



(RESEARCH ARTICLE)



Optimization of isolated and combined pad foundation using computer aided application of finite element approach

Ubi Stanley E. *

Department of Civil Engineering, Cross River University of Technology, Calabar, Nigeria.

Global Journal of Engineering and Technology Advances, 2021, 06(03), 024–041

Publication history: Received on 20 January 2021; revised on 23 February 2021; accepted on 25 February 2021

Article DOI: <https://doi.org/10.30574/gjeta.2021.6.3.0030>

Abstract

Most Finite Element packages provide means to generate meshes automatically. However, the user is usually confronted with the problem of not knowing whether the mesh generated is appropriate for the problem at hand. Since the accuracy of the Finite Element results is mesh dependent, mesh selection forms a very important step in the analysis of isolated and combined footing pad foundation. SAFE is an ultimate tools use in the design of concrete floors and foundation system, hence provide a suitable means for the user. From framing layout all the way through to detail drawing production, SAFE integrate every aspect of engineering design which are in one process easy and intuitive environment. SAFE provides unmatched benefits to the engineer with its truly unique combination of power, comprehensive capabilities, and ease-of-use. In the context of this research, we have plotted graphs showing the relationship between the nodes and displacement with the stress patterns as generated from the software. It is understood from the graph that multiple elements in the process of meshing will make the footing to be at equilibrium. The research also carry the shape deformed diagram which shows the deformation of the footing due to the impose load (stress) on the footing, it also give the bending moment diagram of the footings. The basic structure and analysis of the single and double pad footing foundations have been designed using Finite Element Analysis (FEA) with the failure planes being considered. The results obtained, it is assumed that FEA is an ideal design method that breaks foundation design into basic elements and nodes that shows the action of the loading on the footings.

Keywords: Safe; Finite; Mesh; Foundation; Footing

1. Introduction

The foundation of a structure is the element of that structure which connects it to the ground, and transfers loads from the structure to the ground, it is sometimes considered as either shallow or deep. Hence by so doing, foundation is given a greater value in the recent years till date, Kumar et al (2015). The application of soil mechanics and rock mechanic (geotechnical engineering) in the design of foundation element in a structure. The finite element method is such a widely used analysis-and-design technique that it is essential that undergraduate engineering students have a basic knowledge of the theory and applications of the technique (Hutton, 2015). Hence it is very important in geotechnical engineering in the following ways; mostly failure analysis (what will be the failure plane in the soil mass when the foundation is loaded. How far will it extend? How far would it affect?), for analysis of rock fractures, for analyzing the forces in retaining walls and settlement of foundation, would piling affect the adjacent existing buildings and failure can happen due to pore pressure. Specifically shallow foundation (pad) are used to support individual point load such as that directly from the superstructure to the receiving soil strata, hence the construction of this type of foundation is based upon some certain facts which govern it, this facts are soil type and nature of load as stated by Kumar et al (2015). Foundation is the most important part of the structure and by so doing it must be given a serious attention in the design and construction. Finite Element Analysis is a very accurate means compare to other means in the (analysis) of a footing,

* Corresponding author: Agbor RB
Environmental Biotechnology Unit, Department of Genetics and Biotechnology, University of Calabar, Calabar, Nigeria.

or foundation. Finite element analyses have also proved very useful for planning instrumentation studies by showing where objects can be located to best advantage through meshing, which is the primary aim of finite element. Fattah et al., (2017) state that this type of foundation is often used when the structural load will not cause excessive settlement of the underlying soil layers. In general, shallow foundations are more economical to construct than deep foundations. Pad foundations are used to support an individual point load such as that due to a structural column. Kharagpur (2017) pad foundations are shallow foundation used to support an individual point load such as that which is transfer from the super structure due to the structural column. They are spread footings, single footings or double footings which are may be circular, square or rectangular. They usually consist of a block or slab of uniform thickness, but they may be stepped if they are required to spread the load from a heavy column. Farrokhzad, et al (2011), a node is a specific point in the finite element at which the value of the field variable is to be explicitly calculated. Exterior nodes are located on the boundaries of the finite element and may be used to connect an element to adjacent finite elements. Nodes that do not lie on element boundaries are interior nodes and cannot be connected to any other element. Soil is considered by the engineer as a complex material produced by the weathering of the solid rock. Soil is the most important material which is in use for construction of civil engineering structures (Kouzer and Kumar, 2010). Among all parameters, the bearing capacity of soil to support the load coming over its unit area is very important. Principal factors that influence ultimate bearing capacities are type of soil, width of foundation, soil weight in shear zone and surcharge. Structural rigidity and the contact stress distribution do not greatly influence bearing capacity. Bearing capacity analysis assumes a uniform contact pressure between the foundation and the underlying soil. With other factors unchanged the type of failure of soil, depth of foundation and effect of water table also govern the bearing capacity of the soil (Madan et al. 1989). Similarly the effect of depth of footing on bearing capacity of soil is studied. In general, other factors remain constant, bearing capacity of soil goes on increasing as depth or width of foundation increases. In case of local shear failure, amongst different shapes of footing is found to be lowest in comparison with square, circular and rectangular shaped footings. The aim of the study was to model isolated and combined pad footing using computer aided application of finite element approach.

2. Material and methods

2.1. Study Area

This work is based on optimization for single and combined footings (Pad foundation) using Finite Element Analysis. The general techniques and terminology of finite element analysis will be introduced with reference to depict a volume of some material(s) having known physical properties. The volume represents the domain of a boundary value problem to be solved. Note that this implies that an exact mathematical solution is obtained; that is, the solution is a closed-form algebraic expression of the independent variables. In practical problems, the domain may be geometrically complex as is, often, the governing equation and the likelihood of obtaining an exact closed-form solution is very low. Therefore, approximate solutions based on numerical techniques and digital computations are most often obtained in engineering analyses of complex problems. Finite element analysis is a powerful technique for obtaining such approximate solutions with good accuracy. Hence foundation of a structure require a reliable soil to enable it to withstand the proposed structure, necessary measures are taken to ensure that a suitable soil is been chosen in other to acquire a suitable design status.

2.2. Foundation Modelling

The footings were assumed to be founded on clay with un-drained shear strength of 100 kPa. This value is typical residual clay which has vane shear strengths in the range of 70 to 120kPa. For earthquake loading the un-drained shear strength will be greater because of the rate of loading. We observed an increase of about 40% in the un-drained shear strength of this soil when tested at rates of loading comparable to those during a seismic event. Thus the value of 100kPa used herein for earthquake loading was equivalent to the lower end of the range of values found in normal site investigation. Hence the factors considered while modeling this foundation are listed below;

- Energy Dissipation Mechanism and,
- Factor of Safety

2.2.1. Energy dissipation mechanism

The dynamic response of a structure depends on its mechanical properties and the characteristics of the induced excitation. Mechanical properties which are efficient to mitigate the structure's response when subjected to certain inputs might have an undesirable effect during other inputs. Ground motions vary significantly from one another in amplitude, frequency content and duration. Those characteristics are influenced by source mechanism and travel path

and modified by local geological and soil conditions. The ability to dissipate the induced energy is crucial to the earthquake resistance of a structure. Various energy dissipation mechanisms have been proposed to enhance structural response (Oyenuga 2001). These energy dissipation mechanisms can be of various types such as viscous, rigid-plastic, elastoplastic, viscoplastic, or combination of them.

2.2.2. Factor of safety

Where earthquake loads are included, the minimum safety factor for the foundation shall be 1.1. “The 2012 IBC Code and Commentary, volume 2, states the following: “the safety factor for stability of foundations predates the development of load combination.

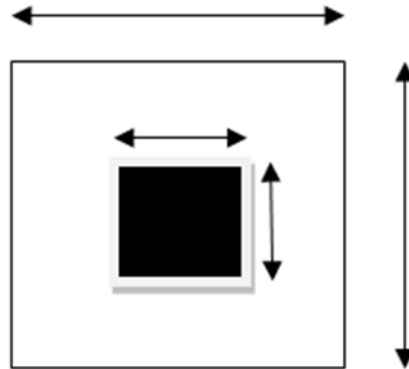


Figure 1 Single footing

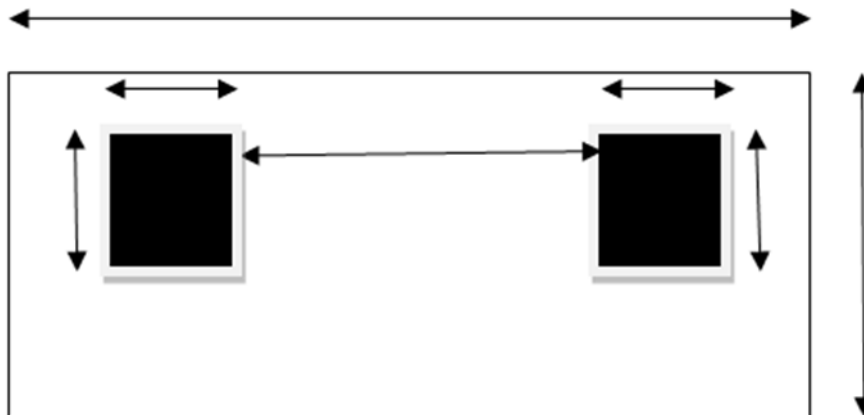


Figure 2 Double footings

2.3. Fem Formulations (Using H - Mesh)

2.3.1. Mesh Generation

Each element in the finite element model is addressed by its number. Also each node is addressed by its number. The inter-connectivity of the elements is determined by the common nodes shared by the elements. In a model with few elements and nodes, the user can manually divide the domain, number each element and node, and keep track of the element connectivity, hence this process is called mesh generation. However, in models with many nodes and elements, the effort required to divide the domain into elements and attend to connectivity is great. It then becomes difficult to accomplish this task without committing errors. However, there are several finite element preprocessors which do this job automatically once the geometry is defined (Bowles 1997). Users can then devote more time to interpreting results. Shephard has reviewed the current trends in mesh generation. Although there are several ways to generate meshes, these methods fall into two broad categories:

2.3.2. Mapping techniques

This type of mesh generation is best suited when the geometry is simple - as in the case of a rectangle or a cuboid. Typically the user needs to choose the number of elements on each of the edges that defines the geometry and the element concentration along the edges. The software then generates the mesh simply by joining nodes on the opposite edges. The software considered here is **SAFE** to generate the mesh.

2.3.3. Free mesh generation

This method of generation is best suited for models with complicated geometry. SUPERTAB has this capability. The model is broken down into sub-areas and sub-volumes. On each of the curves of every sub-area and sub-volume the number of elements and their concentrations are selected. The software then generates a mesh that is consistent with the selected values and satisfies the requirement on the aspect ratios and the distortion factors of the elements

Hence the mesh generation method considered here is mapping techniques, where we choose a rectangular footing base for double footings, with dimension 2000mm by 4050mm and rectangular footing base for single footings with dimension 2400mm by 2800mm. where the maximum mesh size for both footings is 200mm, and was generated with the aid of a software safe, as will be discussed in chapter four of this work.

2.4. Data Analysis

The objective is to develop a practical and efficient procedure of single and double footing (Pad foundation) with computer aided application using Finite Element Approach (David 2004). The thesis is that for many problems the minimization of the trace of the stiffness matrix with respect to the nodal coordinates, leads to a minimization of the potential energy, and as a consequence, provides the optimal grid configuration. The research work is regular shaped concrete footings, with overall dimensions of, for single footing length of 2800mm width of 2400mm with a 450mm column centered, for double footings length of 4050mm width of 2000mm with two columns of 450mm spaced at 2500mm.

3. Procedures in Carrying out Data Analysis using SAFE (software):

3.1. Step 1: Begin a new model

Launch the software and open new model, then it pops out new model initialization form. From the form we choose the design code i.e. B.S8110 and unit in metric. In this Step, the dimensions and basic grid will be defined, which will serve as a guide for developing the model.

- Click the **File menu > New Model** command to access the New Model Initialization form. This form is used to specify the starting point of the model creation: a Blank screen, a screen with a Grid Only, or one of eight templates. Default units also may be selected here, along with the design code and preferences.
- In the Design Data area, select B.S 8110, 1997 from the **Design Code** drop-down list.
- In the Design Data area, verify that the Units are set to Currently US; if not, click the **Modify/Show** button and select the U.S. Defaults on the Units form.
- In the Initial Model area, for single footing click the **single footing** button and for double footings click the **double footings** button to display the footing drop box and edit the following; along X direction, left and right edges distance 0.9m each, along Y direction, top and bottom edge distance 1.2m each, load size 600mm for double footings spacing 2.5m loads, dead 30KN live 45KN for double footings load one and two are the same, footing thickness 500mm, surcharge modulus say 18000KN/M³ then click **ok**.
- Save your model often! Click the **File menu > Save** or **save as** command. Specify the directory in which to save the model. For this research, the file name as footing design.

3.2. Step 2: Define the material

In this Step, material and section properties for the footings (area object), columns, (line object) are defined.

- Click the **Define menu > Materials** command to access the Materials.

- Highlight **4000Psi** in the Materials area, and click the **Modify/Show Material** button to display the Material Property Data form. That form lists the properties associated with **4000psi** concrete; this is the concrete property that will be used in our model.
- Click the **OK** button to accept this material as defined.
- In the Materials area, highlight **A615Gr60**.
- Click the **Modify/Show Material** button to display the Material Property Data form. This form lists the properties associated with Grade 60 reinforcing; this is the rebar property that will be used in our model.
- Click the **OK** button to accept this material as defined.
- Click the **OK** button on the Materials form to accept all of the defined materials.
- Click the **File menu > Save** command to save your model.

3.3. Define column property

- Click the **Define** menu > **Column Properties** command to access the Column Properties form.
- In the Column Property area, highlight **COL1**
- Click the **Modify/Show Property** button to access the Column Property Data form. Recall that the columns for this research are 450mm square and that the drop panels are 0.9m by 1.2m square.
- In the Column Section Dimensions area, select **Rectangular** from the Column Shape drop-down list
- In the Column Section Dimensions area, type **450mm** into the Parallel to 2-Axis edit box and type **450mm** into the Parallel to 3-Axis edit box.
- Make sure that the **Include Automatic Rigid Zone Area over Column** option is checked. This option restricts deformation of the slab at the column location, which prevents unrealistic peaks in moment distribution from occurring.
- In the automatic drop down area check these **include automatic drop panel over column** option. This option automatically adds a drop panel when a column with this property is drawn.
- In the Automatic Drop Panel Dimensions area, type **900mm** into the Parallel to 2-Axis edit box and type **1200mm** into the Parallel to 3-Axis edit box.
- In the Automatic Drop Panel Dimensions area, select **DROP** from the Slab Property drop-down list.
- Click the **OK** button to leave the Column Property Data form.
- Click the **Add New Property** button to access the Column Property Data form.
- In the General Data area, type **COL2** in the Property Name edit box.
- In the Column Section Dimensions area, select **Rectangular** from the Column Shape drop-down list
- In the Column Section Dimensions area, type **450mm** into the Parallel to 2-Axis edit box and type **450mm** into the Parallel to 3-Axis edit box.
- Make sure that the **Include Automatic Rigid Zone Area Over Column** option is checked
- In the Automatic Drop Panel Dimensions area, make sure that the **Include Automatic Drop Panel Over Column** option is unchecked
- Click the **OK** button to leave the Column Property Data form.
- Click the **OK** button to accept the Column Property definitions

3.4. Step 3: Define static load pattern

In this Step, the dead and live static load patterns are defined. That is, we will name the various types of loads and specify the self-weight multipliers. The loads will be assigned to objects, and the values for the loads specified (uniform dead load of 30kN/m² and live load of 50kN/m²), in Step 8.

- Click the **Define** menu > **Load Patterns** command to access the Load Patterns form
- Note that load patterns **DEAD** and **LIVE** are defined by default

- Recall that the project will be analyzed for the dead load plus the self-weight of the structure. Thus, the Self Weight Multiplier should be set equal to 1 (this will include 1.0 times the self-weight of all members) for the DEAD load. Only the DEAD load pattern should have a non-zero Self Weight Multiplier.
- Click the **OK** button to accept the defined static load patterns

3.5. Step 4: Define load cases

In this Step, the Load Cases are defined. This is where the type of analysis is specified.

- Click the **Define** menu > **Load Cases** command to access the Load Cases form
- With the **DEAD Load** Case Name highlighted, click the **Modify/Show Case** button to display the Load Case Data form. This data form changes based on the type of load case specified.
- In the Load Case Type area, select **Static** from the drop-down list. Modal and Hyper static also are available as load case types
- In the Analysis Type area, select the **linear** option. When working with a Static Load Case Type, the program offers the option to do Linear, Nonlinear (Allow Uplift), Nonlinear (Cracked), or Nonlinear (Long Term Cracked) analysis. For our tutorial example, a Static, Linear analysis will be performed for DEAD and LIVE
- In the Loads Applied area, verify that the load pattern is DEAD with a scale factor of **1**.
- Click the **OK** button to close the Load Case Data form
- Review the LIVE load case, if so desired, by selecting it and using the **Modify/Show Case** button as described for the DEAD load case
- Click the **OK** button to close the Load Cases form
- Click the **File** menu > **Save** command

3.6. Step 5: Draw Object

In this step, column with drop will be drawn with the active window set (i.e., Plan View window active and the snap to points and grid intersections enabled), use the following Action Items to draw columns.

- Click the **Draw** menu > **Draw Columns** command to access the Draw Columns form
- Click in the Property Below drop-down list and select **COL1**. This is the property for the 450mm by 450mm column with a drop panel defined in Step 2.
- Click in the Property Above drop-down list and select **NONE**. There are no columns above the slab.
- Click in the Height Below edit box and type 12, and click in the Height Above edit box and type 0. Note that the units are meter.
- Click in the **Cardinal Point** drop-down list and select 10 (Centroid).
- Locate the mouse cursor at a distance of 0.9m from the left inside the footing and click, for double footing do the same for the two columns with distance of 2.5m apart
- Click on the **Select** menu > **Select > Pointer/Window** command or press the **Esc** key on the keyboard to exit the Draw Columns command.
- Locate the mouse cursor just above and to the left of grid intersection C6, hold down the left mouse button, and drag diagonally to just below and to the right of D5 and release the mouse button. The status bar in the lower left-hand corner should show "2 Points, 1 Lines, 1 Areas, 4 Edges selected." For single footing, and "4 Points, 2 Lines, 2 Areas, 8 Edges Selected." If the selection is not correct, simply click the Select menu > Clear Selection command and try again.
- Click the **Edit** menu > **Delete** command or press the Delete key on the keyboard to remove the columns enclosed in the window.
- Click the **File** menu > **Save** command to save your model.
- Click the **View** menu > **Set Default 3D View** command to display the model in 3D. Note how the columns extend below the slab.
- Click the **View** menu > **Set Plan View** command to return to the Plan View before continuing the project.

3.7. Step 6: Add Design Strip

In this step, design strips will be added to the model. Design strips determine how reinforcing will be calculated and positioned in the slab. Forces are integrated across the design strips and used to calculate the required reinforcing for the selected design code (Satis and Santhian 2010). Typically design strips are positioned in two principal directions: Layer A and Layer B. Similar to the previous sections, ensure that the Plan View is active and the snap to points and grid intersections features are enabled. Add design strips to the model as follows:

- Click the **Edit** menu > **Add/Edit Design Strips** > **Add Design Strips** command to display the Add Design Strips form
- In the Options area, select the **Add Design Strips along Cartesian Grid Lines** option. Make sure that the Include Middle Strip option is checked
- In the Parameters area, click in the **Grid Direction** drop-down list and select **X**.
- Select **A** from the Strip Layer drop-down list.
- Select the **Auto** option. The added design strips will automatically adjust their width to align with adjacent strips
- Click the OK button to leave the Add Design Strips form. Design strips in the X-axis direction should now appear as solid lines.
- Left click on the design strips that lie below grid line 3 to select them; the status bar in the lower left-hand corner should show “4 Design Strips selected.” If the selection is not correct, simply click the Select menu > Clear Selection command and try again
- Left click on the slab (anywhere except at a column or design strip location) to select it; the status bar in the lower left-hand corner should now show “1 Areas, 6 Edges, 4 Design Strips selected.”
- Left click at the left ends of the 4 selected design strips; the status bar in the lower left-hand corner should now show “4 Points, 1 Areas, 6 Edges, 4 Design Strips selected.”
- Click the **Edit** menu > **Align Points/Lines/Edges** command to display the Align Points/Lines/Edges form
- Select the Trim Line/Edge/Tendon/Strip Objects option.
- Click the **OK** button to leave the Align Points/Lines/Edges form. The design strips below grid line 3 should now be trimmed to the edge of the slab.
- Left click on the design strip that lies on grid line 5 to select it; the status bar in the lower left-hand corner should show “1 Design Strips selected.”
- Press the **Delete** key on the keyboard to remove the selected design strip from the model.
- Click the **Draw** menu > **Draw Design Strips** command to display the Draw Design Strips form
- Select **A** from the Strip Layer drop-down list
- Select **Column Strip** from the Strip Design Type drop-down list.
- Type 4.5 into the Start Width Left edit box.
- Type 6 into the Start Width Right edit box.
- Type 4.5 into the End Width Left edit box and type 6 into the End Width Right edit box.
- Left click at grid intersection A5 and at C5, and then click the right mouse button to stop drawing.
- Left click at grid intersection D5 and at F5.
- Press the **Esc** key on the keyboard to leave the Draw command
- Click the **Edit** menu > **Add/Edit Design Strips** > **Add Design Strips** command to display the Add Design Strips form.
- Select **Y** from the Grid Direction drop-down list.
- Click in the **Strip Layer** edit box and select **B** from the drop-down list.
- Click the OK button to leave the Add Design Strips form. Design strips in the Y-axis direction should now appear as solid lines.
- Left click on the design strips that lie to the left of grid line D to select them; the status bar in the lower left-hand corner should show “2 Design Strips selected.”
- Left click on the slab (anywhere except at a column, drop panel, beam or design strip location) to select it; the status bar in the lower left-hand corner should now show “1 Areas, 4 Edges, 2 Design Strips selected.”

- Left click at the bottom ends of the selected design strips; the status bar in the lower left-hand corner should now show “2 Points, 1 Areas, 4 Edges, 2 Design Strips selected.”
- Click the **Edit** menu > **Align Points/Lines/Edges** command to display the Align Points/Lines/Edges form.
- Select the Trim Line/Edge/Tendon/Strip Objects option.
- Click the OK button to leave the Align Points/Lines/Edges form. The Y direction design strips to the left of grid line D should now be trimmed to the edge of the slab.
- The trimming of the design strips was done for display purposes only; the program will automatically ignore the portion of a design strip that extends beyond a slab edge.
- Click the **File** menu > **Save** command to save your model.

3.8. Step 7: Display option

In this Step, the set display options will be used to alter the objects displayed.

- Click the **View** menu > Set Display Options command. When the Set Display Options form displays, uncheck the Design Strip Layer A and Design Strip Layer B check boxes in the Items Present in View area, this action will turn off the display of the design strips.
- Click the OK button to accept the changes, and the model now appears

3.9. Step 8: Assign load

In this Step, the dead and live loads will be assigned to the slab. Ensure that the Plan View is still active, and that the program is in the select mode (**Draw** menu > **Select** > **Pointer/Window** command).

- Select the slab by clicking on it anywhere that is not a beam, wall, column, drop panel or opening. The status bar in the lower left hand corner should show “1 Areas, 4 Edges selected.” click the Select menu > Clear Selection command, and try again in case of mistakes.
- Click the **Assign** menu > **Load Data** > **Surface Loads** command to access the Surface Loads form
- If it is not selected already, select DEAD from the Load Pattern Name drop-down list.
- Select Gravity from the Direction drop-down list in the Load Direction area.
- In the Uniform Loads area, type 30 in the Uniform Load edit box. Note: Additional load patterns may be defined by clicking on the “...” button next to the load pattern name. A “...” button returns you to the form used to define the item in the adjacent drop-down list or edit box, which in this case is the Load Patterns form.
- Click the OK button to accept the dead load assignment. SAFE will display the loads on the model. Use the Assign menu > Clear Display of Assigns command to remove the assignments from the display, if desired
- Click anywhere on the main slab to reselect the slab, or click the Select menu > Get Previous Selection command to select the slab.
- Click the **Assign** menu > **Load Data** > **Surface Loads** command to again access the Surface Loads form.
- Select LIVE from the Load Pattern Name drop-down list.
- Type 50 in the Uniform Load edit box in the Uniform Loads area.
- Click the **OK** button to accept the live load assignment. Again, use the Assign menu > Clear Display of Assigns command to remove the assignments from the display.
- To review the assignments to the slab, right click on the slab anywhere that is not a beam, wall, column, drop-panel or opening to access the Slab-Type Area Object Information form
- Select the Loads tab and note that the **DEAD** Load Pattern has a Load Value of **30KN/M²**, and that the **LIVE** Load Pattern has a Load Value of **50KN/M²**
- Click the **OK** button to close the Slab-Type Area Object Information form.
- Click the **File** menu > **Save** command to save your model.

3.10. Step 9: Run Analysis and Design

In this Step, the analysis and design will be run

- Click the **Run** menu > **Run Analysis & Design** command to start the analysis. The program will create the analysis model from your object-based SAFE model and will display information in the status bar in the lower left-hand corner as the analysis and design proceeds. Additional information about the run may be accessed at a later time using the File menu > Show Input/output Text Files command and selecting the filename with a .LOG extension.
- When the analysis and design are finished, the program automatically displays a deformed shape view of the model, and the model is locked. The model is locked when the Options menu > Lock/Unlock Model icon appears depressed. Locking the model prevents any changes to the model that would invalidate the analysis results

3.11. Step 10: Graphically Review the Analysis Results

In this Step, the analysis will be reviewed using graphical displays of the results

- Click the **View** menu > **Set Default 3D View** command to display the deformed shape for the DEAD load case in 3D.
- Click the **Start Animation** button in the lower right-hand corner of the display to animate the deformed shape. Speed of the animation may be adjusted by using the slider control adjacent to the button. Click the **Stop Animation** button to end the animation
- Click the **Display** menu > **Show Deformed Shape** command to access the Deformed Shape form
- In the Load Case/Load Combination area, select the Load Case option.
- Select LIVE from the Load Case drop-down list.
- Select the **Automatic** option in the Scaling area.
- Check the **Draw Contours** checkbox in the Contour Range area.
- Click the OK button to generate a 3-D deformed shape with contours for the LIVE load case
- Click the Display menu > Show Slab Forces/Stresses command to bring up the Slab Forces/Stresses form
- Select LIVE from the Load Case drop-down list
- Select the Resultant Forces option in the Component Type area.
- Select the M22 option in the Component area.
- Select the Display Contours on Deformed Shape option in the Display Options area
- Click the OK button to generate the moment diagram Note that as you move the cursor over the moment diagram, the values are displayed at the cursor and in the lower left-hand corner of the window.
- Click the **Display** menu > **Show Under-form Shape** command to clear the display of the moment diagram.
- Click the **View** menu > **Set Plan View** command to return to the Plan View.

3.12. Step 11: Design Display

In this Step, design results for the slab and beams will be displayed. Note that the design was run along with the analysis in Step 9. Design results are for the B.S 8110, 1997 code, which was selected in Step 1. Design preferences may be reviewed or changed by going to the Design menu > Design Preferences command (some design preferences are also set on the section property data forms); be sure to re-run the analysis and design (Step 9) if changes to the preferences are made.

- Click the **Display** menu > **Show Slab Design** command to access the Slab Design form
- In the Choose Display Type area, select **Finite Element Based** from the Design Basis drop-down list. This option displays the required reinforcing calculated on an element-by-element basis as intensity contours - integration across the defined design strips is not performed.
- In the Reinforcing Direction and Location area, select the **Direction 2 – Bottom** Rebar option. Direction 2 refers to the object local axis 2 direction.
- In the Show Rebar above Specified Value area, select the **Non** option.
- Click the **OK** button to leave the Slab Design form and display the slab design results for the local axis 2 direction. Again, positioning the cursor anywhere on the slab will result in the display of the reinforcing values at the cursor and in the lower left-hand corner of the **SAFE** window.

3.13. Step 12 Run Detailing

In this Step, detailing will be run and displayed. Detailing may be run only after analysis and design are complete.

- Click the **Detailing** menu > **Detailing Preferences** command to display the Detailing Preferences form. Use this form to set the regional standards, to control how dimensioning is displayed, to manage reinforcing bar notation, and to select the units for material quantity takeoffs.
- Review the settings on this form (we will accept the default selections), and then click the **OK** button to close the form.
- Click the **Detailing** menu > **Footings Preference** command to display the footing preference form.
- Click the **General and Display tab**. On this tab review or alter the rebar curtailment, detailing and callout options, as well as set how sections should be cut. We will accept the default settings.
- Click the **Rebar Selection tab** and review or change the rebar selection rules, preferred sizes, minimums and reinforcing around openings. We will accept the default settings.
- Click the **OK** button to accept the selections and close the form.
- Click the **Detailing** menu > **Drawing Sheet Setup** command to display the Drawing Sheet Setup form. The sheet size, scales, title block and text sizes can be reviewed and changed using this form. We will accept the default settings.
- Click the **OK** button to close the form.
- Review the line thicknesses and styles by clicking the **Detailing** menu > Drawing Format Properties command.
- Click the **OK** button to accept the selections and close the form.
- Now that the detailing preferences and drawing setup options have been reviewed, click the **Run** menu > **Run Detailing** command to generate the detailing drawings. A framing plan is displayed when detailing is complete.
- Click the **Detailing** menu > **Show Detailing** command to access the Display Detailing Item form.
- Select the **Drawing** option.
- Select **Slab Rebar Plan – Bottom Bars** from the Drawing dropdown list.
- Click the **OK** button to leave the Display Detailing Item form and display the selected drawing.
- Clicking on the **Detailing tab** in the Model Explorer and expanding the Views and Drawing Sheets trees also provides access to detailing drawings and component views.
- Click the **Display** menu > **Show Under-formed Shape** command to return to the model.

3.14. Step 13: Create Report

In this Step, a report describing model input and output results will be created.

- Click the **File** menu > **Report Setup** command to display the Report Setup Data form.
- In the Report Output Type area, be sure that the **RTF** File option is selected.
- In the Report Items area, uncheck the **Include Hyperlinked Contents** checkbox.
- Click the **OK** button to leave the Report Setup Data form.
- Click the **File** menu > **Create Report** command to display the Microsoft Word Rich Text File Report form.
- Type **footings (single/double)** in the File name edit box and click the Save button. A report, with a cover written bold SAFE will come up which should be displayed in your word processor, and will be saved to your hard disk, **SAFE**, (2016)

4. Refinement

The user needs to select the number of nodes and elements in the model. The selection may be the one that leads to the best description of the domain geometrically. For example, footing surface of the single and double footings could be modeled by a series of interconnected concrete cube. The larger the number of concrete, the better is the model. The selection may also be based upon intuition, past experience and engineering judgment. The mesh obtained may be

adequate in some cases. In other cases, especially when singularities are present, the mesh may not be adequate to obtain the results to the accuracy desired. In such cases, the meshes need to be refined.

There are three ways of refining a finite element mesh:

- The **H-method**: This method increases the number of elements and hence decreases the element size while keeping the polynomial order of the shape function constant.
- The **P-method**: This method increases the polynomial order of the interpolation function while keeping the number of elements in the model constant.
- The **R-method**: This method redistributes the nodes while keeping the element number and the polynomial order of the interpolation function constant.

But, our concern here is using **H-method** because the object have to be constant i.e. the shape then the element will increase thereby reducing the size of the element to best fit the mesh.

5. Reference Analysis (Safe)

5.1. Overview of Safe

SAFE is an ultimate tools use in the design of concrete floors and foundation system, hence provide a suitable means for the user. From framing layout all the way through to detail drawing production, **SAFE** integrate every aspect of engineering design which are in one process easy and intuitive environment. **SAFE** provides unmatched benefits to the engineer with its truly unique combination of power, comprehensive capabilities, and ease-of-use.

Laying out models is quick and efficient with the sophisticated drawing tools, or use one of the import options to bring in data from CAD, spreadsheet, or database programs. Slabs and foundations can be of any shape, and can include edges shaped with circular and spline curves.

Post-tensioning may be included in both slabs and beams to balance a percentage of the self-weight. Suspended slabs can included flat, two-way, waffle, and ribbed framing systems. Models can have columns, braces, walls and ramps connected from the floor above and below. Wall can be modeled as either straight or curved. Mats and foundations can include nonlinear uplift from the soil springs, and a nonlinear cracked analysis is available for slabs. Generating pattern surface loads is easily done by **SAFE** with an automated option. Design strips can be generated by **SAFE** or drawn in a completely arbitrary manner by the user, with complete control provided for locating and sizing the calculated reinforcement. Finite element design without strips is also available and useful for slabs with complex geometries.

Comprehensive and customizable reports are available for all analysis and design results. Detailed plans, sections, elevations, schedules, and tables may be generated, viewed, and printed from within **SAFE** or exported to CAD packages.

SAFE provides an immensely capable yet easy-to-use program for structural designers, providing the only tool necessary for the modeling, analysis, design, and detailing of concrete slab systems and foundations. (SAFE, 2016).

6. Validation of finite element model (fem) using safe

6.1. Single footing

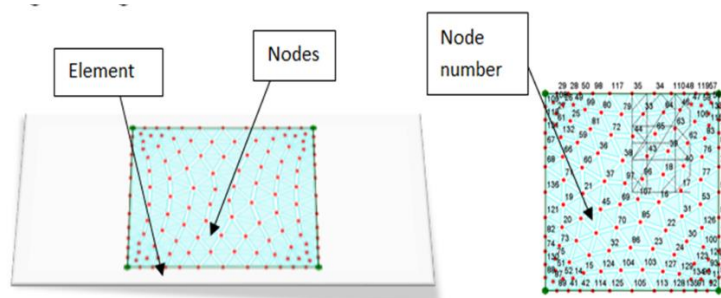


Figure 3 Finite element meshes illustrating the location of the applied load of 80kN

20	0.323634246	57	0.953051656	94	0.575741322		
21	0.3453787	58	0.996698814	95	0.281417769		
22	0.365218318	59	1.040345971	96	0.580760238		
23	0.385057935	60	1.083993129	97	0.670383731		
24	0.404897552	61	1.127640287	98	0.767894092		
25	0.424737169	62	1.199592535	99	0.811750838		
26	0.444576786	63	1.064727102	100	0.856248841		
27	0.464416403	64	0.542222135	101	0.895928076		
28	0.48425602	65	0.30221042	102	0.93560731		
29	0.500941726	66	0.625930478	103	0.975286544		
30	0.517306804	67	0.72638049	104	1.014965778		
31	0.490957497	68	0.888397047	105	1.054645012		
32	0.460664756	69	0.941268859	106	1.094324247		
33	0.222266264	70	0.98888394	107	1.134003481		
34	0.462457226	71	1.036499021	108	1.166365799		
35	0.540250419	72	1.084114102	109	1.198086861		
36	0.625930478	73	1.131729183	110	1.131765475		
37	0.668910906	74	1.179344264	111	1.057557223		

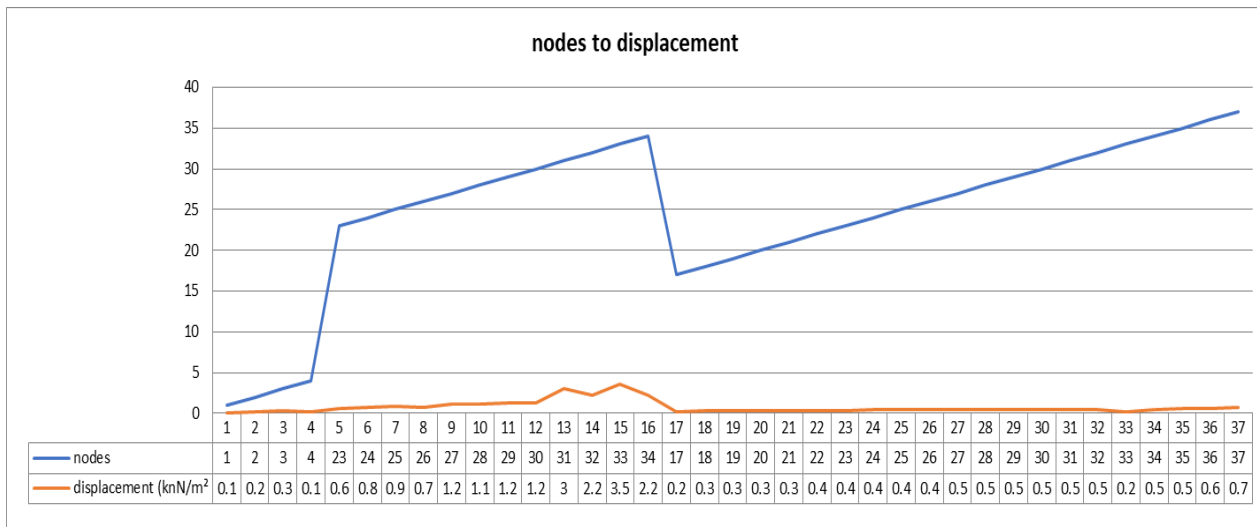


Figure 5 Nodes against displacement for isolated footing

The figure above show the graphical representation of the stated result in table; 1.0, this graph illustrate the displacement with node relationship as the displacement reach it maximum limit with a specified frequency with the node. Showing that at a high node, the displacement increases with increase in the node i.e. for the footing to be at equilibrium it should have a multiple node to enhance a reliable stress on the footing.

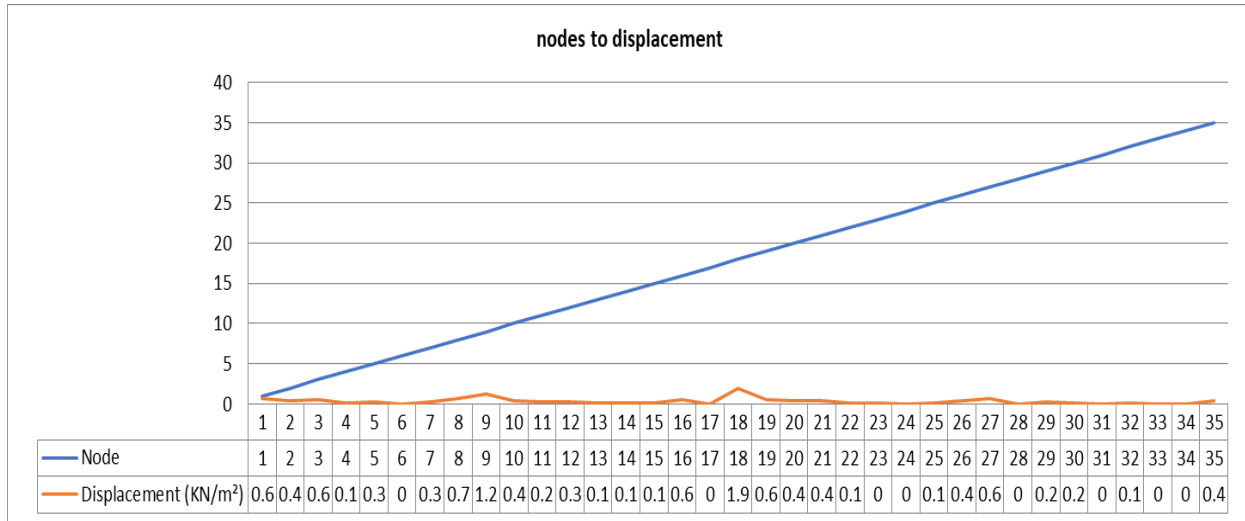


Figure 6 Nodes against displacement for combined footings

The table (2) shows the result of the double footings with the nodal assembly and the displacement at each node as computed from SAFE 2016.

Table; 2 shows the result of the stated Fig. 6 which is the graphical representation of Table 2, here the displacement move at uniform velocity and reaches its destination at the same frequency with the node.

Table 2 Double footings nodal assembly

Assembled Nodal Masses For Double Footings					
Node	Displacement (KN/m ²)	Node	Displacement (KN/m ²)	Node	Displacement (KN/m ²)
1	0.62	36	0.123	69	0.151
2	0.364	43	0.036	70	0.211
3	0.592	44	0.395	71	0.024
4	0.06	45	3.721	72	0.075
5	0.29	46	0.009	73	0.014
6	0.035	47	0.201	74	1.948
7	0.273	48	0.002	74	1.466
8	0.67	46	0.003	75	0.208
9	1.2	48	0.136	76	0.005
10	0.409	49	0.052	77	0.122
11	0.221	37	0.551	78	0.006
12	0.297	38	1.275	79	0.372
13	0.075	39	1.334	80	0.134
14	0.122	40	0.155	81	0.012
15	0.054	41	0.258	82	0.119
16	0.604	42	0.105	83	0.304
17	0.021	50	0.052	84	0.169
18	1.918	51	0.027	85	0.057
19	0.598	52	0.031	86	0.125
20	0.429	53	0.233	87	0.97
21	0.398	54	0.007	88	0.026

22	0.078	55	0.758	89	0.076
23	0.048	56	0.263	90	0.011
24	0.021	57	0.407	91	0.841
25	0.122	58	1.147	92	1.776
26	0.431	59	0.097	93	0.27
27	0.643	60	0.386	94	0.033
28	0.014	61	0.015	95	0.202
29	0.202	62	0.306	96	0.032
30	0.172	63	1.233	97	0.083
31	0.011	64	0.472	98	0.031
32	0.074	65	1.112	99	0.034
33	0.003	66	1.147	100	0.981
34	0.011	67	0.681	101	1.442
35	0.401	68	0.021	102	0.877
103	0.508	117	0.027	110	0.163
104	0.101	106	0.038	111	0.269
105	0.023	107	0.83	112	0.077
115	0.071	108	0.215	113	0.074
116	0.023	109	0.664	114	0.085

7.2. Optimization of the Model

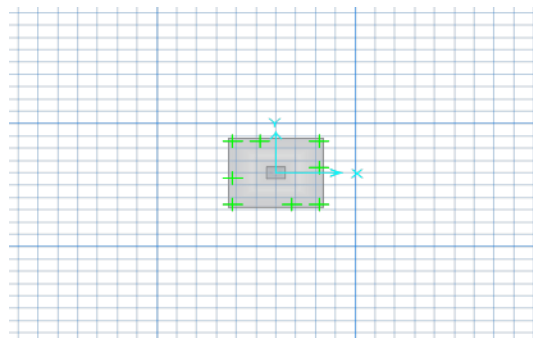


Figure 7 Finite element model for isolated footing

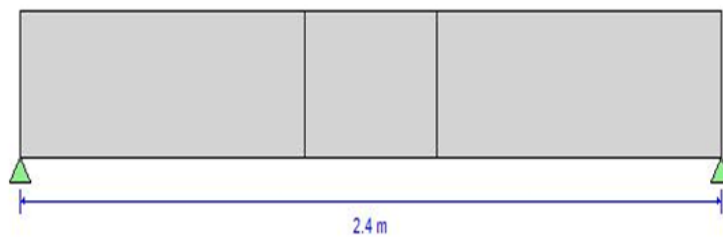


Figure 8 Finite element model for isolated footing

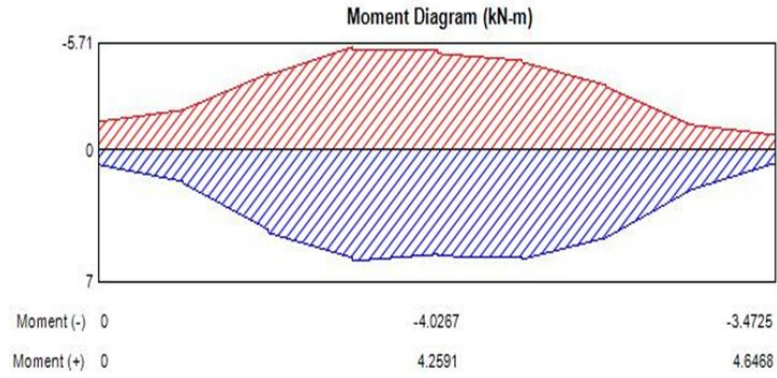


Figure 9 Moment diagram of isolated footing showing the maximum point of stress

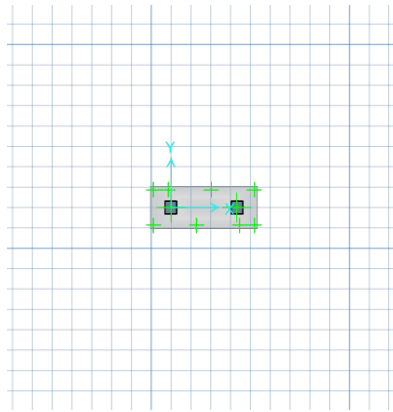


Figure 10 Finite element model for combined footings

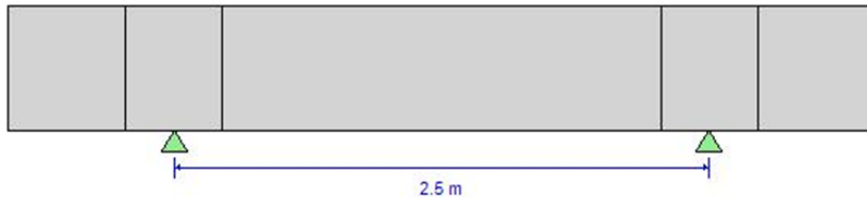


Figure 11 Finite element model for combined footings showing the distance between the columns.

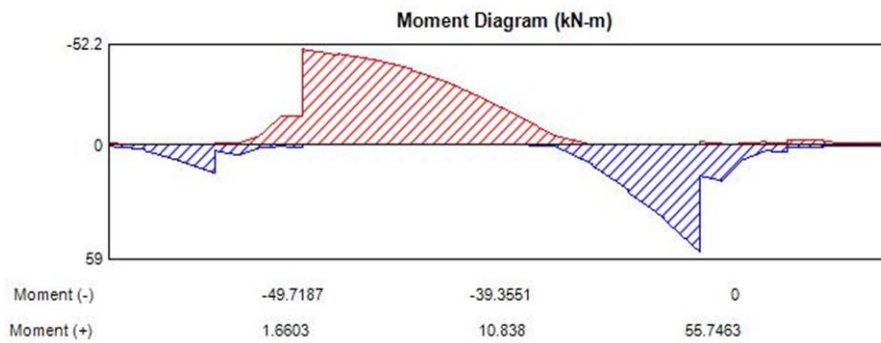


Figure 12 Bending moment diagram for the combined footings

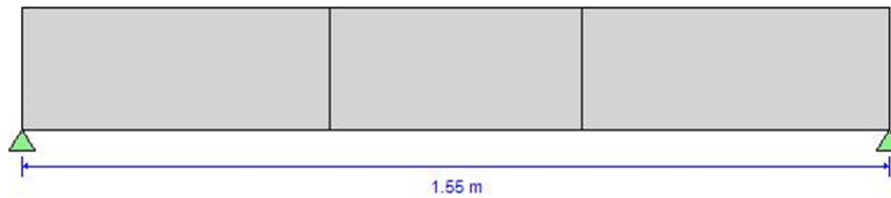


Figure 13 Showing the model at a distance from the column

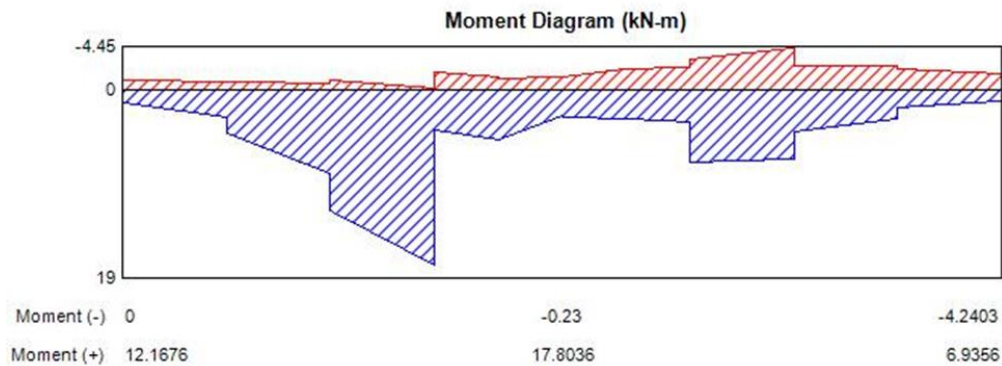


Figure 14 Bending moment diagram of the distance from the column

From the figure stated above in section 4.2 i.e. figure 1.8 – 2.5 it is noted that when the spacing between footings is changed, the maximum moment is always at the footing center and increase with increasing in footing spacing. The reaction of the soil pressure and the applied force on the footing yield equilibrium and state stability in isolated footing.

8. Conclusion

Based on the finite-element analysis of foundations resting on footing embedded in homogeneous linear elastic soil, the following conclusions could be drawn

- The displacement reached its maximum value at the same frequency with a higher node.
- The result indicate that for the footing of foundation considering the meshing spacing ration of (200mm) does not reveal the optimum spacing with regard to displacement.
- When the spacing between footings is changed, the maximum normalize moment factor I_m is always at the footing center and increase with increasing in footing spacing, I_m decrease by about 67% of the footing spacing of 1.675m and increase by about 18% for the 2.5m spacing, This is because the span supported by the footing increases with increasing spacing, leading to greater moments. The increase in I_m with footing length can be attributed to the increase in the unsupported length within the footing, which leads to an increase in the moment.
- The dimensionless displacement factor I_z decreases markedly as the meshing increases, reflecting the increase in displacement with increasing footing spacing.

Recommendations

Based on the analysis of the footing modeling it is deduce that

- For a footing to be suitable and appropriate it is necessary to have a numerous mesh in order to achieve a good analysis.
- For combine footings the spacing should not exceed 2.5m in order to achieve a good stress analysis for the design.

The size of the footing should vary with the magnitude of load and column size that the footing supports.

Compliance with ethical standards

Acknowledgments

All thanks to the Almighty God for the financial grace and wisdom for embarking and completing this research. I sincerely appreciate all authors whose research articles were cited.

Disclosure of conflict of interest

There was no conflict of interest during the research.

References

- [1] Bowles JE. Foundation Analysis and Design, 5th edn. McGraw-Hill, New York, NY, USA. 1977.
- [2] David VH. Fundamental of Finite Element Analysis. New York: McGraw-Hill. 2004.
- [3] Abdel-Fattah M, Saber M, Kantoush SA, Khalil MF, Sumi T, Sefelnasr AM. A Hydrological and Geomorphometric Approach to Understanding the Generation of Wadi Flash Floods. *Water*. 2017; 9(7): 553.
- [4] Farrokhzad F, Choobbasti AJ. Assessing the Load Size Effect in the Soil (Under Single Foundation) Using Finite Element Method. *International Journal of Soil Science*. 2011; 6(3): 209-216.
- [5] Hutton JL, Eccles MP, Grimshaw JM. Ethical issues in implementation research: a discussion of the problems in achieving informed consent. *Implement Sci*. 2008; 3: 52.
- [6] Kharagpur. Version 2 CE IIT Foundations - Theory and Design. 2017.
- [7] Kouzer KM, Kumar J. Ultimate bearing capacity of a footing considering the interference of an existing footing on sand. *Geotech. Geol*. 2010; 28: 457-470.
- [8] Kumar J, Bhattacharya P. Bearing capacity of interfering multiple strip footings by using lower bound finite element limit analysis. *Comput. Geotech*. 2015; 37: 731-736.
- [9] Madan G. Kittur, Ronald L. Huston. (November 1989) Mesh Refinement in Finite Element Analysis by Minimization of the Stiffness Matrix Trace.
- [10] Oyenuga VO. Simplified Reinforced Concrete Design, Asros Limited, Lagos. 2001.
- [11] SAFE. Safe manual, version 16.0.1. CSC, IP, Inc. 2016.
- [12] Satish KSR, Santhian KAR. Design of Steel Structure. 2010.
- [13] Fattah A, M Sharafi M, Masoudi R, Shahverdi A, Esmaeili V, Najafil A. Carnitine in rooster semen cryopreservation: flow cytometric, biochemical and motion findings for frozen-thawed sperm. *Cryobiology*. 2017; 74: 148-153.