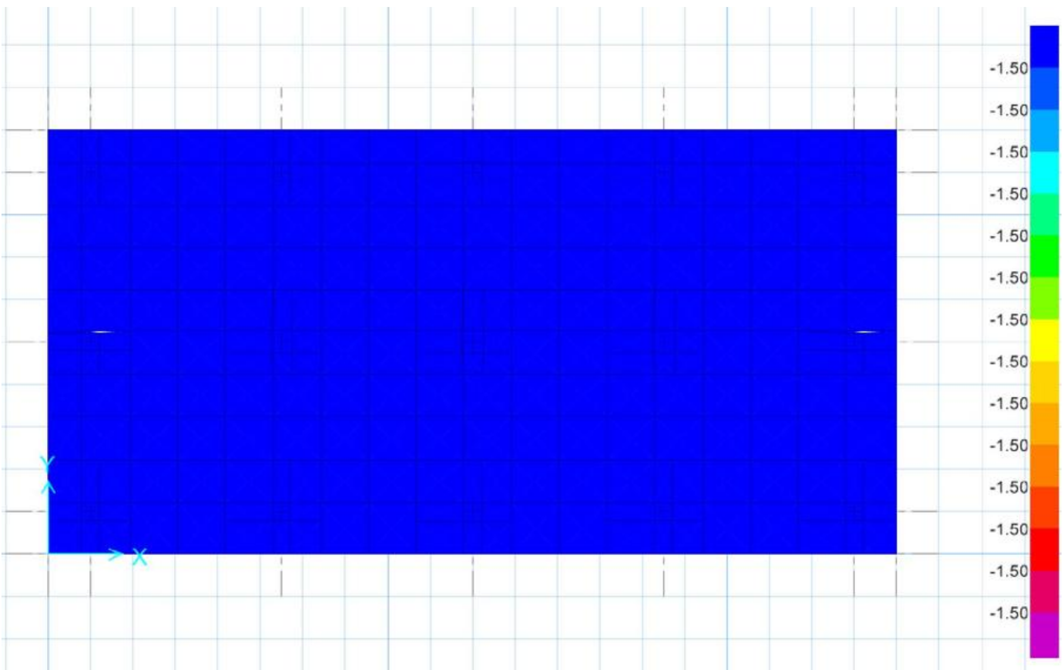


Figure 249

If you click on '**Display**' > '**Show Punching Shear Design**', the following will be displayed on the screen.

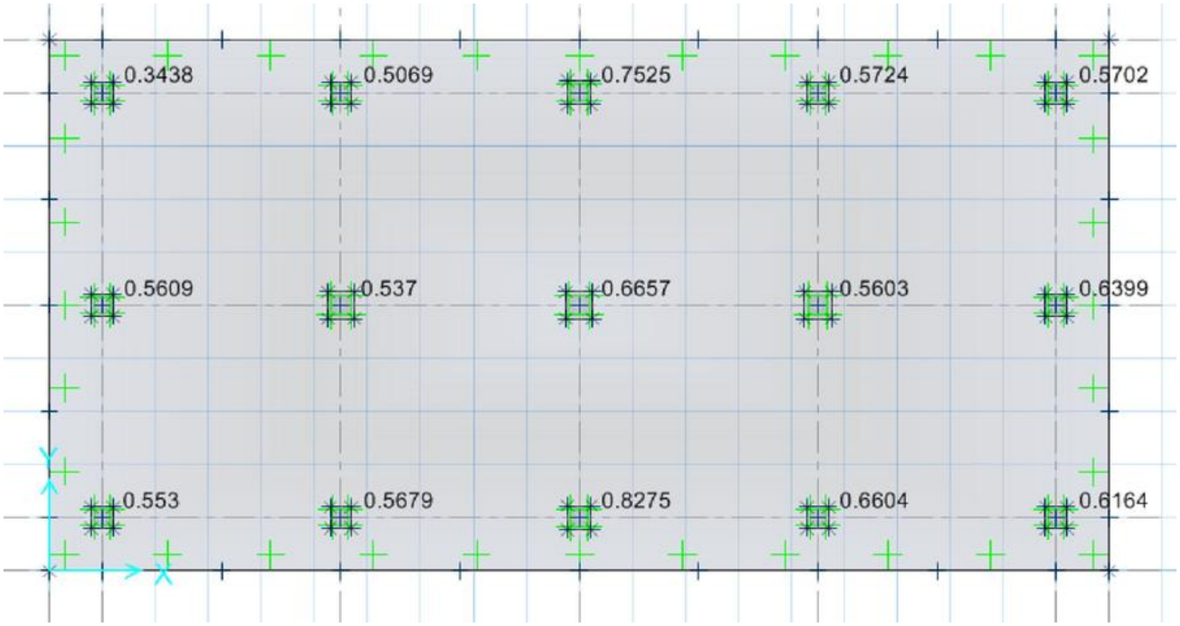


Figure 250.

It can be seen from the above punching shear design diagram that all the punching shear ratios are acceptable.

Through the '**Display**' menu, relevant quantities can be displayed on the screen. For instance, the '**Display**'>'**Show Strip Forces**' command or by simply clicking the '**F8**' key, the following window will be displayed.

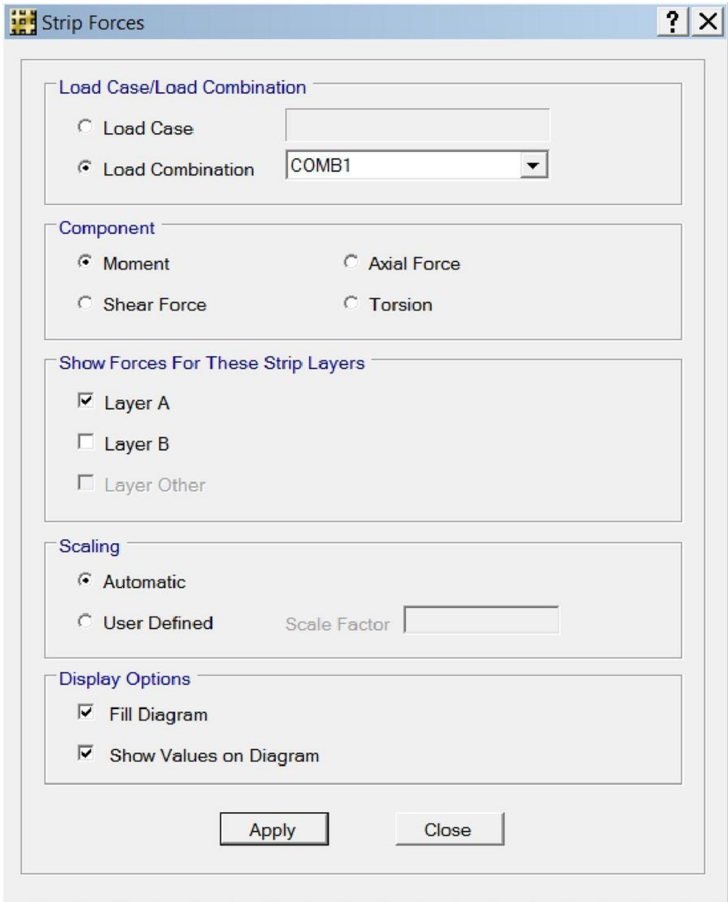


Figure 251

In this window, the '**Load Case/Load Combination**' box provides with radio buttons to select for which particular load case or load combination that we want to display the output. The '**Component**' box contains four radio buttons to select which quantity to display. The '**Show Forces For These Strip Layers**' box allows us to select the strip layer for which the quantity is displayed. Both strip layers can be selected at the same time. From the '**Scaling**' box, we can select whether automatic scaling or user defined scaling is used while displaying the diagram. The '**Display Options**' box allows us to fill or not to fill the diagram and to display or not to display the values on the diagram. For the preferences shown in Fig. 251, the following diagram will be displayed.

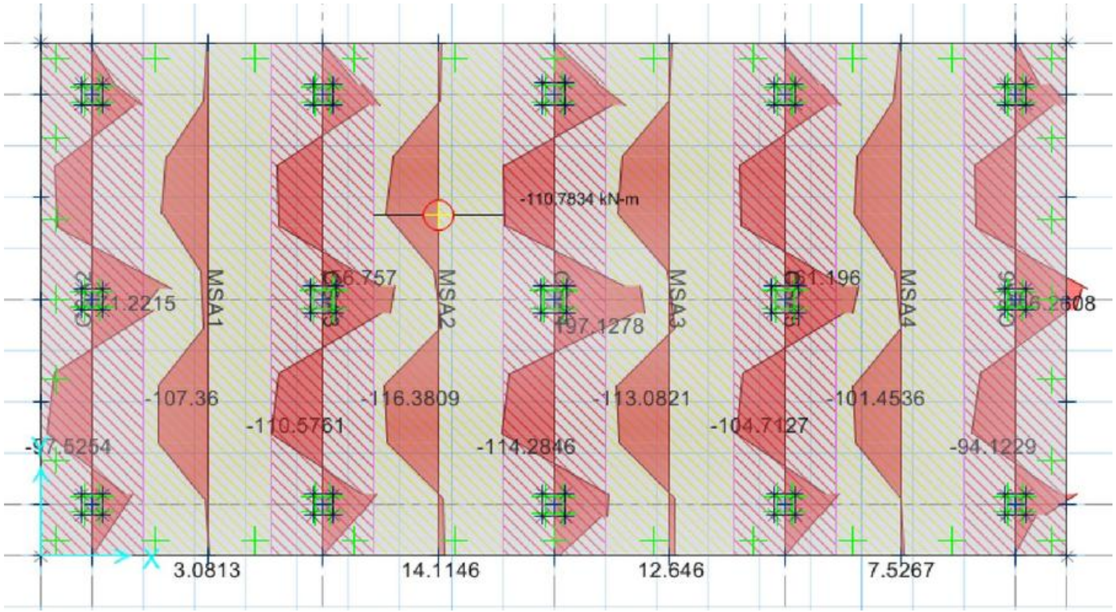


Figure 252

The '**Display**'>'**Show Slab Design…**' command results in the window shown in Fig 254. In this window, several options can be set in order to display the footing design in the way we wanted to. The '**Choose Display Type**' box allows us to select the '**Design Basis**' between '**Strip Based**' and '**Finite Element Based**'. Unless some differences in the way the design is displayed, there is no difference in the amount of reinforcement between these selections. Through the '**Display Type**' in the same box, it can be selected whether to display flexural reinforcement or shear reinforcement. This box also allows us to impose or not to impose minimum reinforcement during the design. The '**Rebar Location Shown**' box allows us to select which reinforcement, top or bottom or both, to be displayed. The '**Reinforcing Display Type**' box allows us to set the manner in which the amount of reinforcement is displayed. The option whether to show the reinforcing envelop diagram and the reinforcing extent can be set by the check boxes in the '**Reinforcing Diagram**' window. The strip layer direction for which the amount of reinforcement is displayed can be chosen from the '**Choose Strip Direction**' box. The '**Display Options**' box allows us whether to display output in filled diagram or not and whether the values at controlling stations will be displayed or not. If we want to display the amount of reinforcement above some specified reinforcement bar area or spacing, we can use the options in the '**Show Rebar Above Specified Value**' box. When the '**Typical Uniform Reinforcing Specified Below**' radio button is selected, the '**Typical Uniform Reinforcing'** box get activated. In this box, we can set a specific value above which the

reinforcement amount will be displayed. The reinforcement diagram output, for the options set in Fig. 253, will be shown below in fig. 254.

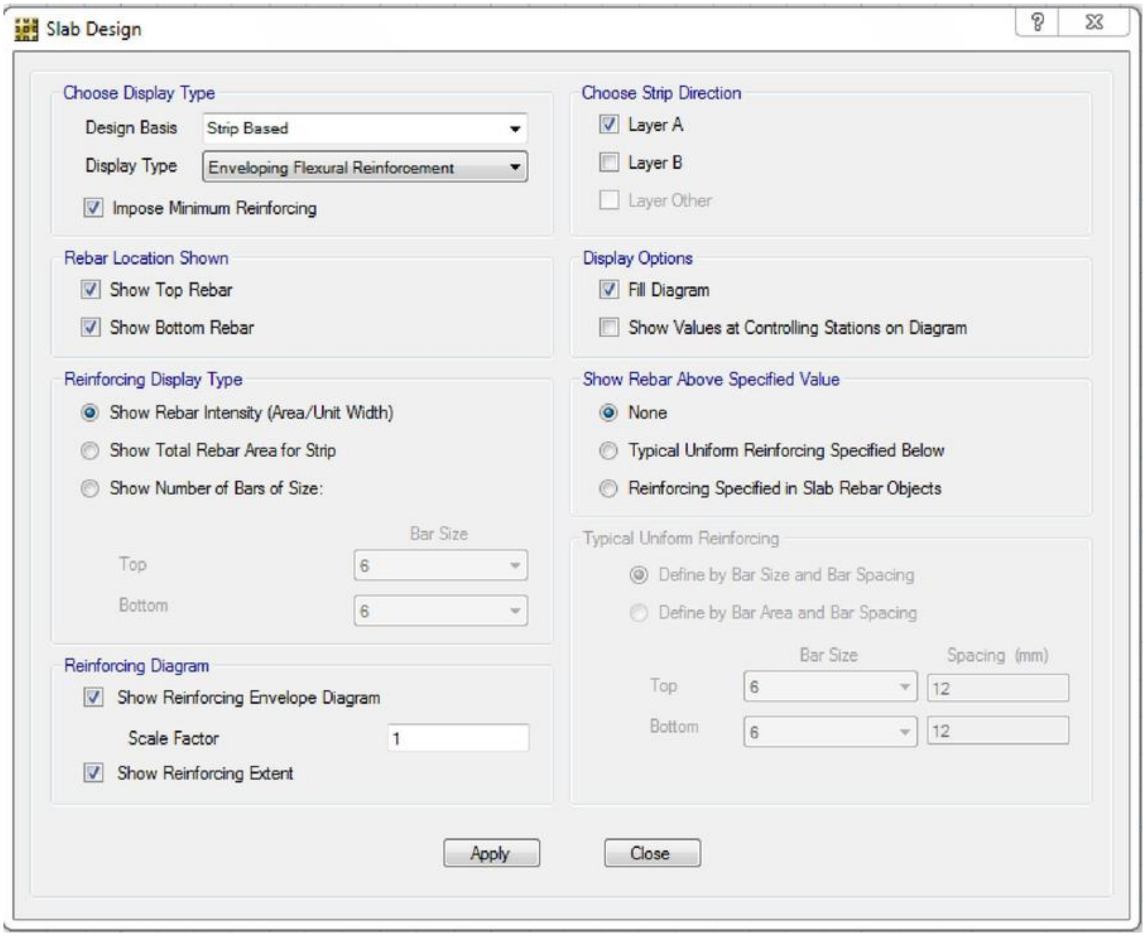


Figure 253

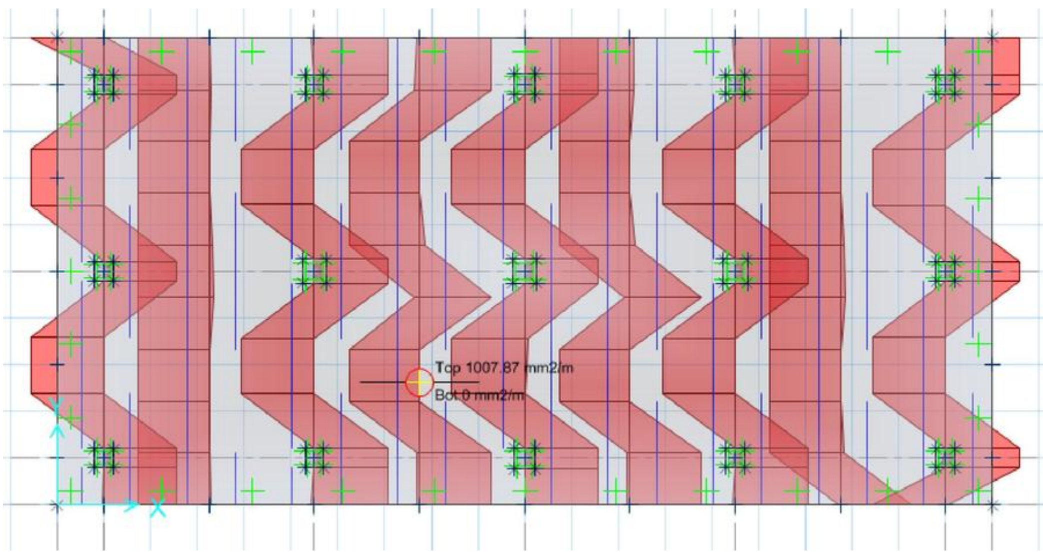


Figure 254

The design outputs can also be displayed in tabular format by clicking on the '**Show Tables…**' menu item from the '**Display**' menu or by just clicking on the equivalent icon  from the tool bar below the menu bar and the following window will pop up.

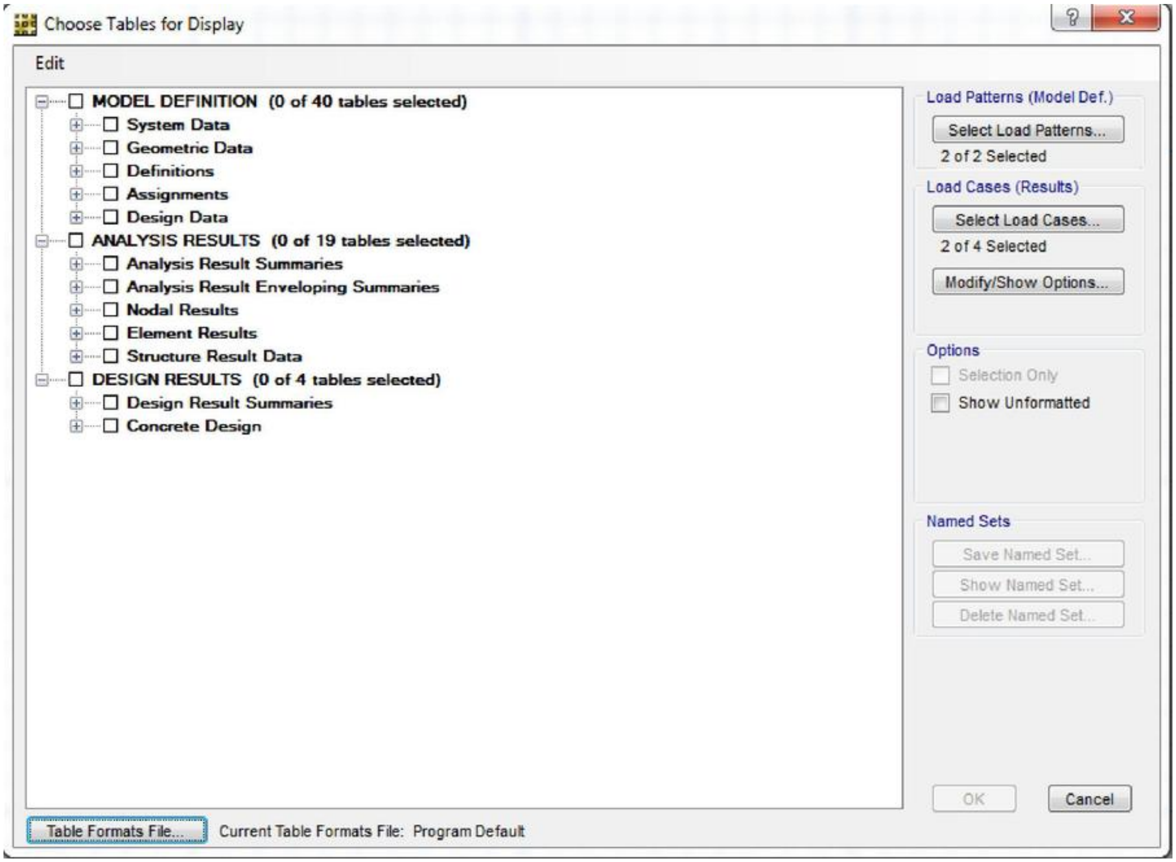


Figure 255

In this window, we can select any of the model definitions or analysis results or design results and press '**OK**' to display the quantity which we want to have a look at. By using the right hand side buttons in the window, the load patterns and the load cases can be selected.

STEP 10: Detailing

After running the analysis and after checking that the results are reasonable, the detailing will be done. However, before running the detailing, the detailing preferences can be set from the '**Detailing**' menu. From the '**Detailing Preferences…**', the likes of dimensional units and material quantity units can be selected. From '**Slab/Mat Detailing Preferences…**', the likes of rebar curtailment options, the rebar detailing options, rebar selection rules and preferred

rebar sizes can be selected. The '**Drawing Sheet Set-up…**' menu allows us to set-up the contents of the drawing sheet. The '**Drawing Format Properties…**' allows us to set some formats in which the output displayed.

To run the detailing, go to '**Run**' menu and click on '**Run Detailing…**' or simultaneously press '**Shift**' and '**F5**' keys or just click on the run detailing icon  from the tool bar just below the menu bar. Then, the '**Run Detailing Options**' window pops up so that we set the detailing options. Set the detailing options which you want and click '**OK**'.

Once the detailing is run, the detailing can be displayed. The detailing display options can be best accessed from the '**Model Explorer**'. When expanded in full, the '**Detailing**' tab of the model explorer looks like:

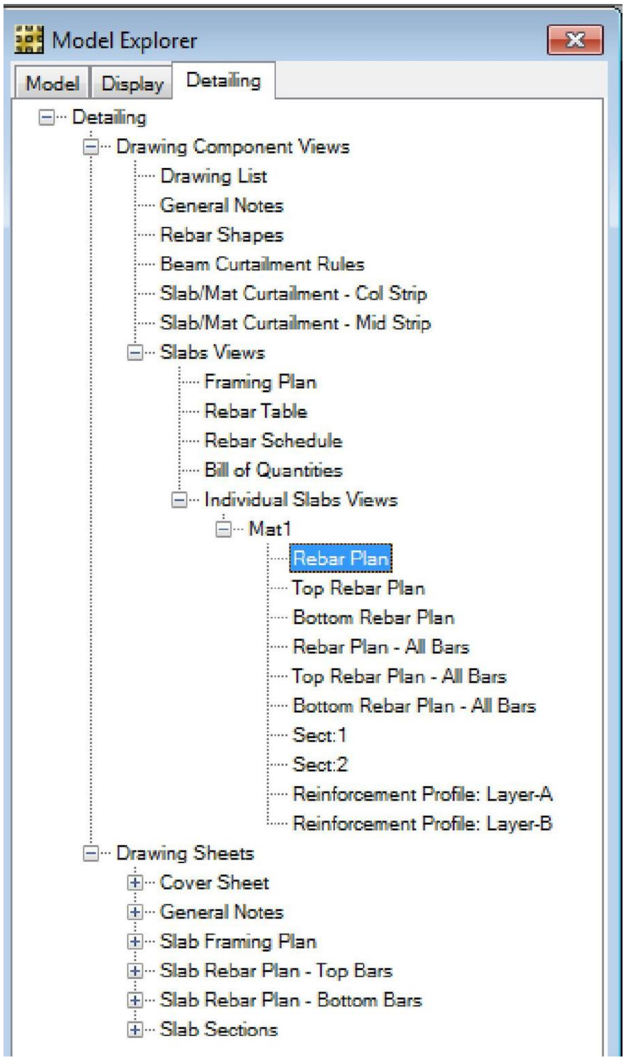


Figure 256

By clicking on any of the options from the detailing tab, a desired detailing can be displayed. For instance, by double clicking on the '**Rebar Plan**', the following detail of footing can be shown.

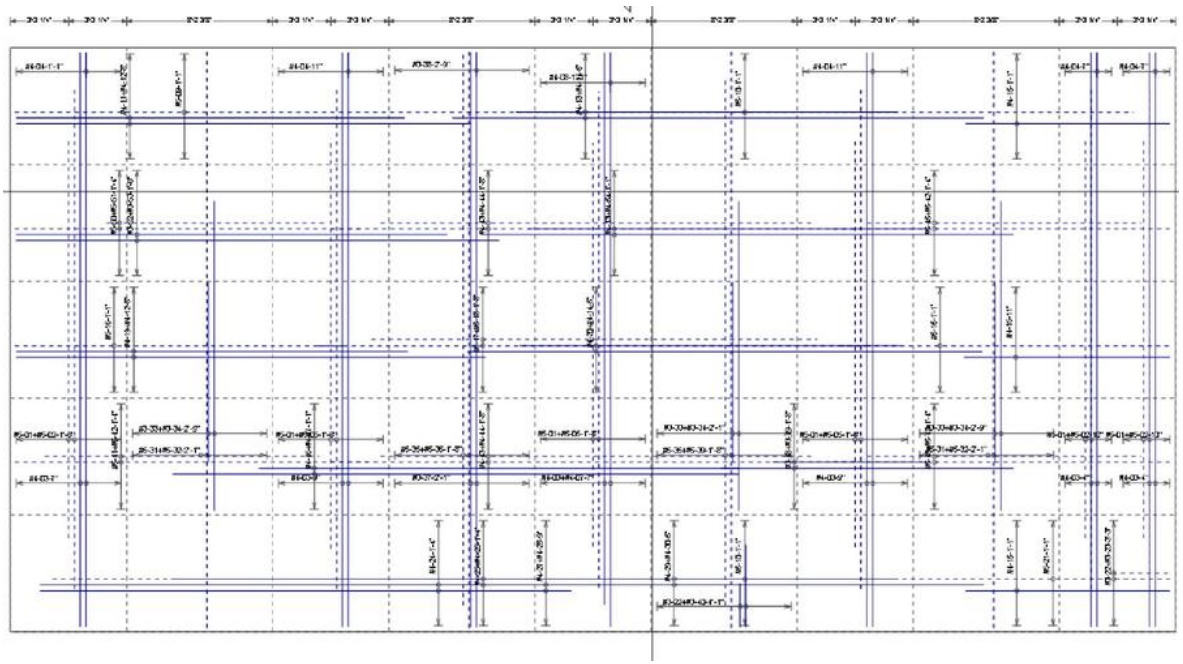


Figure 257

In this detail, the diameter of the reinforcement which is used is 10mm. If you want to change the diameter….

Apart from this detail, other details can also be shown.

STEP 11: Reporting

The last step of foundation design is reporting. Before creating the report, the report preferences should be set up. To do this, go to the 'File' menu and click on 'Report Set-up…' and the following window pops up.

In this '**Report Setup Data**' window, the user preferences regarding the reporting such as the report output type, the report page orientation and the report items can be set along with the load patterns and load combinations. Once the preference is set, the report can be created by clicking on '**Create Report**' command in the '**File**' menu. The '**Advanced Report Writer**' command in the same menu can be used to set some advanced reporting formats.

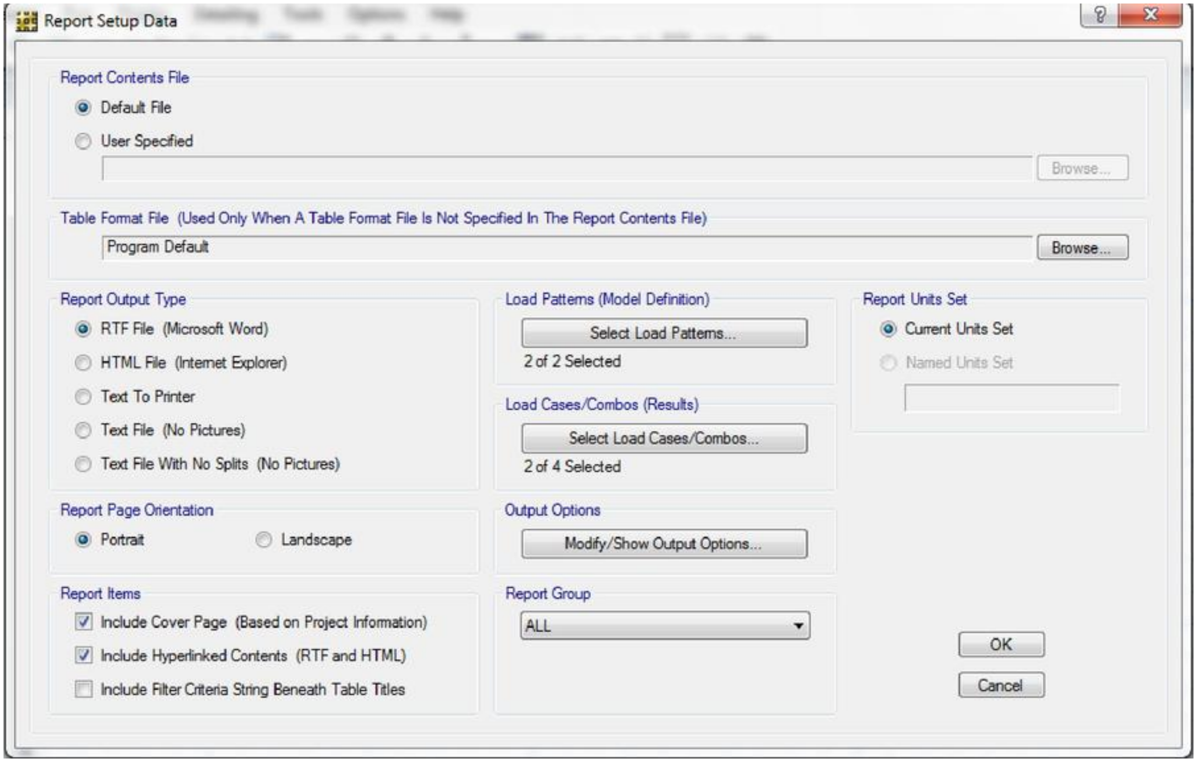


Figure 258

This concludes the tutorial for the design of single footing using a model imported from AutoCAD.

Design of Pile and Pile Cap

Week 8–10

Here is the link to design pile and pile cap

Practice, Review/Reserved Day

Week 12-13

Lab Report Assessment, Self study

Week 14-15

Lab Test, Viva, Quiz, Overall Assessment, Skill Development Test (Competency)

Week 16-17