STAAD.*foundation*

USER'S MANUAL

Research Engineers International
a Bentley Solutions center.

www.Bentley.com

STAAD.*foundation* is a proprietary computer program of Research Engineers, International (REI), a Bentley Solutions center.

RELEASE 4.0

Published January 2009

About STAAD.*foundation*

STAAD.*foundation* is a program from Research Engineers International, for the analysis and design of various types of foundations, such as individual footings, mats, pile caps, combined footing, strip footing, octagonal footing. Plant foundations such as vertical vessel foundation and heat exchanger foundation have been introduced to serve specific needs of plant industry.

Table of Contents

STAAD.*foundation* User's Manual

Section 1 System Requirements, Installation and Start-up 1 - 1

1.1 Introduction 1 - 2
1.2 Hardware Requirements 1 - 4
1.3 Installation 1 - 5
1.5 Running STAAD.*foundation* 1 - 10

Section 2 Theoretical Basis 2 - 1

2.1 Introduction to Finite Element Analysis 2 - 2
2.2 Element Load Specification 2 - 6
2.3 Theoretical Basis 2 - 7
2.4 Element Local Coordinate System 2 - 11
2.5 Output of Element Forces 2 - 12
2.6 Sign Convention of Element Forces 2 - 14
2.7 STAAD.*foundation* Program Theory 2 - 19

Section 3 Quick Tour 3 - 1

3.1 Introduction 3 - 2
3.2 Starting a New Project 3 - 3
3.3 Entering Support Coordinates 3 - 9
3.4 Defining the Loads 3 - 11
3.5 Using Jobs to Specify Design Constraints 3 - 19
3.6 Entering Design Parameters 3 - 23
3.7 Performing an Isolated Footing Design 3 - 24
3.8 Importing Structural Geometry and Analysis Results from STAAD.*Pro* 3 - 29
3.9 Creating a New Job for a Mat Foundation 3 - 33
3.10 Setting Up the Grid and Defining the Mat Boundary 3 - 35
3.11 Creating a Mesh 3 - 38
3.12 Specifying Slab Thickness 3 - 43
3.13 Defining Soil Properties 3 - 44
3.14 Analyzing the Slab 3 - 45
3.15 Slab Design 3 - 56
3.16 Pile Cap Example 3 - 70
3.17 Entering Pile Data 3 - 72
3.18 Entering Pile Cap Design Parameters 3 - 78
3.19 Performing Pile Cap Design and Viewing Results 3 - 80
3.20 Exporting Drawings to CAD 3 - 84
3.21 Creating Strip Footing Job 3 - 88
3.22 Strip Footing design parameters 3 - 91
3.23 Creating Octagonal Footing Job 3 - 95

3.24 Entering Octagonal footing design parameters 3 - 97
3.22 Conclusion 3 - 98

Section 4 STAAD.*foundation* Graphical Environment 4 - 1

4.1 Introduction 4 - 2
4.2 STAAD.*foundation* Screen Organization 4 - 3
4.3 The Navigator Control 4 - 6
4.4 Global Data 4 - 8
4.4.1 The Project Info Page 4 - 10
4.4.1.1 General Info 4 - 11
4.4.1.2 Review History 4 - 13
4.4.2 The Foundation Plan Page 4 - 15
4.4.2.1 Grid Setup 4 - 16
4.4.2.2 Column Positions 4 - 28
4.4.2.3 Column Dimensions 4 - 29
4.4.3 The Loads and Factors Page 4 - 30
4.4.3.1 Create New Load Case 4 - 31
4.4.3.2 Add a Column Reaction Load 4 - 33
4.4.3.3 Add a Point Load (for Mat only) 4 - 34
4.4.3.4 Add a Quadrilateral Load (for Mat only) 4 - 35
4.4.3.5 Add a Circular Pressure Load (for Mat only) 4 - 38
4.4.3.6 Add a Line Load (for Mat only) 4 - 40
4.4.3.7 Add a Uniform Load (member load) 4 - 42
4.4.3.8 Add a Concentrated Load (member load) 4 - 43
4.4.3.9 Add a Trapezoidal Load (member load) 4 - 44
4.4.3.10 Load Assignment Methods 4 - 45
4.4.3.11 Load Combination 4 - 47
4.4.3.12 Remove Load Case 4 - 50
4.4.3.13 Safety Factor 4 - 51
4.5 Job Setup 4 - 52
4.5.1 Create a New Job 4 - 53
4.5.2 Edit Current Job 4 - 57
4.6 Local Data 4 - 58
4.6.1 Isolated Footing 4 - 59
4.6.1.1 Concrete and Rebar 4 - 60
4.6.1.2 Cover and Soil 4 - 62
4.6.1.3 Footing Geometry 4 - 65
4.6.1.3 Design 4 - 68
4.6.2 Pile Cap 4 - 71
4.6.2.1 Pile Layout (Predefined) 4 - 72
4.6.2.2 Pile Layout (Parametric) 4 - 79
4.6.2.3 Design Parameters 4 - 84
4.6.2.4 Design 4 - 87
4.6.2.4.1 Layout Drawing 4 - 90
4.6.2.4.2 Detail Drawing 4 - 91
4.6.3 Mat Foundation 4 - 92
4.6.3.1 Default analysis properties 4 - 94
4.6.3.2 Physical Beam Table 4 - 97

4.6.3.3 Pile Layout 4 - 98
4.6.3.3.1 Pile Position Table 4 - 99
4.6.3.3.2 Recangular Pile Arrangement Wizard (Parametric) 4 - 100
4.6.3.3.3 Circular Pile Arrangement Wizard (Parametric) 4 - 102
4.6.3.4 Mesh generation 4 - 104
4.6.3.4.1 Adding Mesh Region 4 - 105
4.6.3.4.1.1 Using Polyline 4 - 106
4.6.3.4.1.2 Add a reectangular region 4 - 107
4.6.3.4.1.3 Add a circular region 4 - 109
4.6.3.4.1.4 Regular Polygon 4 - 111
4.6.3.4.2 Meshing Setup 4 - 113
4.6.4 Analysis properties 4 - 118
4.6.5 Mat slab design dption 4 - 124
4.6.5.1 Analyze 4 - 125
4.6.5.2 Displacement 4 - 128
4.6.5.3 Disp(Displacement) summery 4 - 131
4.6.5.4 (Support)Reaction(s) 4 - 132
4.6.5.5 (Support)Reaction Summery 4 - 133
4.6.5.6 Contact Area 4 - 134
4.6.5.7 Plate Stresses 4 - 135
4.6.5.8 Plate Stresses Summery 4 - 136
4.6.5.9 Pile Reaction 4 - 137
4.6.5.10 Output View Options 4 - 138
4.6.6 Slab Design 4 - 144
4.6.6.1 Moment Envelope Generation 4 - 145
4.6.6.2 Design Parameters 4 – 149
4.6.6.3 Reinforcing zones 4 - 152
4.6.6.4 Section Design Along a Line 4 - 157
4.6.6.5 Calculation Sheet 4 - 162
4.7 Combined Footing 4 - 163
4.7.1 Creating Combined footing Job 4 - 164
4.7.2 Creating the Combined Footing 4 - 166
4.7.3 Defining the Design Parameters 4 - 169
4.7.3.1 Concrete and Rebar 4 - 170
4.7.3.2 Cover and Soil 4 - 172
4.7.3.3 Footing Geometry 4 - 174
4.7.3.4 Design 4 - 178
4.8 Octagonal Footing 4 - 180
4.8.1 Creating Octagonal footing Job 4 - 181
4.8.2 Defining the Design Parameters 4 - 183
4.8.2.1 Design Parameters 4 - 184
4.8.2.2 Footing Geometry 4 - 187
4.8.2.3 Design 4 - 189
4.9 The Menu Commands 4 - 188
4.9.1 File Menu 4 - 189
4.9.2 Edit Menu 4 - 199
4.9.3 View Menu 4 - 200
4.9.4 Tools Menu 4 - 203
4.10 The Toolbars 4 - 209

4.10.1 File Toolbar 4 - 210
4.10.1.1 File Toolbar 4 - 214
4.10.1.2 Print Toolbar 4 - 217
4.10.1.3 Import Toolbar 4 - 223
4.10.1.4 Save Picture Toolbar 4 - 226
4.10.1.5 Change Job Toolbar 4 - 227
4.10.1.6 Change Current Load Case Toolbar 4 - 228
4.10.1.7 Tools Toolbar 4 - 229
4.10.1.8 Loading Toolbar 4 - 235
4.10.1.9 View option Toolbar 4 - 240
4.10.1.10 Scale Setup Toolbar 4 - 243
4.10.1.11 Unit Setup Toolbar 4 - 245
4.10.2 Help Toolbar 4 - 247
4.10.3 Rotate Toolbar 4 - 249
4.10.4 Zoom Toolbar 4 - 253
4.10.5 Select Toolbar 4 - 255

Section 5 Plant Foundation 5 - 1

5.1 Introduction 5 - 2
5.1.1 Creating a New Plant Setup Job 5 - 3
5.2 Vertical Vessel Foundation 5 - 5
5.2.1 Geometry Page 5 - 6
5.2.2 Primary Load Page 5 - 8
5.2.3 Time Period Page 5 - 9
5.2.4 Wind Load Generation Page 5 - 12
5.2.5 Seismic Load Generation Page 5 - 15
5.2.6 Load Combination Page 5 - 19
5.2.7 Design Parameter Page 5 - 21
5.2.8 Foundation Type Page 5 - 23
5.2.9 Finish and Design 5 - 24
5.3 Heat Exchanger Foundation 5 - 25
5.3.1 Exchanger Geometry Page 5 - 26
5.3.2 Footing Geometry Page 5 - 29
5.3.3 Primary Load Page 5 - 32
5.3.4 Wind Load Generation Page 5 - 34
5.3.5 Seismic Load Generation Page 5 - 38
5.3.6 Load Combination Page 5 - 42
5.3.7 Design Parameter Page 5 - 44
5.3.8 Finish and Design 5 - 47

Section 6 Indian Verification Problems 6 - 1

6.1 Indian Verification Problem 1 6 - 2
6.2 Indian Verification Problem 2 6 - 7
6.3 Indian Verification Problem 3 6 - 12
6.4 Indian Verification Problem 4 6 - 16
6.5 Indian Verification Problem 5 6 - 21

System Requirements, Installation and Start-up

Section 1

This section includes discussion on the following topics:

- Introduction
- Hardware Requirements
- Installation
- Running STAAD.*foundation*

1.1 Introduction

Thank you for your purchase of STAAD.*foundation*. STAAD.*foundation* is an exhaustive analysis, design, and drafting solution for a variety of foundations that include general foundation types such as isolated, combined footings, mat foundations, pile caps and slab on grade and plant foundation such as vertical vessel foundation and heat exchanger foundation. A part of the STAAD.*Pro* family of products, STAAD.*foundation* is a cost-saving downstream application that enables engineers to analyze and design a foundation. STAAD.*foundation* can automatically absorb the geometry, loads and reactions from a STAAD.*Pro* model and accurately design isolated, pile cap, strip footing, true mat foundations and even perform pile arrangements for a pile cap.

STAAD.*foundation* not only analyzes and designs a myriad of foundation configurations, but will also produce production quality reports and detailed 3D rendering of your foundation structures. With full OpenGL graphics, engineers can clearly see the displaced shape, stress distribution, reinforcement layout and force diagrams of their supporting structure. All models use physical objects including physical beams, physical slabs, automatic meshing, load distributions, and support generation. STAAD.*foundation* designs the physical slabs rather than individual elements.

For mat foundation designs, STAAD.*foundation* utilizes a true finite element design using the individual element stresses rather than using column strips. STAAD.*foundation* can be used in a stand-alone mode or can be used in conjunction with STAAD.*Pro* where the support reactions from the main model and associated load cases are automatically brought in.

Because STAAD.*foundation* provides a total solution for your foundation needs, a built-in project management system enables line and span of control, revision records and multi-job functionalities. This helps you reduce cost in assembling the technical and managerial information for your foundation. Full step-by-step calculations are also provided in XML form (where possible) to verify each and every output provided by the program. These verification checks can be easily shared with your clients for approval.

We hope you enjoy your experience with STAAD.*foundation*. If you have any questions or problems with the program, please visit our product page at http://www.bentley.com/Staad.foundation or email us at support@bentley.com.

1.2 Hardware Requirements

The following requirements are suggested minimums. Systems with increased capacity provide enhanced performance.

- PC with Intel-Pentium / AMD processor.
- Graphics card and monitor with 1024x768 resolution, 256 color display (16-bit high color recommended).
- 128 MB RAM or higher.
- Windows 98/ NT 4.0 or higher operating system. Windows 2000/XP Preferred. Running it on Windows 95 systems is not recommended, as performance may be degraded.
- Sufficient free space on the hard disk to hold the program and data files. The disk space requirement will vary depending on the modules you are installing. A typical minimum is 500MB free space.

Note: Additional RAM, disk space, and video memory will enhance the performance of STAAD.*foundation*. The user must have a basic familiarity with Microsoft Windows systems in order to use the software.

1.3 Installation

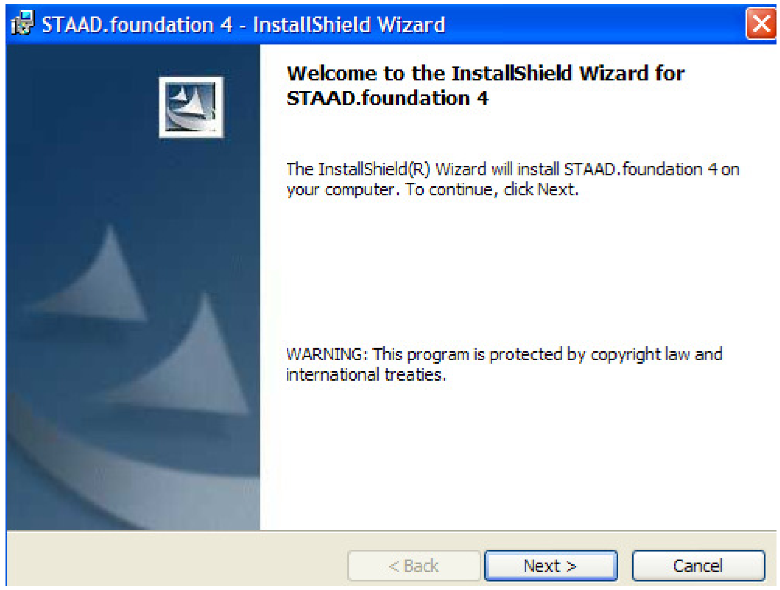
To install STAAD.foundation 4.0, ensure you have logged in your machine with an account that has administrative privileges. If you are unable to log in with a suitable account, then contact your network administrator to login and perform the installation.

It is to be noted that, before installing STAAD.foundation 4.0, you must install "Bentley IEG License Service, Version 2.0.7" using the MSI package "BentleyIEGLicenseService.2.0.7.msi". This MSI package is available at the Bentley SELECT download site as the pre-requisite for STAAD.foundation and can be downloaded from the same location as STAAD.foundation. If you can locate any updated version (later than 2.0.7) of this component, you may use that package instead of "BentleyIEGLicenseService.2.0.7.msi".

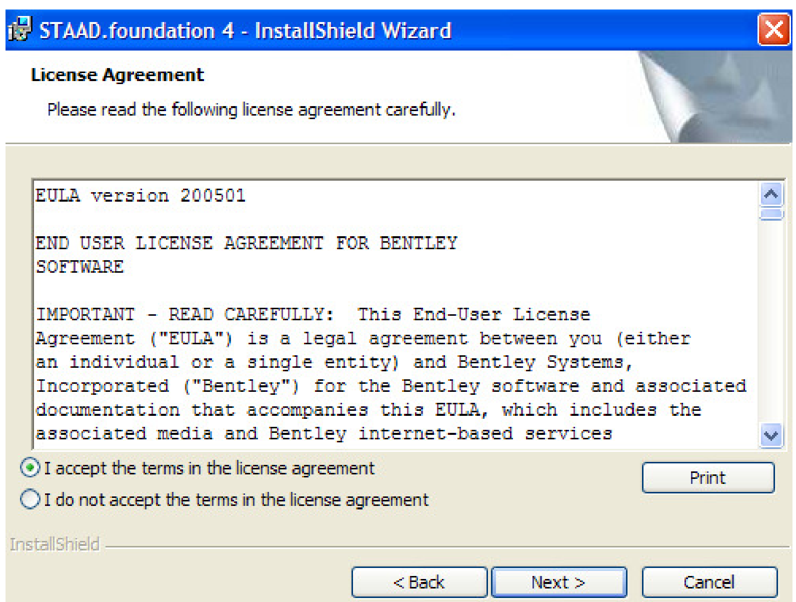
Locate STAAD.foundation 4.0 installation image on local or network drive and double click on the installation startup MSI package (***STAAD.foundation 4.msi***) or double click on the installation startup program (***Setup.exe***) available within the **Install** subfolder of the installation image.

While installing STAAD.foundation, please follow all of the installation interaction dialogs and enter necessary information. Following dialogs will appear in sequence.

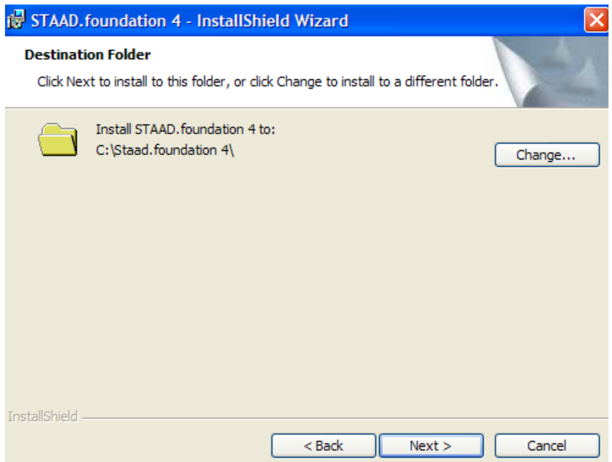
Follow the instructions on the subsequent dialog boxes. The following steps are for assistance on the more significant dialogs. Those that are not illustrated here are self explanatory.



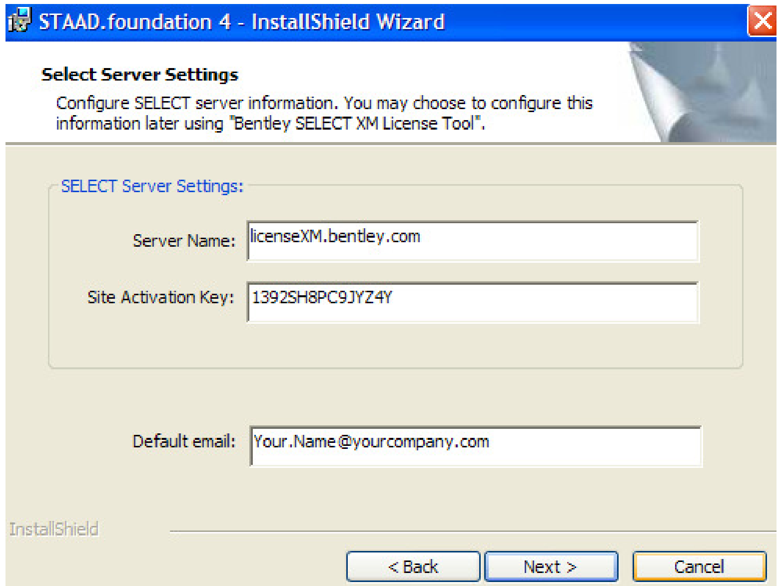
The first screen is a welcome screen. Click on Next button to continue installation.



Next screen is the license agreement. By default the option is set to “I do not accept the terms in the license agreement” and Next button is grayed. Select “I accept the terms in the license agreement” option to continue with the installation.

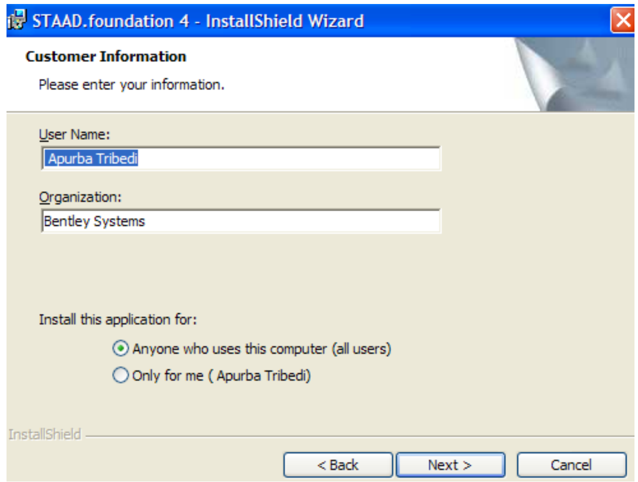


Next screen allows user to choose destination folder. By default the destination folder is set as “C:\Staad.foundation 4”. Click on “*Change”* button to change the folder location.

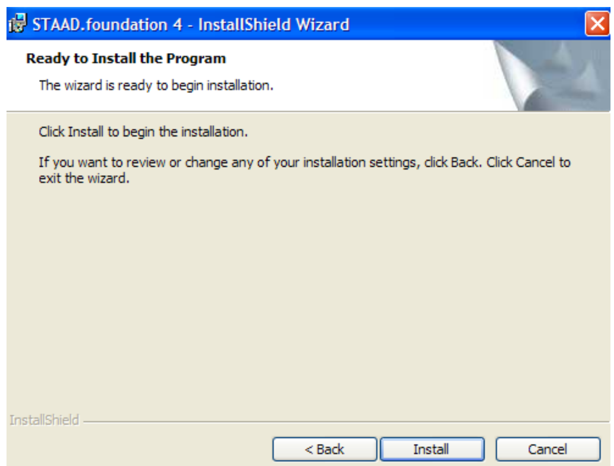


When asked for SELECT Server name and site activation key, please enter the information if you have those. For Standalone workstation set server name to **LocalHost** and activation key to **1**. For Bentley hosted or deployed (local) SELECT installations discussed later in this guide, you will need to use the proper server name and activation key. In case of Bentley hosted server both server name and activation key is provided by Bentley. For deployed (local) SELECT server installations activation key is provided by Bentley and server name is the name of your local SELECT server. You may also choose to configure these information later.

A trial license is installed with software, which allows you to run STAAD.foundation for a period of up to 15 days. In case you did not enter the server name and activation key during installation, you must configure the server information using the Bentley SELECT XM License Tool within 15 days. The process is described under the heading " **b) Adding the Bentley SELECT Server activation code**" of this document



Next screen allows user to input User Name and Organization and the option to choose whether program will be installed for current user or all users.

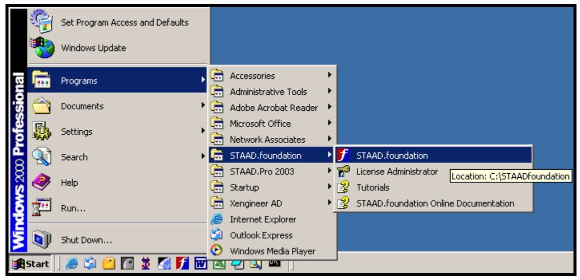


Clicking on "Install" button in next string will start installing the program.

After the installation is complete, please restart your machine for any changes made to take effect.

1.4 Running STAAD.*foundation*

Click on the STAAD.*foundation* icon from the STAAD.*foundation* program group as shown below.



The STAAD.*foundation* screen appears as shown below.

If you're a first time user unfamiliar with STAAD.*foundation*, we suggest that you go through the Quick Tour presented in Section 3 of this manual.

N
o
t
e
s

Theoretical Basis

Section 2

This section includes discussion on the following topics:

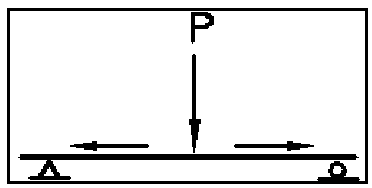
- Introduction to Finite Element Analysis
- Element Load Specification
- Theoretical Basis
- Element Local Coordinate System
- Output of Element Forces
- Sign Convention of Element Forces
- STAAD.*foundation* Program Theory

2.1 Introduction to Finite Element Analysis

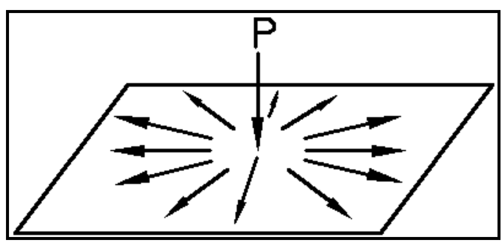
If you want to model a surface entity like a wall, a roof or a slab, where the load is distributed in more than one direction, you need a surface entity to carry that kind of loading. The kind of entity that is used to model a beam or a column cannot be used to model a slab. We need to use another kind of structural entity known as a *finite element*. In a finite element analysis, you take a wall or a slab and subdivide it into smaller parts consisting of triangles or quadrilaterals.

Finite elements are often referred to as plates. In our discussion, we may use these two words interchangeably.

The difference between a beam and a plate is a load that is applied to a beam can only go in two directions: towards one end, or the other, or both.

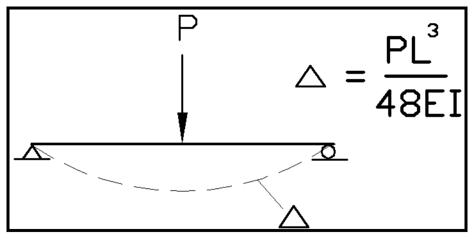


In a plate, there is more than one path for the load to flow.



A plate can be 3-noded (triangular) or 4-noded (quadrilateral). The thickness of an element may be different from one node to another. All nodes of a 4-noded plate must lie in the same plane. If the four nodes of a quadrilateral element do not lie on one plane, you should replace the quadrilateral element with two triangular elements.

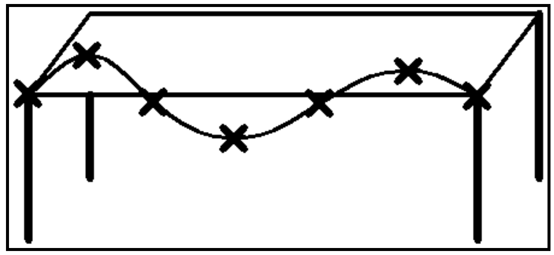
It is not possible to accurately model the behavior of a slab using just a single element. Why not? One reason is you can determine the displacements in the finite element only at the corner nodes. With a beam, if you know the displacements at the ends, you can use secondary analysis techniques like the moment-area method to determine the displacements at intermediate points.



In a plate, there are no equations you can use to determine the displacement at some arbitrary point within the 3 or 4 corners of the element. Therefore, if you would like to know the displacements at some interior points of the slab, or if you would like to know the deformed shape along the edges of the slab, it is necessary to model the slab using a series of plate elements in such a manner that the points of interest become nodes of the elements.

Similarly, you can accurately determine the stresses only at the center of the element. The only way to find the stresses at other points is to interpolate values at points between the centers of adjacent elements.

Suppose you had a slab supported by a frame, and under load it had a deflected shape something like that shown in the figure below.



In order to obtain deflection information that would allow you to plot the deflected shape, you would need to at least know the deflections at the points of maximum deflection, at the end points, and at a few intermediate points, as shown by the X's in the figure. The more points you have, the more accurately you can model the deflected shape. On the other hand, you would not want hundreds of points either, since it would make your structure too cumbersome to analyze. You need to exercise judgment in selecting the number of elements you use to model a slab, enough to accurately model the behavior of the slab under load, but not so many as to make the model difficult to work with.

Another situation in which you would need more than one plate element to model a slab would be when you want to know the stresses in a slab caused by some type of point loading. You would want to have quite a few elements in the vicinity of where the point loading occurs in order to determine the stress distribution in the slab caused by the concentrated load.

As a result, rather than using just a single element or a few elements, a series or matrix of finite elements is often needed to model the behavior of a wall or slab. This series of elements is commonly referred to as a *mesh*. Once you have created a mesh, incorporated it into a model, and used it as a basis for further developing the model, it can be difficult to go back later and change the size (i.e. the ‘density’) of the mesh. Here are some suggestions that may help you determine the mesh size that you need.

- Try to predict the approximate deflected shape of the plate or slab. For example, a simply supported plate deflects like a bowl. If you cut a section that intersects the middle of its edges, the longitudinal section as well as the transverse section both look like a "U". How many points does one need to represent the U? Probably four points for each half of the "U" would be a minimum number needed to be able to visualize the deflected shape. Four points would mean there are three elements on each half of the "U', thus six elements each in the local X and Y directions would be required. If the edges of the element are fixed or monolithic with a concrete beam, the deflected shape is more like an inverted hat. In this case, one would perhaps need nine or more points to represent the deflected shape. That means eight or more elements in that direction.

- Do you have concentrated forces on the surface of the element? If so, you need to have a finer mesh around that region in order to visualize the deflected shape or the stresses at that location. How many elements are needed is hard to say. But, for example, one can estimate a circular area around the concentrated load point, divide that circle into say 30 degree pie-shaped segments, thus obtaining 12 triangular elements around a circle whose center is the location of the point load.

- Do you have holes in the plate? You need a finer mesh around the holes. Again, there is no easy guideline for how many elements there should be. Your engineering judgment is often the best guideline.

2.2 Element Load Specification

The following load specifications are available:

1) Joint loads at element nodes in global directions.

2) Concentrated loads at any user specified point within the element in global or local directions.

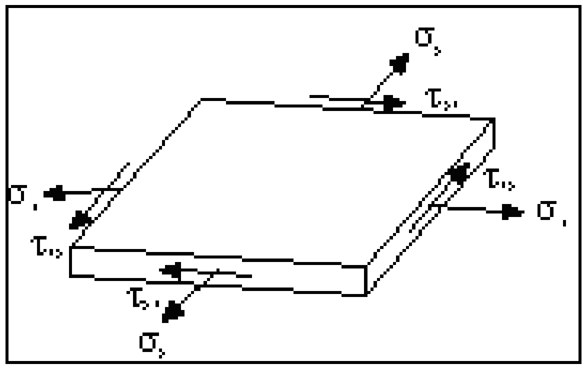
3) Uniform pressure on an element surface in global or local directions.

4) Partial uniform pressure on a user specified portion of an element surface in global or local directions.

5) Linearly varying pressure on an element surface in local directions.

2.3 Theoretical Basis

The STAAD plate finite element is based on hybrid finite element formulations. A complete quadratic stress distribution is assumed. For plane stress action, the assumed stress distribution is as follows.

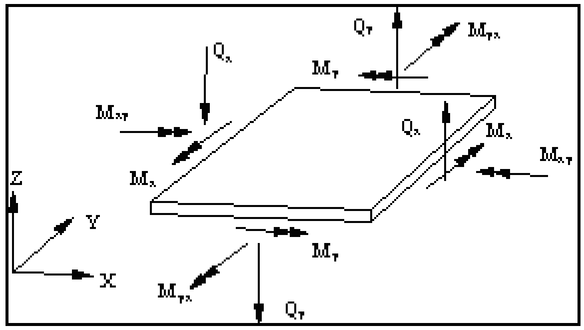


Complete quadratic assumed stress distribution:

$$\begin{pmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{pmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 & 0 & x^2 & 2xy & 0 \\ 0 & 0 & 0 & 1 & x & y & 0 & y^2 & 0 & 2xy \\ 0 & -y & 0 & 0 & 0 & -x & 1 & -2xy & -y^2 & -x^2 \end{bmatrix} \begin{pmatrix} a_1 \\ a_2 \\ a_3 \\ \vdots \\ a_{10} \end{pmatrix}$$

a_1 through a_{10} = constants of stress polynomials.

The following quadratic stress distribution is assumed for plate bending action:



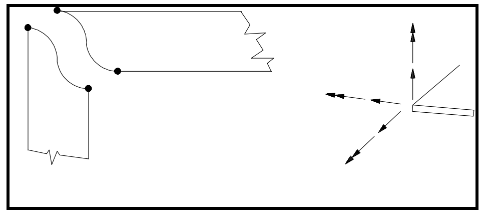
Complete quadratic assumed stress distribution:

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \\ Q_x \\ Q_y \end{Bmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 & 0 & 0 & 0 & x^2 & xy & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y & 0 & 0 & 0 & 0 & 0 & xy & y^2 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & -xy & 0 & 0 & -xy \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & 0 & -x \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 & -y & 0 & x & y \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \\ \vdots \\ \vdots \\ a_{13} \end{Bmatrix}$$

a_1 through a_{13} = constants of stress polynomials.

The distinguishing features of this finite element are:

1) Displacement compatibility between the plane stress component of one element and the plate bending component of an adjacent element which is at an angle to the first (see Figure below) is achieved by the elements. This compatibility requirement is usually ignored in most flat shell/plate elements.



2) The out of plane rotational stiffness from the plane stress portion of each element is usefully incorporated and not treated as a dummy as is usually done in most commonly available commercial software.

3) Despite the incorporation of the rotational stiffness mentioned previously, the elements satisfy the patch test absolutely.

4) These elements are available as triangles and quadrilaterals, with corner nodes only, with each node having six degrees of freedom.

5) These elements are the simplest forms of flat shell/plate elements possible with corner nodes only and six degrees of freedom per node. Yet solutions to sample problems converge rapidly to accurate answers even with a large mesh size.

6) These elements may be connected to plane/space frame members with full displacement compatibility. No additional restraints/releases are required.

7) Out of plane shear strain energy is incorporated in the formulation of the plate-bending component. As a result, the elements respond to Poisson boundary conditions that are considered to be more accurate than the customary Kirchoff boundary conditions.

8) The plate-bending portion can handle thick and thin plates, thus extending the usefulness of the plate elements into a multiplicity of problems. In addition, the thickness of the plate is taken into consideration in calculating the out of plane shear.

9) The plane stress triangle behaves almost on par with the well-known linear stress triangle. The triangles of most similar flat shell elements incorporate the constant stress triangle that has very slow rates of convergence. Thus the triangular shell element is very useful in problems with double curvature where the quadrilateral element may not be suitable.

10) Stress retrieval at nodes and at any point within the element.

2.4 Element Local Coordinate System

The precise orientation of local coordinates is determined as follows:

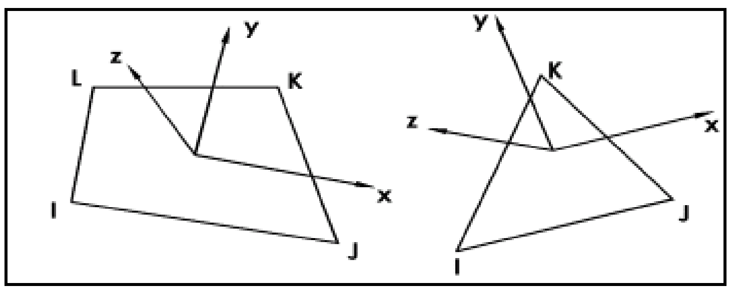
1) The vector pointing from I to J is defined to be parallel to the local X-axis.

2) The cross product of vectors IJ and IK defines a vector parallel to the local Z-axis, i.e., z = IJ x IK.

3) The cross product of vectors z and x defines a vector parallel to the local Y-axis, i.e., y = z x x.

4) The origin of the axes is at the center (average) of the 4 joint locations (3 joint locations for a triangle).

The sign convention of output force and moment resultants is illustrated in Section 2.6.



2.5 Output of Element Forces

ELEMENT FORCE outputs are available at the following locations:

A. Center point of the element.
B. All corner nodes of the element.
C. At any user specified point within the element.

The following is a list of the items included in the ELEMENT STRESS output:

SQX, SQY	Shear stresses (Force/ unit len./thk.)
SX, SY, SXY	Membrane stresses (Force/unit len./thk)
MX, MY, MXY	Bending moments per unit width (Moment/unit len.)
SMAX, SMIN	Principal stresses (Force/unit area)
TMAX	Maximum shear stress (Force/unit area)
ANGLE	Orientation of the principal plane (Degrees)
VONT, VONB	Von Mises stress, where

$$VM = 0.707\sqrt{(SMAX - SMIN)^2 + SMAX^2 + SMIN^2}$$

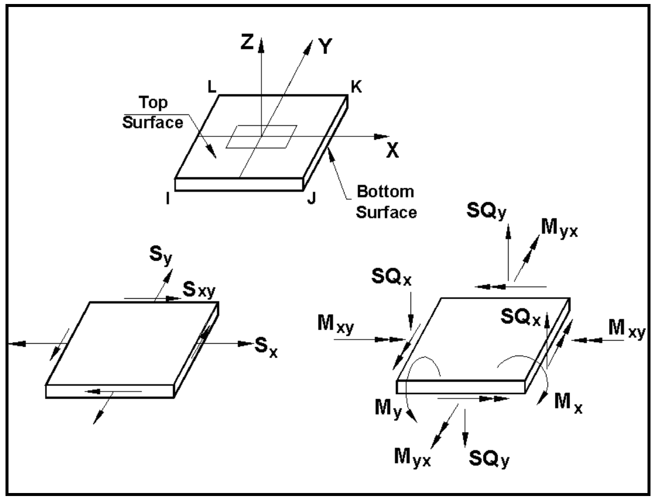
TRESCAT, TRESCAB Tresca stress, where

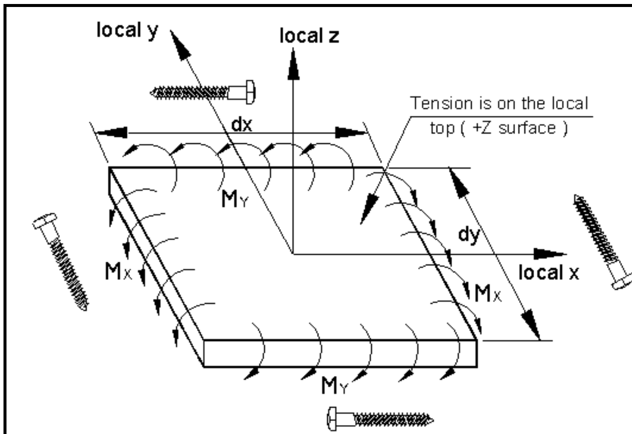
TRESCA = MAX[|(SMAX-SMIN)| , |(SMAX)| , |(SMIN)|]

Note:
1. **All element stress output is in the local coordinate system. The direction and sense of the element stresses are explained in Section 2.6.**
2. **To obtain element stresses at a specified point within the element, the user must provide the coordinate system for the element. Note that the origin of the local coordinate system coincides with the center node of the element.**
3. **Principal stresses (SMAX & SMIN), the maximum shear stress (TMAX), the orientation of the principal plane**

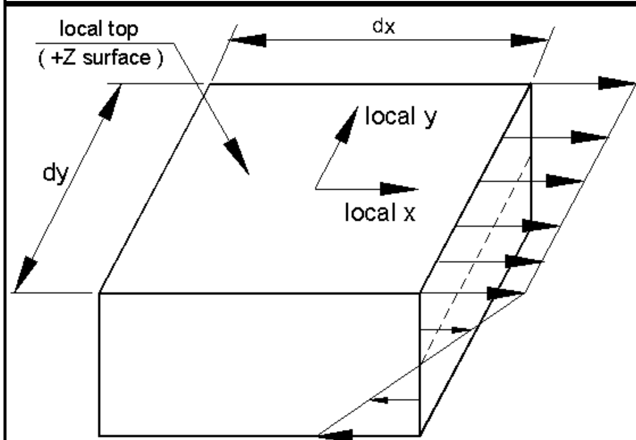
(ANGLE), the Von Mises stress (VONT & VONB), and the Tresca stress (TRESCAT & TRESCAB) are also printed for the top and bottom surfaces of the elements. The top and the bottom surfaces are determined on the basis of the direction of the local Z-axis.

2.6 Sign Convention of Element Forces

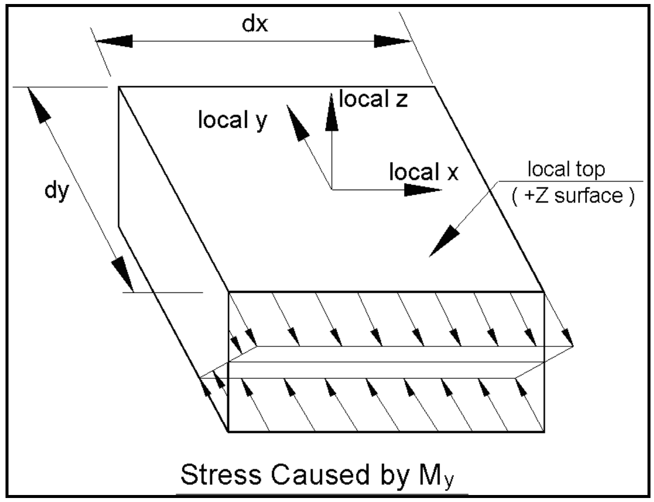


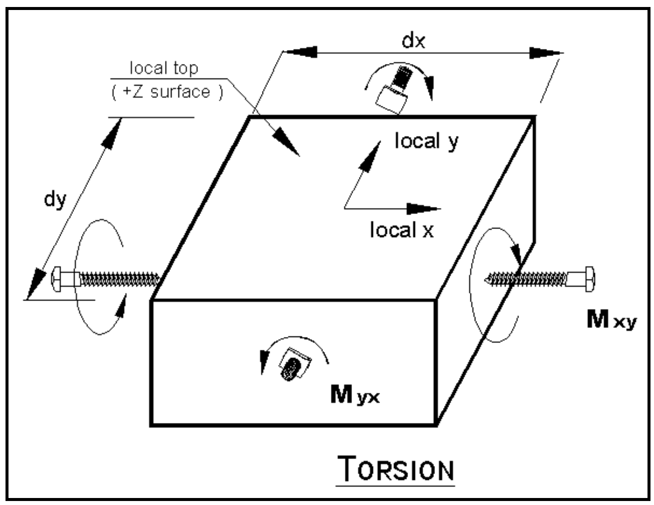


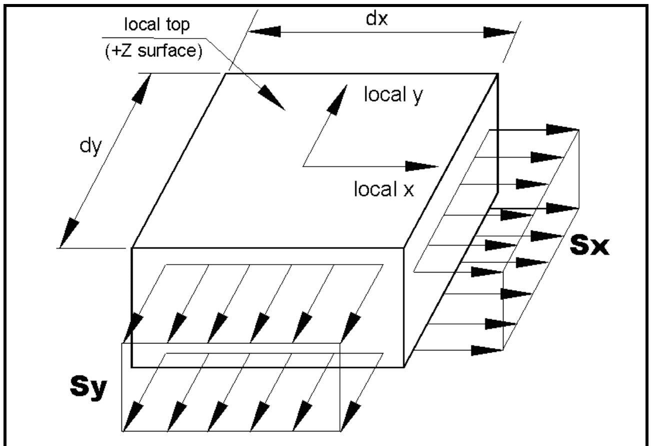
M_x is the Bending Moment on the local x face, the local x-face is the face perpendicular to the local x-axis;
M_y is the Bending Moment on the local y face, the local y-face is the face perpendicular to the local y-axis



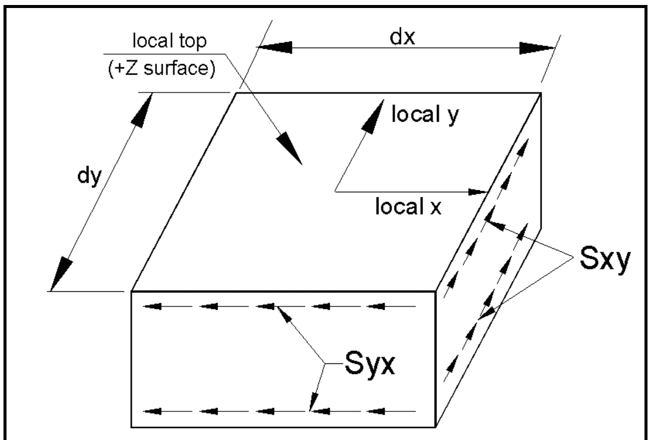
STRESS CAUSED BY M_X


dx
dy
local y
local z
local x
local top
(+Z surface)
Stress Caused by My

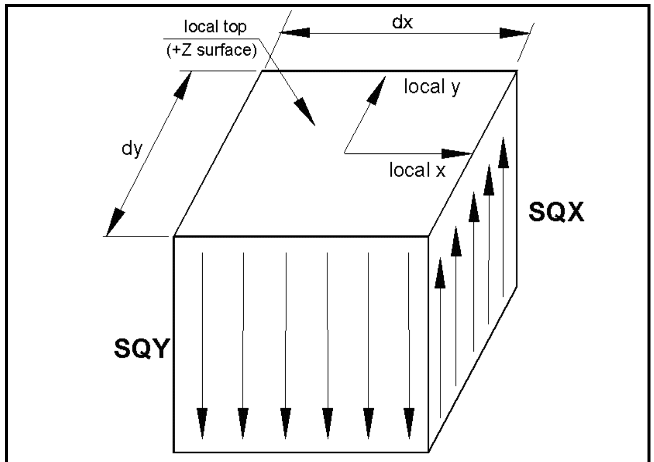

dx
dy
local top
(+Z surface)
local y
local x
Mxy
Myx
TORSION



Membrane Stresses Sx and Sy



In plane shear stresses Sxy and Syx



Out-of plane shear stresses (SQX and SQY)

2.7 STAAD.*foundation* Program Theory

STAAD.*foundation* performs structural design of foundations in accordance with the ACI 318-05 Code. The available foundation types are: isolated spread footing, pile cap, strip footing, mat foundation and octagonal footing.

1. Isolated Spread Footing

The program uses the following criteria:

a. Soil bearing capacity,
b. Shear and flexural strength of footing (no shear reinforcing assumed),
c. Compressive and flexural strength of pedestal

Step 1 - Determine footing plan geometry based on loading and bearing resistance of the soil.

Stress distribution under the footing is assumed to be linear. For eccentrically loaded footings, the stresses may become tensile under part of the foundation. In such cases the program sets stress values in uplift zones to zero and calculates new values elsewhere for the revised equilibrium condition. The final plan dimensions of the footing are established iteratively from the condition that the maximum stress should not exceed the factored bearing resistance of the soil.

Step 2 - Calculate footing thickness based on structural capacity in shear and bending.

Structural design of the footing consists of the following:

a. Punching shear check, in accordance with Section 11.12.2, at a distance of **d**/2 from the pedestal. The critical section comprises four straight-line segments, parallel to the corresponding sides of the pedestal.

b. One-way shear (beam action), in accordance with Sections 11.1 through 11.5, at a distance of **d** from the face of the pedestal, in both orthogonal directions. The critical plane is assumed to extend over the entire width/length of the footing.
c. Bending, in accordance with Sections 15.4.2 and 10.3.4, with the critical planes located at both orthogonal faces of the pedestal and extending across the full width/length of the footing.

2. Pile Cap

The program produces the following design output:

a. Required pile quantity and layout to satisfy loading applied to the footing, based on bearing, uplift and lateral pile capacity,
b. Geometry of the pile cap based on shear and bending strength requirements at critical sections of the footing.

Step 1 - Pile Arrangement

The user provides the following pile properties: capacity (bearing, uplift, and lateral), diameter, spacing, and edge distance. Based on these parameters, the program determines the required pile configuration as well as plan dimensions of the footing from the condition, that the force transferred to any pile should not exceed its capacity. For a general case of vertical and horizontal forces, and bending moments acting on the cap, that stipulation is equivalent to satisfying the following two equations:

$$\mathbf{H}_{pile} >= \mathbf{H}_{appl} / \mathbf{N}$$
$$\mathbf{V}_{pile} >= \mathbf{V}_{appl} / \mathbf{N} + \mathbf{Mx}_{appl} * \mathbf{Ry} / \mathbf{Ix}_{g} + \mathbf{My}_{appl} * \mathbf{Rx} / \mathbf{Iy}_{g}$$

Where:

$\mathbf{H}_{pile}$ - single pile horizontal capacity
$\mathbf{V}_{pile}$ - single pile vertical capacity
$\mathbf{H}_{appl}$ - total horizontal load applied
$\mathbf{V}_{appl}$ - total vertical load applied
N - total number of piles in footing
$\mathbf{Mx}_{appl}$ - applied bending moment about X-axis
$\mathbf{My}_{appl}$ - applied bending moment about Y-axis
Rx - distance from Y-axis to the farthest pile
Ry - distance from X-axis to the farthest pile
$\mathbf{Ix}_{g}$ - pile group moment of inertia about X-axis
$\mathbf{Iy}_{g}$ - pile group moment of inertia about Y-axis

Note: X and Y-axes above are centroidal axes of the pile group, $\mathbf{Ix}_{g}$ and $\mathbf{Iy}_{g}$ are calculated treating each pile as a unit, and are equal $\Sigma(1*y_i^2)$ and $\Sigma(1*x_i^2)$, respectively.

The program includes a library of possible pile layouts for quantities from 1 to 25 piles. Based on the user input, the program recommends the most economical (least number of piles) layout. The user may select any other layout/quantity if desired, however. In addition, changing the coordinates of individual piles may modify the selected pile layout. Alternatively, the user may input the entire configuration by hand.

The layout recommended by the program is guaranteed to satisfy the load/capacity ratio for all piles. Should the user-modified or manually input layouts result in pile overstressing, the program will flag this deficiency in the design output.

Step 2 - Design of Pile Cap

Proportioning of the pile cap involves satisfying the shear (one and two way) and bending requirements at applicable critical sections, in accordance with Chapter 15 of ACI 318-02.

One way shear is checked in two areas:

i. At outer piles, with the critical section located at a min. distance **d** from the face of a corner pile or faces of a pile group along the edge of the footing,
ii. At the distance **d** from two orthogonal faces of the pedestal.

The critical shear plane is assumed along a shortest straight line connecting free edges of the footing. The design is then performed for the total pile reaction force on one side of the shear plane, in accordance with Sections 11.1 through 11.5.

Two way shear is checked in three areas:

i. At outer piles, with the critical section located at a min. distance **d**/2 from the face of a corner pile or faces of a pile group along the edge of the footing. The critical plane is assumed to be positioned along a straight and curved line, so that the total section length is minimized.
ii. At the distance **d**/2 around the pedestal. The section comprises four straight-line segments, parallel to corresponding sides of the column.
iii. At the distance **d**/2 around a pile.

The design is performed for the total pile reaction force acting within the perimeter of the critical section, in accordance with Sections 11.12.2 through 11.12.6.

Flexure is checked for critical planes located at both faces of the pedestal. The bending moment is calculated as an aggregate of moments due to pile reactions on one side of the plane.

Determination of an individual pile contribution to the forces at a critical section is based on whether the pile is outside this section (full reaction value assumed), inside the section (reaction ignored), or at an intermediate location (partial reaction assumed), as per Section 15.5.4.

3. Mat (Raft Foundation)

Analysis and design of mats is based on finite element method (FEM) coupled with slab-on-elastic-subgrade principles. First, the user creates a finite element model of the proposed mat foundation. This may be accomplished in one of two ways:

Importing a STAAD file of the superstructure, thus providing reference points for initial mat set-up and load information, and defining boundaries of the mat, or
Creating the foundation slab from scratch and inputting loading information manually.

Modeling of the foundation involves meshing of the slab to generate grid of finite elements. As with any FEM project, the denser the grid (smaller elements), the more precise results will be obtained. In addition to the slab, the raft may include a number of beams between the column locations. Since the beams would normally be part of the foundation, the slab polygonal meshing algorithm accounts for the presence of the beam and ensures that they become continuously integral with the slab. New nodes are purposely created on the centerline of the beam and the beam is split between those points into a number of segments.

Once the mat is defined and all material/soil properties are input, the program may proceed with the analysis of the structure. It is performed by the state-of-the-art STAAD Analysis Engine. Realistic soil response is achieved by employing non-linear (compression only) spring supports to model subgrade reactions. Pile reactions, if present, are proportional to linear displacements of the supported node and include both compression and tension (uplift).

The program calculates internal forces and deflections for all slab and beam elements of the foundation. This information is then used in the design stage of the program to:

Establish the required top and bottom flexural reinforcing in two orthogonal directions, check punching shear capacity at column locations.

The flexural design is done in accordance with Chapter 10 of the Code. The reinforcement areas are computed for a notional band one unit of length wide.

The program allows the designer, as an option, to use the Wood-Armer equations for reinforcement calculations, as follows:

Mx, My, and Mxy are fetched or calculated, as described above. They are used to compute the values of design moments, Mxd and Myd.

For top reinforcement, the program computes:

$Mx1 = Mx + abs(Mxy)$
$My1 = My + abs(Mxy)$
$Mx2 = Mx + abs(Mxy^2 / My)$
$My2 = My + abs(Mxy^2 / Mx)$

If both Mx1 and My1 are positive, Mxd = Mx1 and Myd = My1.
If both Mx1 and My1 are negative, Mxd = 0 and Myd = 0.
If Mx1 is negative and My1 positive, Mxd = 0 and Myd = My2.
If My1 is negative and Mx1 positive, Mxd = Mx2 and Myd = 0.

For bottom reinforcement:

Mx1 = Mx - abs(Mxy)
My1 = My - abs(Mxy)
Mx2 = Mx - abs(Mxy^2 / My)
My2 = My - abs(Mxy^2 / Mx)

If both Mx1 and My1 are positive, Mxd = 0 and Myd = 0.
If both Mx1 and My1 are negative, Mxd = Mx1 and Myd = My1.
If Mx1 is negative and My1 positive, Mxd = Mx2 and Myd = 0.
If My1 is negative and Mx1 positive, Mxd = 0 and Myd = My2.

Mxd and Myd are then used in lieu of Mx and My for calculations of the required reinforcing. Use of the modified bending moments brings about more accurate distribution of the reinforcing, better matching critical areas of the slab.

Flexural design notes:

Reinforcement calculations for slab panels are based on Chapter 10 of ACI 318-02. The minimum-reinforcing ratio complies with the limits prescribed for shrinkage and temperature reinforcement in Section 7.12. Maximum spacing of rebar is 18 in. The maximum reinforcing ratio corresponds to the net tensile strain at nominal strength equal to 0.004 (Clause 10.3.5). Strength reduction factor is established in accordance with Section 9.3.2.

Punching shear design notes:

Design for two-way shear is carried out in accordance with Section 11.12. The unbalanced moment transfer by eccentricity of shear is based on Clause 11.12.6. Shear strength of concrete is based on Clause 11.12.2.1. Strength reduction factor used is 0.75, in accordance with Section 9.3.2.

The program computes shear stress values at four corners of the rectangular critical section located at the distance of d/2 from edges of a column. The calculations include the unbalanced moment transfer effect, if applicable, in accordance with 11.12.6.2.

4. Strip Footing design

The program uses the following criteria:

d. Soil bearing capacity,
e. Shear and flexural strength of footing (no shear reinforcing assumed),
f. Compressive and flexural strength of pedestal

Step 1 - Determine footing plan geometry based on loading and bearing resistance of the soil.

Stress distribution under the footing is assumed to be linear. For eccentrically loaded footings, the stresses may become tensile under part of the foundation. In such cases the program sets stress values in uplift zones to zero and calculates new values elsewhere for the revised equilibrium condition. The final plan dimensions of the footing are established iteratively from the condition that the maximum stress should not exceed the factored bearing resistance of the soil.

Step 2 - Calculate footing thickness based on structural capacity in shear and bending.

Structural design of the footing consists of the following:

d. Punching shear check, in accordance with Section 11.12.2, at a distance of **d**/2 from the pedestal. The critical section comprises four straight-line segments, parallel to the corresponding sides of the pedestal.
e. One-way shear (beam action), in accordance with Sections 11.1 through 11.5, at a distance of **d** from the face of the pedestal, in both orthogonal directions. The critical plane is assumed to extend over the entire width/length of the footing.
f. Bending, in accordance with Sections 15.4.2 and 10.3.4, with the critical planes located at both orthogonal faces of the pedestal and extending across the full width/length of the footing.

Notes

Quick Tour

Section 3

This section includes discussion on the following topics:

- Introduction
- Starting a New Project
- Entering Support Coordinates
- Defining the Loads
- Using Jobs to Specify Design Constraints
- Entering Design Parameters
- Performing an Isolated Footing Design
- Importing Structural Geometry and Analysis Results from STAAD.*Pro*
- Creating a New Job for a Mat Foundation
- Setting up the Grid and Defining the Mat Boundary
- Creating a Mesh
- Specifying Slab Thickness
- Defining Soil Properties
- Analyzing the Slab
- Slab Design
- Pile Cap Example
- Entering Pile Data
- Entering Pile Cap Design Parameters
- Performing Pile Cap Design and Viewing Results
- Exporting Drawings to CAD
- Strip footing design example
- Entering strip footing design parameters
- Design strip footing and review results

3.1 Introduction

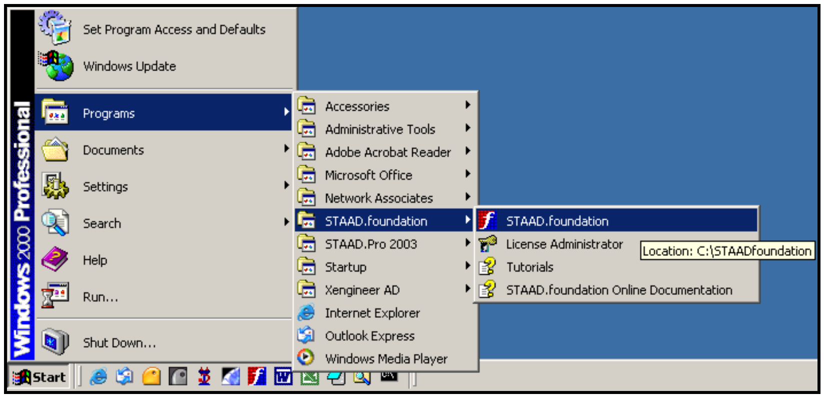
This *Quick Tour* is a set of short example exercises that illustrate how to use STAAD.*foundation* to design several different types of foundations. The procedure for importing support co-ordinates and forces/moments on the individual supports from STAAD.*Pro* is also discussed.

In STAAD.*foundation,* you start out by creating a *Project* to hold all your physical information, such as column locations, loads, etc. This physical information represents the structure that the foundation is intended to support. Unless the design of the structure is modified, these physical conditions generally remain constant throughout the life of the foundation design project. Your Project also contains *Jobs*, which are sets of constraints needed to tell the program how to perform a foundation design. Each project may contain multiple jobs, making it easy for you to evaluate different design scenarios for a given set of physical conditions.

In general discussion, the names of commands, dialog boxes, toolbar buttons or other program controls are indicated in *italics*. When it is intended that you perform a specific action, the names of the menu commands, dialog box labels, data values you are expected to input, etc., are indicated in **bold** type.

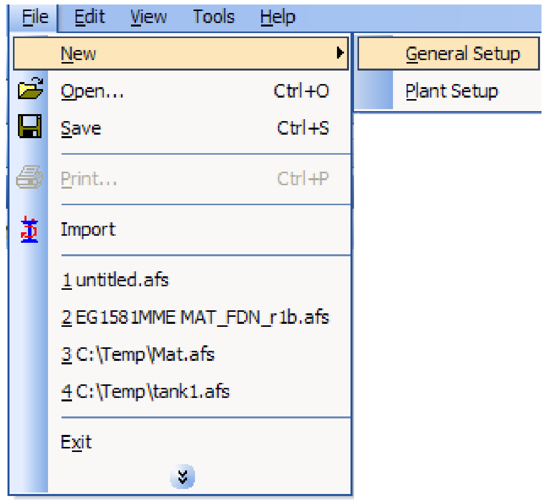
3.2 Starting a New Project

To start STAAD.*foundation*, first click on the Windows **Start** button. Next, select the **Programs** option, select the **STAAD.*foundation*** program group, and then click on the **STAAD.*foundation*** program icon.

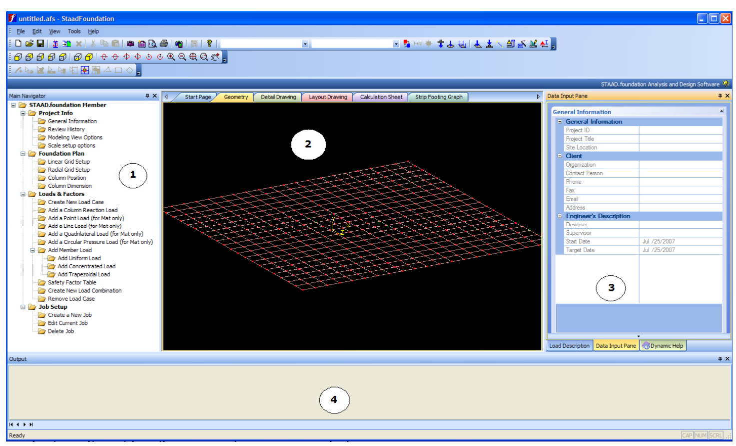


STAAD.*foundation* will launch with the option to create a new project for general foundation or plant foundation.

You may start a project from the *File* menu. You may start a *New* file, *Open* an existing file or *Import* an analyzed file from STAAD.*Pro*.



To start a new project, pull down the **File** menu and select the **New** menu command. Then select foundation type as **General Setup**. Program will add additional tabs in the main window to support general foundation design setup as shown in the following figure.



This introduces the multi-pane window environment with tabbed views of STAAD.*foundation*. As seen in the figure above, a project is divided into 4 separate panes.

The left-most pane (1) is called the *Main Navigator*. It contains the tree by which user will navigate to the different pages within STAAD.*foundation*. The different pages contain forms or grids used to provide input data. The *Main navigator* is arranged from top to bottom in a logical design sequence. If you start at the top tree leaf and work your way down, you will be able to input all the data needed to perform a successful design.

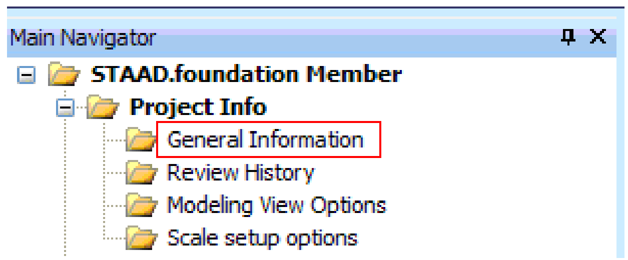
Note: STAAD.*foundation* consists of two sets of data, global and local. Global data such as column reactions and column positions is shared throughout a project among both similar and different jobs. Local data such as design parameters is used only within a specific job type. For example, an Isolated Footing job type has local data within the design parameters group.

The middle pane (2), located to the immediate right of the *Main Navigator* pane, is called the *Main View. It has multiple tabs for graphical display of the model, layout drawing, detail drawing and calculation sheet. For strip footing design it will have an additional tab to display Shear Force and Bending Moment graphs for design load cases.*

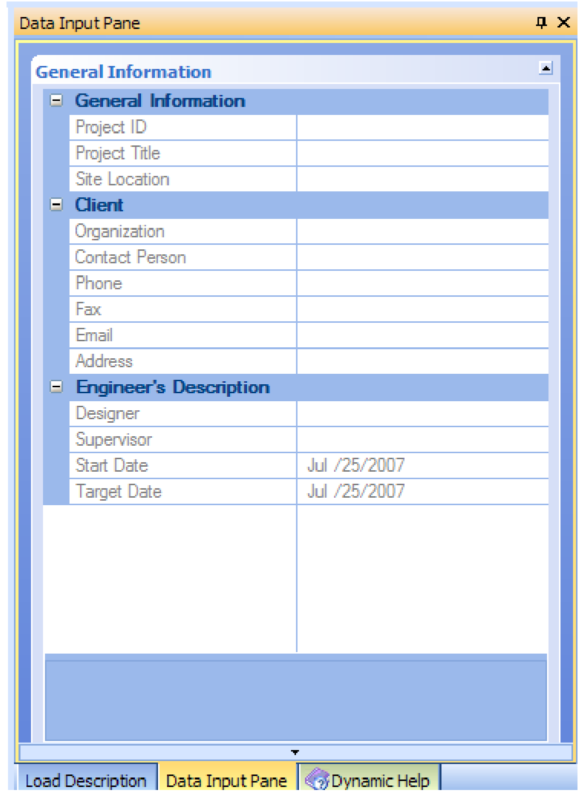
The right-most pane (3) is called the *Data Area*. It also has 2 tabs. One for loading and the other one to contain the forms that provide input data or display output.

The bottom pane (4) is called *Output Area.* It will list design progress while designing or analyzing a foundation and will display output tables like "Output Summary Table" for Isolated, pile cap, strip footing design and analysis output tables like displacement, plate stress etc. for mat foundation.

The default page that opens when starting a new project is the *General Information* page. The *General Information* page may also be invoked by clicking on the *General Information* leaf under Main Navigator tab.

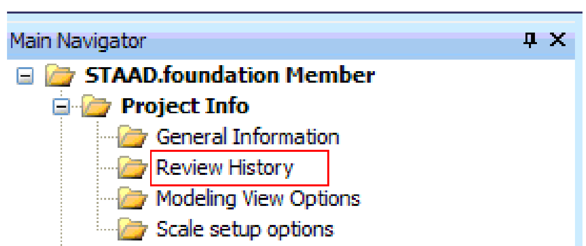


The *General Information* page opens a form in the *Data Area* pane that allows you to store project-related data.

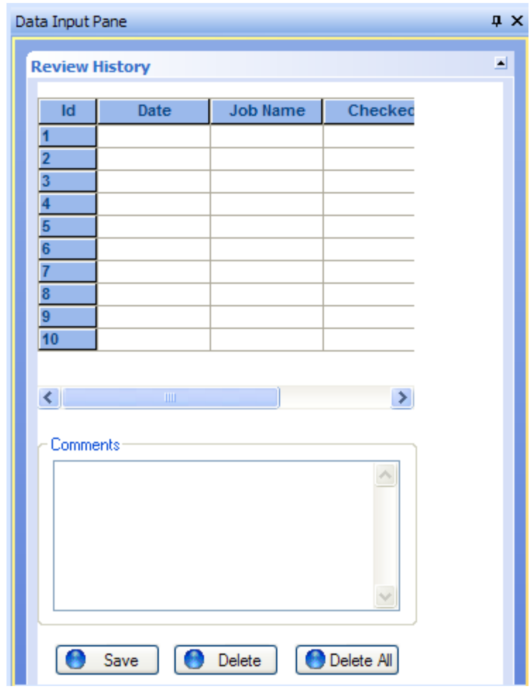


The information inputted on the *General Information* page can later be used in reports and drawings.

In addition to the *General Information* page, there is also another leaf called *Review History*. Invoke the page by clicking on the **Review History** under the *Project Info* group.



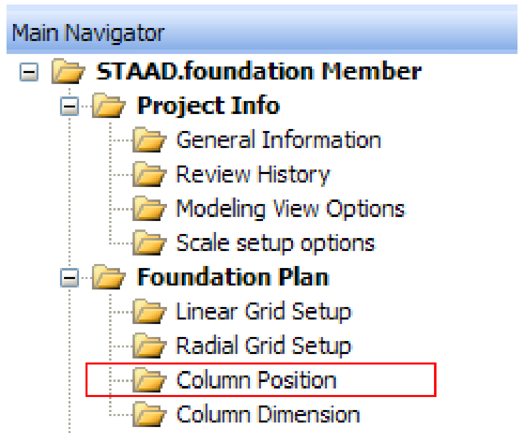
A form will be displayed in the *Data Area* pane that allows you to keep track of the progress of a project.



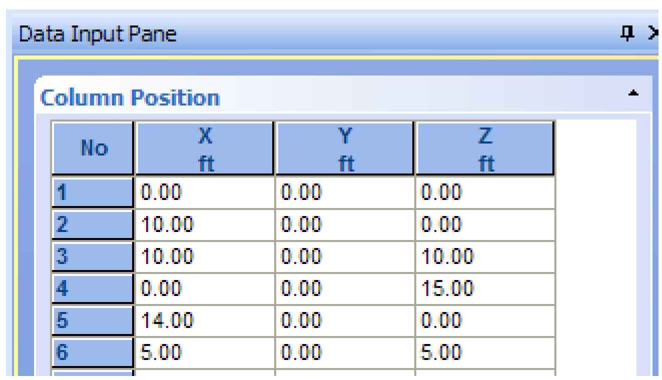
A date, job name, checked by and comments may be entered for each revision of a project. The comments for a given revision will be shown in the *Comments* box if you select the respective revision in the table. The *Save* button should be clicked after inserting any new revision history data to update the revision history table. If you want to delete a particular revision history record, simply select the record by clicking on the respective *Id* and click on the *Delete* button. If you want to delete the entire revision history, click on *Delete All*.

3.3 Entering Support Coordinates

To enter the coordinates for supports that construct the foundation plan of a project, click on the leaf called **Column Position** under **Foundation Plan** group in *Main Navigator* pane.



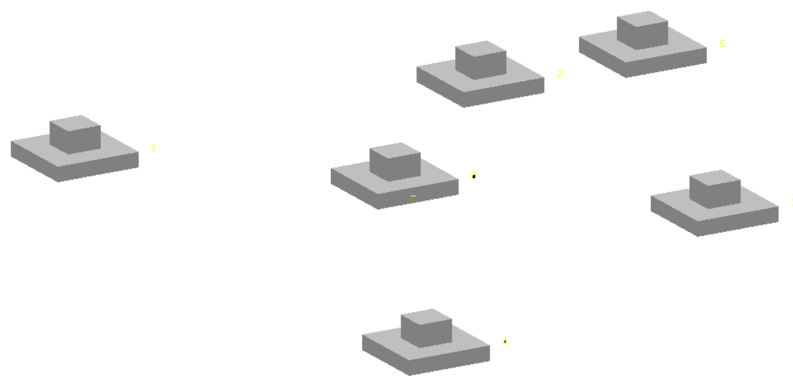
A table allowing you to input the coordinates of supports will be displayed in the *Data Area* pane.



Input the support coordinates **(0,0,0)**, **(10,0,0)**, **(10,0,10)**, **(0,0,15)**, **(14,0,0)** and **(5,0,5)** for Nodes 1, 2, 3, 4, 5 and 6 respectively. Please make sure length unit is set as “ft”. The tab key or the arrow keys may be used to move from one cell to the next in the table. The supports along with their respective node numbers are displayed in the *Graphics Window*.

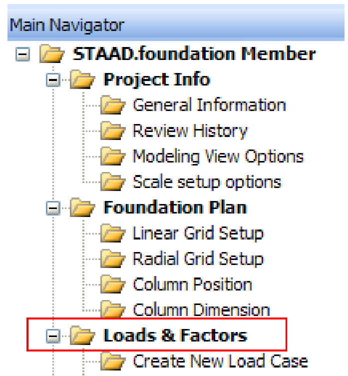
To change/set current length unit please click on the "Set Input/Output Unit" in the toolbar.

Note: Supports will not be shown in the *Graphics Window* until you click on a cell outside of the row you are currently in.

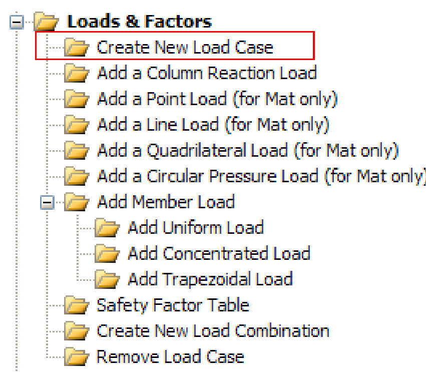


3.4 Defining the Loads

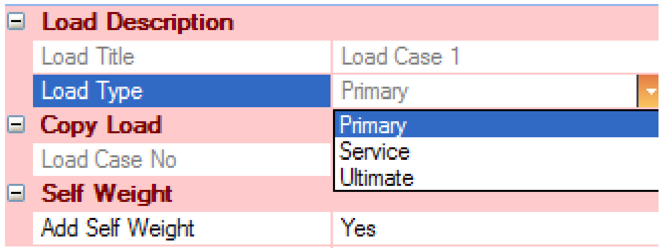
In order to define loads, please click on the **Loads & Factors** group in the *Main Navigator* pane.



By default, the *Load Description* page will open in the *Data Area* pane. The *Load Description* page allows you to define loads for load cases, as well as assign loads. To create a load case, click on the **New Load Case** leaf under the *Loads & Factors* group in Main Navigator.

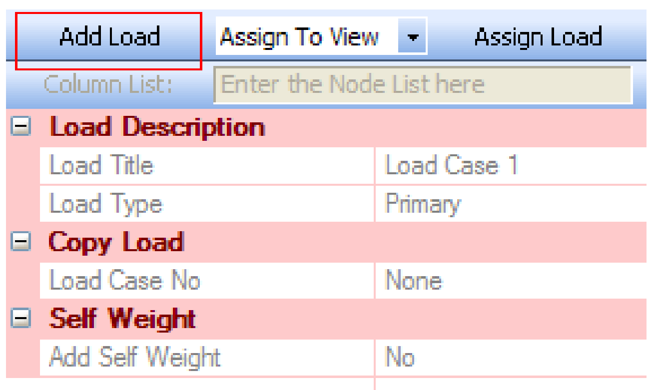


A form under the load description area will appear allowing you to create a new load case.



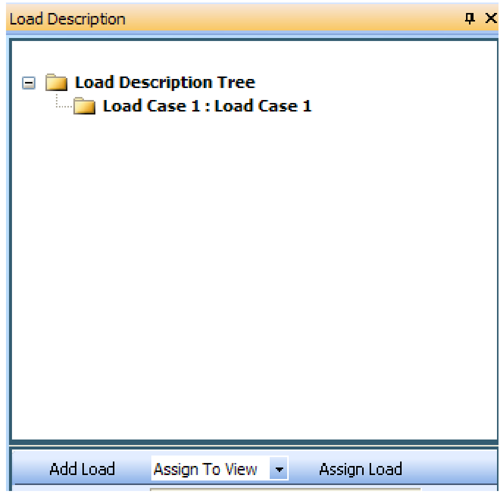
Enter the title "**Load Case 1"** in the *Load Title* field. The *Load Title* allows you to give each load case a descriptive name to help identify between load cases. Leave the *Load Type* set as **Primary**.

> **Note: Three load types are available: Primary, Service, and Ultimate. Primary loads can be further used to create combination loads. Service loads are not factored and are used for soil bearing pressure checks. Ultimate loads are factored and are used for shear and reinforcement design.**

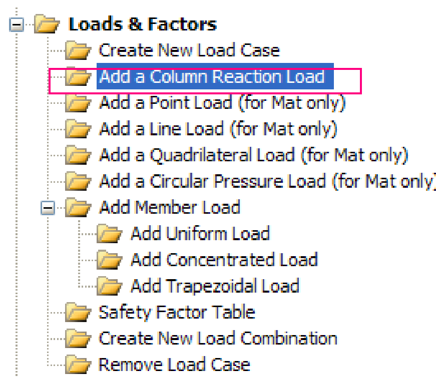


While creating a new load case, load items from an existing load case can be copied. As we don't have any defined load case yet we leave that field as "None" and finally choose "No" for add self weight field.

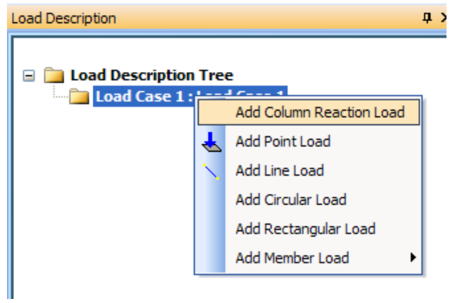
Now click on the **"Add Load"** button to have the load case created.

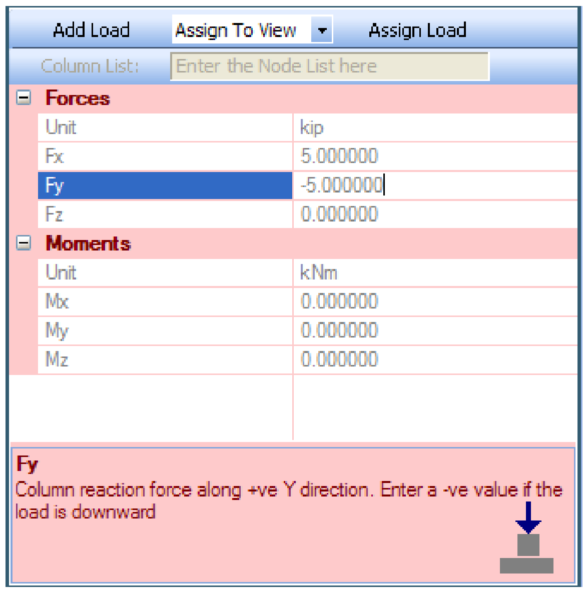


The load case we just created will now appear in the list box in the *Data Area* pane. We will now specify the loads imposed on our foundation by the columns. To do this, click on the "**Add a column Reaction Load**" leaf under Loads and Factors group.



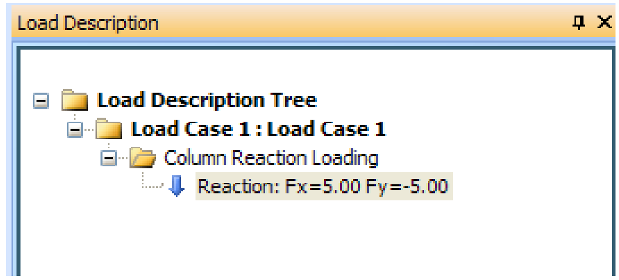
Alternatively you can right click on "Load Case 1: Load Case 1" string in load description area and click on the **Add a Column Reaction Load** menu.



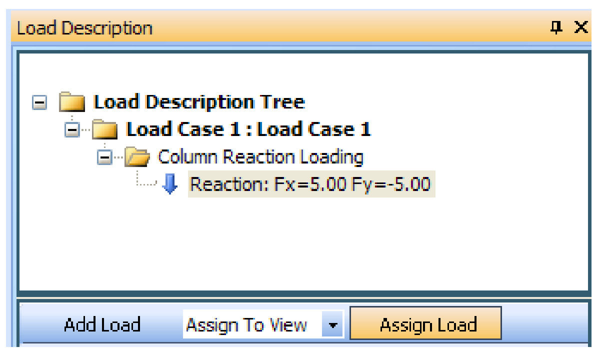


A form will appear allowing you to create a nodal load. Enter a value of **5** in the *Fx* field, and a value of **–5** in the *Fy* field. Then click on the **Add Load** button to accept the load input.

Note: Negative and positive values follow the sign conventions of the axis system. Negative values are downward, compressive forces and positive values are upward, tensile forces.



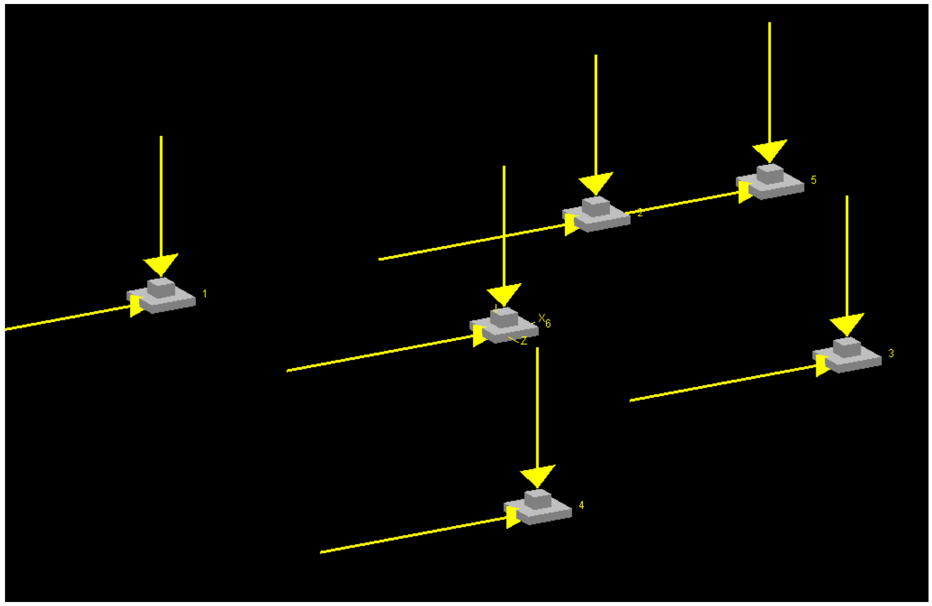
The load will now appear under the *Column Reaction Loading* folder in the *Load Description* pane. We will now assign the load to all the supports we created earlier. First select the load in the *Data Area* pane by clicking on it. Then select **Assign To View** for the *Assignment Method*. Finally, click on the **Assign** button to have the load assigned to all the supports in the project.



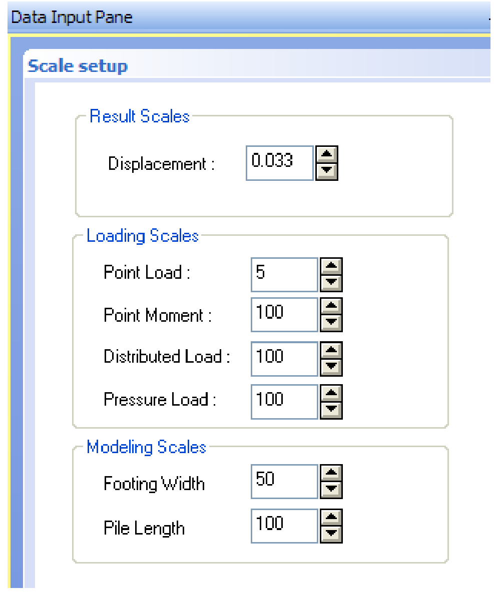
Note: Alternatively, we could have selected all the supports in the *Graphics window* by clicking on them and then selected

***Assign To Selection*. Or, we could have selected *Assign To Edit List* and then typed in the list of nodes for each support.**

The assigned loadings will be displayed on the nodes as illustrated in the figure below.



If you are not able to see the loads properly, it may because the scaling value for the display is either too small or too big. To change the scale value, click on the **Scale Setup Options** leaf under "Project Info" group in Main Navigator pane.



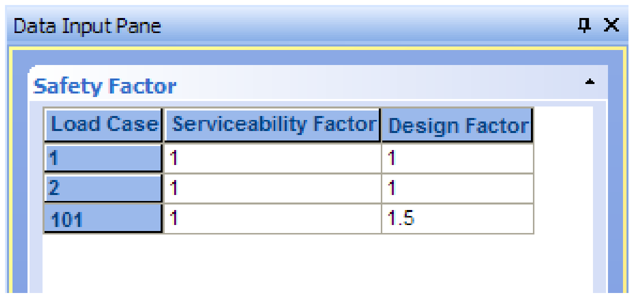
Now increase the value for *Point Load* in order to make the nodal loads more visible. Any changes to the values in the *Scales* tab will become effective immediately in view pane.

We will now repeat the process we just went through to create a second *Load Case*. First click on the **New Load Case** leaf under the *Loads & Factors* group in *Main Navigator* pane. Next, input "**Live Load**" for the *Load Title* and select **No** for self weight and then click on **Add Load**. Now click on the **Add a Column Reaction** leaf, input a value of **10** for *Fz* and click on **Add Load**. Finally, select **Assign To View** and click on **Assign**.

If you have multiple load cases and want to combine them, you can use the *Load Combination* feature. To bring up the *Load Combination* feature, click on the **Create New Load Combination** leaf under the *Loads & Factors* group in *Main navigator* pane.

To define serviceability and design factors for each load case in a project, you may use the *Safety Factor Table*. To bring up the *Safety Factors* page, click on the **Safety Factors** leaf under Loads and Factors group in *Main Navigator* pane.

A table allowing you to input serviceability and design factors for each load case will be displayed in the *Data Area* pane.

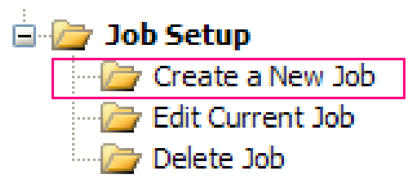


Load Case	Serviceability Factor	Design Factor
1	1	1
2	1	1
101	1	1.5

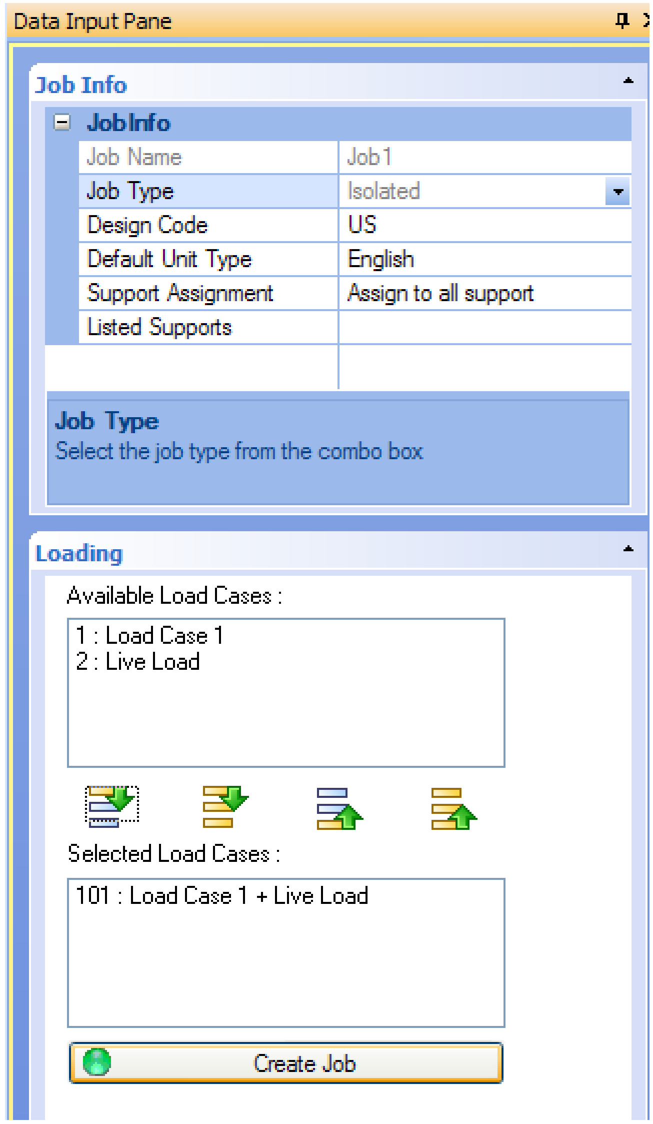
By default, STAAD.*foundation* will assign values for the safety factors depending on the load type. Refer to section 4.3.3.2 for a detailed explanation of the default values. The default values can be changed by inputting new values into the table like any spreadsheet. The tab key or arrow keys may be used to move from one cell to the next in the table. The serviceability factor will be applied when checking the base pressure of a foundation (geotechnical design). The design factor will be used for design.

3.5 Using Jobs to Specify Design Constraints

Now that all the global project data has been inputted, you have the ability to design the foundation using *Isolated Supports*, *Pile Caps*, *Strip Footing* or you could support the entire structure on a single *Mat Foundation*. You will not have to create separate input files for entering all this information. All you have to do is to create separate jobs under the same project. In order to create a job, click on **Create a New Job** leaf under the job setup group in *Main Navigator* pane.



A form to create a new job will open in the *Data Area* pane.



The job types may be to design for *Isolated*, *Pile Cap, Mat Foundation, Strip footing and Octagonal footing*. We can assign the job to all the supports or we can type in the list of supports to be included in the job.

The design codes may be US, British or Indian.

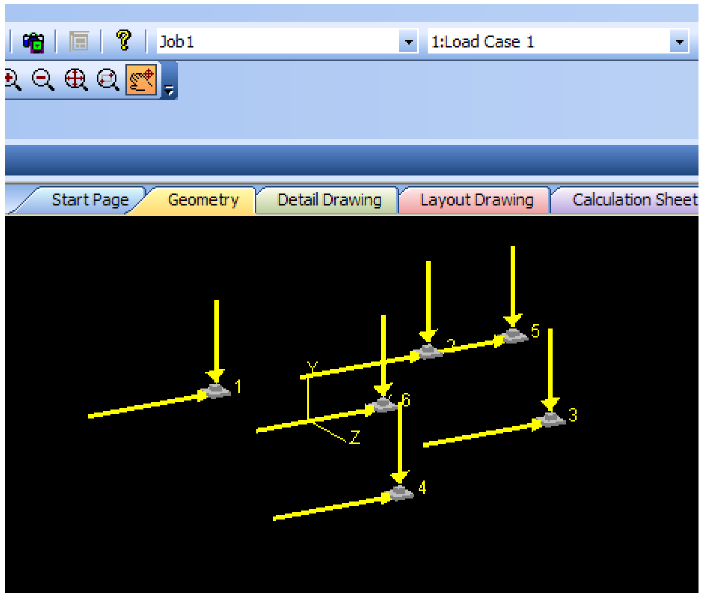
The default unit type may be English or SI. This denotes the units in which the actual calculation will be performed. The reports etc. of course will be shown as per the users' choice of force and displacement units.

For this *Quick Tour* example, enter the job name **Job1** in the *New Job Name* edit box. In the *Job Type* drop-down menu, select the **Isolated** job type. Under the *Supports In This Job* category, check to ensure that the **All Supports** radio button is activated by default.

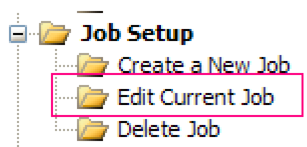
In this *Create a New Job* form we have another group called ***Loading***. Using this page we could have several jobs of the same type (e.g. *Isolated* footing) having different design load cases.

We will take all the loadings for this job. Click the button to move all the load cases over to the Selected Load Cases list on the bottom of the page. Click the **Create Job** button. A new job is created.

After the job is input, the graphics display window looks like the following figure.



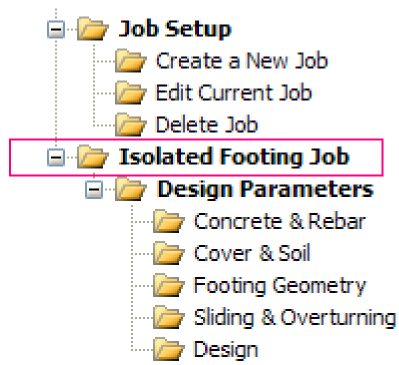
When there are multiple jobs and load cases in a project, by changing the job numbers and the load case numbers in the respective combo boxes, we can change the display of supports and loadings in the window. We may also change the job settings of the job shown in the job combo box by clicking the *'**Edit Current Job**'* leaf under job setup group in *Main Navigator* pane.



3.6 Entering Design Parameters

When you begin a new project, only the *Project Info*, *Foundation Plan*, *Loads and Factor and Job Setup* groups will appear in the *Main Navigator pane*. The first three groups allow you to specify the physical model upon which the foundation design is to be performed. A fourth group (Job Setup) allows you to create a new job or edit an existing job. It is only when you create a New *Job* (a set of constraints for the program to use in performing a foundation design) that *groups* related to the current design process will appear.

Now, take a look at the *Main Navigator* pane. A new group called "**Isolated Footing Job**" is created. This group allows you to enter design parameters like footing geometry, concrete cover, soil parameters etc. As *Design Parameters* forms are self-explanatory, we will not discuss them in this Quick Tour.

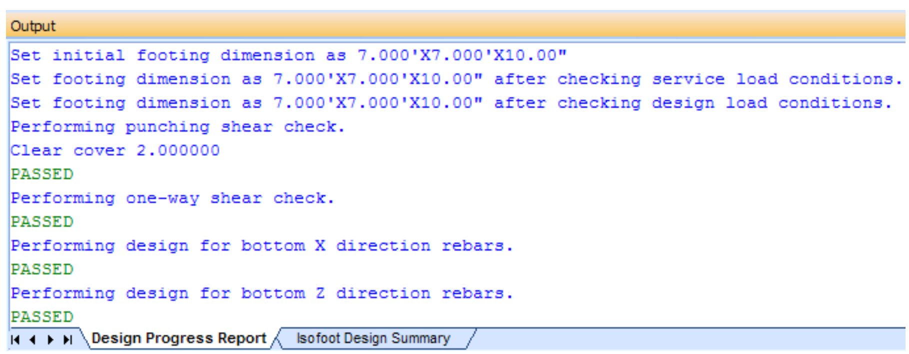


Note: STAAD.foundation gives the user flexibility to check an existing foundation by specifying footing geometry like Length, Width and Thickness or design a new foundation where the program will calculate footing dimension.

3.7 Performing an Isolated Footing Design

Now click on the **"Design"** leaf under "**Design Parameters**" group in *Main Navigator* pane to design the footing.

Look at the *Output* pane, it will display a series of messages as the program performs the footing design.



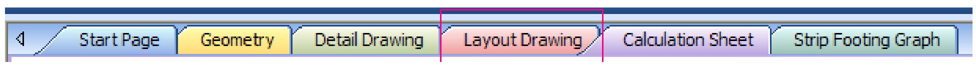
When the design is complete, the program will automatically display a *Design Summary* table in the ***Output*** pane.

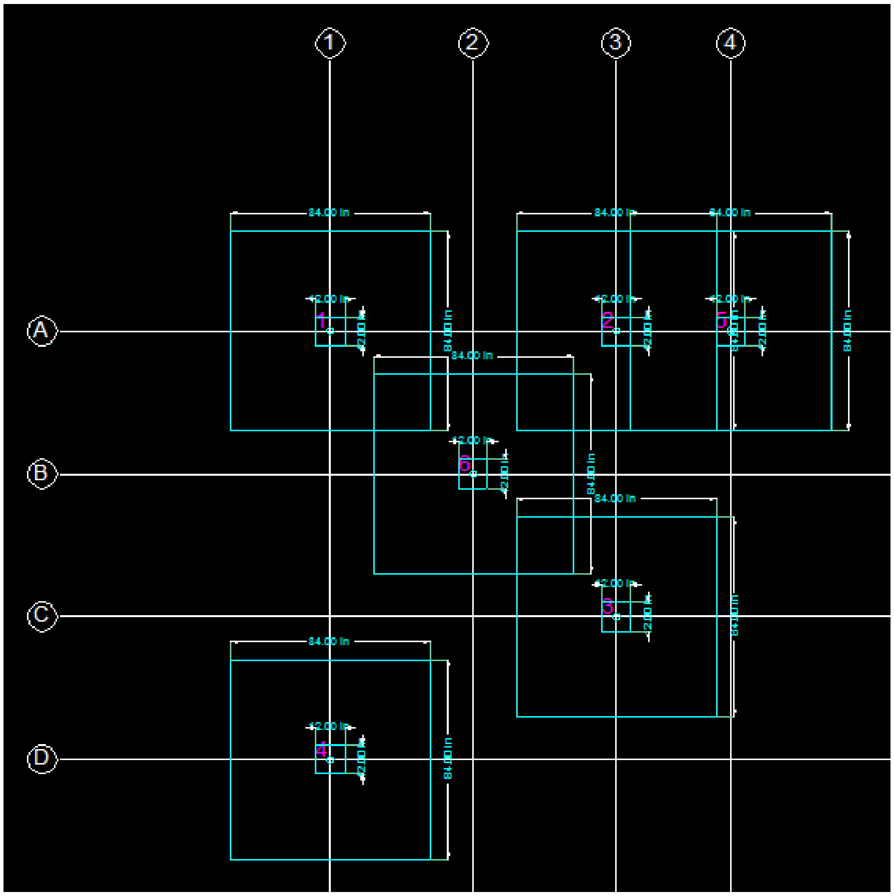
Output

FootNo	GrpID	Footing Dim			Footing Reinfo			
		Length ft	Width ft	Thickness ft	Main Steel		Secondary Steel	
					Bar	Spacing	Bar	Spacing
1	1	7.00	7.00	0.83	#6	15.85 in	#6	15.85 in
2	2	7.00	7.00	0.83	#6	15.85 in	#6	15.85 in
3	3	7.00	7.00	0.83	#6	15.85 in	#6	15.85 in
4	4	7.00	7.00	0.83	#6	15.85 in	#6	15.85 in
5	5	7.00	7.00	0.83	#6	15.85 in	#6	15.85 in
6	6	7.00	7.00	0.83	#6	15.85 in	#6	15.85 in

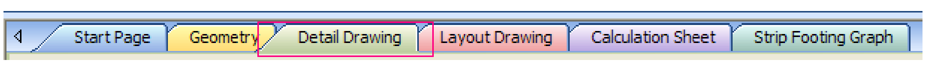
Design Progress Report | Isofoot Design Summary

Click on the “Layout Drawing” tab in view pane area to check footing overlap. It produces a layout of analyzed footings drawn to scale, complete with a title block for the drawing.





Click on the "Detail Drawing" tab in view pane area to see footing detailed drawing. The detailed drawing displays a schematic diagram of the footing elevation and reinforcement plan.



A click on the *'Calculation Sheet'* tab in view pane brings up the design calculation of the footings. It displays step by step calculation with relevant code clause numbers and equations.

Critical Axial design Load, Pu = 0.00 kip, Load Case # 2

From ACI Cl.11.12.2.1, β_o for column =	$2 \times (B_{col} + D_{col} + 2 \times d_{eff}) =$	216.00	in
Equation 11-35, V_{c1} =	$\left(2 + \frac{4}{\beta_c}\right) \times \beta_o \times d_{eff} \times \sqrt{1000 \times F'_c} =$	3442.58	kip
Equation 11-36, V_{c2} =	$\left(2 + 40 \times \frac{d_{eff}}{\beta_o}\right) \times \beta_o \times d_{eff} \times \sqrt{1000 \times F'_c} =$	5610.13	kip
Equation 11-37, V_{c3} =	$4 \times \beta_o \times d_{eff} \times \sqrt{1000 \times F'_c} =$	2295.05	kip
Punching shear strength, V_c =	minimum of (V_{c1}, V_{c2}, V_{c3}) =	2295.05	kip

This calculation sheet is web-enabled for real time checking. Hard copies can also be made from this sheet.

A project once created can be saved and re-opened later using the *File* | *Save* and *File* | *Open* options. STAAD.*foundation* files are saved with .afs extension.

3.8 Importing Structural Geometry and Analysis Results from STAAD.*Pro*

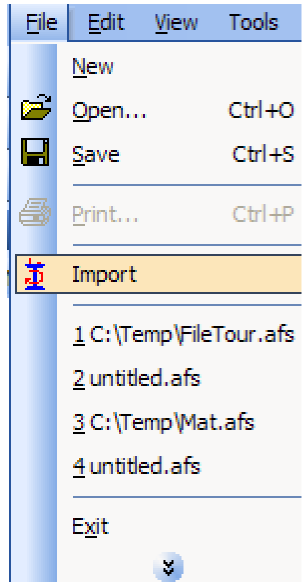
In most cases the forces and moments on the foundation are given by the analysis of the superstructure. To ensure a seamless and efficient integration with the analysis software, STAAD.*foundation* includes an *"Import"* facility. This option allows us to import the support co-ordinates and forces/moments on the individual supports from a structural analysis software program.

At present we have the facility to import analysis data from STAAD.*Pro* structural analysis software. Thus by default the control goes to the folder where STAAD.*Pro* example files are located. If you do not have STAAD.*Pro* installed in your machine, please do not use the import facility now. The ability to import analysis data from other software programs will be provided in a future release of STAAD.*foundation*.

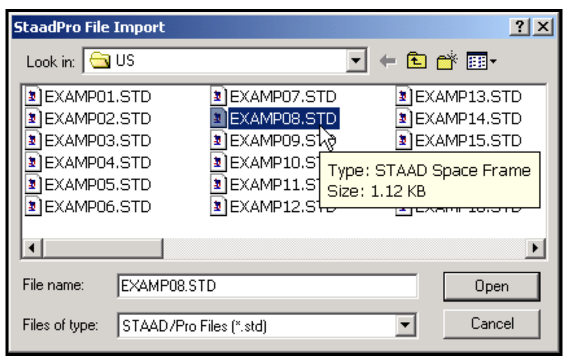
Note: You must first perform an analysis and design on a STAAD.*Pro* model in STAAD.*Pro* before importing the model into STAAD.*foundation*.

Let us import STAAD.*Pro* US Example No. 8 to STAAD.*foundation* and use the imported geometry and support reactions to design a *mat* foundation for the structure. You can only import a STAAD.*Pro* model that has been successfully analyzed, because you will want to have the support reactions available for the foundation design. So, if you have not already run the analysis for STAAD.*Pro* U.S. Example No. 8 open the example in STAAD.*Pro* (C:\Spro2007\STAAD\Examp\US\Examp\Examp08.std), run the analysis, and then return to this Quick Tour.

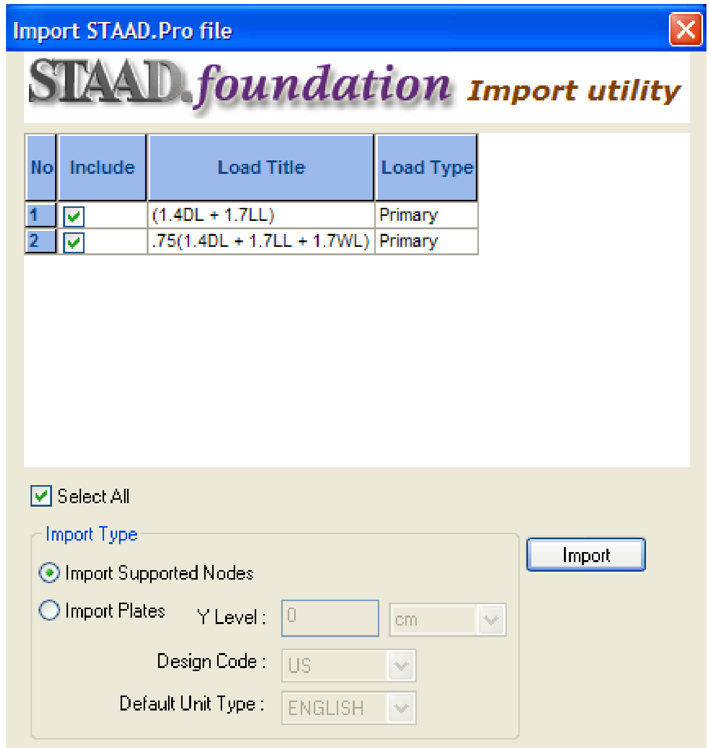
Pull down the **File** menu and select the **Import** command.



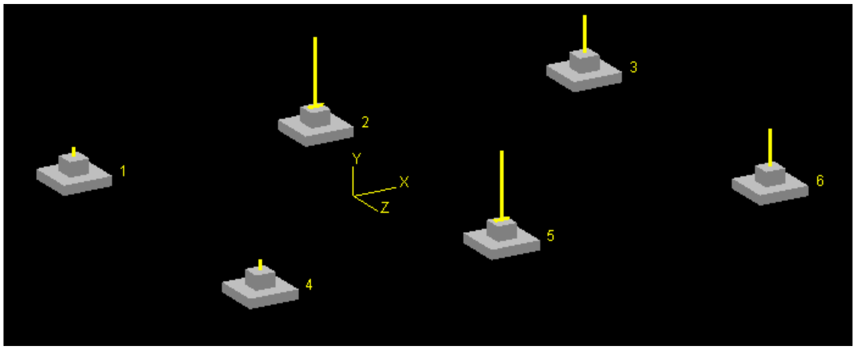
A file manager dialog box labeled *STAADPro File Import* will be displayed.



Locate the STAAD.*Pro* US Example No. 8 file and highlight it. Then click on **Open.**

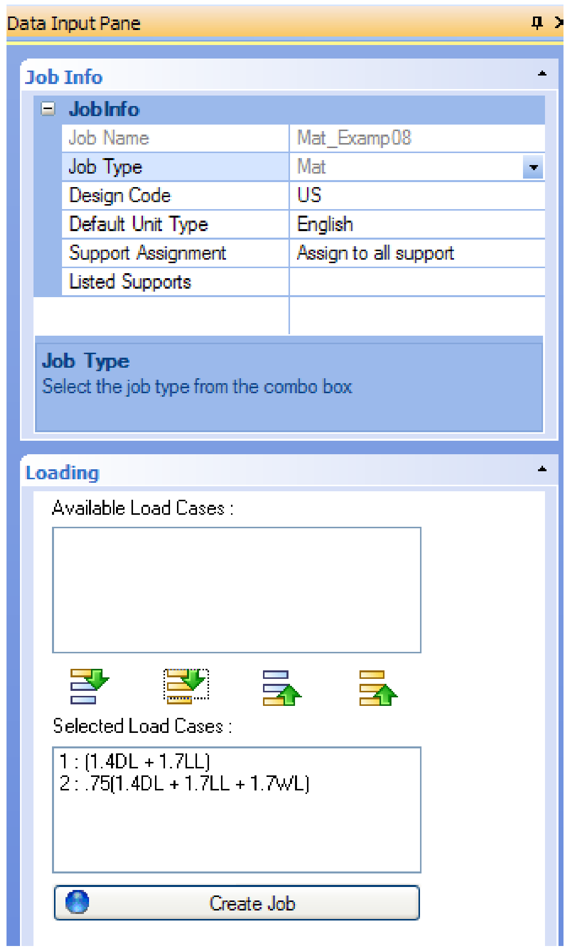


Click on **Import** in the resulting dialog box. The support coordinates will be imported to STAAD.*foundation* as shown in the graphics display window. Notice that you did not have to create a new project. STAAD.*foundation* did not overwrite the project you already had open, or add any new data to it. Instead, the program has created a new project with the default project name *Untitled*.


1
2
3
4
5
6
Y
X
Z

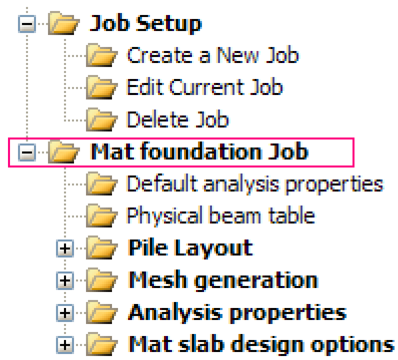
3.9 Creating a New Job for a Mat Foundation

Click on the **Create a new Job** leaf under *Job Setup* in *Main Navigator* pane to create a new job for designing the mat foundation. The *Create a New Job* form will open in the data pane area. Choose Job type as Mat foundation and design code as US.



The job will be assigned to all the supports. We then include all the loadings on it from the *Loading* group of the same property form. Now click on "Create Job" to create a new mat foundation job.

If we look at the Main Navigator pane we see a set of new groups not seen in our previous project. These groups are related to mat foundation analysis and design.

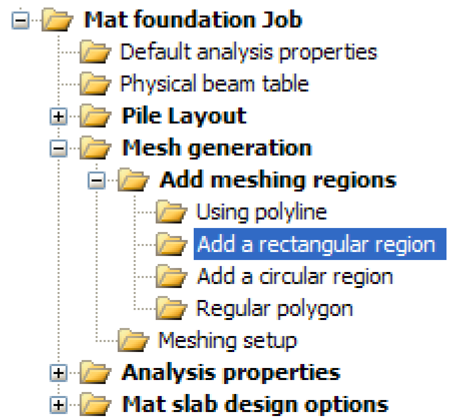


The *Meshing* generation and *Analysis properties* groups are particular to the mat foundation. We also see the Pile Layout group which is to create pile arrangement in case the mat slab is supported by piles instead of soil. These groups only appear when a Mat Foundation job type is active.

For our example, mat slab will be supported by soil; in other words it's a slab on grade problem.

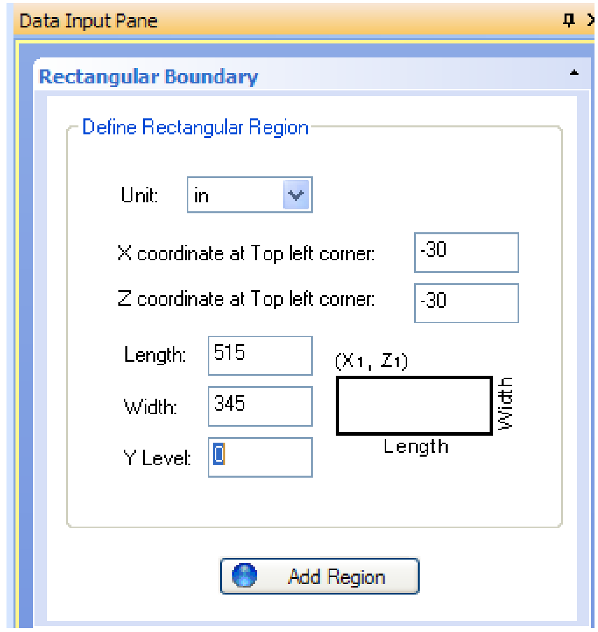
3.10 Defining the Mat Boundary

Now we would like to define the boundary of the mat. To do this we expand *"Mesh generation"* group and then click on "Add a rectangular region" leaf.

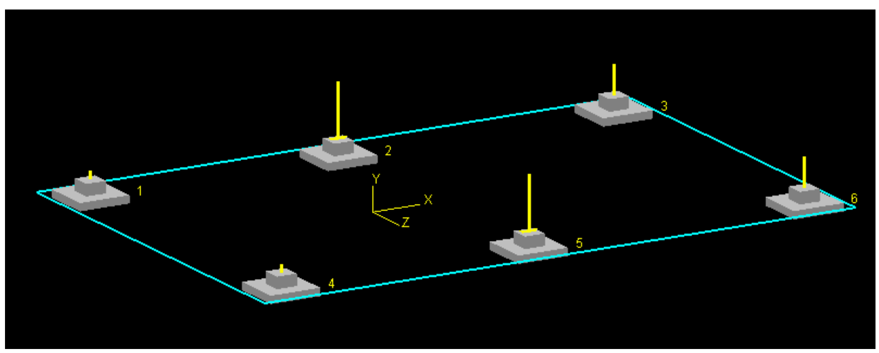


The following form opens up in data area pane. Set unit as "inch" and input X1,Z1 as (-30,-30). Then enter *Length* as 515 inch and *Width* as 345 inch. Keep Y level as 0.0 as our support columns have same Y level.

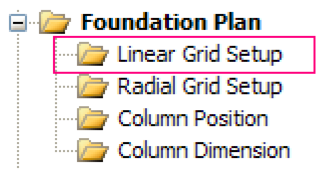
Now, click on "Add Region" button to create the *Mat* boundary.



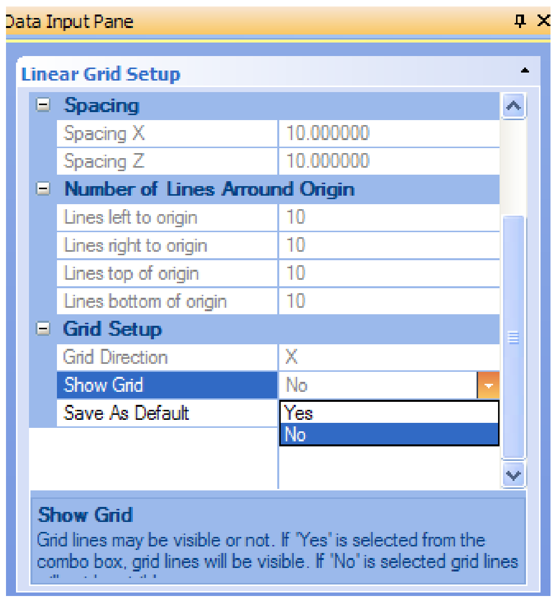
This action will create the boundary in the graphics display window of view pane.



If your screen shows a grid, you may want to switch it off by clicking on ***Linear Grid* Setup** leaf under *Foundation Plan* group in *Main Navigator* pane.



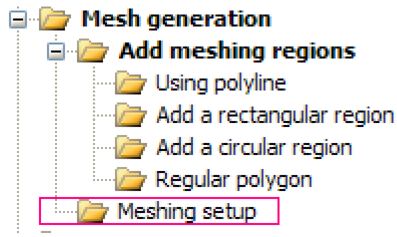
The following form will open in data area pane. Choose ***Show Grid*** as "No" to switch off the grid.



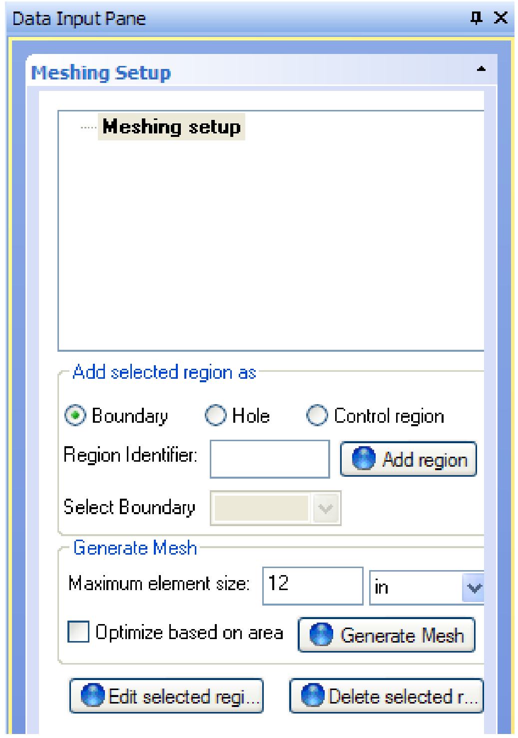
Now it would be a good idea to save your model, since you have done a substantial amount of work to get to this point. Pull down the **File** menu and select the **Save** command.

3.11 Creating a Mesh

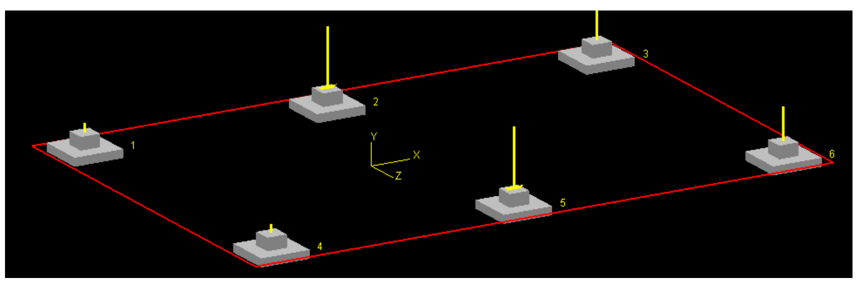
Now we are ready to add the boundary and create the mesh. Click on the *Meshing Setup* leaf under *Mesh generation* group.



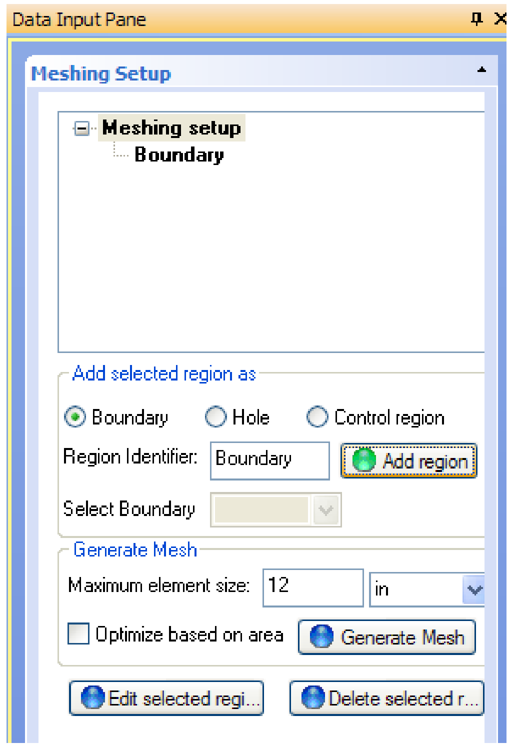
The *Meshing Setup* page will be displayed in the *Data Area* pane.



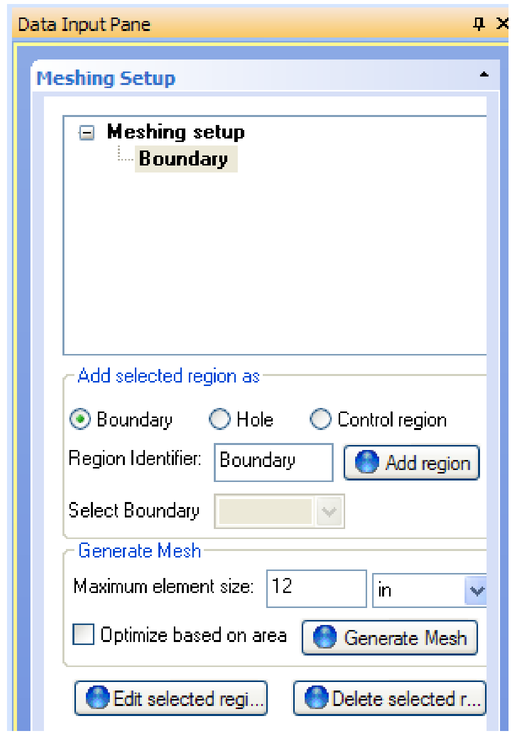
Click in the Graphics Window with the cursor on a line of the boundary. The boundary changes color.



Make sure the **Boundary** radio button is checked and give a title for the boundary in *Region Identifier* edit box. Click on "***Add Region***" button to add the boundary. You will see that the *Region Identifier* name you entered is now listed under the *Meshing Setup* heading in the tree.

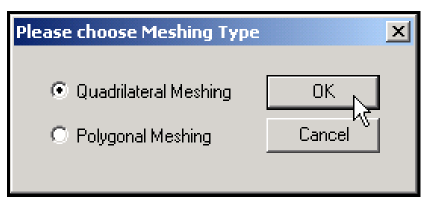


We may also choose the number of divisions for the mesh and specify locations of holes. Let us specify an element size of 12 inches. Enter a value of **12** in the Maximum *Element Size* edit box. In this example project we will not create any holes in the mesh. We are ready to create the mesh. In the list box in the *Data Area* pane, highlight the Mesh Identifier Name for your mesh boundary.

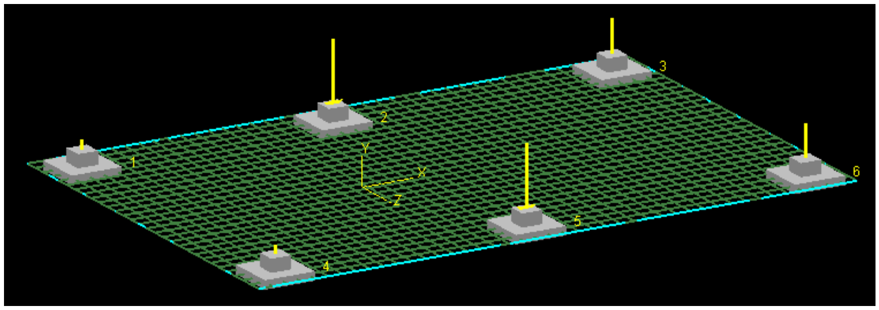


Click on the **Generate Mesh** button to generate mesh.

The program will display a dialog box asking you to choose either a Quadrilateral Mesh or a Polygonal Mesh.

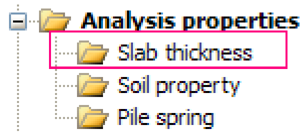


Verify that the *Quadrilateral Meshing* radio button is selected by default, and then click the OK button. STAAD.*foundation* will create the mesh and display it in the graphics window.

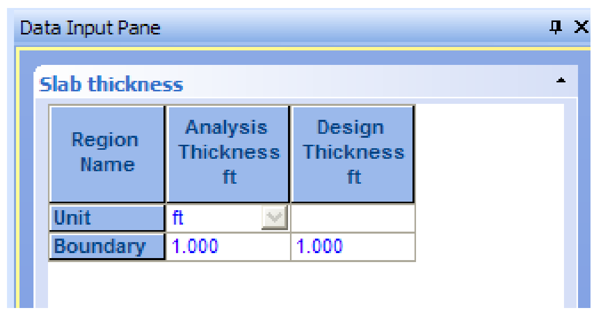


3.12 Specifying Slab Thickness

As this is a physical modeling system, slab thickness and soil properties are automatically assigned to the slab with default values. To change slab thickness click on the ***Slab Thickness*** leaf under ***Analysis Properties*** group.



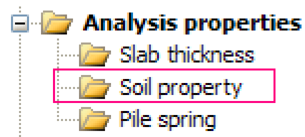
The *Slab Thickness* page will be displayed in the *Data Area* pane.



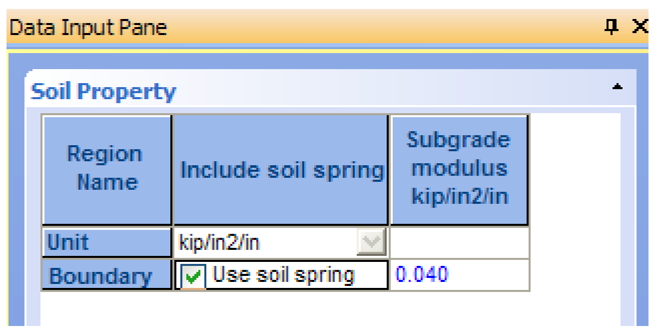
By default it will show default slab thickness. We can change this thickness to our desired values. For our example, we will use default values. There are two types of thickness which are analysis thickness and design thickness. Analysis thickness will be used to analyze the slab and design thickness will be used while designing the slab. This is particularly important in modeling a pedestal, where you may want to use excess thickness for stiffness modeling but want to use slab thickness for design

3.13 Defining Soil Properties

Like thickness, soil property is also automatically created assigned with the slab, the only thing we need to do is to activate the flag. To change or activate soil property click on the ***Soil Property*** leaf under ***Analysis Properties*** group.

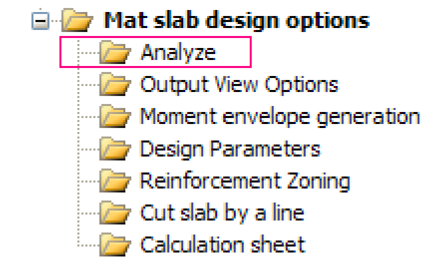


The *Soil Properties* page will be displayed in the *Data Area* pane. Check on the check box for soil spring.

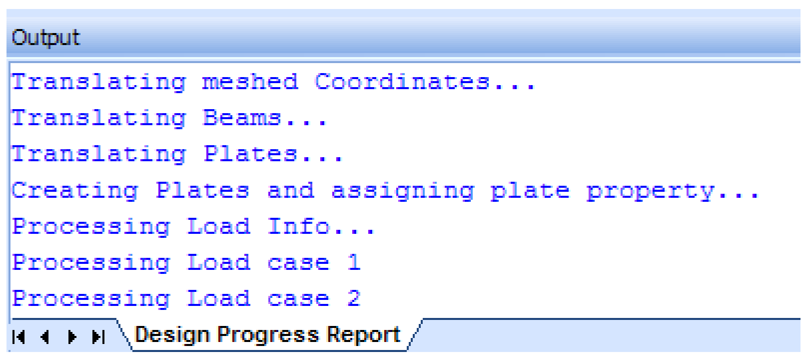


3.14 Analyzing the Slab

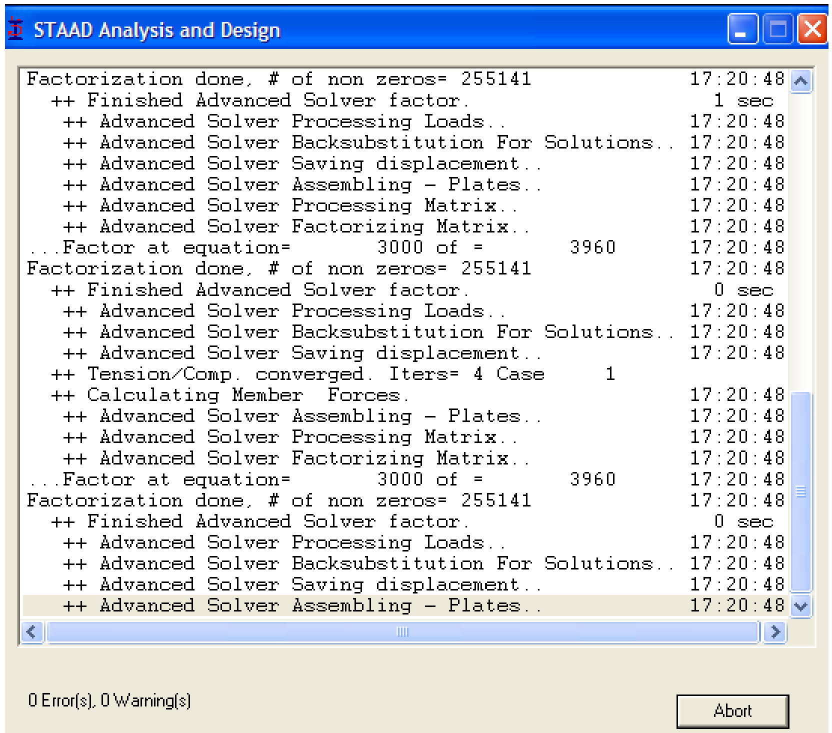
We are ready to analyze the slab. Save your work one more time: pull down the **File** menu and select the **Save** command.
Click on the ***Analyze*** leaf under **"Mat Slab design Options"** group to analyze the mat.



Design progress report tab under ***Output*** group will be populated with progress messages while the program creates an analytical model to analyze.

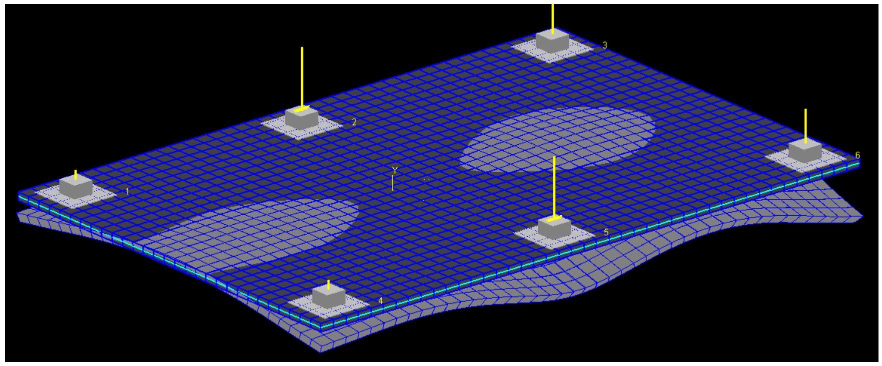


Then another window will come up and show the analysis progress messages and status. When the analysis is completed that window will automatically disappear.

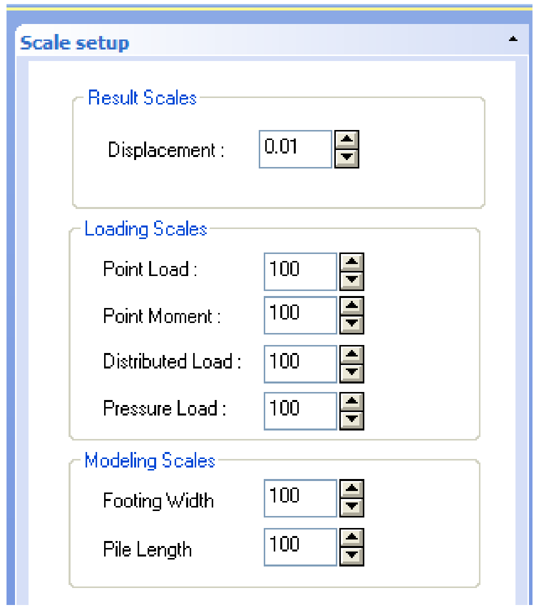


When you see the messages have stopped scrolling, look for a final message, "Analysis is completed" which indicates that the analysis has been successfully performed.

By default the deformed plates showing the node displacements appear in the graphics display window.

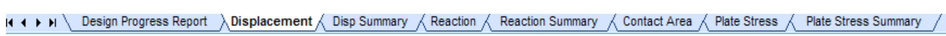


If the slab's deformed shape is not apparent in your graphics display, you may need to change the scaling values. Click on the toolbar for changing scale which will bring up ***Scale Setup*** page in data area pane.



Under the *Result Scales* category, decrease the *Displacement* value to increase the amount of deflection shown. Why do you decrease it to increase the deflection? The *Displacement* value in the dialog box is the actual displacement of the structure per unit distance on the graphic diagram. Therefore, if you reduce the amount of actual structural deflection required to display a unit distance of deflection on the diagram, you will see a larger apparent displacement on the diagram.

After a successful analysis, the program will add several tables in the output pane below.



Click on the *Displacement* tab to view nodal displacement for current selected load case.

Output

Node	Dx in	Dy in	Dz in	Rx Deg	Ry Deg	Rz Deg
1	0.000000	-0.296130	0.000000	-0.002332	0.000000	0.000834
2	0.000000	-0.285943	0.000000	-0.002315	0.000000	0.000864
3	0.000000	-0.274945	0.000000	-0.002268	0.000000	0.000940
4	0.000000	-0.262900	0.000000	-0.002221	0.000000	0.000999
5	0.000000	-0.250547	0.000000	-0.002180	0.000000	0.000962
6	0.000000	-0.239450	0.000000	-0.002141	0.000000	0.000791
7	0.000000	-0.231245	0.000000	-0.002110	0.000000	0.000500
8	0.000000	-0.227239	0.000000	-0.002094	0.000000	0.000120

This table lists the node displacement for the three translational and three rotational degrees of freedom. Click on "Disp Summary" tab to view node displacement summary for all six degrees of freedom among all load cases. Please note, maximum positive displacement in Y direction is 0.049418 in and maximum negative displacement is .792751 in.

Output

	Node	Load Case	Dx in	Dy in	Dz in	Rx Deg	Ry Deg	Rz Deg
Max Dx	1	1	**0.000000**	-0.296130	0.000000	-0.002332	0.000000	0.000834
Min Dx	1	1	**0.000000**	-0.296130	0.000000	-0.002332	0.000000	0.000834
Max Dy	693	1	0.000000	**0.049418**	0.000000	0.000150	0.000000	-0.000102
Min Dy	1320	1	0.000000	**-0.792751**	0.000000	0.005146	0.000000	-0.003932
Max Dz	1	1	0.000000	-0.296130	**0.000000**	-0.002332	0.000000	0.000834
Min Dz	1	1	0.000000	-0.296130	**0.000000**	-0.002332	0.000000	0.000834
Max Rx	1144	1	0.000000	-0.523888	0.000000	**0.006002**	0.000000	-0.003256
Min Rx	220	1	0.000000	-0.519242	0.000000	**-0.005586**	0.000000	-0.003260
Max Ry	1	1	0.000000	-0.296130	0.000000	-0.002332	**0.000000**	0.000834
Min Ry	1	1	0.000000	-0.296130	0.000000	-0.002332	**0.000000**	0.000834
Max Rz	1302	1	0.000000	-0.472672	0.000000	0.004098	0.000000	**0.003573**
Min Rz	1316	1	0.000000	-0.579236	0.000000	0.004439	0.000000	**-0.004835**

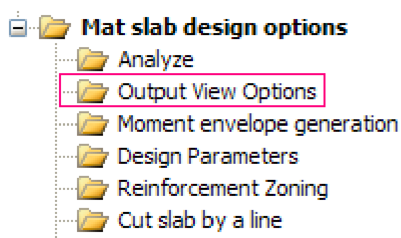
Please click on the 'Support Reactions' tab to view soil pressure for the current load case. To know maximum reaction among all load cases please click on "Reaction Summary" tab.

Output

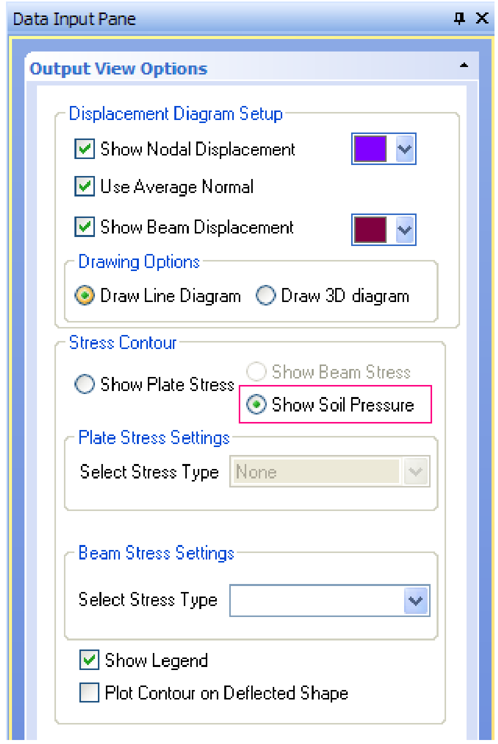
	Node	Load Case	Fx kip	Fy kip	Fz kip	Mx kip-in	My kip-in	Mz kip-in
Max Fx	1192	1	**1.496520042419**	1.239615082741	-0.256951004	0	0.003219750	0
Min Fx	1230	1	**-1.91767001152**	3.298341035843	-0.284902006	0	-0.000755416	0
Max Fy	1275	1	0	**3.902505636215**	0	0	0	0
Min Fy	471	1	0	**0**	0	0	0	0
Max Fz	154	1	0.1348859965801	2.467761039734	**0.2048510015**	0	-0.001639820	0
Min Fz	1230	2	-1.218780040741	2.879165887833	**-5.640240192**	0	0.856886982	0
Max M	1	1	0	0.421932607889	0	**0**	0	0
Min Mx	1	1	0	0.421932607889	0	**0**	0	0
Max M	1192	2	1.244420051575	1.088517308235	-2.414170026	0	**4.461840152**	0
Min My	136	1	1.06895005703	1.036438703537	0.1835380047	0	**-0.002299840**	0
Max M	1	1	0	0.421932607889	0	0	0	**0**
Min Mz	1	1	0	0.421932607889	0	0	0	**0**

Design Progress Report | Displacement | Disp Summary | Reaction | Reaction Summary | Contact Area

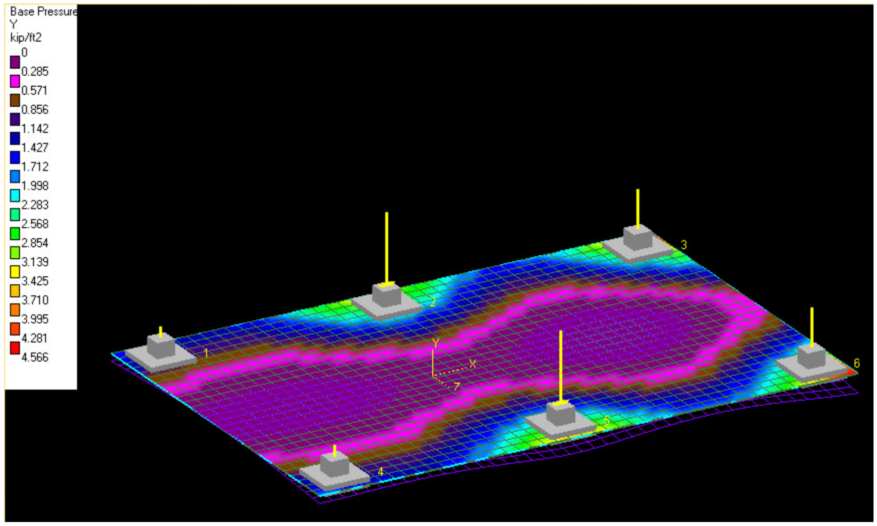
To view soil pressure contour, please click on the "Output View Options" leaf under "Mat slab design options" group in main navigator pane.



A form called "Output View Options" will appear in data input pane. Please select "Show Soil Pressure" radio to view soil pressure contour.



A soil pressure legend will be displayed at the left of the view pane along with the soil pressure contour. Please note, the maximum soil pressure for load case 1 is 4.556 kip/ft2. Also, minimum soil pressure is 0.0 which means that some part of the mat has lost contact with the soil and the program has distributed the pressure of that portion to the rest of the mat slab.



We can easily verify the slab's loss of contact with the soil by reviewing "Contact Area" table. Please note for both load cases more than 80% of total area is in contact with the soil.

Output

Load Case	Area in Contact (ft²)	% of Total Area	Area Out of Contact (ft²)	% of Total Area
1	1034.478480	83.84121906	199.3758122	16.15878093
2	1028.541729	83.36006411	205.3125633	16.63993588

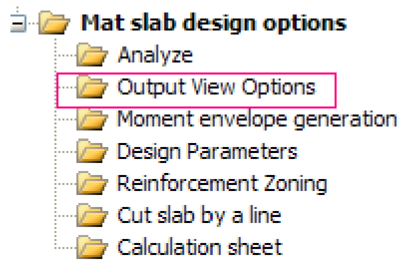
To review plates stresses please click on the "Plate Stress" and "Plate Stress Summary" pages. Please note that the stress summary page displays a maximum value 68.636 kip-ft/ft. Please note that all plate stress values are based on plate local axis system.

Output

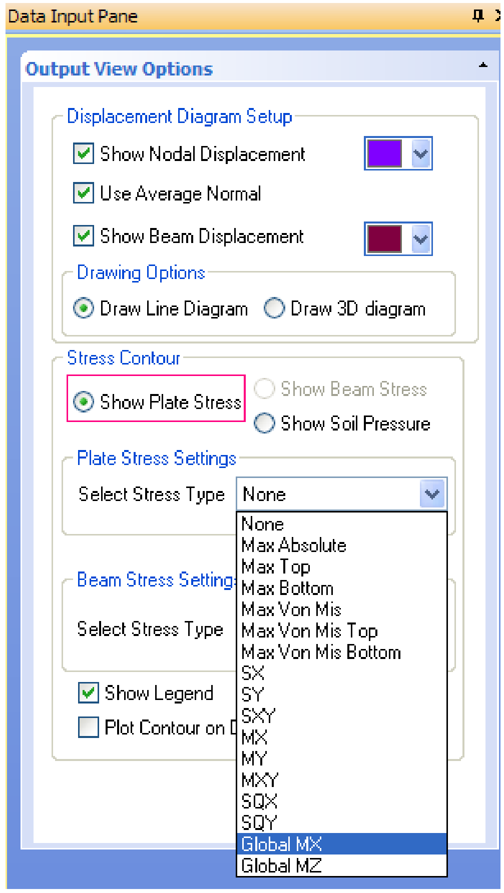
	Plate	Load	SQX kip/ft2	SQY kip/ft2	SX kip/ft2	SY kip/ft2	SXY kip/ft2	MX kip-ft/ft	MY kip-ft/ft	MXY kip-ft/ft
Max SX	1	1	0.514779	0.948151	**0.000000**	0.000000	0.000000	1.022040	1.121768	-0.813953
Min SX	1	1	0.514779	0.948151	**0.000000**	0.000000	0.000000	1.022040	1.121768	-0.813953
Max SY	1	1	0.514779	0.948151	0.000000	**0.000000**	0.000000	1.022040	1.121768	-0.813953
Min SY	1	1	0.514779	0.948151	0.000000	**0.000000**	0.000000	1.022040	1.121768	-0.813953
Max SX	1	1	0.514779	0.948151	0.000000	0.000000	**0.000000**	1.022040	1.121768	-0.813953
Min SX	1	1	0.514779	0.948151	0.000000	0.000000	**0.000000**	1.022040	1.121768	-0.813953
Max M	1182	1	3.509669	-41.619241	0.000000	0.000000	0.000000	**68.636673**	24.623075	-0.748366
Min M	1237	1	2.589893	0.984670	0.000000	0.000000	0.000000	**-30.788416**	-0.870419	-1.726139
Max M	107	1	4.902564	-1.069600	0.000000	0.000000	0.000000	65.682480	**33.505085**	0.631877
Min M	774	1	0.748960	4.190650	0.000000	0.000000	0.000000	-0.676226	**-26.271280**	-2.859145
Max M	211	1	6.616501	-5.318206	0.000000	0.000000	0.000000	-7.500260	-4.694698	**13.465384**
Min M	1158	2	15.832063	5.545430	0.000000	0.000000	0.000000	4.200468	1.543481	**-15.353448**

Displacement | Disp Summary | Reaction | Reaction Summary | Contact Area | Plate Stress | Plate Stress Summary

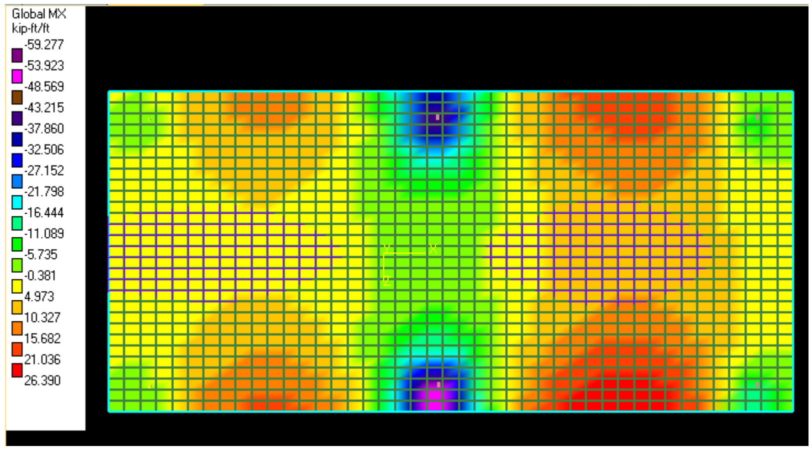
To view plate stress contours please click on the "Output View Options" leaf under "Mat slab design options" group



A form will appear at data area pane. Please select "Show Plate Stress" radio button and then choose "Global Mx" stress type.



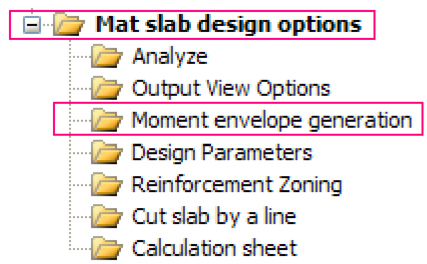
The screen will look like the following figure. Please note that this contour is based on the global X axis.


Global MX
kip-ft/ft
-59.277
-53.923
-48.569
-43.215
-37.860
-32.506
-27.152
-21.798
-16.444
-11.089
-5.735
-0.381
4.973
10.327
15.682
21.036
26.390

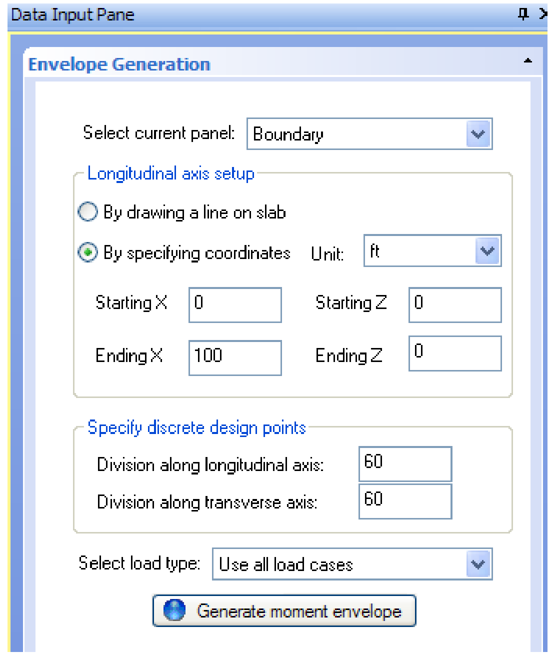
3.15 Slab Design

Now we go for designing the slab. Slab design in STAAD.foundation has three distinct parts. First step is to generate moment envelope. Next step is to design the slab and the last step is to create reinforcement zones for reinforcement layout.

Please click on the ***Moment envelope generation*** leaf under **"**Mat slab design options" group in main navigator pane.

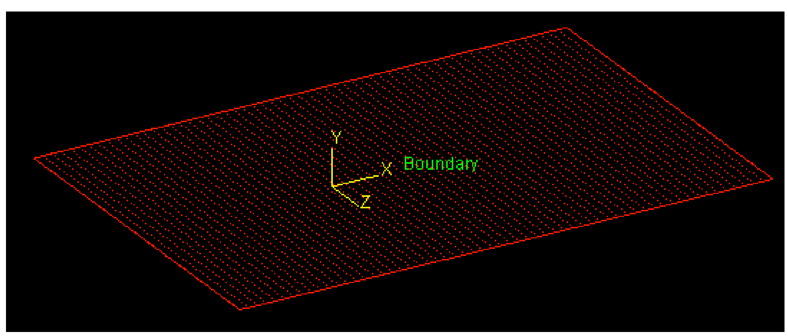


The moment envelope generation dialog box will appear in the *Data Area* pane as shown in the following figure. Here we need to setup slab longitudinal axis. We can setup the axis by simply clicking at two points on the view or typing (x, z) coordinate. As our slab is rectangular and parallel to global axis system we will use Global X axis as slab longitudinal axis. A X axis can be defined by (0,0) and (100,0) coordinates.

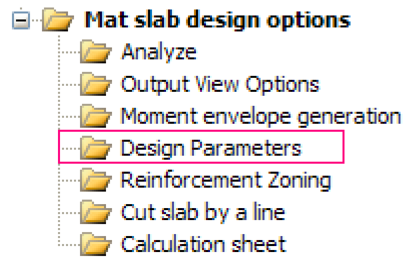


Select "**Use all load cases**" from "**Select load type**" drop down list and click on the "Generate Moment Envelope" button to generate moment envelope.

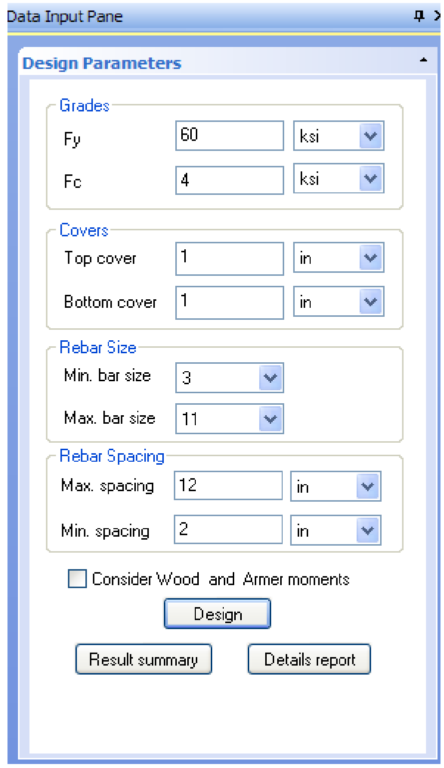
Program will generate a finite number of discrete points which in turn will be used as design points. The design grid appears in the graphics window like the following figure.



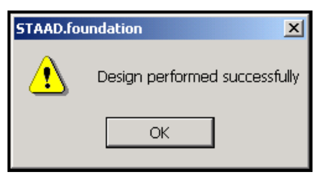
The next step is to set design parameters and design slab. Please click on the "Design Parameters" leaf under "Mat slab design options" in main navigator pane.



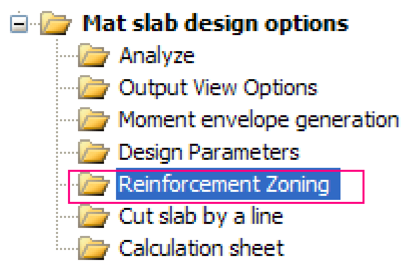
A form with the design parameters will appear in the *Data Area* pane. For this example we will use all default values. Click on the "Design" button to design the slab.



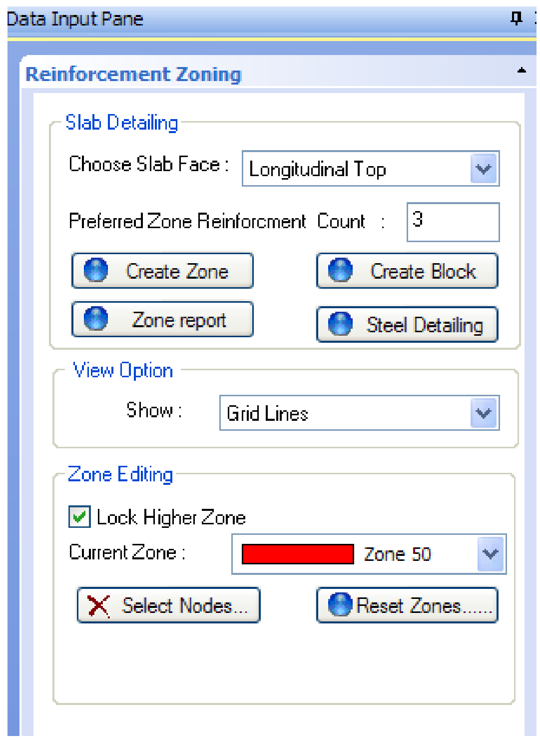
The program will perform the slab design. When the design operation is completed, a message box will appear.



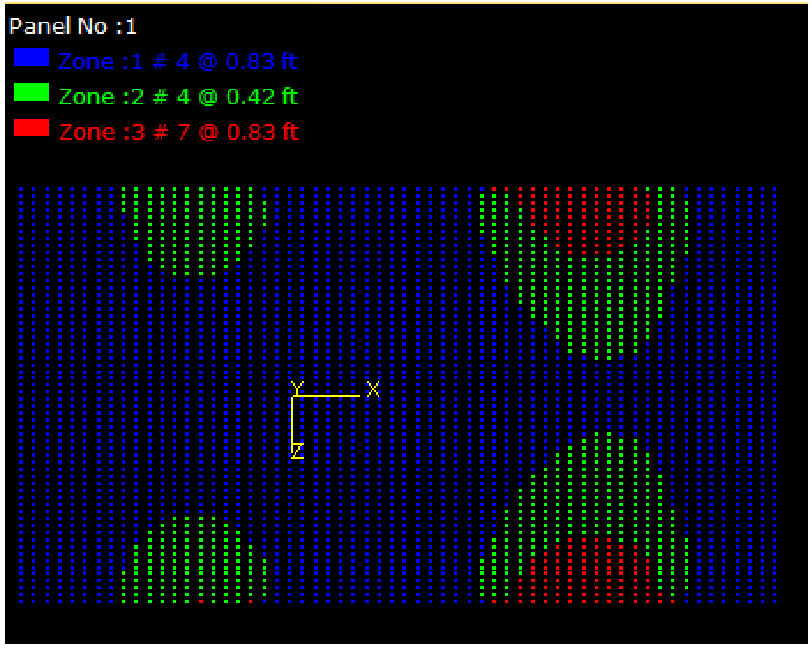
The next step is to create reinforcing zones. Click on the "Reinforcement Zoning" leaf under "Mat slab design options" in main navigator pane.



A form will appear in data area pane which will look like following.

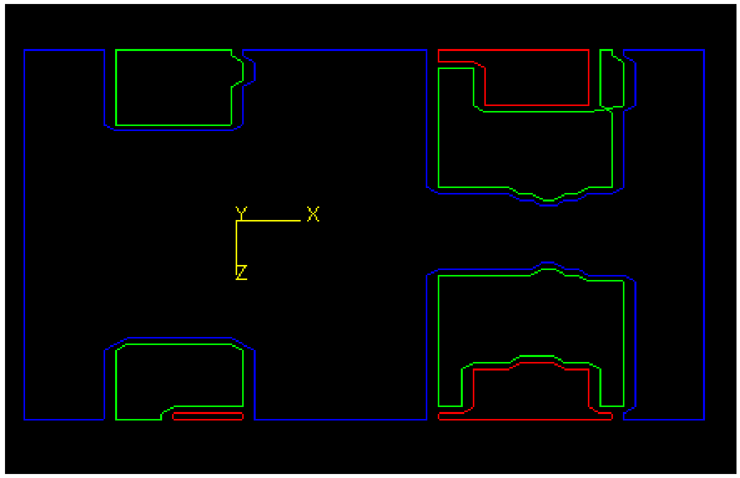


By default, slab face is set as "Longitudinal Top". Use "preferred zone reinforcement count" as 3. Now click on "***Create Zone***" button. The program will plot colored dots on each design point based on required reinforcement. We can call this ***reinforcement contour*** plot.



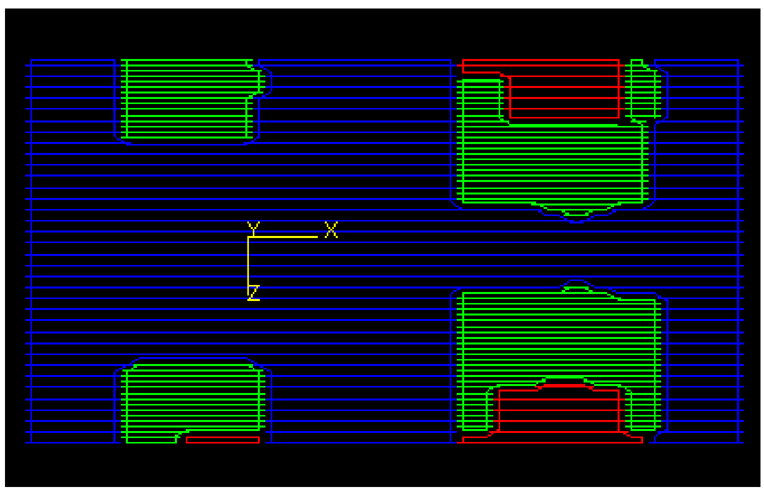
Please note that the slab is divided in three different reinforcing zones where Zone 1 is blue and lowest zone. Often times “Zone 1” represents the minimum reinforcing zone. “Zone 3” is red and represents highest reinforcing zone.

It is evident from the graphics that the reinforcement blocks are not regular shaped polygon. STAAD.foundation has an in-built tool to create a regular shape from this irregular shape. Click on the *Create Block* button. STAAD.*foundation* will divide the slab into block-shaped areas, based on the reinforcement zones generated by the *Create Reinforcing Zones* command.

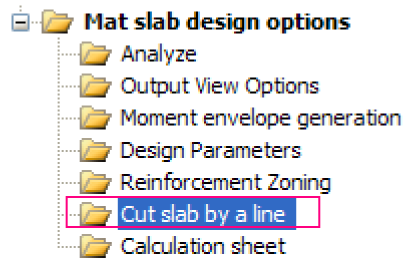


These rectangular areas are created to allow a practical layout of the various sizes of reinforcing steel.

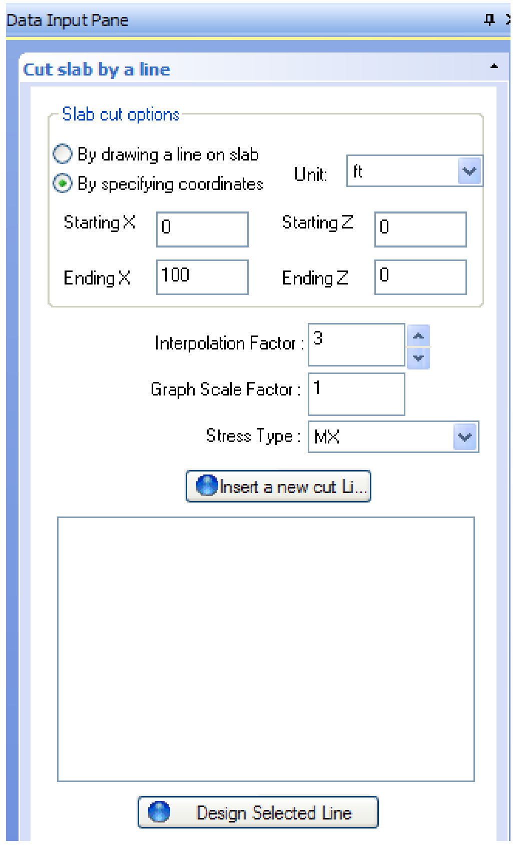
Click on the *Steel Detailing* button. The following screen will come up showing the reinforcement steel details of the three zones in the plan view.



We may also cut the slab by any user-defined line and view your desired stress value (Max absolute/Max VonMises/SX/SY/MX/MY etc.) along that line. Click on the "***Cut slab by a line***" leaf under "Mat slab design options" group in main navigator pane.



A new form will appear in the *Data Area* pane.



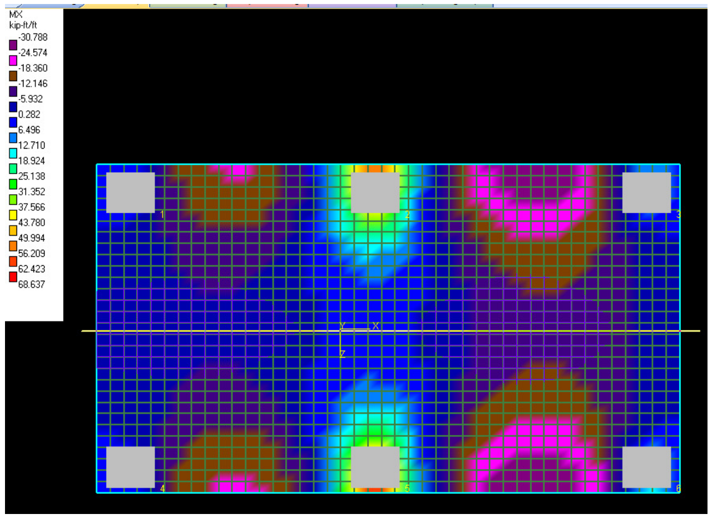
Under the *Slab cut options* category, select the **"By drawing a line on slab"** radio button, and then click in the graphics window. This will allow you to create a section in the plan view of your model.

Enter a value of **3** in the *Interpolation Factor* edit box.

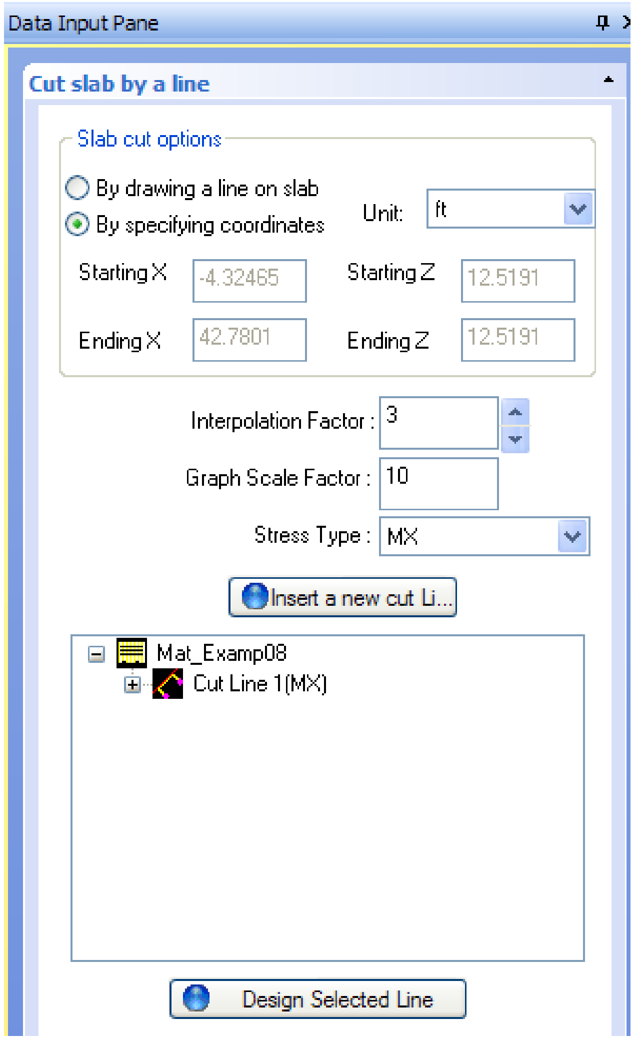
In the *Stress Type* drop-down menu, select **MX** to look for Mx moment along the cut line.

Now, draw a section line on the plan view of your model along which you wish to see the graph of the maximum absolute stress.

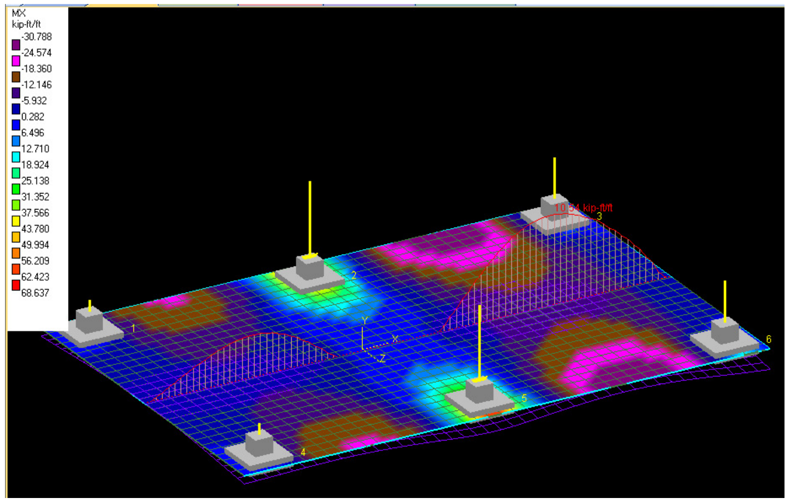
Click your mouse cursor at the beginning point of the line, drag your cursor to the end point, and then click again.



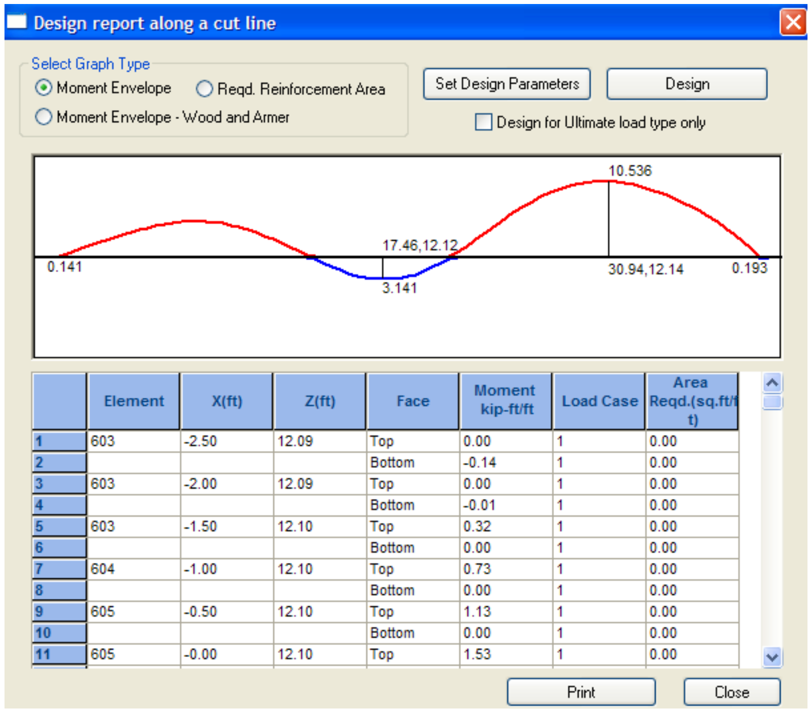
Note that the coordinates of Start and End points now have values. Click on "Insert a new cut line" button. A graph will be shown in the view pane.



Increase graph scale factor to 10 and now the screen will look like following.

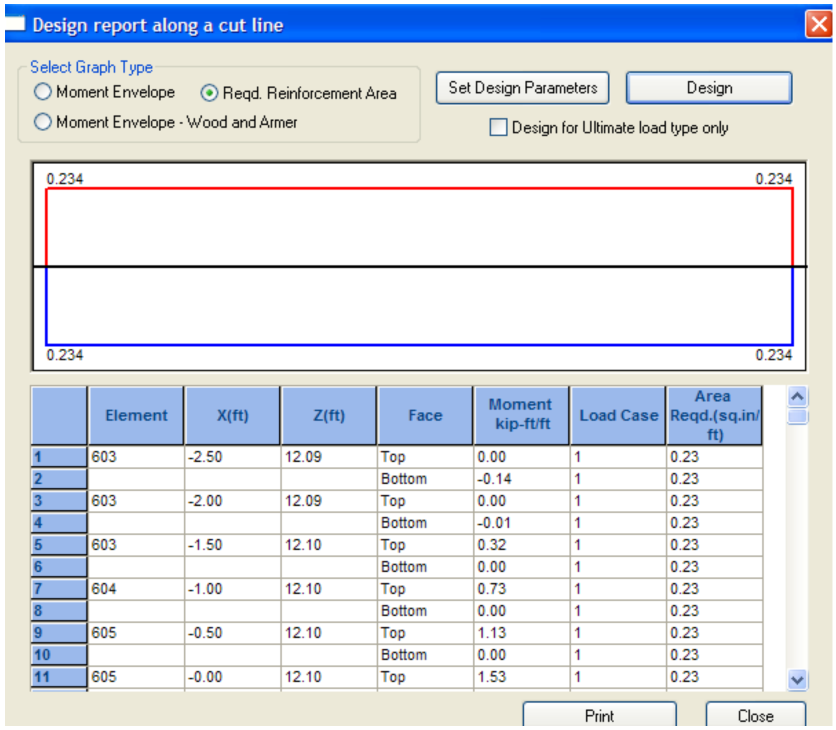

MX
kip-ft/ft
-30.788
-24.574
-18.360
-12.146
-5.932
0.282
6.496
12.710
18.924
25.138
31.352
37.566
43.780
49.994
56.209
62.423
68.637

Now click the button labeled **Design Selected Line**. A dialog box labeled *Design Report Along a Selected Line* will be displayed.



	Element	X(ft)	Z(ft)	Face	Moment kip-ft/ft	Load Case	Area Reqd.(sq.ft/ft)
1	603	-2.50	12.09	Top	0.00	1	0.00
2				Bottom	-0.14	1	0.00
3	603	-2.00	12.09	Top	0.00	1	0.00
4				Bottom	-0.01	1	0.00
5	603	-1.50	12.10	Top	0.32	1	0.00
6				Bottom	0.00	1	0.00
7	604	-1.00	12.10	Top	0.73	1	0.00
8				Bottom	0.00	1	0.00
9	605	-0.50	12.10	Top	1.13	1	0.00
10				Bottom	0.00	1	0.00
11	605	-0.00	12.10	Top	1.53	1	0.00

Click on the **Design** button to calculate the required reinforcement area for each element along the cut line.



	Element	X(ft)	Z(ft)	Face	Moment kip-ft/ft	Load Case	Area Reqd.(sq.in/ft)
1	603	-2.50	12.09	Top	0.00	1	0.23
2				Bottom	-0.14	1	0.23
3	603	-2.00	12.09	Top	0.00	1	0.23
4				Bottom	-0.01	1	0.23
5	603	-1.50	12.10	Top	0.32	1	0.23
6				Bottom	0.00	1	0.23
7	604	-1.00	12.10	Top	0.73	1	0.23
8				Bottom	0.00	1	0.23
9	605	-0.50	12.10	Top	1.13	1	0.23
10				Bottom	0.00	1	0.23
11	605	-0.00	12.10	Top	1.53	1	0.23

Select the **Close** button to dismiss the *Design Report Along a Selected Line* dialog box.

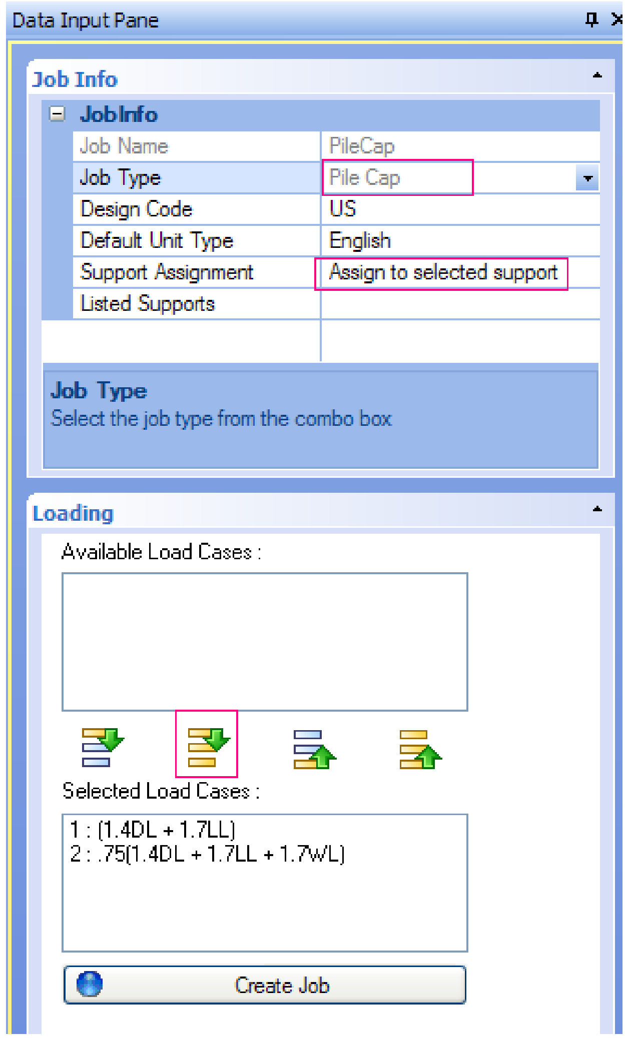
Save your work. Pull down the **File** menu and select the **Save** command.

3.16 Pile Cap Example

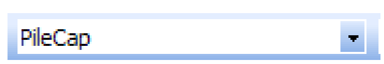
Now let us create a new job inside this same project to illustrate the process for designing a pile cap.

Click on the "Create a new Job" leaf under "Job Setup" group in main navigator pane. The *Create a New Job* form will open in data area pane. Enter job name as "PileCap". Choose Job type as "Pile Cap" and design code as US. Select support node "**1**" in main view. Support assignment type will be automatically switched to "Assign to selected support".

Transfer both load cases to "Selected load cases" by clicking  button. Now click on "Create Job" button to create a new job to design footing 1 as pile cap.

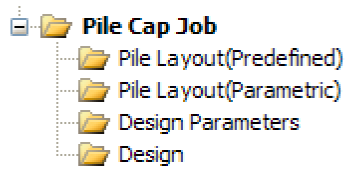


Enter a name for the job in the *New Job Name* edit box. Notice that your New Job Name now appears in the drop-down menu in the Jobs toolbar.

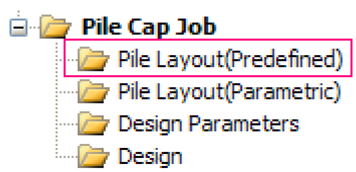


3.17 Entering Pile Data

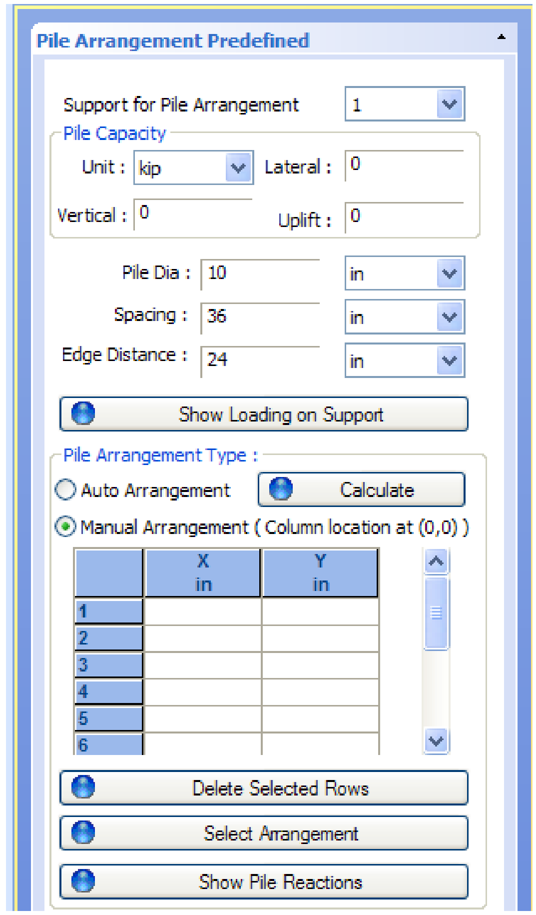
Also, please note main navigator tree is now changed with pile cap related controls. For pile cap jobs, a unique group called *Pile Cap Job* will be created in the *main navigator pane*.



We first need to create pile arrangement for pile cap. To create pile arrangement please click on the "Pile Layout(Predefined)" leaf.



The Pile Arrangement page will be displayed in the *Data Area* pane as shown in following figure.



The combo box labeled *Support for Pile Arrangement* lists the support numbers in the pile cap job. We will select the support number and input the vertical, lateral and uplift pile capacities for each support. The pile diameter, spacing and distance of the edge from the corner piles are also input.

Let us input data for Support No. 1. Leave the *Support for Pile Arrangement* drop-down menu set to **1**.

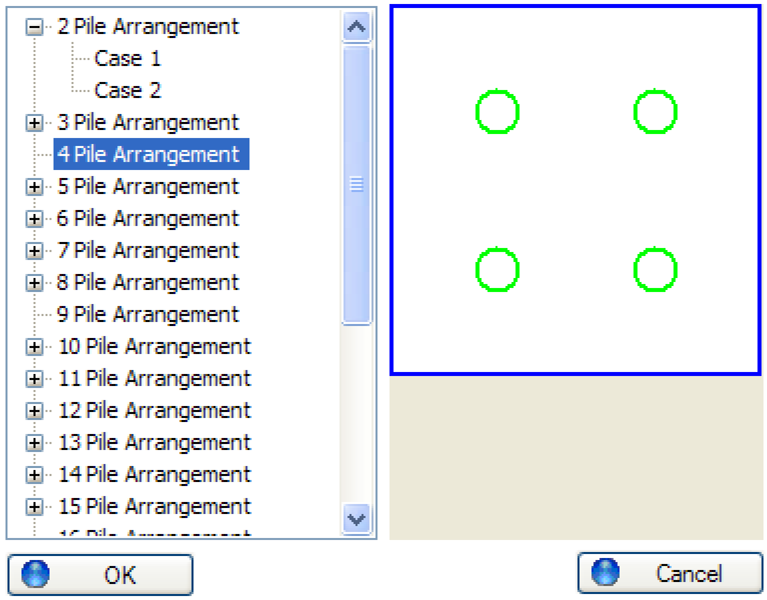
Under the *Pile Capacity* category, set the **Unit** drop-down menu to **kip**. Enter a value of **60** kips in the **Vertical** edit box, and a value of **40** kips in the **Lateral** and **Uplift** edit boxes.

Enter a value of **10 in**. for the **Pile Dia**. Enter a value of **36 in**. for the **Spacing**. Enter a value of **24 in**. for the **Edge Distance**.
The total loading on the support is shown if we click on the *Show Loading On Support* button.

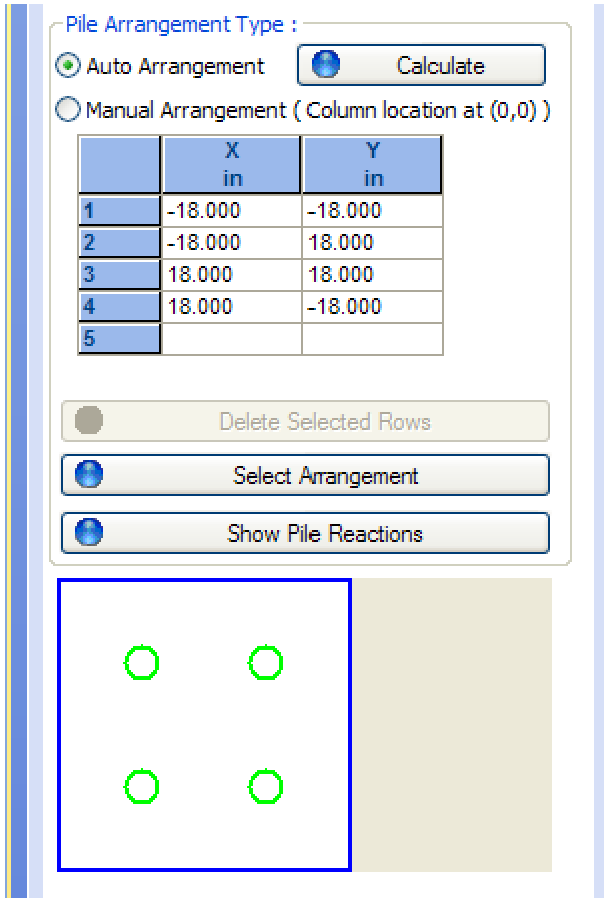
Load Table for Support No. 1

Load Case	Fx kip	Fy kip	Fz kip	Mx kip-in	Mz kip-in
1	-4.058242321	-68.03285980	-0.696796894	-33.31498336	209.5984191
2	-2.699549674	-47.24615097	5.542748928	450.1890258	130.1803131

We may either chose the *Auto Arrangement* or we may go for a manual input of the co-ordinates of the piles. If we choose **Auto Arrangement** and click on the **Calculate** button, all possible pile arrangements corresponding to the pile loads in all the load cases are shown according to the BOCA standard. Please note the program automatically calculates all possible arrangements which satisfy pile capacity criteria.



We go through the tree controls and choose the arrangement suitable to us. For this example we will choose "4 pile arrangement". As we click on the **OK** button, the diagram showing the pile arrangements is transferred to the dialog box in the *Data Area* pane showing the input for pile arrangements. Please notice that the pile co-ordinates have come up in the corresponding table.



If we click on the button for "Show Pile Reactions", the reaction on each pile shows up.

Pile Reaction Table for Support ...

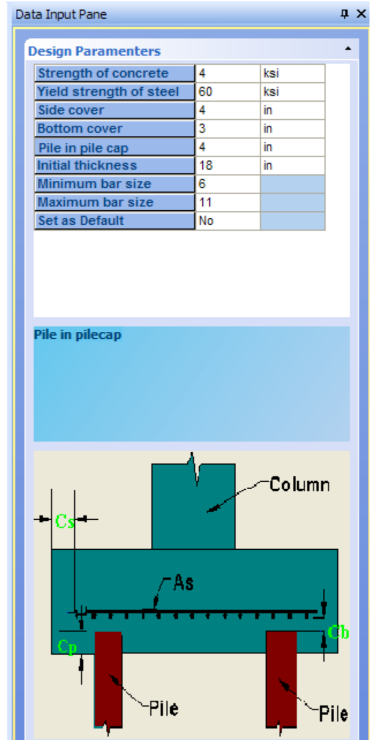
	Vertical kip	Lateral kip	Uplift kip
Load Case = 1			
1	-19.457	1.029	0.000
2	-20.382	1.029	0.000
3	-14.560	1.029	0.000
4	-13.634	1.029	0.000
Load Case = 2			
1	-19.872	1.584	0.000
2	-7.367	1.653	0.000
3	-3.751	1.507	0.000
4	-16.256	1.430	0.000

Depending on the pile arrangement diagram we may decide whether to go on with this arrangement or not. If we decide not to go on with the arrangement we would again click on *Calculate*. Otherwise we click on the *Select Arrangement* button to select the arrangement.

3.18 Entering Pile Cap Design Parameters

After the pile arrangement is selected, the design for the pile cap is begun. The form for input of design parameters is invoked by clicking the **Design Parameters** leaf under "Pile Cap Job" group in main navigator pane.

The *Design Parameters* page will be displayed in the *Data Area* pane.

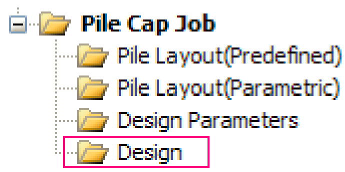


Let us accept the default parameters provided by the program. Check to make sure that the default values displayed on the *Design Parameters* form in your program match those shown in the figure above:

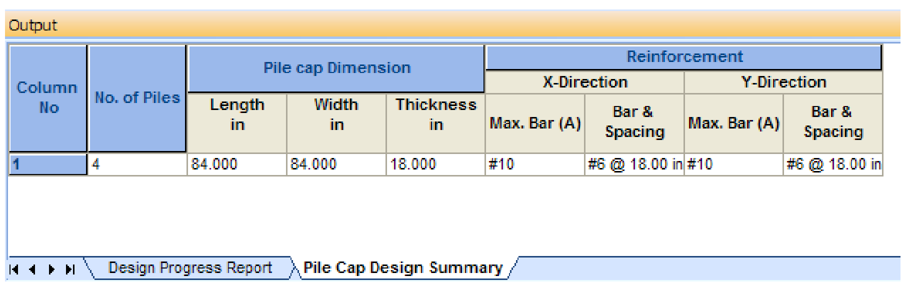
Strength of Concrete: 4 ksi
Yield Strength of Steel: 60 ksi
Minimum Bar Size: 6
Maximum Bar Size: 11
Side cover (Cs): 4 in
Bottom Cover (Cb): 3 in
Pile in Pile Cap (Cp): 4 in.
Initial thickness: 18 in

3.19 Performing Pile Cap Design and Viewing Results

Now that the design parameters are entered, we are ready to perform the design. Click on the **Design** leaf under "Pile Cap Job" group in main navigator pane to perform the design.



The program performs the pile cap design. When it is finished, a results table appears in the *Output* pane showing the pile cap dimensions and the bar size and spacing in the longitudinal and transverse directions.



Output

Column No	No. of Piles	Pile cap Dimension			Reinforcement			
					X-Direction		Y-Direction	
		Length in	Width in	Thickness in	Max. Bar (A)	Bar & Spacing	Max. Bar (A)	Bar & Spacing
1	4	84.000	84.000	18.000	#10	#6 @ 18.00 in	#10	#6 @ 18.00 in

Design Progress Report | Pile Cap Design Summary

The figure above shows results for only one of the six supports in the project because pile arrangements were selected for only support 1.

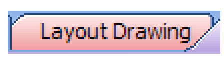
Program will automatically open Calculation sheet of the designed pile cap as shown below.

Check for Two-Way Shear

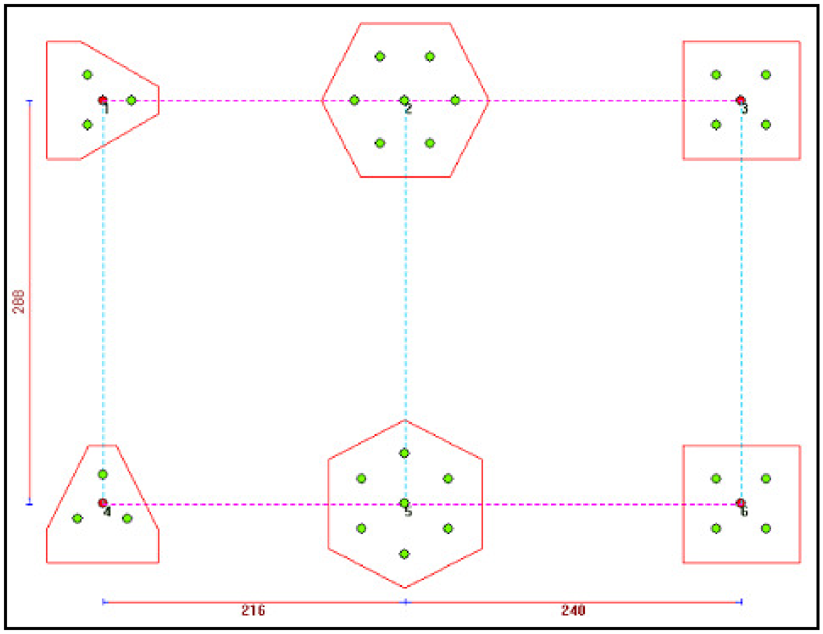
Pile No.	Two-way Shear (kip)
1	-19.457
2	-20.382
3	-14.560
4	-13.634
TOTAL	-68.033

Design Shear for Two-Way Action	$S_t =$	-68.033	kip
Beta =	$\frac{C_L}{C_W} =$	1.000	
$B_0 =$	$2(C_L + C_W + 2d) =$	92.000	
Equation 11-33 : $V_{C1} =$	$B_0 d\left(2 + \frac{4}{beta}\right)\sqrt{F_c} =$	384.027	kip
Equation 11-34 : $V_{C3} =$	$B_0 d\left(2 + \frac{40d}{B_0}\right)\sqrt{F_c} =$	434.117	kip
Equation 11-35 : $V_{C2} =$	$4B_0 d\sqrt{F_c} =$	256.018	kip
$V_C =$	minimum of $(V_{C1}, V_{C2}, V_{C3}) =$	256.018	kip
	S_t <= 0.75 V_C...	hence, safe	

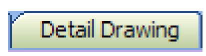
Click on the **Layout Drawing tab** to view layout drawing**.**



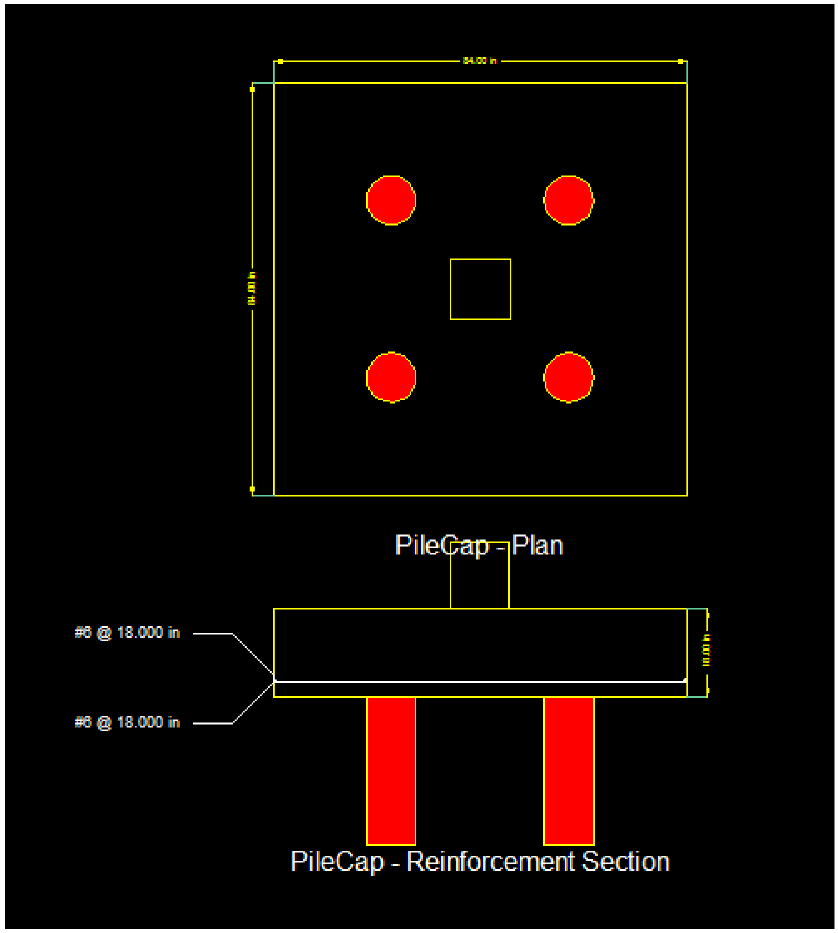
The layout of the pile cap job will show the different supports along with the pile arrangements and number of piles pertaining to them.



Click on the **Detailed Drawing** sub-page button.



The following screen appears showing the plan of the pile cap, the details of the reinforcement bar for the pile cap and the front view of the pile cap along with the column for the support selected.



3.20 Exporting Drawings to CAD

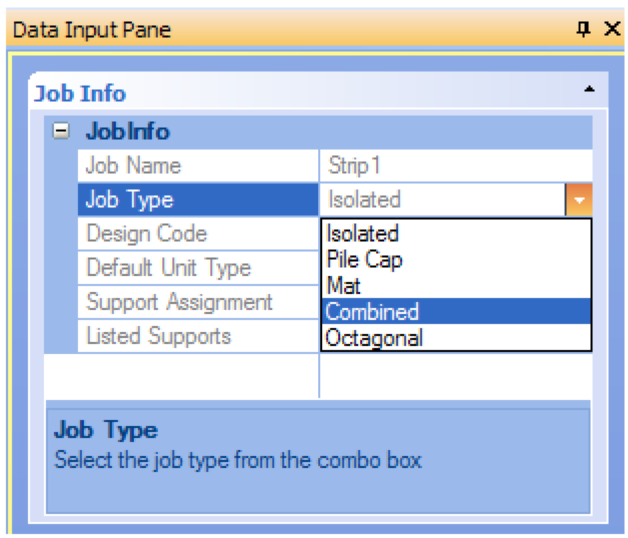
Clicking on the *Save drawing As...* button gives the option to save the drawing in many formats including DXF and DWG.

3.21 Creating Strip Footing Job

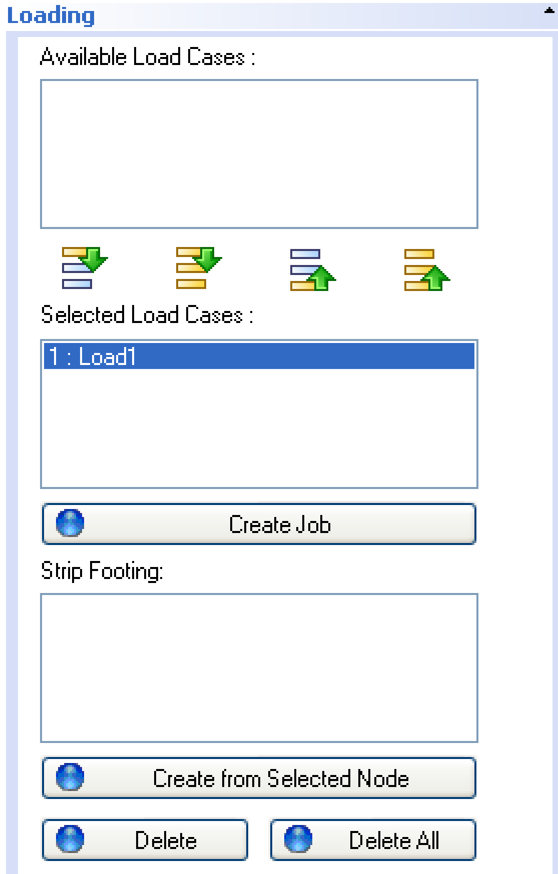
Now let us create a new job inside this same project to illustrate the process for designing a combined footing.

Click on the "Create a new Job" leaf under "Job Setup" group in main navigator pane. The *Create a New Job* form will open in data area pane. Enter job name as "Strip1". Choose Job type as "Combined" and design code as US.

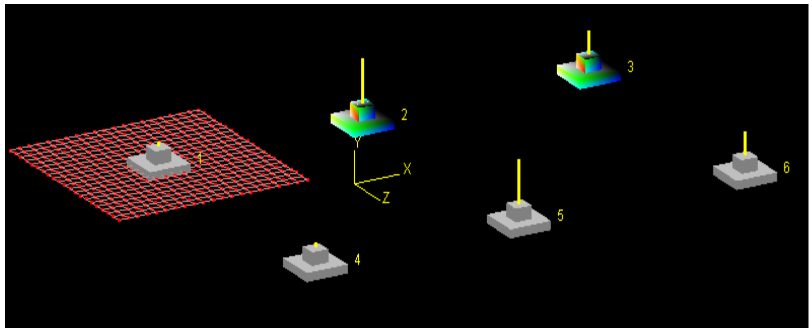
Transfer both load cases to "Selected load cases" by clicking  button. Now click on "Create Job" button to create a new combined footing job.



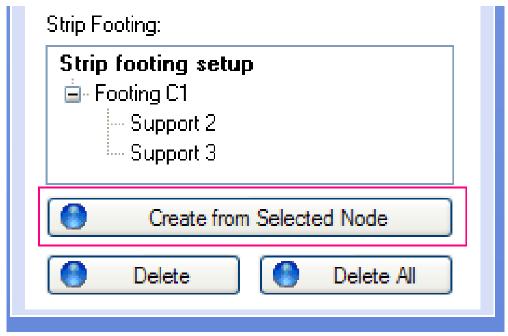
Please note that there are some new controls in the job creation page which will be used to setup and assign strip footing.



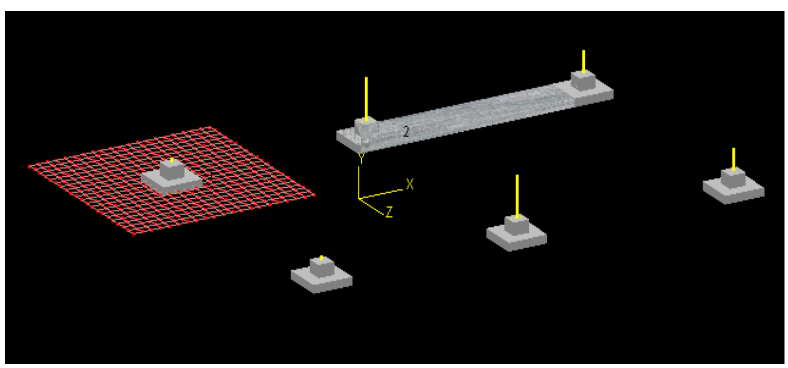
Now, with mouse select node 2 and 3 in view pane. Nodes will be shown as selected as shown below.



Click on the ‘Create from Selected Node’ button. A tree view showing the support assignment will appear.

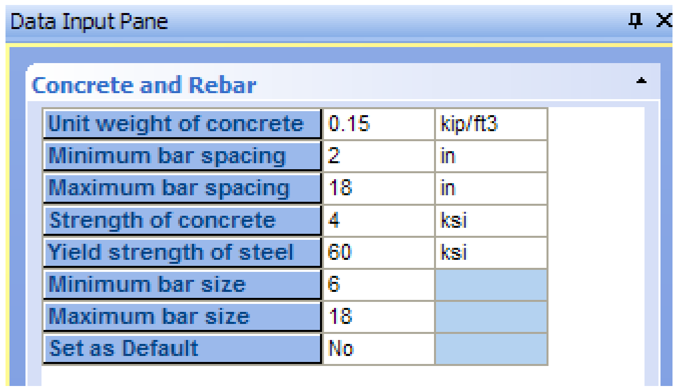


Note the view pane also shows the combined footing.

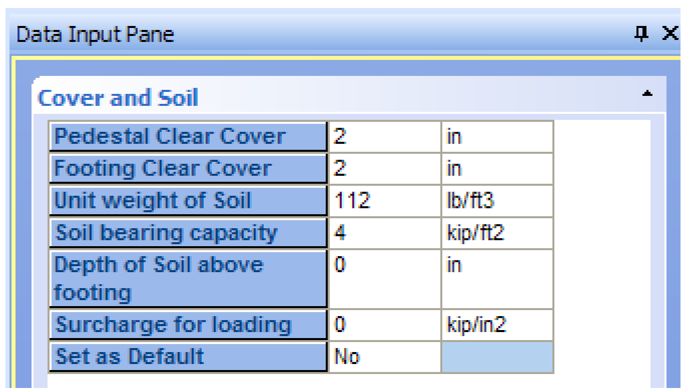


3.22 Strip Footing design parameters

Now we need to input suitable design parameters. The input for 'Concrete & Rebar' & 'Cover & Soil' are as same as isolated footing. We will use default values for those two sections of design parameters as shown below.

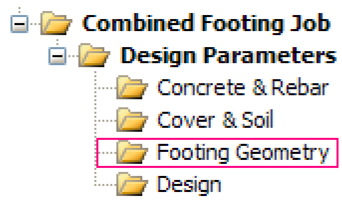


Unit weight of concrete	0.15	kip/ft3
Minimum bar spacing	2	in
Maximum bar spacing	18	in
Strength of concrete	4	ksi
Yield strength of steel	60	ksi
Minimum bar size	6	
Maximum bar size	18	
Set as Default	No	

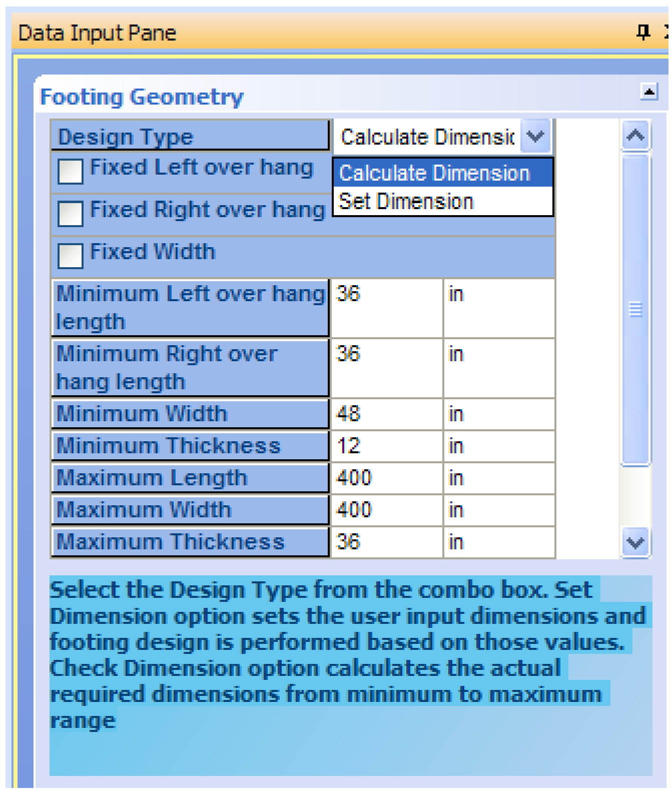


Pedestal Clear Cover	2	in
Footing Clear Cover	2	in
Unit weight of Soil	112	lb/ft3
Soil bearing capacity	4	kip/ft2
Depth of Soil above footing	0	in
Surcharge for loading	0	kip/in2
Set as Default	No	

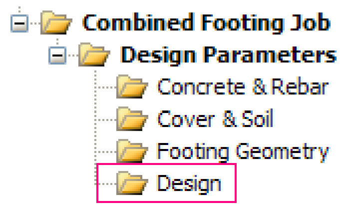
But the footing geometry page for combined footing is unique. Click on the "Footing Geometry" leaf under "Design Parameters" group in main navigator pane.



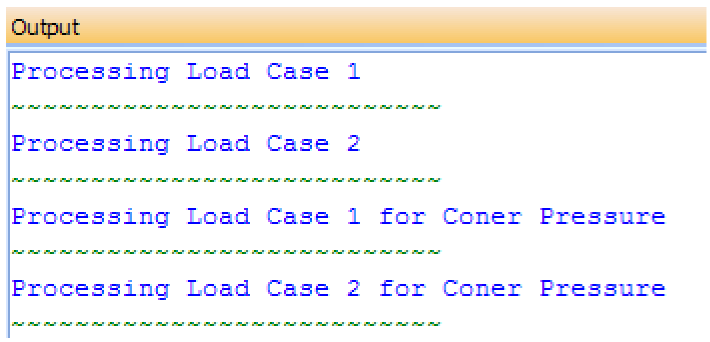
This will bring up geometry page where user has option to limit footing size along length and width. User has option to check or calculate footing dimensions. For this example we will use default values as shown below.



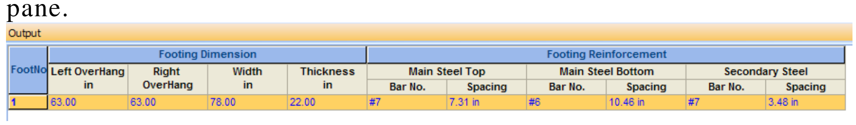
Now click on 'Design' leaf under "Design Parameters" group in main navigator pane to design the combined footing.



Design progress will be shown in Output pane.

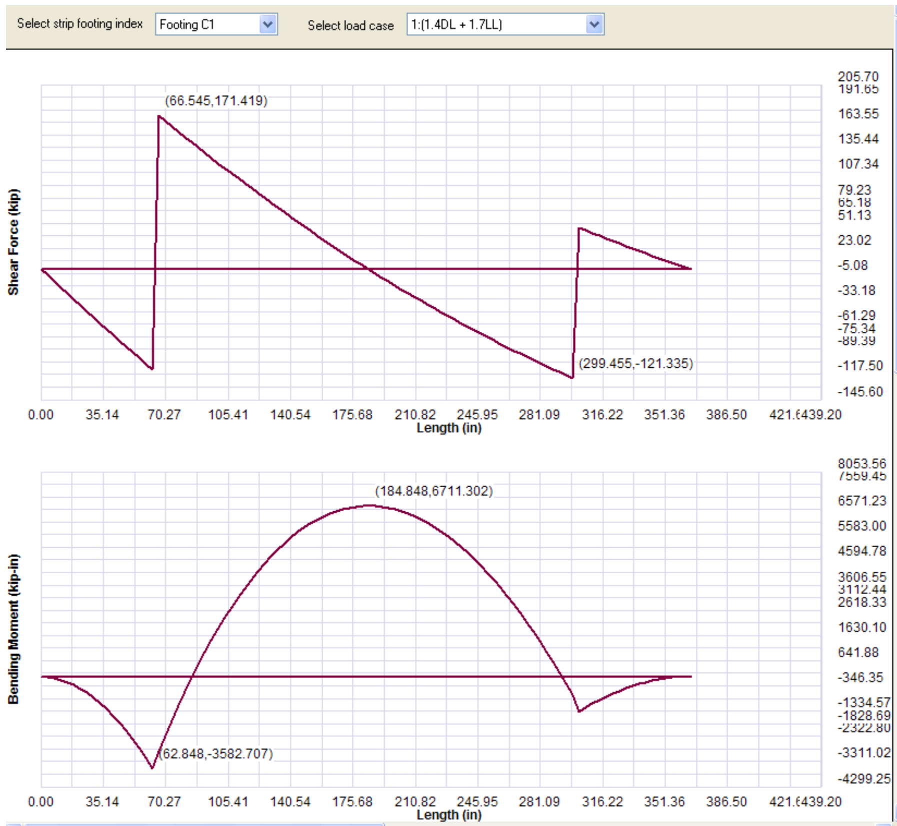


After design is complete a summary table will appear in output pane.


Output

FootNo	Footing Dimension				Footing Reinforcement					
	Left OverHang in	Right OverHang	Width in	Thickness in	Main Steel Top		Main Steel Bottom		Secondary Steel	
					Bar No.	Spacing	Bar No.	Spacing	Bar No.	Spacing
1	63.00	63.00	78.00	22.00	#7	7.31 in	#6	10.46 in	#7	3.48 in

Detail drawing and layout drawing will be shown in corresponding tabs. Other than that, a BM & SF diagram will be generated for the strip footing in the 'Strip Footing Graph' pane as shown below.

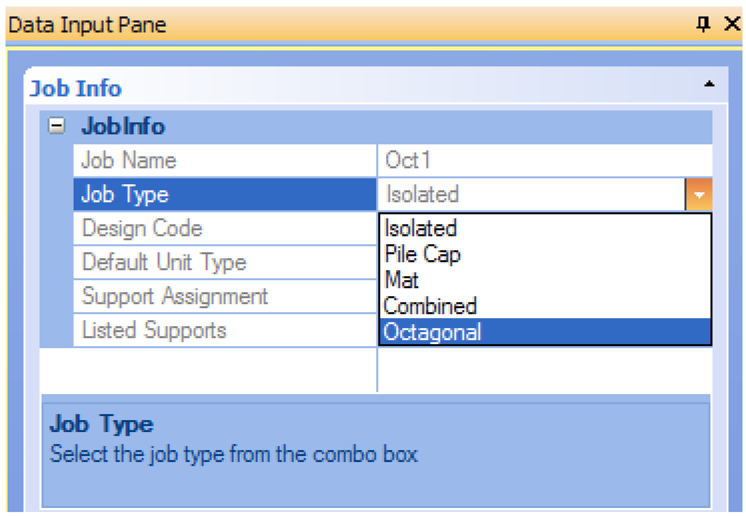


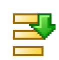
Select strip footing index
Footing C1
Select load case
1:(1.4DL + 1.7LL)
Shear Force (kip)
(66.545,171.419)
(299.455,-121.335)
Length (in)
Bending Moment (kip-in)
(184.848,6711.302)
(62.848,-3582.707)
Length (in)

3.23 Creating Octagonal Footing Job

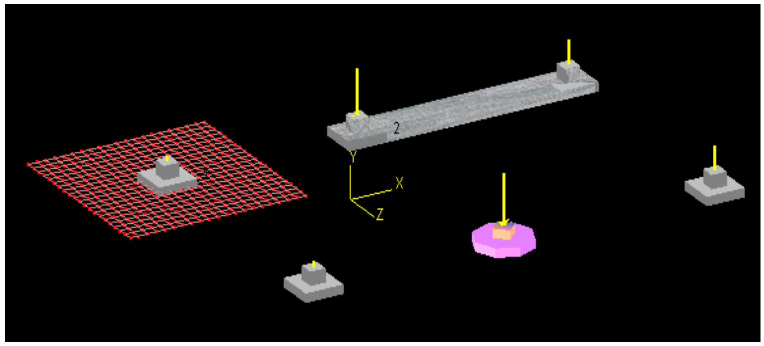
Now let us create a new job inside this same project to illustrate the process for designing an octagonal footing.

Click on the "Create a new Job" leaf under "Job Setup" group in main navigator pane. The *Create a New Job* form will open in data area pane. Enter job name as "Oct1". Choose Job type as "Octagonal" and design code as US. Select support node "5" in main view. Support assignment type will be automatically switched to "Assign to selected support".



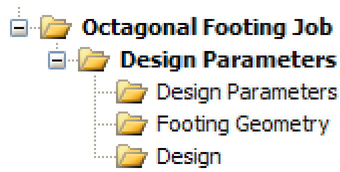
Transfer both load cases to "Selected load cases" by clicking  button. Now click on "Create Job" button to create a new octagonal footing job.

Also note, the main view now shows octagonal shape for support number 5.

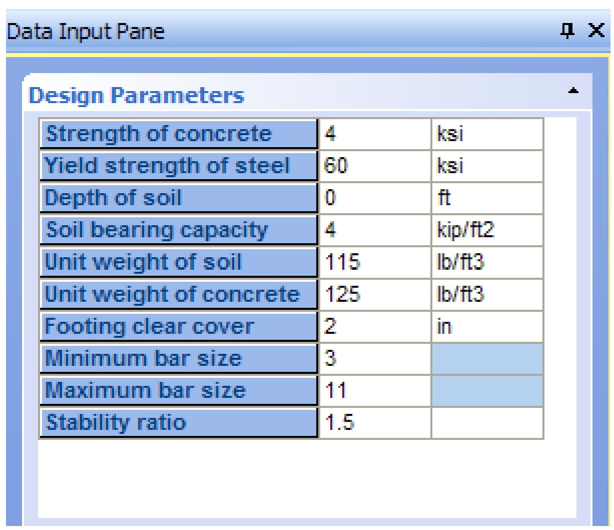

2
Y
X
Z

3.24 Entering Octagonal footing design parameters

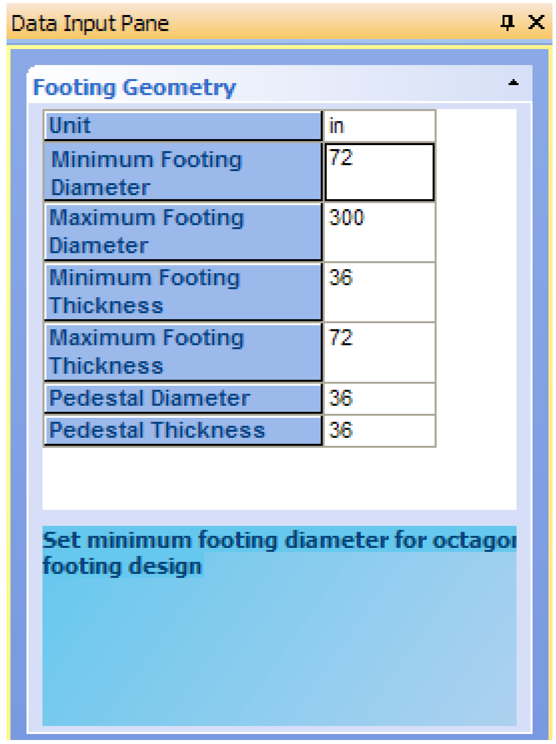
Please note main navigator tree is now changed with octagonal footing related controls. For octagonal footing jobs, a unique group called *Octagonal footing Job* will be created in the *main navigator pane*.



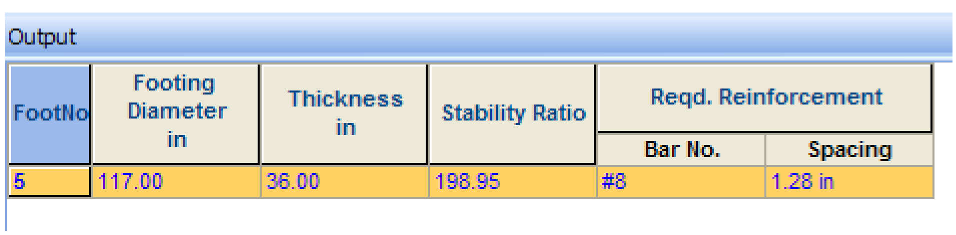
Click on *Design Parameters* leaf under "Design parameters" group to change design parameters. For this example we will use default values as shown below.



Click on *Footing Geometry* leaf under 'Design Parameters" group to change values related to geometry. For this example we will use default values as shown below.



Now click on *Design* leaf to design the octagonal footing. After design is completed a design summary table will be shown in Output pane.


Output

FootNo	Footing Diameter in	Thickness in	Stability Ratio	Reqd. Reinforcement	
				Bar No.	Spacing
5	117.00	36.00	198.95	#8	1.28 in

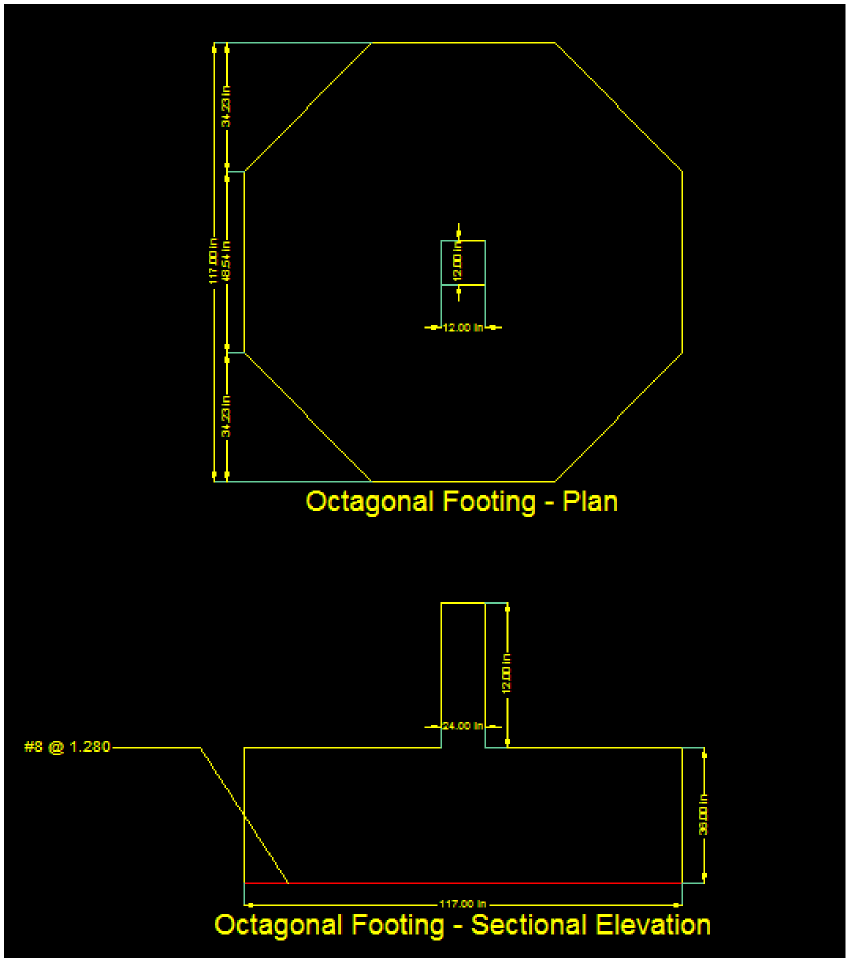
Calculation sheet will be opened automatically as shown below.

Octagonal Foundation 5

Geometrical Description	
Pedestal Geometry	
Unit :	in
Height :	36.000
Diameter :	36.000
Footing Geometry	
Unit :	in
Minimum Footing Diameter :	72.000
Maximum Footing Diameter :	300.000
Minimum Footing Depth :	36.000
Maximum Footing Depth :	72.000

Design Parameters	
Minimum Stability Ratio :	1.500
Soil Depth :	0.000 ft
Cover :	2.000 in
F_c :	4.000 ksi
F_y :	60.000 ksi
Bar Type :	Metric
Minimum Bar Diameter :	0
Maximum Bar Diameter :	8
Concrete Density :	125.000 lb/ft3
Soil Density :	115.000 lb/ft3
Allowable Bearing Pressure :	4.000 kip/ft2

We may now want to visit layout drawing and detailed drawing pages to see those drawings.


34.23 In
117.00 In
48.54 In
34.23 In
12.00 In
12.00 In
Octagonal Footing - Plan
12.00 In
24.00 In
#8 @ 1.280
117.00 In
Octagonal Footing - Sectional Elevation

3.25 Conclusion

We hope you have enjoyed this Quick Tour of STAAD.*foundation's* features and facilities. If you would like additional assistance in learning how to use STAAD.*foundation*, there are many resources available to you. Within the Online Help facility, you will find documentation describing the program theory and a detailed description of every command in the program. You may also view a number of animated movie files that demonstrate how to perform various tasks.

Additional STAAD.*foundation* learning resources are available at Bentley Systems, Inc. web site at http://www.bentley.com/en-US/Products/STAAD.foundation/.

Finally, we strongly encourage you to take advantage of Bentley's technical support service. Our support staff is most eager and willing to help you learn to use the program correctly.
You may contact our STAAD.*foundation* technical support staff by sending e-mail to the following address:
support@reiusa.com

Write down your questions and attach your STAAD.*foundation* project file, if you think it would be helpful (the STAAD.*foundation* project file is appended with the extension AFS – the current input file name always appears in the title bar at the top of your STAAD.*foundation* program window). Most technical support e-mails are answered the same day they are received.
Thank you for purchasing STAAD.*foundation*. We hope you enjoy using the program and hope that it adds value and efficiency to your engineering endeavors. If you have any comments regarding the program, or suggestions on how it could be improved to better serve your needs, we would very much like to hear from you.

N
o
t
e
s

STAAD.*foundation* Graphical Environment

Section 4

General Foundation

This section includes discussion on the following topics:

- Introduction
- Screen Organization (GUI)
- Navigator Controls
- Menu Commands
- Toolbars

4.1 Introduction

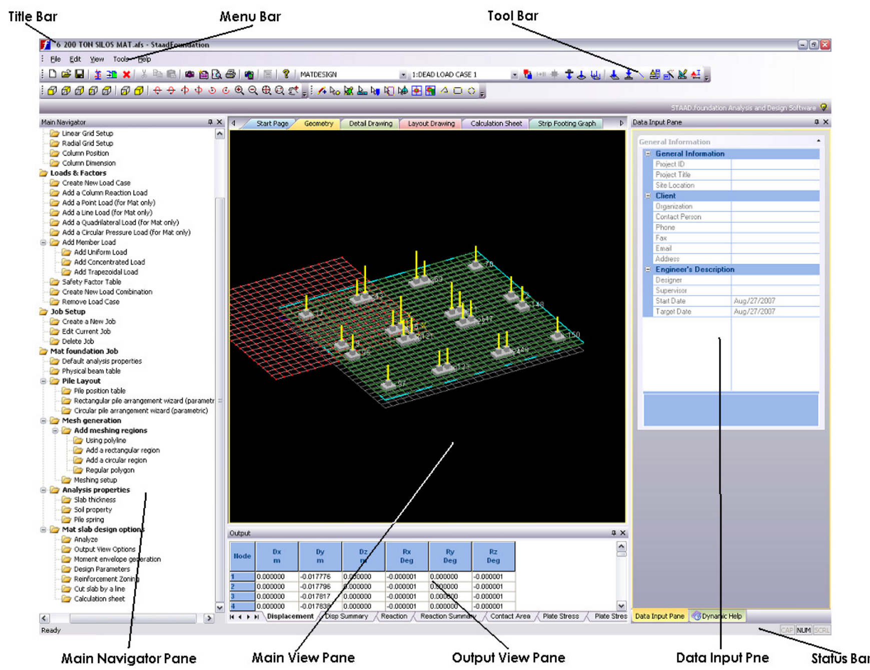
This section provides an overview of STAAD.*foundation's* graphical user interface (GUI). STAAD.*foundation* combines the menu-driven functionality of the Windows environment with the user-friendly split window functionality available in programs like Microsoft Outlook.

In STAAD.*foundation,* you start out by creating a *Project* to hold physical information, such as column locations, column dimensions, piles, beams, and loads. The physical information represents the structure that a foundation is intended to support. Unless the design of the structure is modified, these physical conditions generally remain constant throughout the life of a foundation design project. Your project also contains *Jobs*, which are sets of constraints needed to tell STAAD.*foundation* how to perform a foundation design. Each project may contain multiple jobs, making it easy for you to evaluate different design scenarios for a given set of physical conditions.

Once a project is created, it can be saved and re-opened later using the *File* | *Save* and *File* | *Open* menu commands. Project files are saved with an .afs extension.

4.2 STAAD.*foundation* Screen Organization

STAAD.*foundation's* GUI uses a split window interface. The split window interface divides the screen into three panes: the *Page Control* pane, the *Data Area* pane, and the *Main View* or *Graphics Window* pane. In addition to the three panes, the interface also contains a *Title Bar*, *Menu Bar*, *Toolbar*, and *Status Bar*. The seven elements of the STAAD.*foundation* GUI are identified in the figure below.



Title Bar

Located at the top of the screen, the *Title Bar* displays the file name of the project that is currently open & active.

Menu Bar

Located just below the *Title Bar*, the *Menu Bar* gives you access to all the facilities of STAAD.*foundation*. Many of the same functions are also available in the *Toolbar* and *Page Control* pane.

Users who are familiar with STAAD.*foundation* and its commands usually find that the *Menu Bar* is the most efficient way to quickly access the commands they need.

A complete description of the *Menu Bar* commands is provided in Section 4.4 of this manual.

Toolbar

Located below the *Menu Bar* by default, the dockable *Toolbar* gives you access to the most frequently used commands. Each button in the toolbar offers *Tool Tip* help. As you move the mouse cursor over a button, the name of the button – called a *Tool Tip* – appears above or below the button. To control the appearance of the toolbar or create your own customized toolbar, use the *View* | *Toolbar* menu command. To control the appearance of *Tool Tips*, use the *View* | *Tool Tip Options* menu command.

A complete description of the *Toolbar* is provided in Section 4.5 of this manual.

Main Navigator Pane

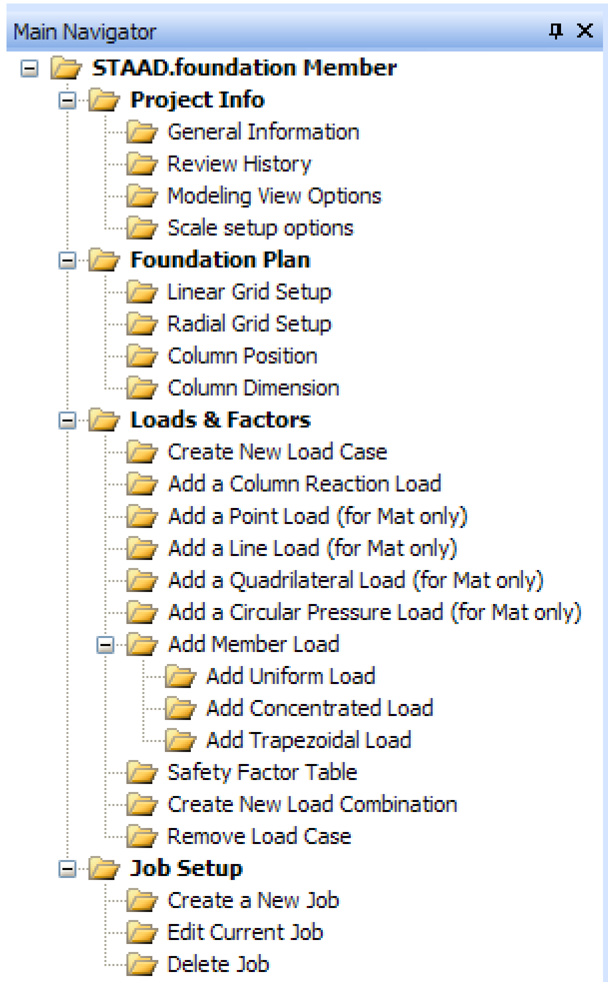
Located at the left side of the screen, the *Main Navigator* pane is a tree control. Clicking on any leaf of the tree opens a new page on the *Data Input* pane that allows you to perform specific tasks for a project. The organization of the *Main Navigator* leaves, from top to bottom, represents the logical sequence of operations for a project, such as definition of footing positions, specification of loads and factors, mesh generation, and so on. If you go through the pages from top to bottom and enter all the data that is relevant to a project, you will end up with a successful model.

A detailed discussion of the facilities included in the *Main Navigator* pane is provided in Section 4.3 of this manual.

Data Input Pane

Located in the right of the screen, the *Data Input* pane is where you enter all relevant data for a project. The *Data Input* pane contains different forms, dialog boxes, tables, and list boxes depending on the type of operation you are performing. For example, when you click on the *Column Dimension* leaf, a *Column Dimension* table is displayed in the *Data Input* pane; when you click on the *Plate Thickness* button, the contents of the *Data Input* pane change to display the *Plate Thickness* dialog box.

4.3 The Navigator Controls



The *Main Navigator* pane handles the program flow and display of forms, tables, dialog boxes etc. for entering your project data. It is organized in a logical order, allowing you to complete a project by working from the top to the bottom.

It's primarily a tree control where the whole tree is divided in several groups. The basic division is *Global* and *Local* data. Information which will be used all through the project is called global data. Column positions, column dimension and loading are global data.

Data related to specific type of job like *Isolated* footing are known as local data. Design parameters, footing geometry are examples of local data.

The *Main navigator* pane represents all the major steps required to complete a foundation project. The tree leaves under any group on navigational tree manage the display of forms that appear in the *Data Area* pane.

4.4 Global Data

The main navigator pane ***Global Data*** groups are as followings

- Project Info
  - General Information
  - Review History
  - Modeling view options
  - Scale setup options

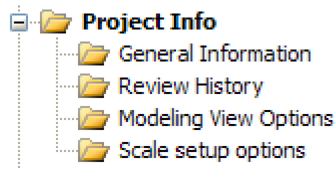
- Foundation Plan
  - Linear grid setup
  - Radial grid setup
  - Column Position
  - Column Dimension

- Loads and Factors
  - Create a new load case
  - Add a Column Reaction Load
  - Add a Point Load (for Mat only)
  - Add a Line Load (for Mat only)
  - Add a Quadrilateral Load (for Mat only)
  - Add a Circular Pressure Load (for Mat only)
  - Add Member Load
    - Add Uniform Load
    - Add Concentrated Load
    - Add Trapezoidal Load
  - Safety Factor Table
  - Create New Load Combination
  - Remove Load Case

- Job Setup
  - Create a New Job
  - Edit Current Job
  - Delete Job

Note: STAAD.*foundation* does not display the entire tree all the time. Rather, it only displays groups and leaves that are relevant to the current status of the project. For example, when you begin a new project, only the *Project Info*, *Foundation Plan, Loads and Factors and Job Setup group*s will appear in the *Navigator* pane. These four groups allow you to specify the physical model upon which the foundation design is performed. It is only when you specify a *Job* (a set of constraints for STAAD.*foundation* to use in performing a foundation design) that the *groups* related to the design will appear.

4.4.1 The Project Info Group



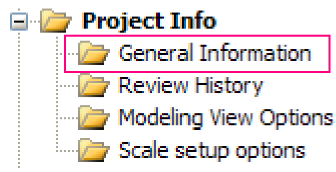
The *Project Info* group allows you to input general info about a project, as well as create a revision history for a project. The *Project Info* group is active by default when you open a new or existing project. Using this group you can scale objects for better visibility or can switch on/off objects as needed.

Use of the *Project Info* group is optional. It is provided for your convenience. You can store relevant general information regarding a project and also create a revision history. Later on, you can instruct STAAD.*foundation* to display this information in reports and drawings.

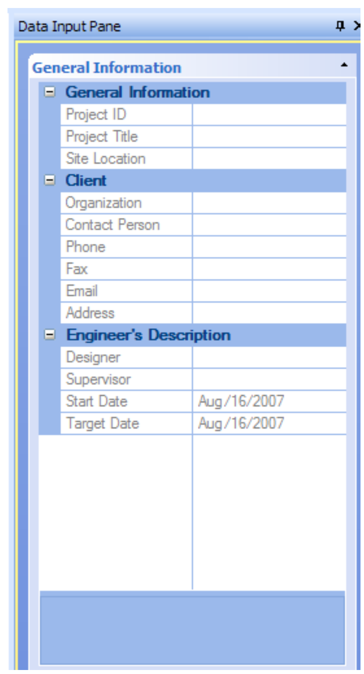
The *Project Info* group contains the following elements:

- General Information
- Review History
- Modeling View Options
- Scale Setup Options

4.4.1.1 General Info



The *General Information* leaf opens a form in the *Data Area* pane that allows you to store general information regarding a project. The information you input in the *General Information* form can later be used in reports and drawings



The *General Information* form contains the following three groups of information:

- General Information
- Client
- Engineer's Description

General Information

The fields contained in this group box allow you to input an *ID*, *Title*, and *Site Location* for a project.

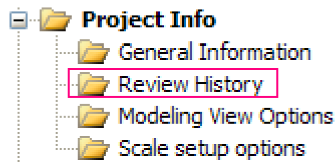
Client

The fields contained in this group box allow you to input information pertaining to the client of a project such as *Organization*, *Contact Person*, *Phone*, *Fax*, *E-mail*, and *Address*.

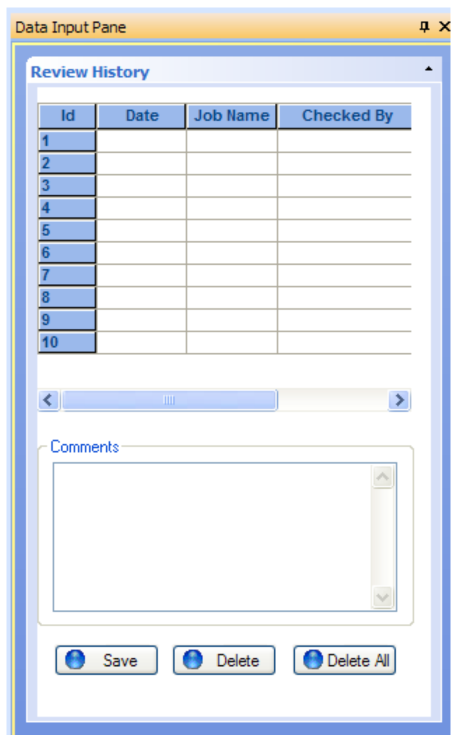
Engineer's Description

The fields contained in this group box allow you to input information pertaining to the engineer of a project such as *Designer*, *Supervisor*, *Start Date*, and *Target Date*.

4.4.1.2 Review History



Clicking on *Review History* leaf opens a review history form in *Data Area* pane that allows you to keep track of the progress of a project.



The *Review History* form allows you to input a *Date*, *Job Name*, *Checked By Name*, and *Comments* for each revision of a project. Each new revision is given a unique *ID Number*, starting from 1.

To add a revision, first input the information for *Date*, *Job Name*, and *Checked By*. Then input any comments about the revision in the *Comments* field. Finally, click on the *Save* button to keep the changes you have made. To view the *Comments* for a given revision, select the revision from the table.

The *Review History* form contains the following three commands buttons:

- Save
- Delete
- Delete All.

Save

The *Save* button saves any changes made to the revision table and comments field.

Delete

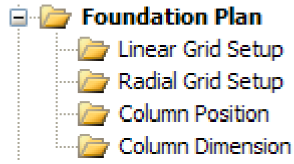
The *Delete* button removes the currently selected revision from the revision table.

Delete All

The *Delete All* button removes all revisions from the revision table.

Note: Deleting a revision from the revision table also deletes the *Comments* that were stored with the deleted revision.

4.4.2 The Foundation Plan Page

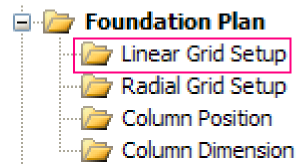


The *Foundation Plan* page allows you to specify basic information on support, such as *Column Positions*, *Column Dimension*. It also allows creating a grid to be used for defining column position, pile position, mat boundary etc.

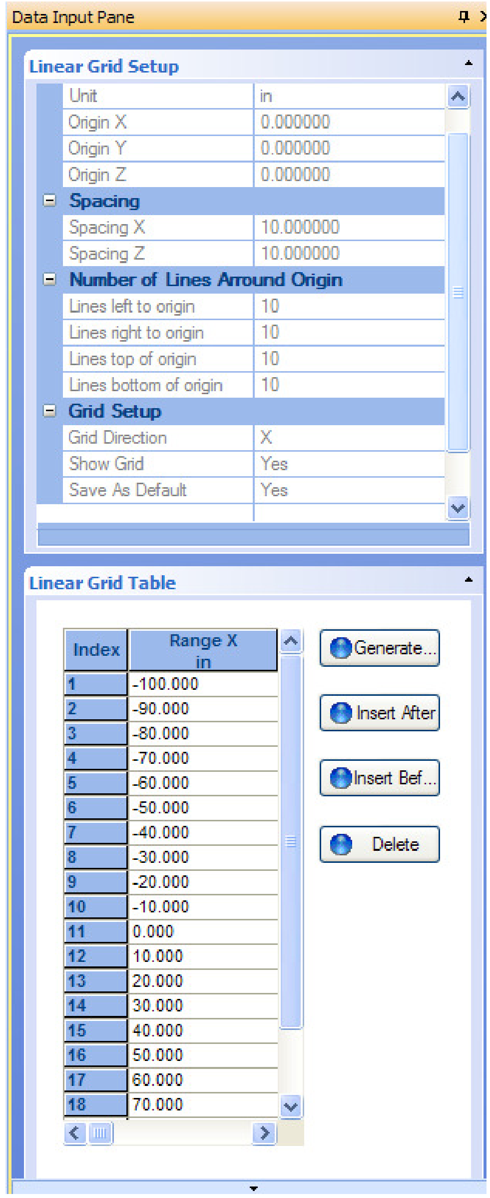
The *Foundation Plan* page contains the following sub-pages:

- Linear Grid Setup
- Radial Grid Setup
- Column Positions
- Column Dimensions

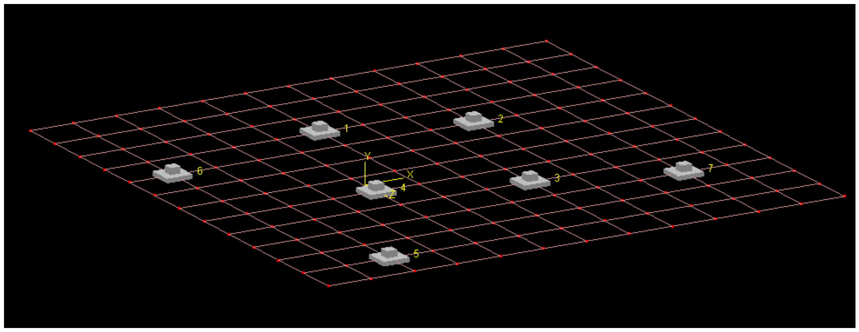
4.4.2.1 Grid Setup



Clicking on the *Linear Grid Setup* leaf opens a form in the *Data Area* pane that allows you to define a linear grid which will be displayed in the *Graphics Window* for you to create foundation geometry on.



You may use the form to draw a grid in the *Graphics Window*. The grid allows you to specify your foundation geometry by snapping to the intersections of the grid lines. You can control the location of the grid origin with respect to the global coordinate system. You can also specify the number of grid lines, and the spacing between lines. The grid lines may be spaced equally apart, or you can specify the spacing of each individual grid line.



Linear

The *Linear Grid* form allows you to create a linear grid.

Grid Origin

The *Grid Origin* group box allows you to specify the origin of the grid.

<u>Unit</u>

The Unit field allows you to select the current length unit of the grid system. You can change the unit by clicking on that field and selecting desired unit from the drop down list.

<u>Origin X</u>

The *Origin X* field allows you to specify the X-coordinate of the grid origin.

<u>Origin Y</u>

The *Origin Y* field allows you to specify the Y-coordinate of the grid origin.

<u>Origin Z</u>

The *Origin Z* field allows you to specify the Z-coordinate of the grid origin.

Spacing

The *Spacing* group box allows you to specify the spacing between grid lines. The unit measurement used is specified in the *unit* field above in the *Grid Origin* group box.

<u>Spacing X</u>

The *Spacing X* field allows you to specify the spacing between grid lines along the X-axis.

<u>Spacing Z</u>

The *Spacing Z* field allows you to specify the spacing between grid lines along the Z-axis.

Number of Lines around Origin

The *Number of Lines around Origin* group box allows you to specify the number of grid lines to the left, right, top, and bottom of the grid origin.

Left

The *Left* field allows you to specify the number of grid lines to the left of the grid origin.

Right

The *Right* field allows you to specify the number of grid lines to the right of the grid origin.

Top

The *Top* field allows you to specify the number of grid lines above the grid origin.

Bottom

The *Bottom* field allows you to specify the number of grid lines below the grid origin.

Grid Direction

The *Grid Direction* group box allows you to specify in what direction you would like to edit the grid using the table and commands available below.

Direction X

The *Direction X* option allows you to edit the grid lines along the X-axis.

<u>Direction Z</u>

The *Direction Z* option allows you to edit the grid lines along the Z-axis.

Show Grid

The *Show Grid* field toggles the display of the grid in the *Graphics Window*.

Save As Default

Selecting *Yes* in this field will save the grid to be used in future projects.

Generate

The *Generate Grid* command button creates the specified grid in the *Graphics Window*.

Edit Grid Line(s)

The *table* below allows you to edit the grid lines of a grid. You can edit the grid lines by changing the values in the table or by using the command buttons.

Insert After

The *Insert After* command button inserts a grid line after the row selected in the grid line table. The value of that grid line will be automatically calculated by interpolating the values above and below that line.

Insert Before

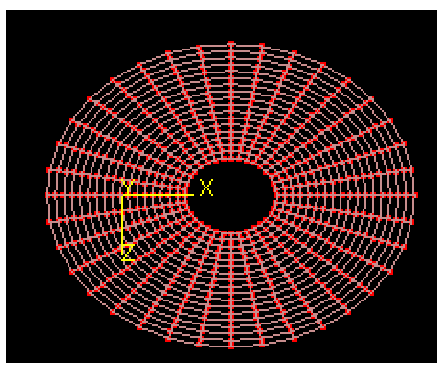
The *Insert Before* command button inserts a grid line before the row selected in the grid line table. The value of that grid line will be automatically calculated by interpolating the values above and below that line.

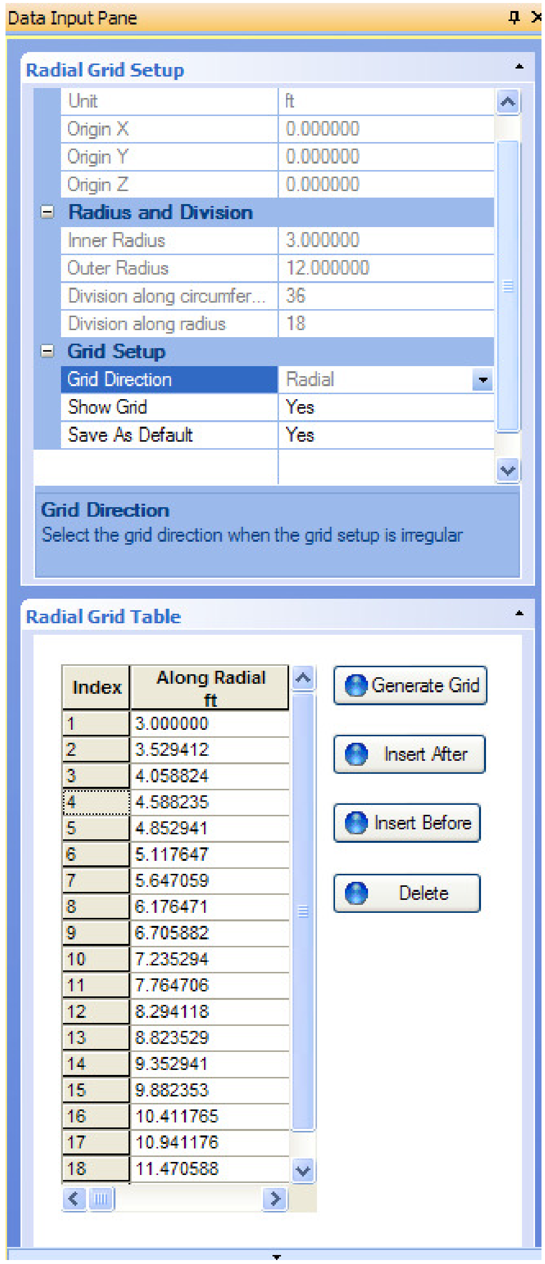
Delete

The *Delete* command button deletes the selected row in the grid line table.

Radial Grid

The *Radial* grid allows you to create a circular grid.




Data Input Pane
Radial Grid Setup
Unit
ft
Origin X
0.000000
Origin Y
0.000000
Origin Z
0.000000
Radius and Division
Inner Radius
3.000000
Outer Radius
12.000000
Division along circumfer...
36
Division along radius
18
Grid Setup
Grid Direction
Radial
Show Grid
Yes
Save As Default
Yes
Grid Direction
Select the grid direction when the grid setup is irregular
Radial Grid Table
Index
Along Radial ft
1
3.000000
2
3.529412
3
4.058824
4
4.588235
5
4.852941
6
5.117647
7
5.647059
8
6.176471
9
6.705882
10
7.235294
11
7.764706
12
8.294118
13
8.823529
14
9.352941
15
9.882353
16
10.411765
17
10.941176
18
11.470588
Generate Grid
Insert After
Insert Before
Delete

Grid Origin

The *Grid Origi*n group box allows you to specify the origin of the grid.

Unit

The Unit field allows you to select the current length unit of the grid system. You can change the unit by clicking on that field and selecting desired unit from the drop down list.

Origin X

The *Origin X* field allows you to specify the X-coordinate of the grid origin.

Origin Y

The *Origin Y* field allows you to specify the Y-coordinate of the grid origin.

Origin Z

The *Origin Z* field allows you to specify the Z-coordinate of the grid origin.

Radius and Division

The *Radius and Division* group box allows you to specify the inner and outer radius of the grid and grid divisions.

<u>Inner Radius of the Grid</u>

The *Inner Radius of the Grid* field allows you to specify the inner radius of the grid using the units selected in the *Change Length Unit* drop-down list box in the *Tools* toolbar.

<u>Outer Radius of the Grid</u>

The *Outer Radius of the Grid* field allows you to specify the outer radius of the grid using the units selected in the *Change Length Unit* drop-down list box in the *Tools* toolbar.

Number of Divisions

<u>Along Circumference</u>

The *Along Circumference* field allows you to specify the number of divisions along the circumference of the grid.

<u>Along Radius</u>

The *Along Radius* field allows you to specify the number of divisions along the radius of the grid.

Show Grid

The *Show Gri*d field toggles the display of the grid in the *Graphics Window*.

Save As Default

Selecting *Yes* in this field will save the grid data to be used in future projects.

Generate

The *Generate Grid* command button creates the specified grid in the *Graphics Window*.

Grid Direction

The *Grid Direction* option allows you to specify in what direction you would like to edit the grid using the table and commands available below.

Circumferential

The *Circumferential* option allows you to edit the grid lines along the circumference of the grid.

Radial

The *Radial* option allows you to edit the grid lines along the radius of the grid.

Edit Grid Line(s)

The *Edit Grid Line(s)* group box allows you to edit the grid lines of a grid. You can edit the grid lines by changing the values in the table or by using the command buttons.

Insert After

The *Insert After* command button inserts a grid line after the row selected in the grid line table.

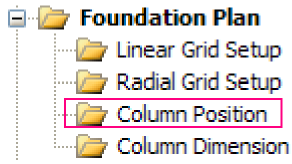
Insert Before

The *Insert Before* command button inserts a grid line before the row selected in the grid line table.

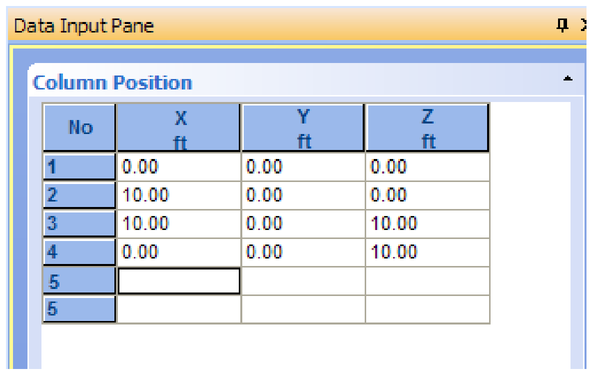
Delete

The *Delete* command button deletes the selected row in the grid line table.

4.4.2.2 Column Positions



The *Column Positions* button opens a spreadsheet table in the *Data Area* pane that allows you to input column positions in Cartesian (XYZ) coordinates.

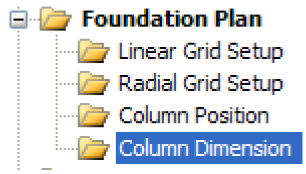


No	X ft	Y ft	Z ft
1	0.00	0.00	0.00
2	10.00	0.00	0.00
3	10.00	0.00	10.00
4	0.00	0.00	10.00
5			
5			

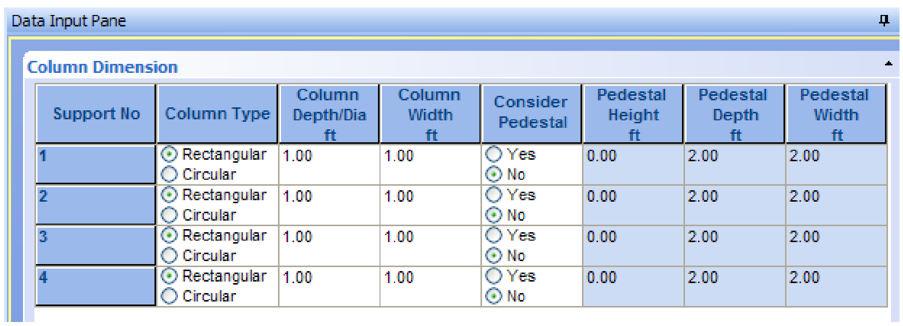
After column coordinates are entered, the columns along with their respective node numbers are displayed in the *Graphics Window*. The tab key or arrow keys may be used to move from one cell to the next in the table. The coordinates in the table can be modified like any spreadsheet. In order to delete a column, select the column in the *Graphics Window* by clicking on it. Then either press the delete key on your keyboard or use the *Menu Bar* command *Edit* | *Delete*.

Note: A column will not be shown in the *Graphics Window* until you hit *Enter* or click outside of the row you are currently in.

4.4.2.3 Column Dimensions



Clicking on the *Column Dimensions* leaf opens a spreadsheet table in the *Data Area* pane that allows you to specify the depth and width of the columns at each support location and pedestal information if any. Column or pedestal dimensions are needed to check punching shear for a mat foundation. For all other footing types these dimensions will be used to calculate critical design forces. The unit used for this form is set through the "*Setup Input/Output unit"* in the toolbar.


Data Input Pane

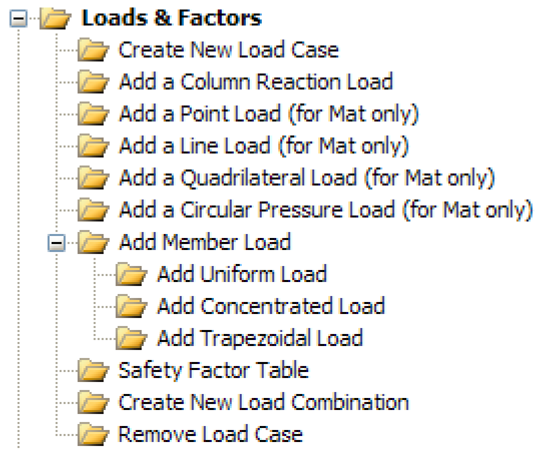
Column Dimension

Support No	Column Type	Column Depth/Dia ft	Column Width ft	Consider Pedestal	Pedestal Height ft	Pedestal Depth ft	Pedestal Width ft
1	Rectangular / Circular	1.00	1.00	Yes / No	0.00	2.00	2.00
2	Rectangular / Circular	1.00	1.00	Yes / No	0.00	2.00	2.00
3	Rectangular / Circular	1.00	1.00	Yes / No	0.00	2.00	2.00
4	Rectangular / Circular	1.00	1.00	Yes / No	0.00	2.00	2.00

If the column type is Circular "Column Width" field will be grayed out. If you have pedestal you can select "Yes" radio button under *Consider Pedestal* field. If you select "Yes" the fields for Pedestal Height, depth and width will be editable.

By default program considers that there is no pedestal.

4.4.3 The Loads & Factors

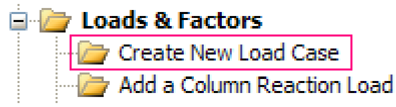


The *Loads & Factors group* allows you to define the loads on a foundation by creating load cases, loads, combination loads, and safety factors for load cases.

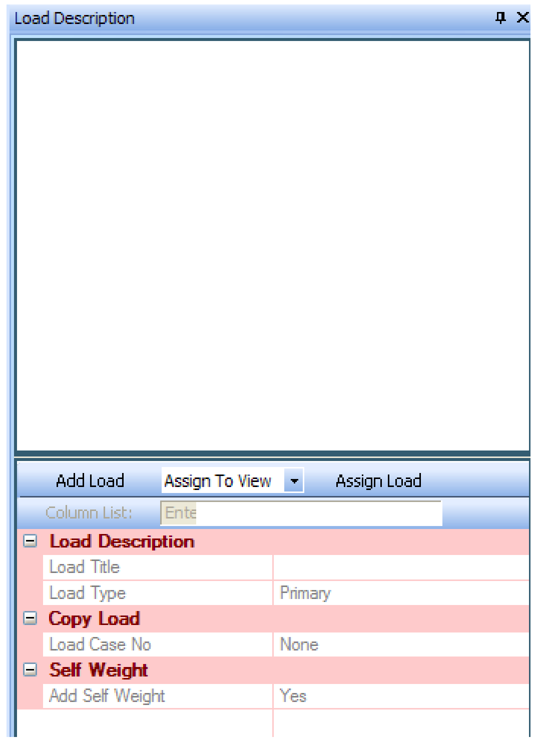
The *Load & Factors* group contains the following elements

- New Load Case
- Add a Column Reaction Load
- Add a Point Load
- Add a Line Load
- Add a Quadrilateral Load
- Add a Circular Pressure Load
- Add Member Load
    - Add Uniform Load
    - Add Concentrated oad
    - Add Trapezoidal Load
- Safety Factor Table
- Create New Load Combination
- Remove Load Case

4.4.3.1 Create New Load Case



Clicking on the *Loads & Factors* leaf or clicking on *Create New Load Case* leaf will open *Load Description* form in data pane area.



To create a new load case you need to input *Load Title*, it can be any string.

Load Type can be one of the followings

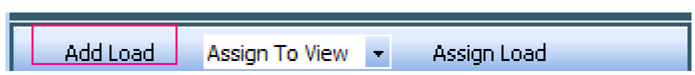
1) Primary
2) Service
3) Ultimate

A Primary load case will be used for both serviceability and factored design. For primary load cases, both the serviceability and design factors will automatically be set to 1. A Service load case will be used only for serviceability checks to calculate footing dimensions. An Ultimate load type will be used for shear checks and reinforcement design.

Copy Load option allows user to copy all load items from a previously defined load case. By default, source load case is set as *None*.

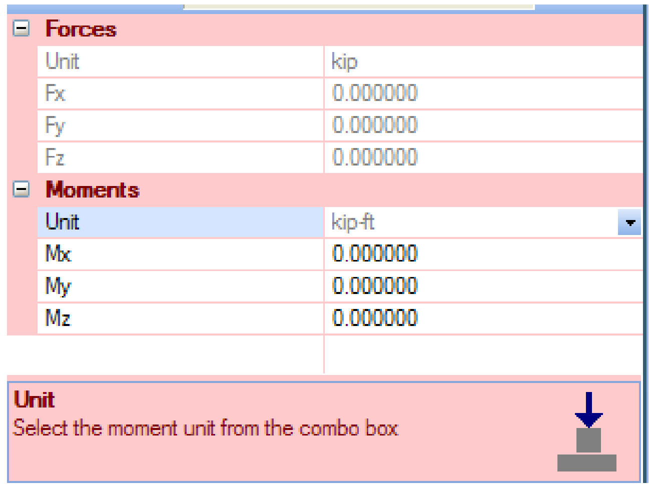
Add Self Weight option is used to add self weight of Mat foundation for analysis. Please note, this option is relevant for Mat foundation design. For all other footing types like Isolated, Combined; the program automatically calculates and adds self weight as appropriate.

After all relevant input is given, click on "***Add Load***" button to add that load.



4.4.3.2 Add a Column Reaction Load

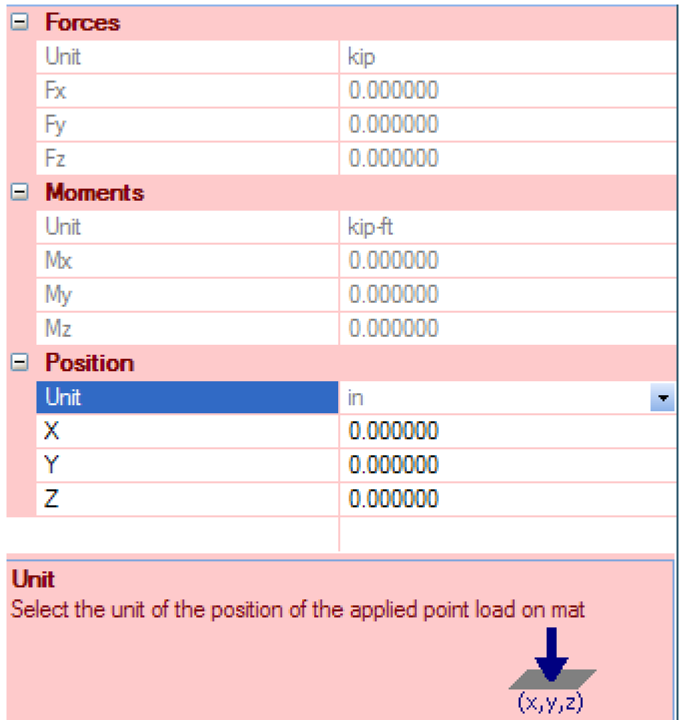
Clicking on the “Add a Column Reaction Load” leaf will open a form in the data area pane which will allow you to create a nodal load acting on support.



To create a *Reaction Load*, first select the *Force* and *Moment Units* to use for the load. Then input the magnitude of the forces (*Fx, Fy, Fz*) and moments (*Mx, My, Mz*). Finally, click on *Add Load* to include the load. Please note, load direction follows right hand rule, so a positive Fy value will create a tensile force.

4.4.3.3 Add a Point Load (for Mat only)

The *Point Load* button opens a dialog box that allows you to create a concentrated load on a mat. *Point loads* are only applicable to mat foundations.

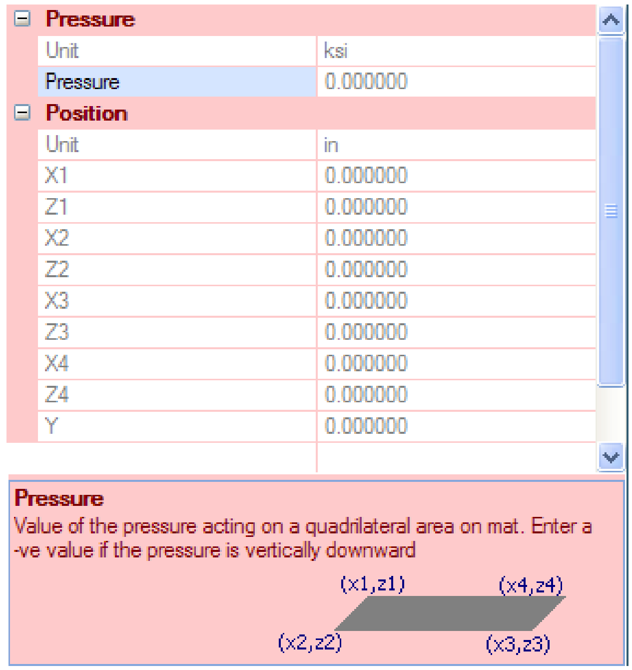


To create a *Point Load*, first select the *Force* and *Moment Units* to use for the load. Then input the magnitude of the forces (*Fx, Fy, Fz*) and moments (*Mx, My, Mz*). Next, input the loading position (*X, Y, Z*) and select the *Unit* measurement for the loading position. Finally, click on *Add Load* button to include the load.

Note: The *Y Loading Position* must correspond to the elevation of the foundation supports.

4.4.3.4 Add a Quadrilateral Load (for Mat only)

Clicking on the Add *Quadrilateral Load* leaf opens a form that allows you to create a *Quadrilateral Load*. *Quadrilateral Loads* are plate pressure load and only applicable to mat foundations.

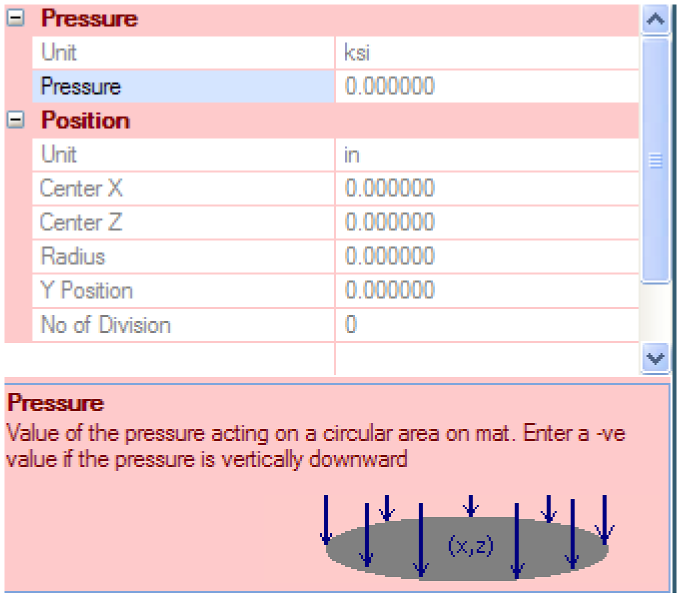


To create a *Quadrilateral Load*, first select the *Dimension* and *Pressure Units* to use for the load. Then input the magnitude of the load in the *Pressure* field. Next, input the elevation at which the load is applied in the *Y Coord* field. Now, define the area or "footprint" of the load by inputting the coordinates of the quadrilateral figure (*x1, x2, x3, x4, z1, z2, z3, z4*). Finally, click on *OK* to accept the load.

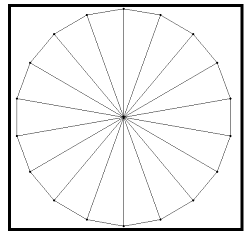
Note: The *Y Coordinate* must correspond to the elevation of the foundation supports.

4.4.3.5 Add a Circular Pressure Load (for Mat only)

Clicking on the Add *Circular Load* leaf opens a form in data pane area that allows you to create a *Circular Load*. *Quadrilateral Loads* are plate pressure load and only applicable to mat foundations.



STAAD.*foundation* does not actually create a true circular boundary for a *Circular Load*. Instead, STAAD.*foundation* simulates a circle through the use of pie-shaped wedges as shown in the figure below.

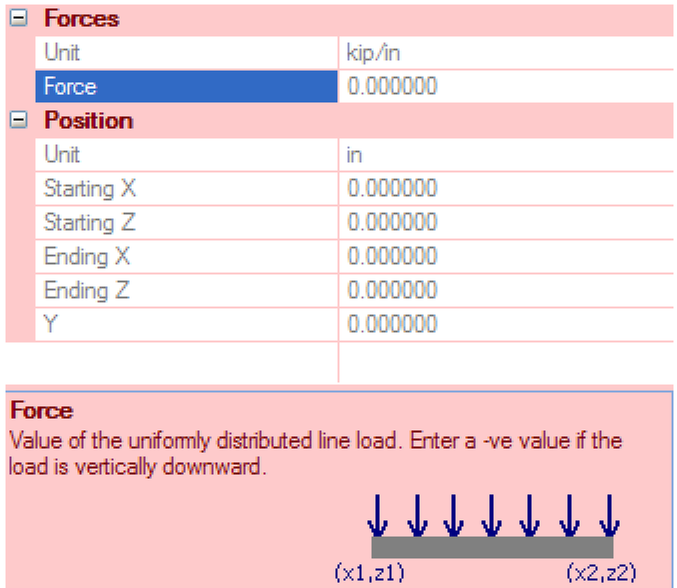


To create a *Circular Load*, first select the *Dimension* and *Pressure Units* to use for the load. Then input the magnitude of the load in the *Pressure* field. Next, input the X and Z-coordinates of the center of the circle in the *Center X* and *Center Z* fields. Now input the length of the radius of the circle in the *Radius* field. Then input the elevation at which the load is applied in the *Y Pos* field. Next, input the number of pie-shaped wedges to use in simulating the circular boundary in the *No. of Divisions Field*. Finally, click on *Add Load button* to accept the load.

Note: The *Y Position* must correspond to the elevation of the foundation supports.

4.4.3.6 Add a Line Load (for Mat only)

Clicking on the *Add a Line Load* leaf opens a form in data pane area that allows you to create a *Line Load*. *Line Loads* are distributed linear load and only applicable to mat foundations.

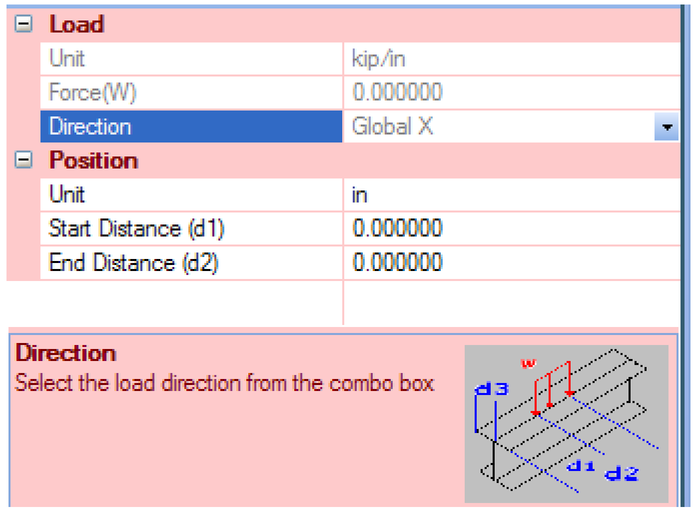


To create a *Line Load*, first select the *Dimension* and *Force Units* to use for the load. Then input the magnitude of the load in the *Force* field. Next, input the X and Z-coordinates of the start and end points of the line in the *Starting X, Starting Z, Ending X, Ending Z* fields. Then input the elevation at which the load is applied in the *Y Pos* field. Finally, click on *Add Load button* to include the load.

Note: The *Y Position* must correspond to the elevation of the foundation supports.

4.4.3.7 Add Uniform Load (member load)

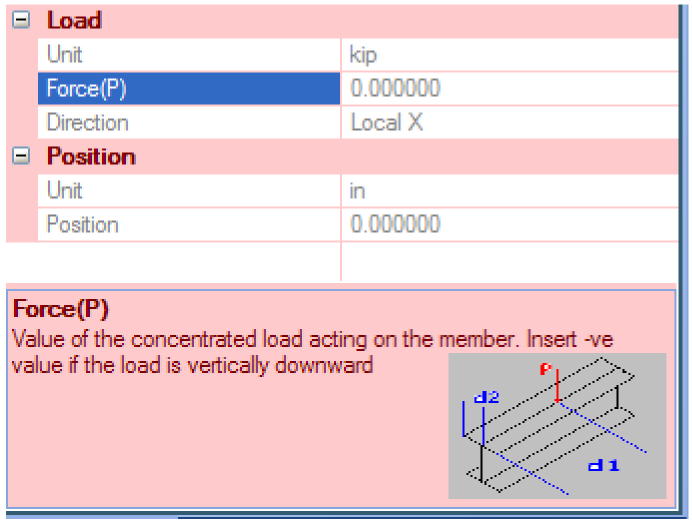
Clicking on the *Add Uniform Load* leaf opens a form in data area pane that allows you to create a uniform load on physical beams.



To create a *uniform beam Load*, first select the *Load* and *Length Units* to use for the load. Next, input the load value with proper sign. Please note, a positive Y value represents load acting upward. Now, select the *Direction* (*Local X, Local Y, Local Z, Global X, Global Y, or Global Z*) in which the load will act upon. If this a partial load input start and end distance. Please note, start and end distance are here in local coordinates. If start and end distance are kept as 0.0 the load will be applied on entire beam. Finally, click on *Add Load* to accept the load.

4.4.3.8 Add Concentrated Load (member load)

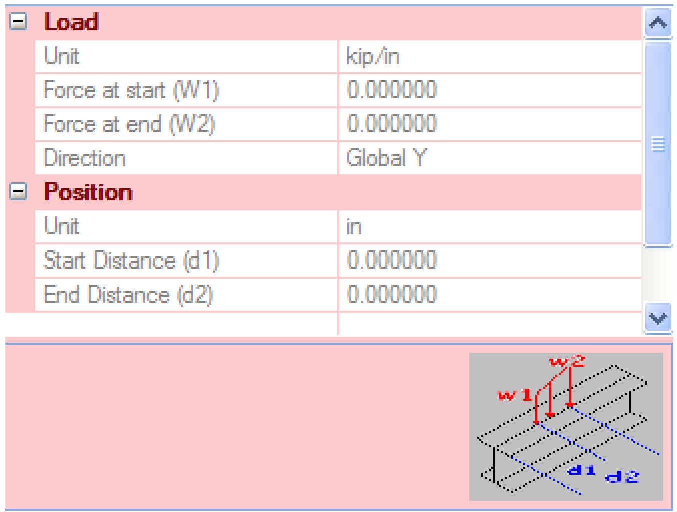
Clicking on the *Add Concentrated Load* leaf opens a form in data area pane that allows you to create a concentrated load on physical beams.



To create *concentrated load* acting on a beam, first select the *Load* and *Length Units* to use for the load. Next, input the load value with proper sign. Please note, a positive Y value represents load acting upward. Now, select the *Direction* (*Local X, Local Y, Local Z, Global X, Global Y, or Global Z*) in which the load will act upon. If load is acting at the middle of the beam you don't need to input *position* parameter. Please note that *position* here is in local coordinates. Finally, click on *Add Load* to accept the load.

4.4.3.9 Add Trapezoidal Load (member load)

Clicking on the *Add Trapezoidal Load* leaf opens a form in data area pane that allows you to create a trapezoidal load on physical beams.

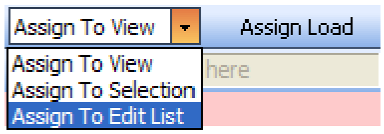


To create *trapezoidal load* acting on a beam, first select the *Load* and *Length Units* to use for the load. Next, input the load values with proper sign at start and end. Now, select the *Direction* (*Local X, Local Y, Local Z, Global X, Global Y, or Global Z*) in which the load will act upon. Now input start and end distance of the load. Please note, *distances* here are in local coordinates. Finally, click on *Add Load* to accept the load.

4.4.3.10 Load assignment methods

The *Assignment Method* utility allows you to choose the method of assignment and contains the following methods and commands:

- Assign to View
- Assign to Selection
- Assign to Edit List
- Assign Load



Assign to View

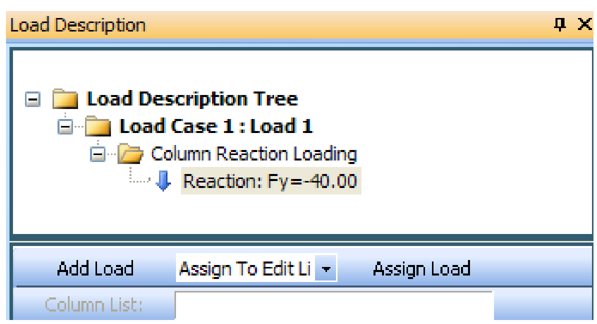
The *Assign to View* option assigns the selected load to all relevant objects in the *Graphics Window*.

Assign to Selection

The *Assign to Selection* option assigns the selected load to only those relevant objects that are selected in the *Graphics Window*.

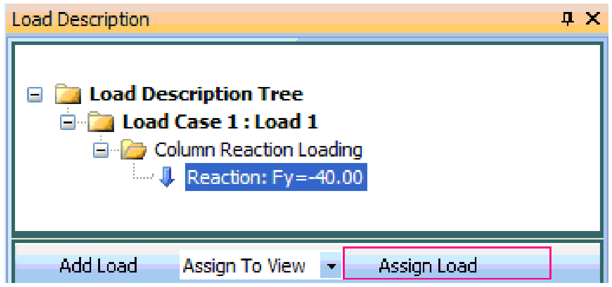
Assign to Edit List

The *Assign to Edit List* option assigns the selected load to only those objects that are inputted in the column list edit box.



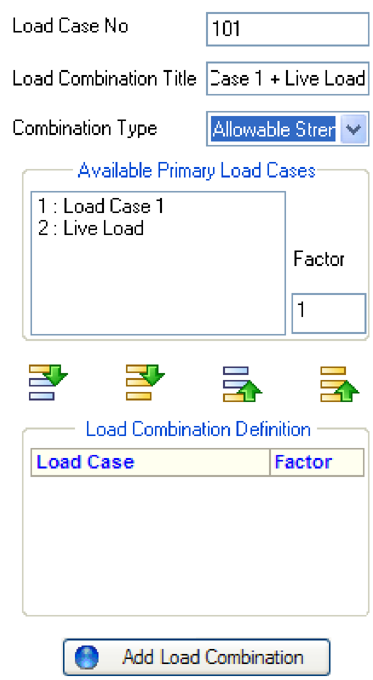
Assign Load

The *Assign* command button assigns the selected load using the *Assignment Method* chosen.



4.4.3.11 Load Combination

Clicking on “Create New load Combination” leaf under “Loads and Factors” group will bring a form at the bottom of the load description pane allowing you to create factored algebraic load combinations.

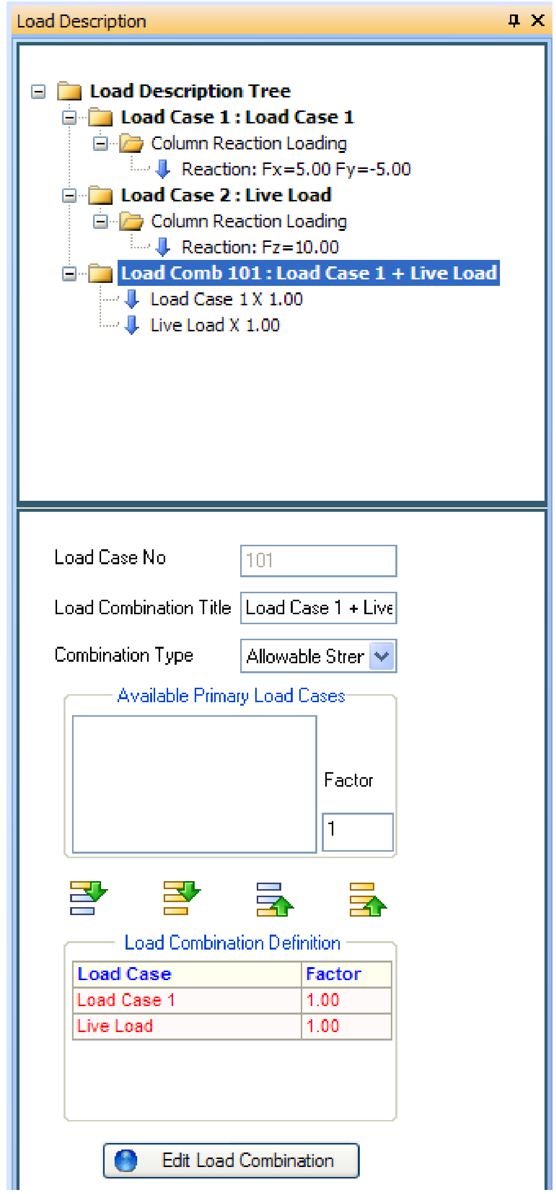


The load case number is automatically incremented with each new load combination. Enter a description for the new combined load such as “**Dead Load + Live Load**”.

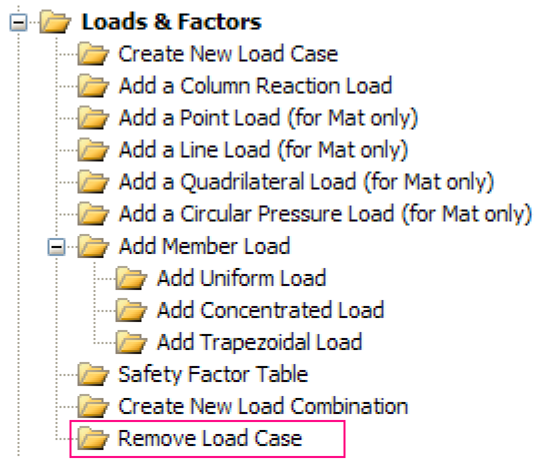
The *Load Combinations form* will lists all defined Primary Load Cases for the foundation in the list box on the top. The *Factor* box on the right indicates the factor with which the selected Primary Load Cases are to be multiplied.

To include a Primary Load Case, first select the load case from the list. Enter the multiplication factor in the *Factor* field. Click the  button to include the Primary Load Case in the *Load Combinations* specification. Continue for all primary load cases to be combined. Use the  button to include all Primary Load Cases, which will be multiplied by the specified factor. To remove a Primary Load Case from the *Load Combinations*, select the load case in the grid on the bottom and click the  button. To remove all Primary Load Cases, click on the button. 

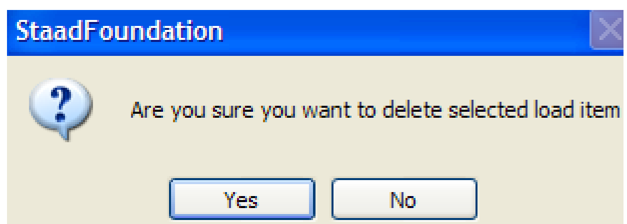
Click **Add Load Combination** button to create the load combination.


Load Description
Load Description Tree
Load Case 1 : Load Case 1
Column Reaction Loading
Reaction: Fx=5.00 Fy=-5.00
Load Case 2 : Live Load
Column Reaction Loading
Reaction: Fz=10.00
Load Comb 101 : Load Case 1 + Live Load
Load Case 1 X 1.00
Live Load X 1.00
Load Case No
101
Load Combination Title
Load Case 1 + Live
Combination Type
Allowable Strer
Available Primary Load Cases
Factor
1
Load Combination Definition
Load Case
Factor
Load Case 1
1.00
Live Load
1.00
Edit Load Combination

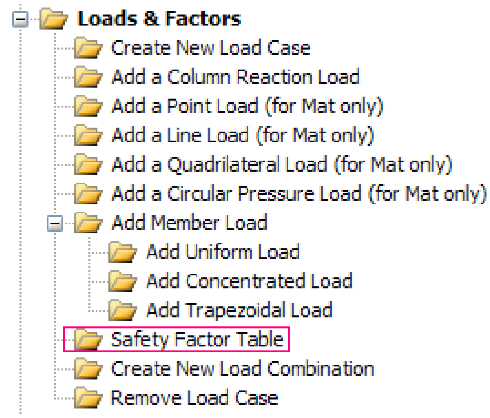
4.4.3.12 Remove Load Case



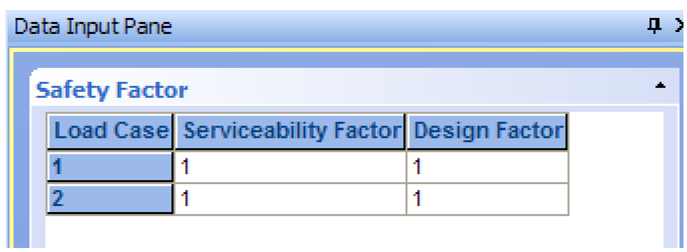
To remove *an entire Load Case* or a specific load item select that load item or load case in *load description* window and click on "*Remove Load Case*" leaf under *Loads and Factors group*. A message box will appear to confirm delete operation, click on "Yes" to delete the selected load item or load case. Click on "No" to cancel the process.



4.4.3.13 Safety Factors



The *Safety Factors* button opens a spreadsheet table in the *Data Area* pane that allows you to assign serviceability and design factors for each load case in a project.

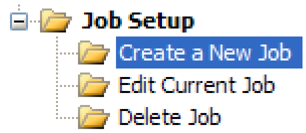


By default, STAAD.*foundation* will assign values for the safety factors depending on the load type. Refer to Section 4.4.3.1 of this manual for a detailed explanation of the default values. The default values can be changed by inputting new values into the table like any spreadsheet. The tab key or arrow keys may be used to move from one cell to the next in the table. The serviceability factor will be applied when checking the base pressure of a foundation (geotechnical design). The design factor will be used for design shear and reinforcement design..

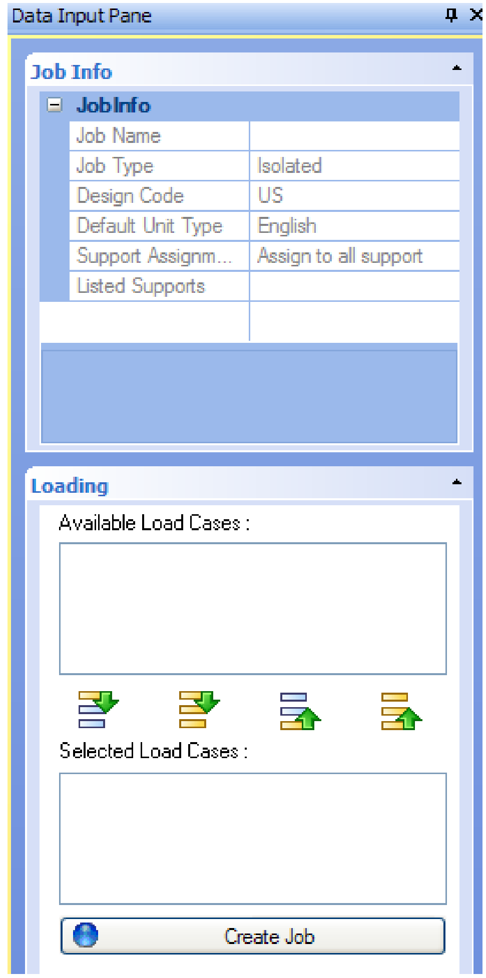
4.5 Job Setup

The link between global and local data is J*ob Setup* where user can create different footing job types. User can create as many jobs as needed. Multiple jobs with same footing type are also allowed.

4.5.1 Create a New Job



To create a new job click on *Create a New job* leaf under *Job Setup* group which will open a form in data area pane.



Job setup form has two separate groups. The group on top is to define Job type, unit etc. and the group at the bottom is to associate loading with the job.

Job Name: Job name is used to uniquely identify each job. You can enter any string here.

Job Type: It is used to define the foundation type for the new job. In current version we support 5 different types of footing which are

- Isolated
- Pile Cap
- Combined
- Mat
- Octagonal

Design Code: It is used to define concrete code to be used. Current version supports 3 country codes which are

- ACI 318-05
- BS 8110
- IS 456-2000

Default Unit Type: It is used to setup default design parameters of the job. We support both FPS and SI unit systems. User can choose any combination of design code and default unit type. In other words user can choose US design code with SI unit system.

Support Assignment: It is used to assign supports to a job. There are three assignment methods

- Assign to All Supports
- Assign to Selected Supports
- Assign to Listed Supports

Selecting first option *Assign to all supports* will assign all supports to the current job. Selecting second option will assign all selected supports in the main view to the current job.

If we select third option which is *Assign to Listed Supports,* the last field in the first group *Listed Support* becomes active and you can type the support numbers to be assigned to the current job.

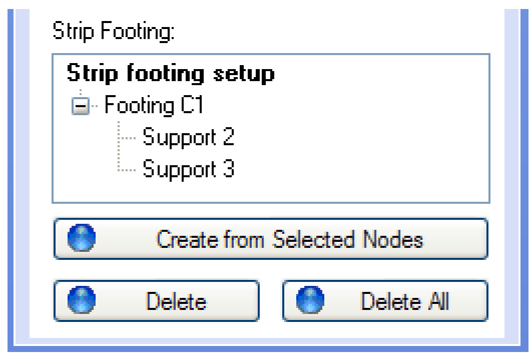
The bottom group is to assign loads to the current job. All load cases will be shown in available list box. To include a load case, first select the load case from the list and then click the

 button. Continue for all load cases to be included. Use the

button to include all load cases, which will be multiplied by the specified factor. To remove a load case from the *Load Combinations*, select the load case in the grid on the bottom and click the button. To remove all load cases, click on the button.

Finally click on "Create Job" to create a new job. Please note, new job will be set as current job and will be shown in "Change Job" dropdown toolbar.

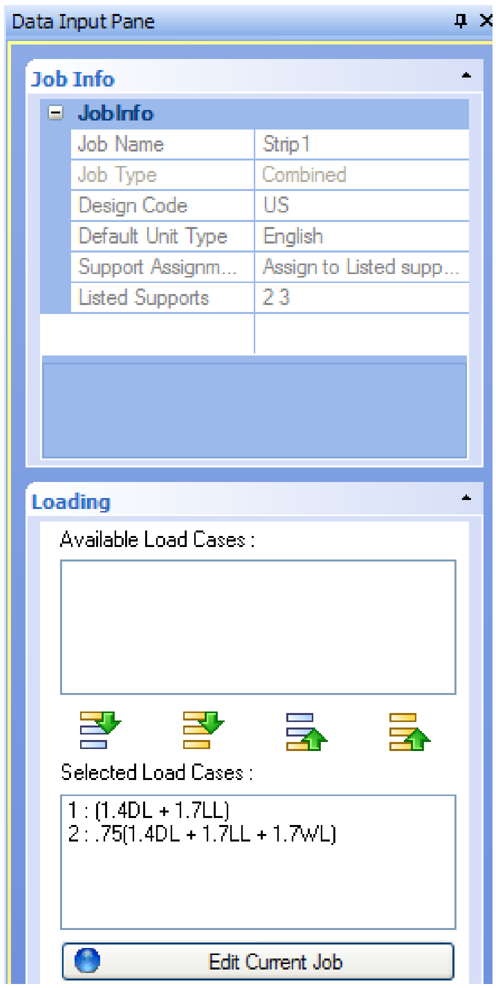
Strip/Combined footing: If we select *Job Type* as combined, a new set of controls appear at the bottom of the *Job Setup* form. Those controls will initially be grayed out. Click on "Create Job" to activate those controls.



Select two or more collinear supports in main view and then click on "Create from Selected Nodes" button to add those supports as strip footing.

4.5.2 Edit Current Job

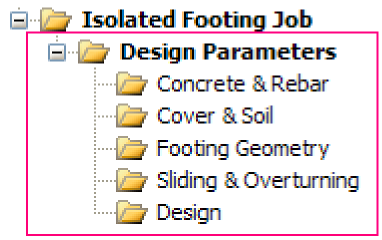
Click on Edit Current Job leaf under Job Setup group to edit the current job setup. All the fields in job setup can be edited except Job Type. If you need to change the list of assigned support you can use any of the assignment options as discussed earlier. By default support assignment option is set as *Assign to Listed supports*.



4.6 Local Data

Local Data are specific to job types. Each footing type has its own unique local data types. Design parameters such as concrete cover, rebar specifications, soil parameters and footing geometry are typical examples of design parameters. We will discuss local data for each footing type separately. We will start with Isolated footing job.

4.6.1 Isolated footing

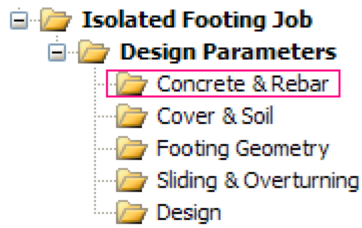


Isolated footing job type has a unique group for local data called *Design Parameters*. The *Design Parameters* group allows you to specify design parameters for an isolated footing and is only active for isolated footing job types.

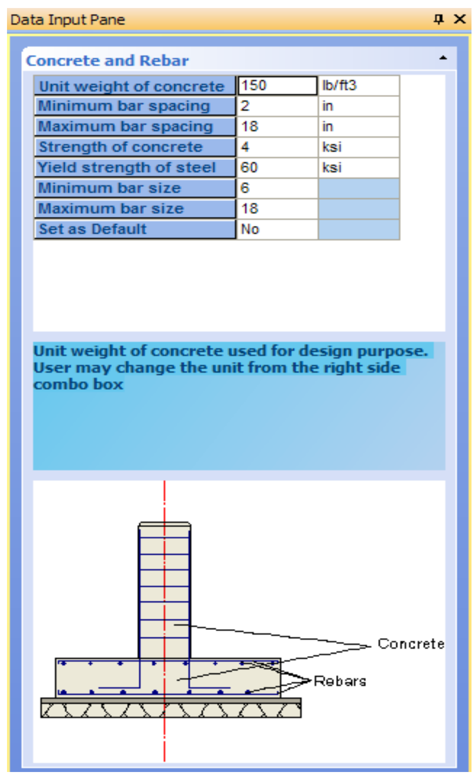
The *Design Parameters* group contains the following elements:

- Concrete and Rebar
- Cover and Soil
- Footing Geometry
- Sliding & Overturning
- Design

4.6.1.1 Concrete and Rebar



Clicking on the *Concrete & Rebar* leaf opens a form in the *Data Area* pane that allows you to input concrete and rebar properties.



The following concrete and rebar properties are available:

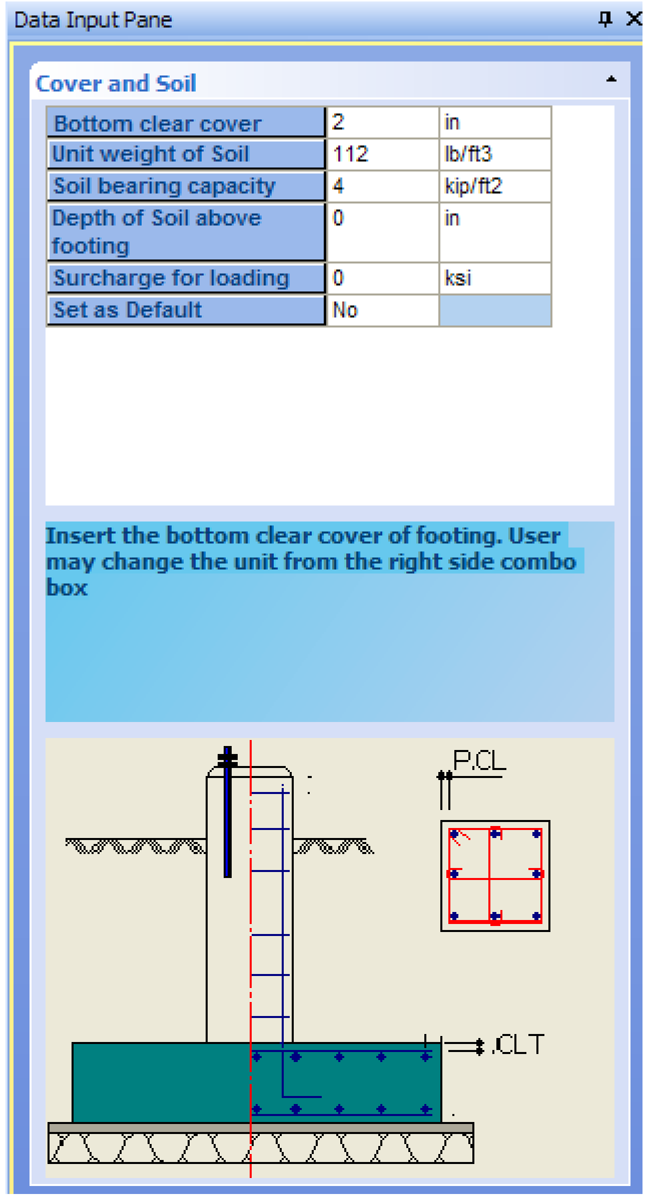
- Unit Weight of Concrete
- Strength of Concrete
- Yield Strength of Reinforcing Steel
- Minimum Bar Size
- Maximum Bar Size
- Minimum Bar Spacing
- Maximum Bar Spacing

Set as Default

The *Set as Default* check box allows you to use the values inputted in the *Concrete & Rebar* form as the default values for future projects.

4.6.1.2 Cover and Soil

Clicking on the *Cover and Soil* leaf under *Design Parameters* group opens a form in the *Data Area* pane that allows you to input cover parameters and soil characteristics.


Data Input Pane
Cover and Soil
Bottom clear cover 2 in
Unit weight of Soil 112 lb/ft3
Soil bearing capacity 4 kip/ft2
Depth of Soil above footing 0 in
Surcharge for loading 0 ksi
Set as Default No
Insert the bottom clear cover of footing. User may change the unit from the right side combo box
P.CL
CLT

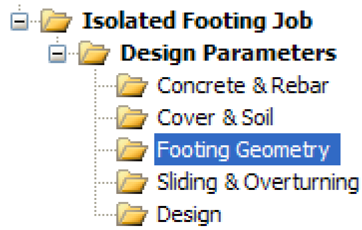
The following cover parameters and soil characteristics are available:

- Bottom Clear Cover
- Unit Weight of Soil
- Soil Bearing Capacity
- Depth of Soil above footing
- Surcharge for Loading

Set as Default

The *Set as Default* check box allows you to use the values inputted in the *Cover and Soil* form as the default values for future projects.

4.6.1.3 Footing Geometry



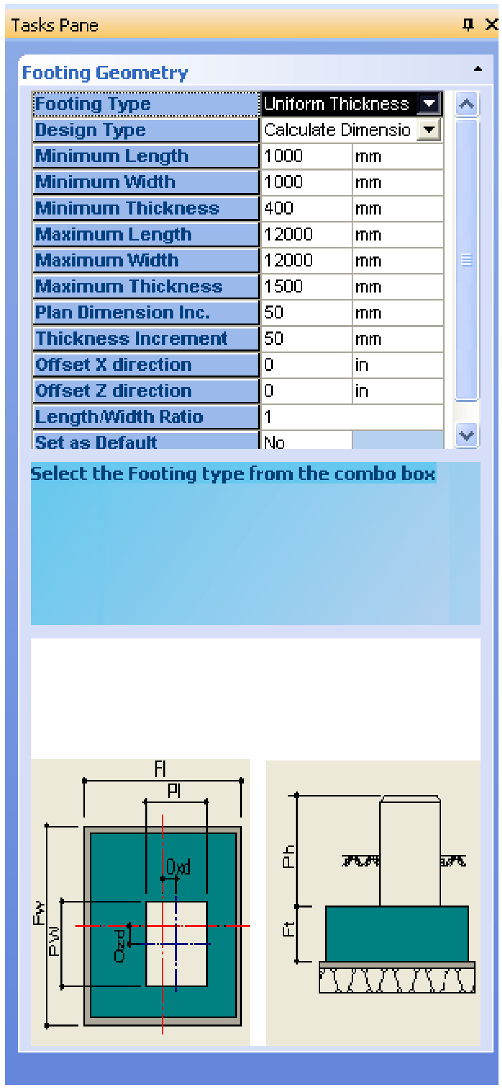
Clicking on the *Footing Geometry* leaf opens a form in data area pane that allows you to input isolated footing geometry.
The following footing details are available:

- Thickness
- Length
- Width
- Offset in both X and Z direction
- Length/Width Ratio
- Footing Type

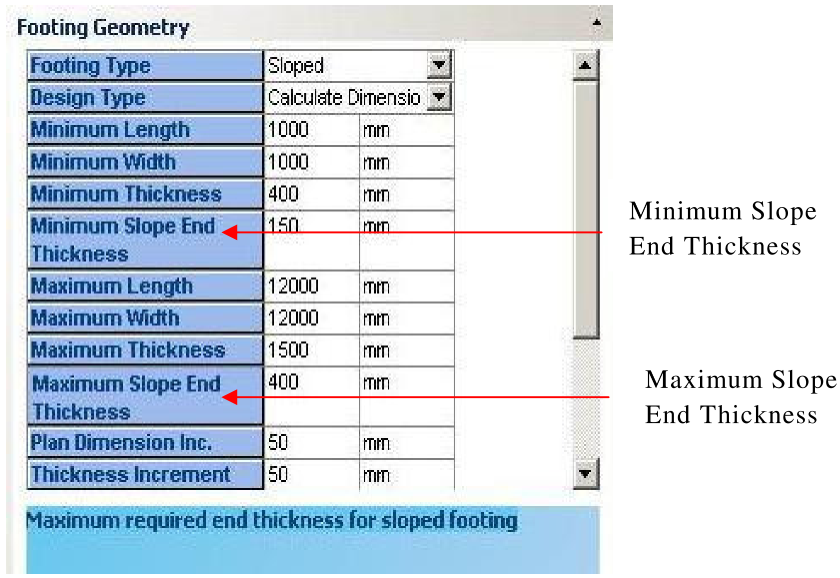
The *Footing Geometry* provides you with the option to instruct STAAD.*foundation* to calculate the footing dimensions, or you can check the footing dimension by specifying fixed values. You can specify a desired minimum and maximum for *Thickness*, *Length*, and/or *Width,* as well as an increment for *Thickness, Plan Dimension, and Length/Width Ratio* by entering the desired values in the corresponding fields. STAAD.*foundation* will calculate any value left unspecified for you

Footing Type

Two types of Footing are used; *Uniform Thickness* and *Sloped*. By default *Uniform Thickness* is been set as shown below.



When you choose *Sloped* footing type the dialog will appear as shown below and two new fields will be added, *Minimum Slope End Thickness* and *Maximum Slope End Thickness*.

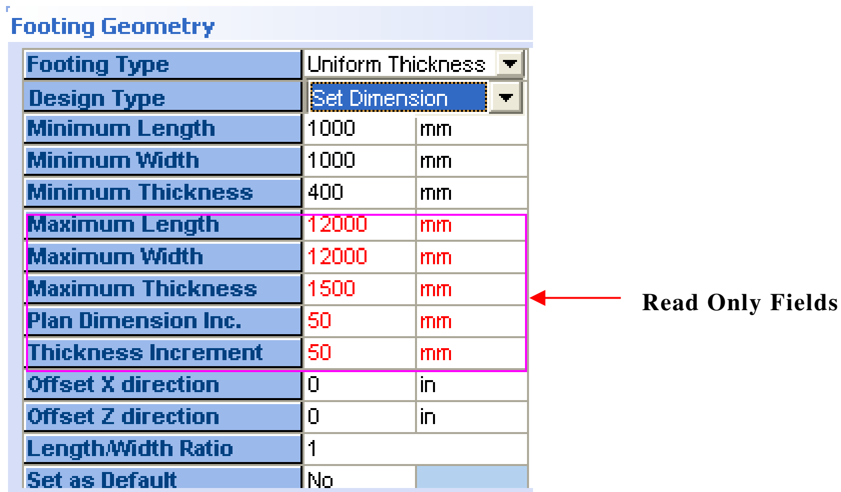


<u>Design Type</u>

In the design type field, two design types are given, *Calculate dimension* and *Set dimension.*

Calculate Dimension Option calculates the actual required dimension from minimum and maximum range and *Set dimension* option sets the user input dimension.

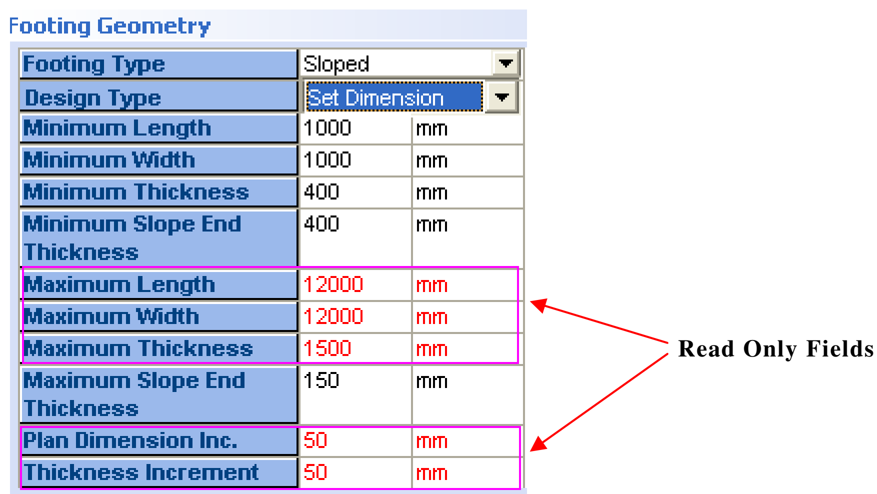
If you choose Set Dimension and Uniform thickness footing type, the following fields will be read only.



Footing Geometry

Footing Type	Uniform Thickness	
Design Type	Set Dimension	
Minimum Length	1000	mm
Minimum Width	1000	mm
Minimum Thickness	400	mm
Maximum Length	12000	mm
Maximum Width	12000	mm
Maximum Thickness	1500	mm
Plan Dimension Inc.	50	mm
Thickness Increment	50	mm
Offset X direction	0	in
Offset Z direction	0	in
Length/Width Ratio	1	
Set as Default	No	

If you choose Set Dimension and Sloped footing type, the following fields will be read only.



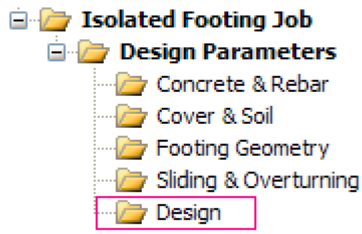
Footing Geometry

Footing Type	Sloped	
Design Type	Set Dimension	
Minimum Length	1000	mm
Minimum Width	1000	mm
Minimum Thickness	400	mm
Minimum Slope End Thickness	400	mm
Maximum Length	12000	mm
Maximum Width	12000	mm
Maximum Thickness	1500	mm
Maximum Slope End Thickness	150	mm
Plan Dimension Inc.	50	mm
Thickness Increment	50	mm

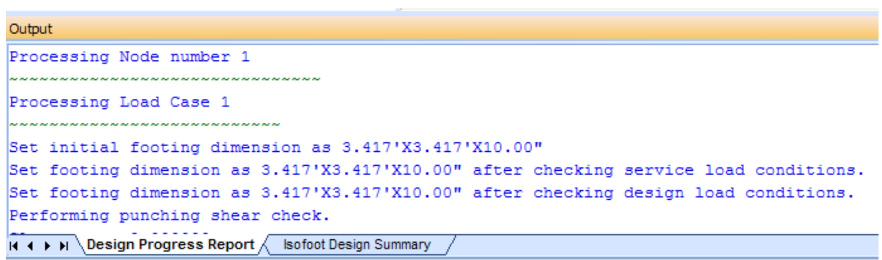
<u>Set as Default</u>

The *Set as Default* field allows you to use the values inputted in the *Footing Geometry* form as the default values for future projects.

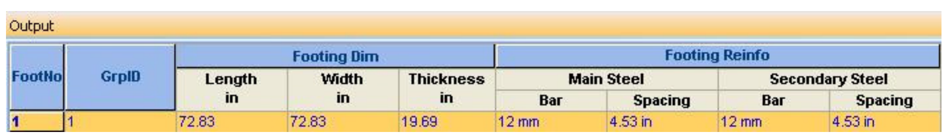
4.6.1.4 Design



Click on *Design* leaf to design all the footings associated in the current job. Program will list all design progress messages including warning and error messages in the bottom output pane. It will help user to understand and review design progress.



After design is completed a new tab called *Isofoot Design Summary* will appear in output pane. Click that tab to view design summary table. You can copy this table by selecting it and then by pressing Ctrl+C key which then can be pasted in *Microsoft Excel* or *Word*.


Output

FootNo	GrpID	Footing Dim			Footing Reinfo			
		Length in	Width in	Thickness in	Main Steel		Secondary Steel	
					Bar	Spacing	Bar	Spacing
1	1	72.83	72.83	19.69	12 mm	4.53 in	12 mm	4.53 in

If you choose *uniform thickness* footing type, the *Isofoot Design Summary* table will be as given above.

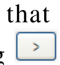
If you choose *Sloped* footing type, the *Isofoot Design Summary* table, a new column. *Slope End Thickness* will be added as shown below,

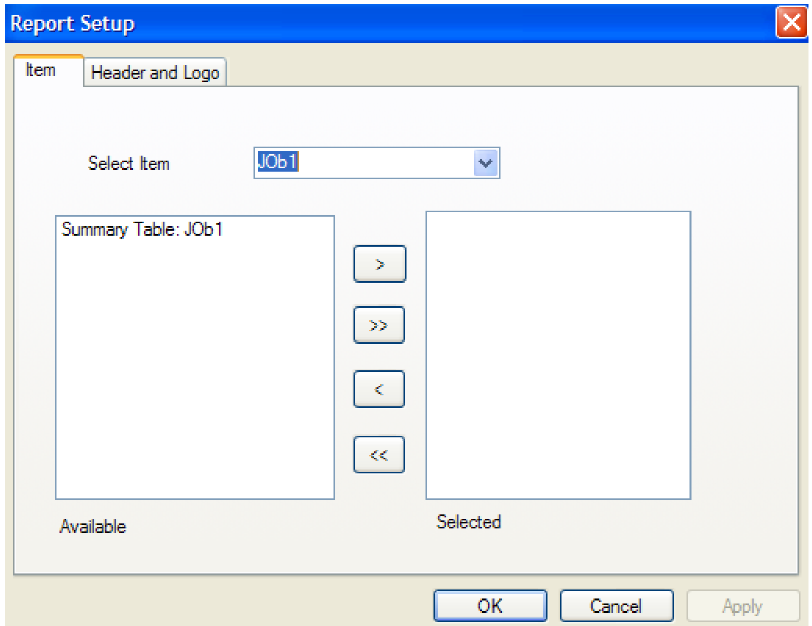
FootNo	GrpID	Footing Dim				Footing Reinfo			
		Length mm	Width mm	Thickness mm	Slope End Thickness	Main Steel		Secondary Steel	
						Bar	Spacing	Bar	Spacing
1	1	1850.00	1850.00	500.00	250.00	12 mm	115.00 mm	12 mm	115.00 mm

Slope End Thickness

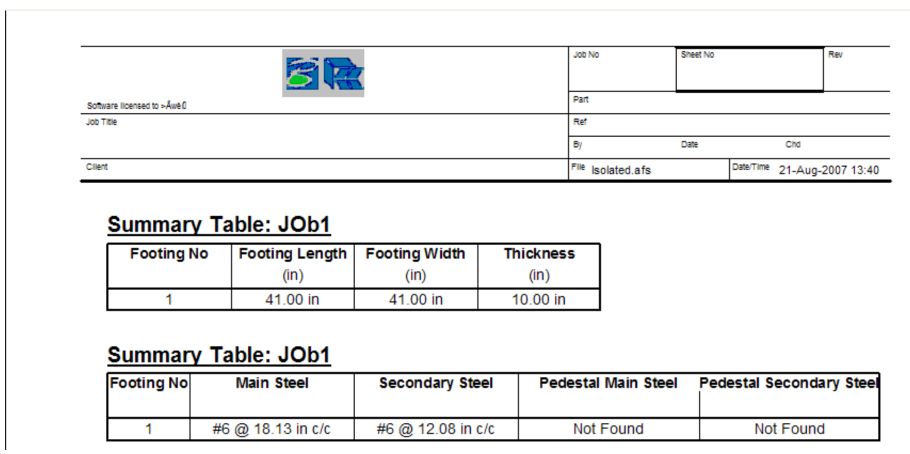
Program will automatically open calculation sheet which presents detailed step by step calculation with relevant code clause numbers, equations and corresponding calculated values. Calculation sheet is organized in a logical manner which shows program flow.

After successful design, you can open *Report Setup* to print design summary table. Switch to *Geometry* view in main view area to activate toolbar. Now click on *Report Setup* icon in toolbar.

It will open a report setup dialog box where under current job a summary table will be available. Select that item by clicking that item in left list box and bring it over to right side by clicking  button. Now click on OK and the dialog box will be dismissed.



Now click on Print Preview icon in toolbar to see print preview and then click on Print button to print the summary table.



Software licensed to

Job Title

Client

Job No | Sheet No | Rev

Part

Ref

By | Date | Chd

File Isolated.afs | Date/Time 21-Aug-2007 13:40

Summary Table: JOb1

Footing No	Footing Length (in)	Footing Width (in)	Thickness (in)
1	41.00 in	41.00 in	10.00 in

Summary Table: JOb1

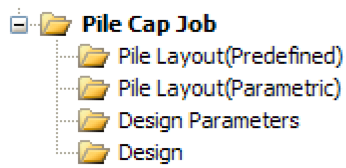
Footing No	Main Steel	Secondary Steel	Pedestal Main Steel	Pedestal Secondary Steel
1	#6 @ 18.13 in c/c	#6 @ 12.08 in c/c	Not Found	Not Found

4.6.2 Pile Cap

Pile Cap job type has a unique group for local data called *Pile Cap Job*. The *Pile Cap Job* group allows you to specify pile arrangement for each pile cap and design parameters and is only active for pile cap job types.

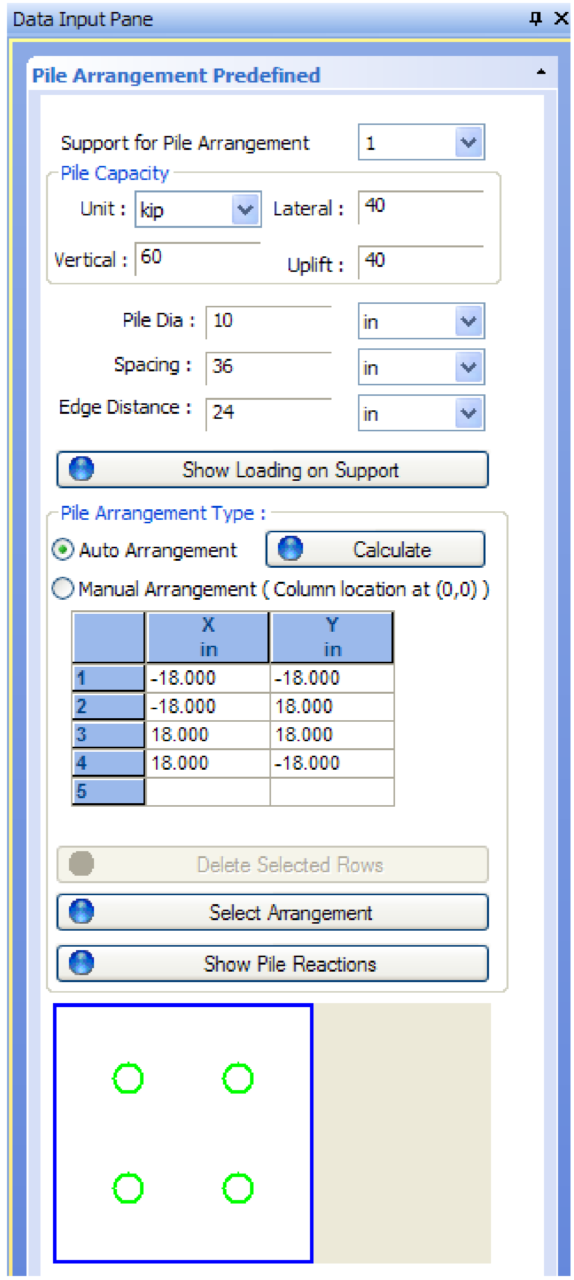
The *Pile Cap Job* group contains the following elements:

- Pile Layout (Predefined)
- Pile Layout (Parametric)
- Design Parameters
- Design



4.6.2.1 Pile Layout (Predefined)

Clicking on *Pile Layout (Predefined)* leaf opens a form in the *Data Area* pane that allows you to specify pile arrangement for a pile cap. Predefined page has a set of predefined pile layout and program can automatically choose the best possible pile arrangement.


Data Input Pane
Pile Arrangement Predefined
Support for Pile Arrangement
1
Pile Capacity
Unit :
kip
Lateral :
40
Vertical :
60
Uplift :
40
Pile Dia :
10
in
Spacing :
36
in
Edge Distance :
24
in
Show Loading on Support
Pile Arrangement Type :
Auto Arrangement
Calculate
Manual Arrangement (Column location at (0,0))
X in
Y in
1 -18.000 -18.000
2 -18.000 18.000
3 18.000 18.000
4 18.000 -18.000
5
Delete Selected Rows
Select Arrangement
Show Pile Reactions

The following pile arrangement data and commands are available:

- Support for Pile Arrangement
- Pile Capacity
- Pile Diameter
- Spacing
- Edge Distance
- Show Loading on Support
- Pile Arrangement Type
- Calculate
- Delete Selected Rows
- Show Pile Reactions
- Select Arrangement

Support for Pile Arrangement

The *Support for Pile Arrangement* drop-down list box allows you to select a support from the current job for which you would like to input pile arrangement.

Pile Capacity

The *Pile Capacity* group box allows you to input the forces that a pile is meant to bear.

Unit

The *Unit* drop-down list box allows you to select the force unit used for *Pile Capacity*.

Lateral

The *Lateral* field allows you to specify the lateral force a pile is meant to bear.

Vertical

The *Vertical* field allows you to specify the vertical force a pile is meant to bear.

Uplift

The *Uplift* field allows you to specify the uplifting force a pile is meant to bear.

Pile Diameter

The *Pile Dia.* field allows you to specify the diameter of a pile. User can choose appropriate unit from the drop down list at right.

Spacing

The *Spacing* field allows you to specify the spacing between piles. User can choose appropriate unit from the drop down list at right.

Edge Distance

The *Edge Distance* field allows you to specify the distance between the edges of a pile. User can choose appropriate unit from the drop down list at right.

Show Loading On Support

The *Show Loading on Support* button opens a table displaying the total loading on the support for each load case selected under *Support for Pile Arrangement*.

Load Table for Support No. 1

Load Case	Fx kip	Fy kip	Fz kip	Mx kip-in	Mz kip-in
1	-4.058242321	-68.03285980	-0.696796894	-33.31498336	209.5984191
2	-2.699549674	-47.24615097	5.542748928	450.1890258	130.1803131
3	-6.757791996	-115.2790145	4.845952033	416.8740539	339.7787170

Pile Arrangement Type

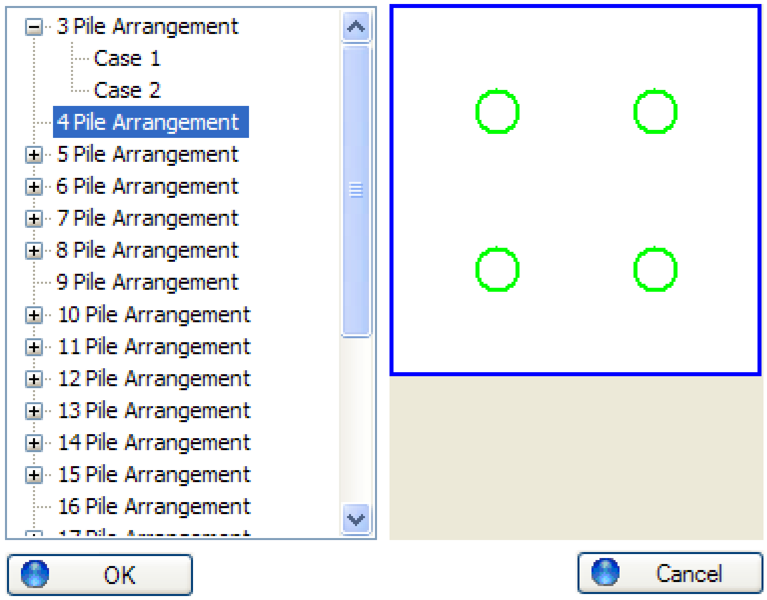
The *Pile Arrangement Type* group box allows you to input the coordinates for a pile arrangement or have STAAD.*foundation* calculate a pile arrangement automatically.

Auto Arrangement

The *Auto Arrangement* radio option allows you to have STAAD.*foundation* calculate the pile arrangement. In order to have STAAD.*foundation* calculate the pile arrangement, select *Auto Arrangement* and click on the *Calculate* button. A window will appear displaying all possible pile arrangements corresponding to the pile loads in all the load cases according to the BOCA standard.

Calculate

The *Calculate* button opens a window displaying all possible pile arrangements corresponding to the pile loads in all the load cases according to the BOCA standard when the *Auto Arrangement* radio option is selected.



Go through the tree controls and choose the desired pile arrangement. After you have chosen the desired pile arrangement, click on the *OK* button. The pile coordinates of the selected pile arrangement will be displayed in the table in the *Data Area* pane. In addition, the diagram of the pile arrangement will be displayed in the *Data Area* pane.

Manual Arrangement

The *Manual Arrangement* radio option allows you to enter the pile arrangement manually by inputting the pile cap coordinates in the table in the *Data Area* pane.

Note: These are local coordinates for the footing, relative to the center of the footing.

Delete Selected Rows

The *Delete Selected Rows* button allows you to delete a row in the table of pile cap coordinates when using the *Manual Arrangement* mode. To delete a row, select the row you would like to delete from the table and then click on *Delete Selected Rows*.

Show Pile Reactions

The *Show Pile Reactions* button opens a table displaying the reaction on each pile.

Pile Reaction Table for Support No. 1

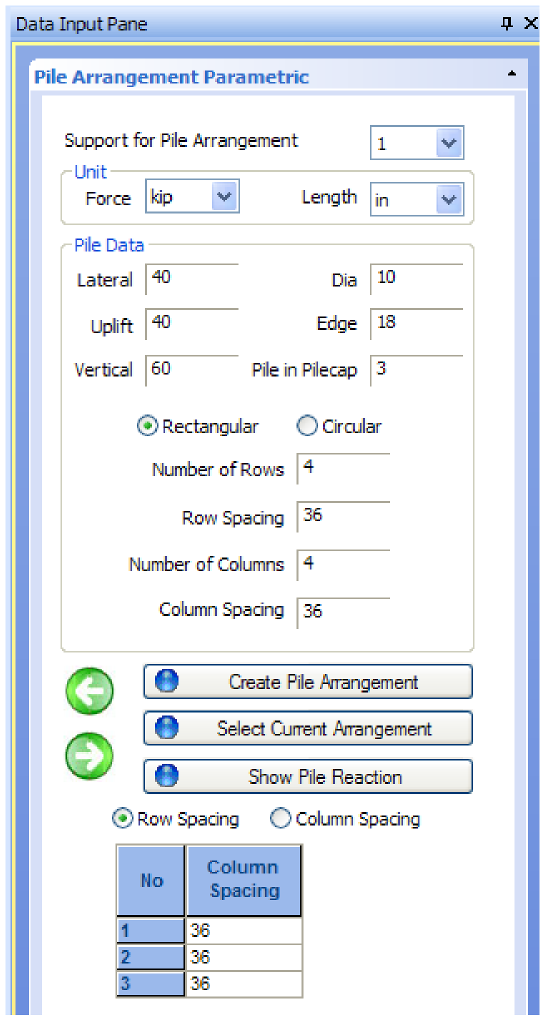
	Vertical kip	Lateral kip	Uplift kip
Load Case = 1			
1	-14.560	1.029	0.000
2	-13.634	1.029	0.000
3	-19.457	1.029	0.000
4	-20.382	1.029	0.000
Load Case = 2			
1	-3.751	1.584	0.000
2	-16.256	1.653	0.000
3	-19.872	1.507	0.000
4	-7.367	1.430	0.000
Load Case = 3			
1	-18.311	2.063	0.000
2	-29.891	2.195	0.000
3	-39.329	2.101	0.000
4	-27.749	1.963	0.000

Select Arrangement

The *Select Arrangement* button allows you to select the current pile arrangement for the design of the pile cap. If you do not want to use the current pile arrangement, recalculate the arrangement or input the pile coordinates again manually.

4.6.2.2 Pile Layout (Parametric)

Clicking on *Pile Layout (Parametric)* leaf opens a form in the *Data Area* pane that allows you to specify pile arrangement for a pile cap. Parametric page allows you to input rectangular and circular pile arrangement. If circular arrangement is chosen, the program will design that pile cap as *Octagonal Pile Cap*.



The following parametric pile arrangement data and commands are available:

- Support for Pile Arrangement
- Pile Capacity
- Pile Diameter
- Spacing
- Edge Distance
- Pile Arrangement Type (Rectangular or Circular)
- Number of Rows
- Row spacing
- Number of Columns
- Column spacing
- Create Pile Arrangement
- Select Current Arrangement
- Show Pile Reaction
- Row Spacing
- Column Spacing

Support for Pile Arrangement

The *Support for Pile Arrangement* drop-down list box allows you to select a support from the current job for which you would like to input pile arrangement.

Pile Capacity

The *Pile Capacity* group box allows you to input the forces that a pile is meant to bear.

Unit

The *Unit* drop-down list box allows you to select the force unit used for *Pile Capacity* and length unit used for *spacing, diameter, edge distance etc..*

Pile Capacity

Lateral

The *Lateral* field allows you to specify the lateral force a pile is meant to bear.

Vertical

The *Vertical* field allows you to specify the vertical force a pile is meant to bear.

Uplift

The *Uplift* field allows you to specify the uplifting force a pile is meant to bear.

Pile Diameter

The *Pile Dia.* field allows you to specify the diameter of a pile.

Spacing

The *Spacing* field allows you to specify the spacing between piles.

Edge Distance

The *Edge Distance* field allows you to specify the distance between the edges of a pile.

Arrangement Type

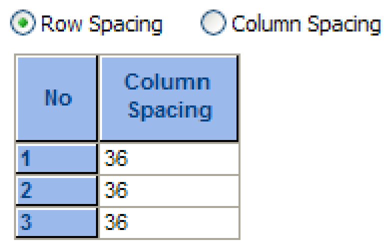
Pile arrangement can be either rectangular or circular. Pile cap having circular arrangement will be design as octagonal pile cap.

Rectangular Arrangement

Rectangular arrangement needs following inputs

- Number of Rows
- Number of Columns
- Row Spacing
- Column Spacing

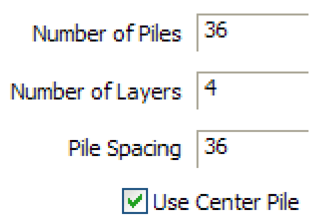
By default program will create symmetric pile arrangement from the above input but user can change the default setup by editing the table below. Both row and column grid lines can be adjusted by selecting appropriate radio button.



Circular Arrangement

Circular arrangement needs following inputs

- Number of Piles – Total number of piles
- Number of Layers – Number of concentric circles
- Pile Spacing – minimum spacing between piles
- Use Center Pile – Add a pile at center of pile arrangement



By default, program will try to assign equal number of piles for all concentric circular layers. But that can be changed by editing the table below.

Layers	Piles
1	4
2	6
3	9
4	12

Create Pile Arrangement

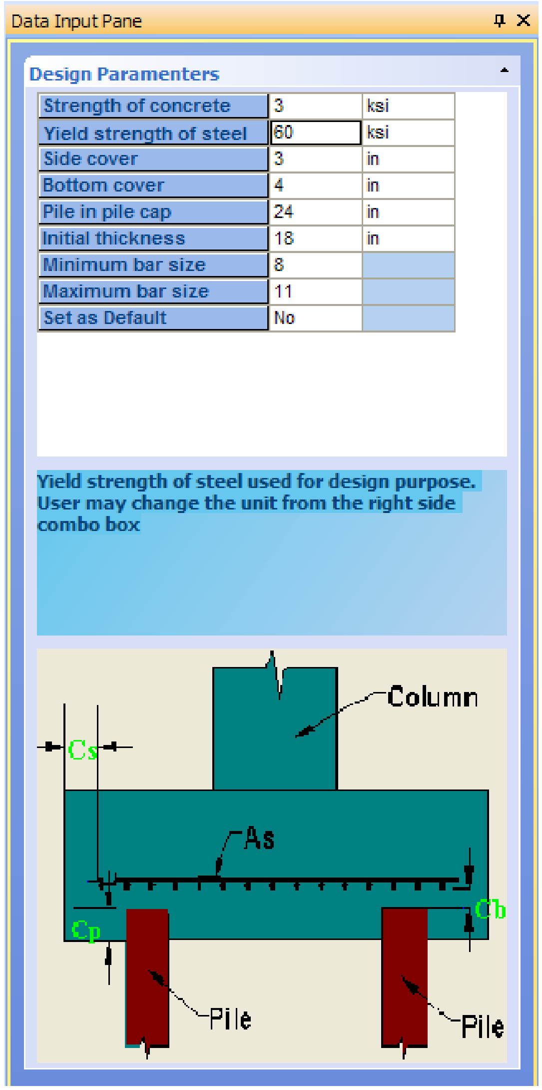
Finally click on the *Create Pile Arrangement* button to create the pile layout. A dialog box will appear at the left of data area pane which will show pile layout drawing and a table for pile coordinates. Pile coordinates in this table are editable.

Select Current Arrangement

When we are satisfied with pile layout, click on *Select Current Arrangement* to select and apply that layout. Program will check the pile reaction against pile capacity to make sure pile reactions do not exceed pile capacity values. A message box will be popped up to inform where the assignment is successful.

4.6.2.3 Design Parameters

Clicking on *Design Parameters* leaf opens a form in the *Data Area* pane that allows you to input standard design control parameters for use in designing pile caps.


Data Input Pane
Design Paramenters
Strength of concrete
3
ksi
Yield strength of steel
60
ksi
Side cover
3
in
Bottom cover
4
in
Pile in pile cap
24
in
Initial thickness
18
in
Minimum bar size
8
Maximum bar size
11
Set as Default
No
Yield strength of steel used for design purpose. User may change the unit from the right side combo box
Column
Cs
As
Cb
Cp
Pile
Pile

The following design parameters are available:

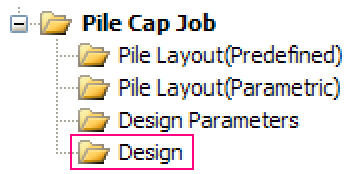
- Strength of Concrete
- Yield Strength of Steel
- Side Cover (Cs)
- Bottom Cover (Cb)
- Pile in Pile Cap (Cp)
- Initial Thickness
- Minimum Bar Size
- Maximum Bar Size

Note: The *Pile in Pile Cap* parameter refers to the length of the pile that is contained within the pile cap, as shown by the Cp parameter in the diagram at the top of the *Data Area* pane.

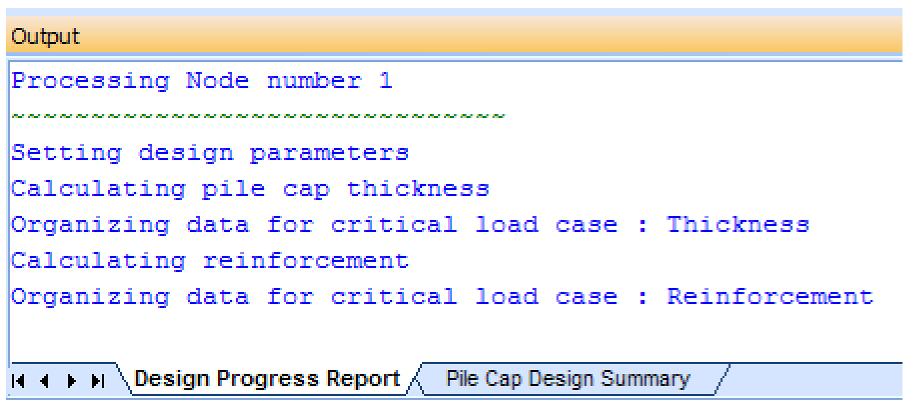
Set as Default

The *Set as Default* check box allows you to use the values inputted in the *Design Parameters* form as the default values for future projects.

4.6.2.4 Design



Click on *Design* leaf to design all the footings associated in the current job. Program will list all design progress messages including warning and error messages in the bottom output pane. It will help user to understand and review design progress.



After design is completed a new tab called *Pile Cap Design Summary* will appear in output pane. Click that tab to view design summary table. You can copy this table by selecting it and then by pressing Ctrl+C key which then can be pasted in *Microsoft Excel* or *Word*.

Output

Column No	No. of Piles	Pile cap Dimension			Reinforcement			
					X-Direction		Y-Direction	
		Length in	Width in	Thickness in	Max. Bar (A)	Bar & Spacing	Max. Bar (A)	Bar & Spacing
1	4	72.000	72.000	38.000	#8	#8 @ 18.00 in	#8	#8 @ 18.00 in

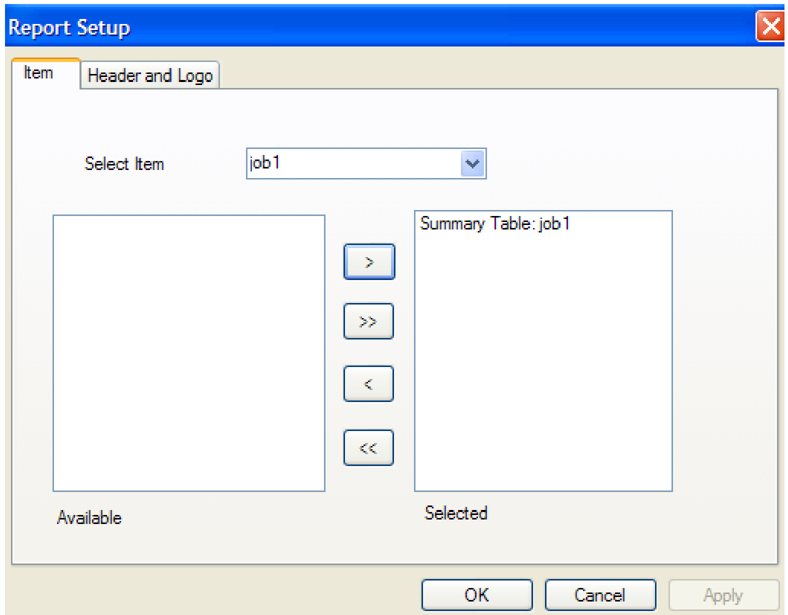
Program will automatically open calculation sheet which presents detailed step by step calculation with relevant code clause numbers, equations and corresponding calculated values. Calculation sheet is organized in a logical manner which shows program flow.

Critical Axial design Load, Pu = 0.00 kip, Load Case # 2

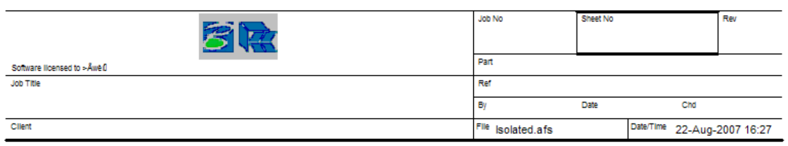
From ACI Cl.11.12.2.1, β_o for column =	$2\times(B_{col}+D_{col}+2\times d_{eff})=$	216.00	in
Equation 11-35, V_{c1} =	$\left(2+\frac{4}{\beta_c}\right)\times\beta_o\times d_{eff}\times\sqrt{1000\times F_c'}=$	3442.58	kip
Equation 11-36, V_{c2} =	$\left(2+40\times\frac{d_{eff}}{\beta_o}\right)\times\beta_o\times d_{eff}\times\sqrt{1000\times F_c'}=$	5610.13	kip
Equation 11-37, V_{c3} =	$4\times\beta_o\times d_{eff}\times\sqrt{1000\times F_c'}=$	2295.05	kip
Punching shear strength, V_c =	minimum of (V_{c1}, V_{c2}, V_{c3}) =	2295.05	kip

After successful design, you can open *Report Setup* to print design summary table. Switch to *Geometry* view in main view area to activate toolbar. Now click on *Report Setup* icon in toolbar.

It will open a report setup dialog box where under current job a summary table will be available. Select that item by clicking that item in left list box and bring it over to right size by clicking  button. Now click on OK and the dialog box will be dismissed.



Now click on Print Preview icon in toolbar to see print preview and then click on Print button to print the summary table.



Job No	Sheet No	Rev
Part		
Ref		
By	Date	Chd
File Isolated.afs	Date/Time 22-Aug-2007 16:27	

Job Title

Client

Summary Table: job1

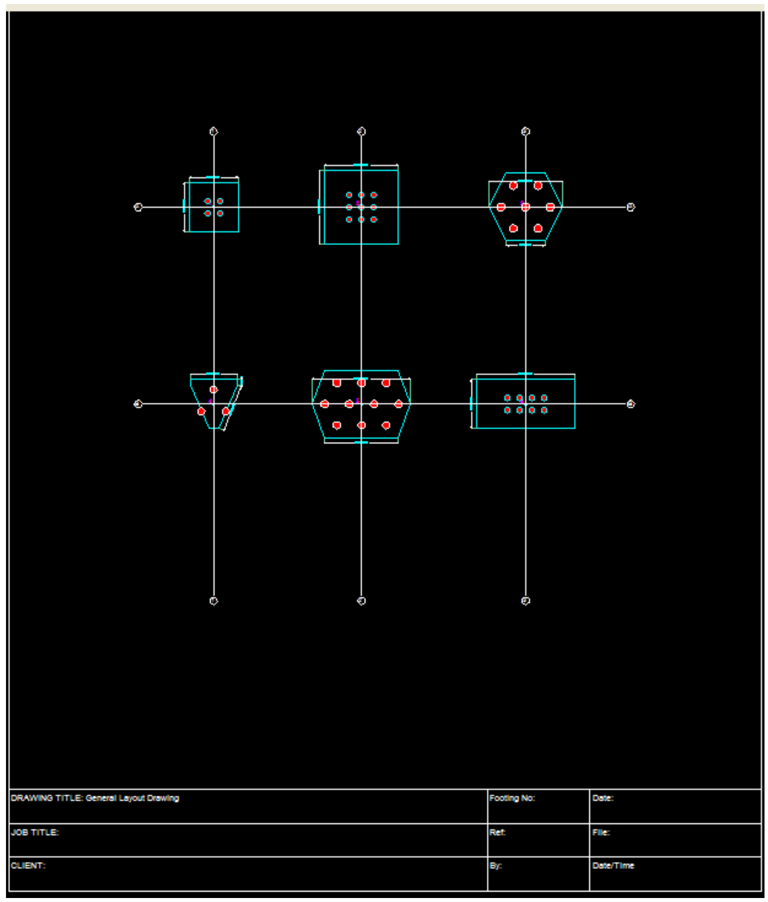
Column no	No. of piles	Footing Length (in)	Footing Width (in)	Thickness (in)
1	4	72.000	72.000	38.000

Summary Table: job1

Footing No	Max bar size (M)	Main Steel	Max bar size (S)	Secondary Steel
1	#8	#8 @ 18.00 in.	#8	#8 @ 18.00 in.

4.6.2.4.1 Layout Drawing

After successful design *Layout drawing* will be automatically drawn to scale, complete with a title block. Switch to Layout Drawing tab to view the layout drawing.

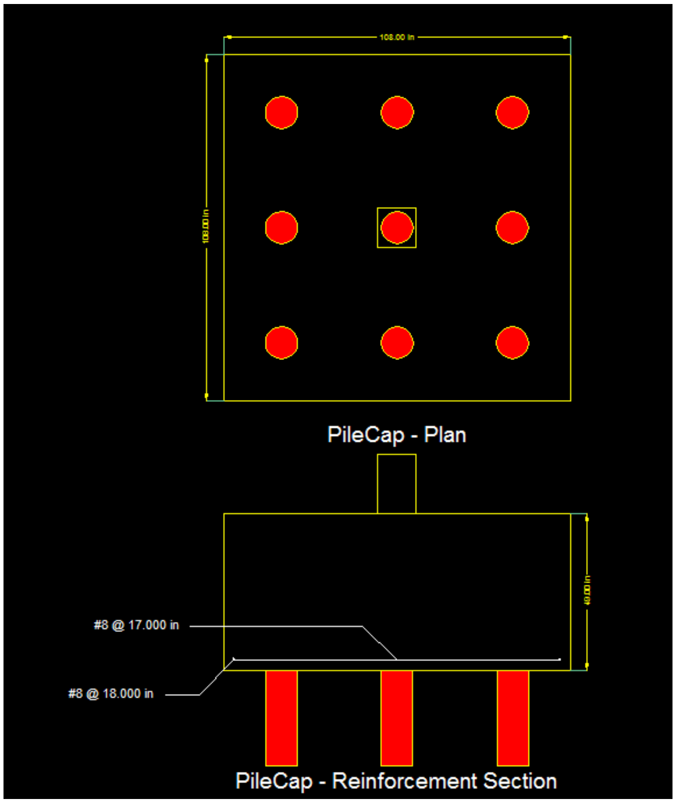


Click on *Save Drawing As* button to save the drawing in different formats including DWG and DXF.

4.6.2.4.2 Detail Drawing

After successful design *Detail drawing* will be automatically drawn complete with a title block. Switch to Detail Drawing tab to view the detailed drawing of each footing designed.

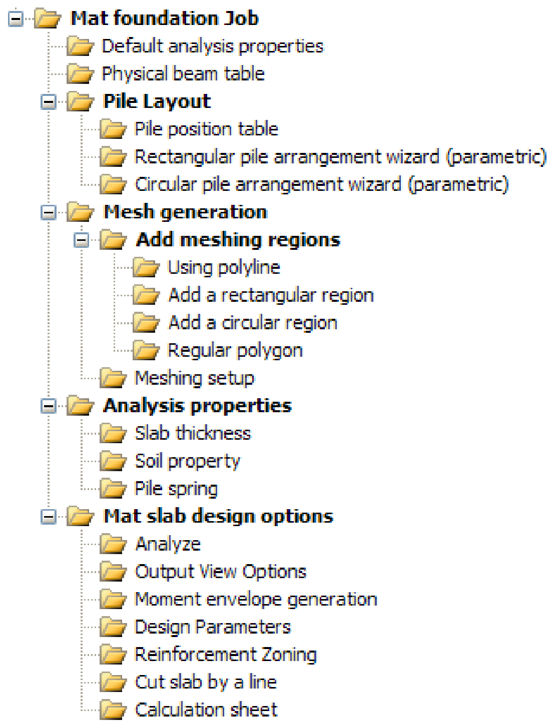
Detailed drawing shows detailed reinforcement and sectional drawing of one footing at once. Select the current footing from drop down list at the top called *Footing No*. The drawing will be automatically refreshed with selection changed.



Like layout drawing, this can be saved in different formats including DWG and DXF by clicking on *Save Drawing As* button.

4.6.3 Mat Foundation

Mat foundation local data group is called *Mat foundation Job*. The *Mat foundation Job* group allows you to create mat boundary, meshing and specify analysis and design parameters to analyze and design mat slab. Mat module uses finite element analysis technique for accurate results.



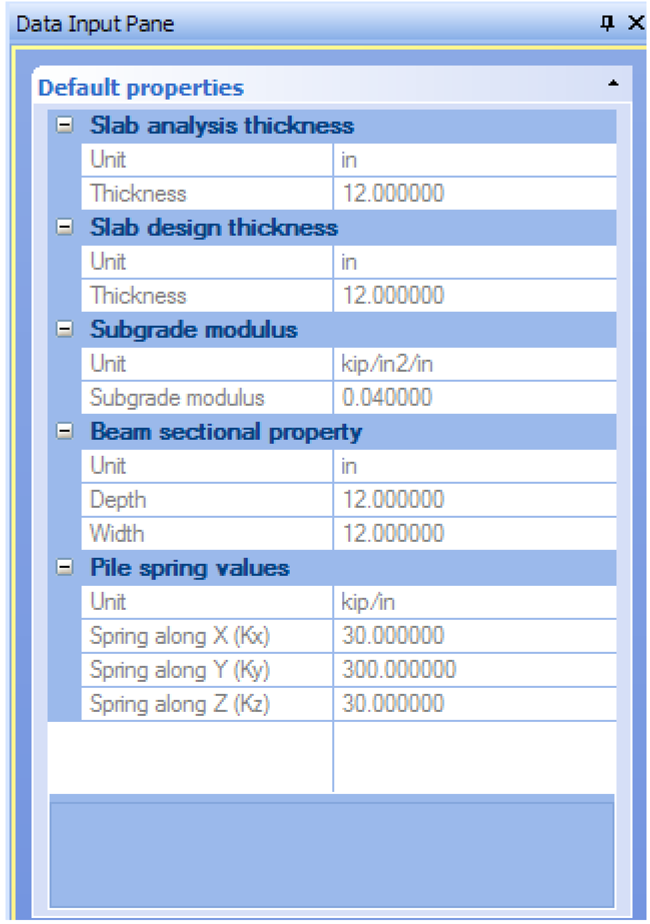
The *Mat foundation Job* group contains following groups:

- Default analysis properties
- Physical beam table
- Pile Layout
- Mesh generation
- Analysis properties
- Mat slab design options

4.6.3.1 Default analysis properties

STAAD.foundation mat foundation module is based on physical modeling environment. So, whenever a physical entity is created, properties associated with that entity will also be created. For example if we create a mat boundary, properties like slab thickness and soil properties will also be created and associated to the newly created boundary automatically. While creating these properties STAAD.foundation takes advantage of default properties setup options.

Clicking on Default analysis properties leaf will bring up a form in data area pane as shown below.



The form has five distinct groups

1. Slab analysis thickness
2. Slab design thickness
3. Subgrade modulus
4. Beam sectional property
5. Pile spring values

Slab analysis thickness

This thickness will be used during the slab FEA analysis. This parameter can have its own unit. This property is especially useful if we you want to simulate pedestal etc. for stiffness analysis but use the actual slab thickness for design. This can also be used to input uncracked thickness for analysis.

Slab design thickness

This thickness will be used during slab design. This parameter can have its own unit. This property is especially useful if we you want to simulate pedestal etc. for stiffness analysis but use the actual slab thickness for design. This can also be used to input cracked thickness for slab design.

Subgrade modulus

Subgrade modulus is a soil property available from geotechnical report. Program uses this value to calculate spring stiffness under each support node by multiplying this value with the nodal tributary area.

Beam sectional property

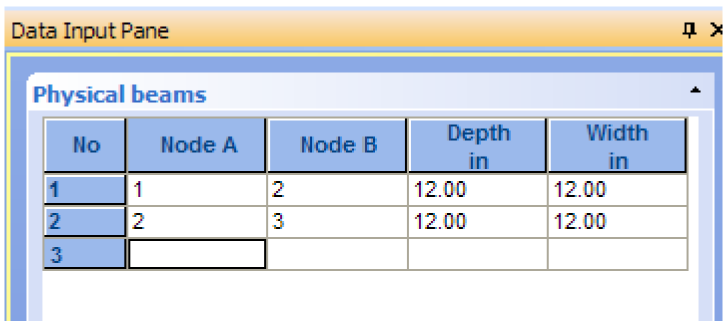
This property will be used to define cross sectional property of the physical beams added to mat foundation. Current version of the program can only have rectangular property.

Pile spring values

If the mat is supported by piles you need to create pile layout by adding piles to mat foundation. Program uses pile as spring support for analysis. So, program needs to know spring constant for those pile supports. Ky represents vertical spring constant. Kx and Kz represent lateral spring constants for respecting X and Z direction.

4.6.3.2 Physical beam table

Beams can be added to mat foundation to model additional stiffness and to transfer loads. Its called *Physical* beam because user don't need to worry about beam connectivity with meshed plates. Program internally will break these physical beams in analytical entities. Physical beams can be created between two support nodes. As you enter two support nodes a physical beam will be created and the default beam sectional property as set in default properties option will be assigned. Those values can be edited as required. The input unit for cross sectional property can be changed by clicking on tool bar icon 



No	Node A	Node B	Depth in	Width in
1	1	2	12.00	12.00
2	2	3	12.00	12.00
3				

After adding a beam the beam will be displayed in main view area.

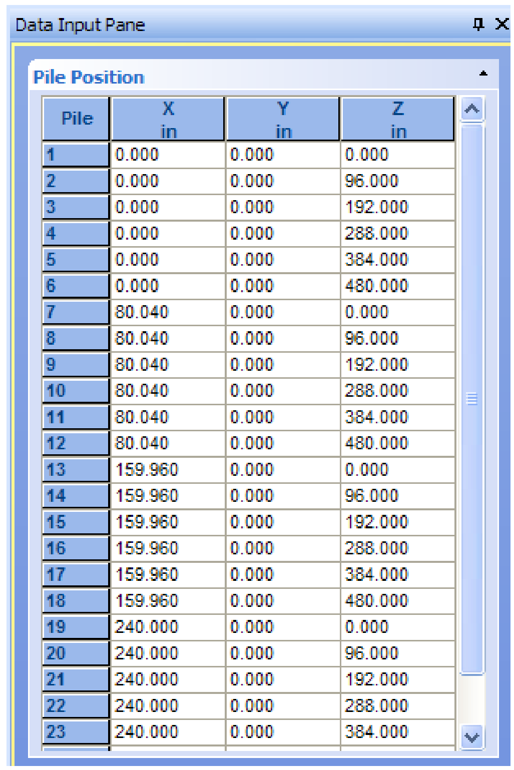
4.6.3.3 Pile Layout

Pile layout group has following elements

1. Pile Position Table
2. Rectangular Pile Arrangement Wizard (Parametric)
3. Circular Pile Arrangement Wizard (Parametric)

4.6.3.3.1 Pile Position Table

This is a grid where we can add piles by specifying their (x,y,z) coordinates. You can add as many piles as needed. Whenever a new pile is created program will automatically create default spring values for that pile. Newly created pile will be displayed in graphics view.



Pile	X in	Y in	Z in
1	0.000	0.000	0.000
2	0.000	0.000	96.000
3	0.000	0.000	192.000
4	0.000	0.000	288.000
5	0.000	0.000	384.000
6	0.000	0.000	480.000
7	80.040	0.000	0.000
8	80.040	0.000	96.000
9	80.040	0.000	192.000
10	80.040	0.000	288.000
11	80.040	0.000	384.000
12	80.040	0.000	480.000
13	159.960	0.000	0.000
14	159.960	0.000	96.000
15	159.960	0.000	192.000
16	159.960	0.000	288.000
17	159.960	0.000	384.000
18	159.960	0.000	480.000
19	240.000	0.000	0.000
20	240.000	0.000	96.000
21	240.000	0.000	192.000
22	240.000	0.000	288.000
23	240.000	0.000	384.000

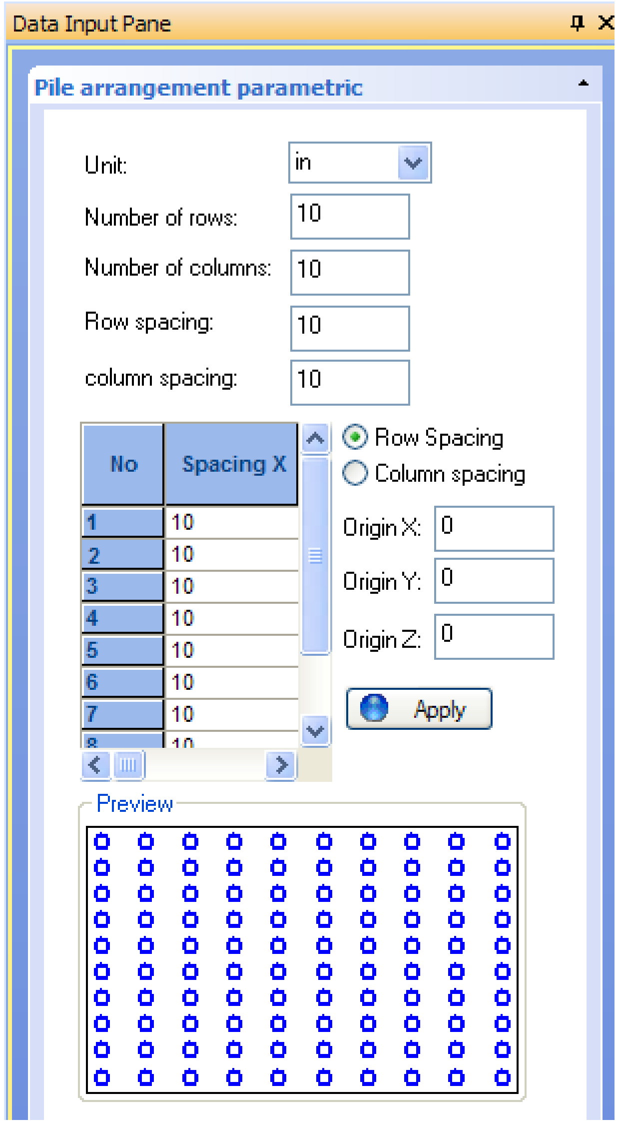
4.6.3.3.2 Rectangular Pile Arrangement Wizard (Parametric)

This is a parametric wizard like input to create rectangular pile layout. Generated pile coordinates will be in local coordinate system where first pile is at 0,0,0 position. You need to move pile group to the right location by inputting Origin X, Origin Y and Origin Z.

The following commands and options are available to generate pile layout.

Unit - Length Unit for row and column spacing
Number of Rows - Specify number of rows in layout grid
Number of Columns - Specify number of columns in layout grid
Row Spacing – Spacing between rows
Column Spacing – Spacing between columns

By default program will create symmetric pile arrangement from the above input but user can change the default setup by editing the table below. Both row and column grid lines can be adjusted by selecting appropriate radio button.



Apply- It will transfer the pile layout to the main view and add piles to the current mat foundation job. Please note, you should input origin to move the layout to the right position.

4.6.3.3.3 Circular Pile Arrangement Wizard (Parametric)

This is a parametric wizard like input to create circular pile layout. Generated pile coordinates will be in local coordinate system where center of the circle is at 0,0,0 position. You need to move pile group to the right location by inputting Origin X, Origin Y and Origin Z.

The following commands and options are available to generate circular pile layout.

Unit

Length Unit for pile spacing

Number of Piles

Total number of piles in pile group

Number of Circular layers

Number of concentric circular pile layers

Pile Spacing

Minimum spacing between two piles in above mentioned length unit

Center Pile

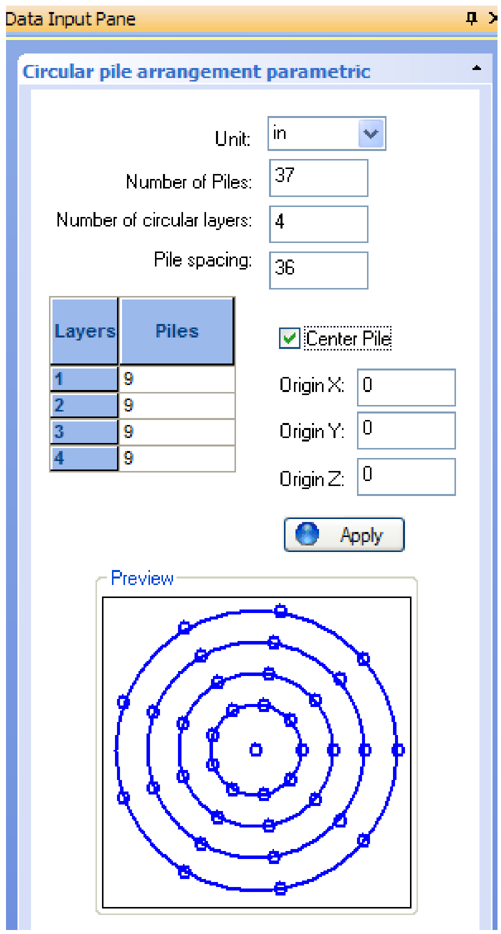
Check this box if you want to add a pile at the center of the circle which is at 0,0,0. If you check this box program will automatically add an extra piles to the total count of number of piles.

By default program will create symmetric pile arrangement from the above input. It will attempt to place equal number of piles to all layers. It will create an additional layer for the remainder of piles.

User can change the default setup by editing the layers table as shown below.

Apply

Click on apply button to transfer pile layout to graphics and add to the current mat foundation job. Please do remember to input appropriate origin coordinates to move the whole pile group to the right position.



4.6.3.4 Mesh generation

As Mat foundation module is based on FEA analysis, program needs to generate plate elements. STAAD.foundation has automatic mesh generation tools and it can generate both quadrilateral and triangular mesh for any shape and size.

It has two categories

1) Add meshing region
2) Meshing setup

4.6.3.4.1 Add meshing region

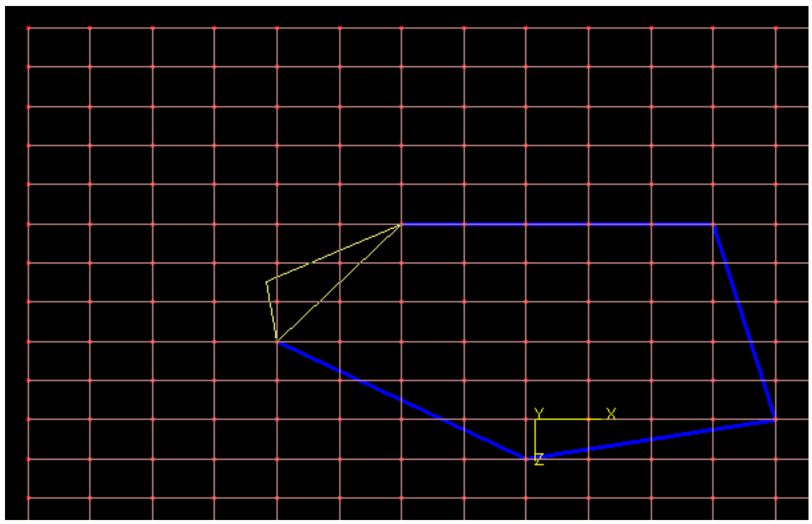
This section is used to create meshing regions which are mat boundary, holes, control regions etc. Four methods are used to create meshing regions which are

- Using polyline
- Add a rectangular region
- Add a circular region
- Regular polygon

4.6.3.4.1.1 Using polyline

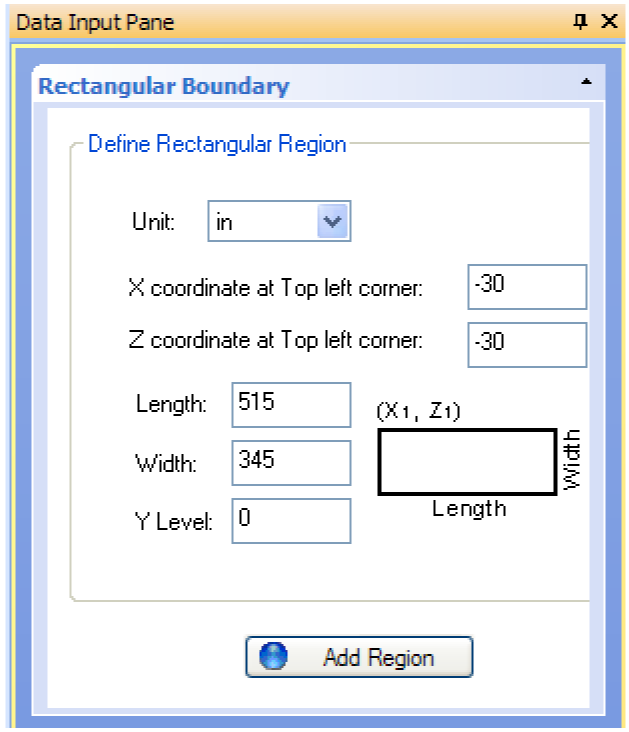
The *Using polyline* option allows you to draw on the grid a mat boundary that represents the edge of a slab. To draw a mat boundary, click in sequence on the points on the grid going in either a clockwise or a counter-clockwise order. Once you have clicked on all the points that define the boundary of your slab, return to your starting point or right-click. You will see a blue closed polygon defining the boundary you have created.

In the figure below blue lines indicate those points are already clicked and yellow lines shows possible closed polygon connectivity.



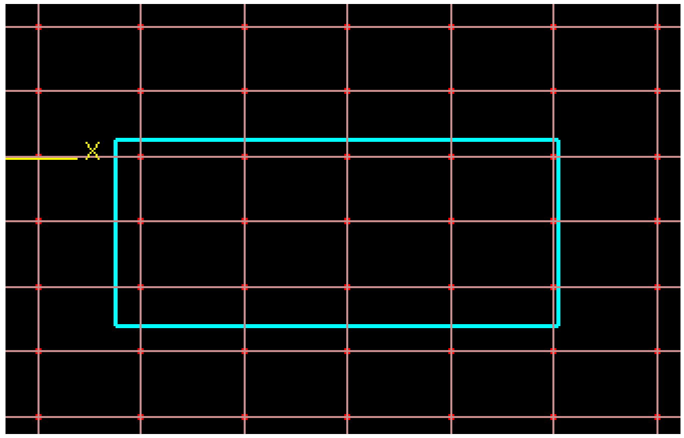
4.6.3.4.1.2 Add a rectangular region

Clicking on *Add a rectangular region* option will open a form in data area pane that allows you to create a rectangular region.



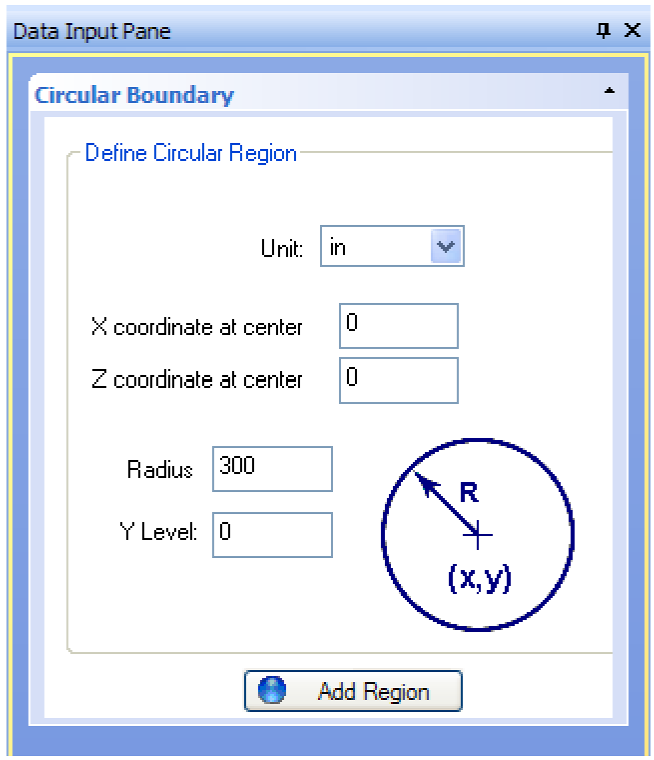
It's a very simple self explanatory form where you need to input coordinates for top left corner on XZ plane and then specify the length and width of the rectangle you are about to create.

Click on *Add Region* and that will create a rectangular region in main view area.


X

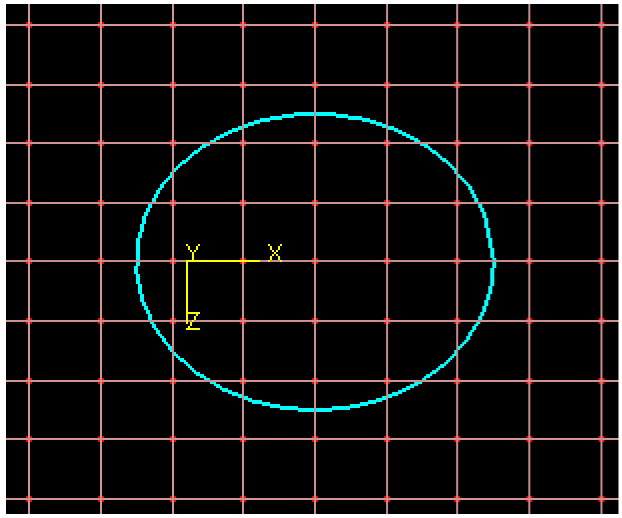
4.6.3.4.1.3 Add a circular region

Clicking on *Add a circular region* option will open a form in data area pane that allows you to create a circular region.



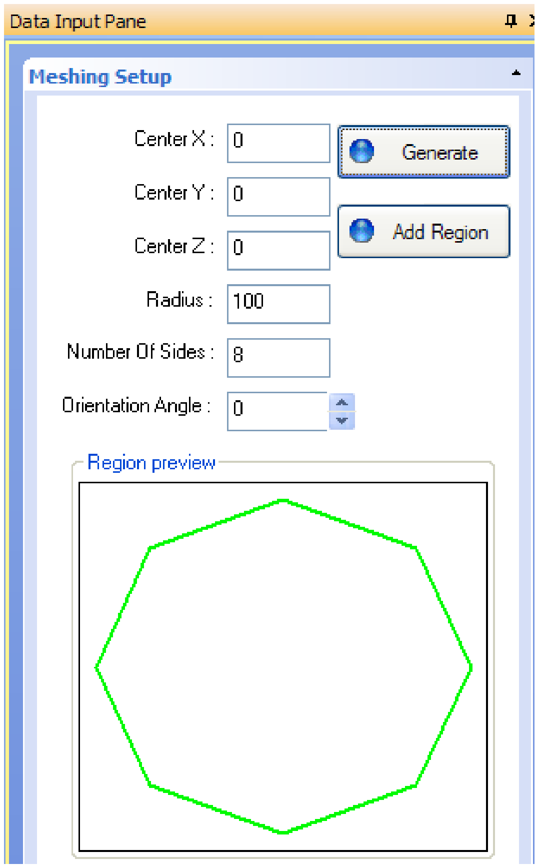
It's a very simple self explanatory form where you need to input coordinates for the center of the circle on XZ plane and then specify radius of the circle.

Click on *Add Region* and that will create a circular region in main view area.


Y
X
Z

4.6.3.4.1.4 Regular Polygon

Clicking on *Regular Polygon* option will open a form in data area pane that allows you to create regular shaped convex polygonal region.



The form generates any sided regular shaped polygon. To generate the polygon you need to input

Center of the polygon

Specify X,Y,Z coordinate of polygon center

Radius

Circular radius of the polygon where radius is the distance measured between center and each vertex of the polygon.

Number of sides

Number of polygon sides. For example enter 8 for an octagonal shaped polygon.

Orientation Angle

This is the rotation angle of the polygon. Change the angle to get your desired orientation.

Generate

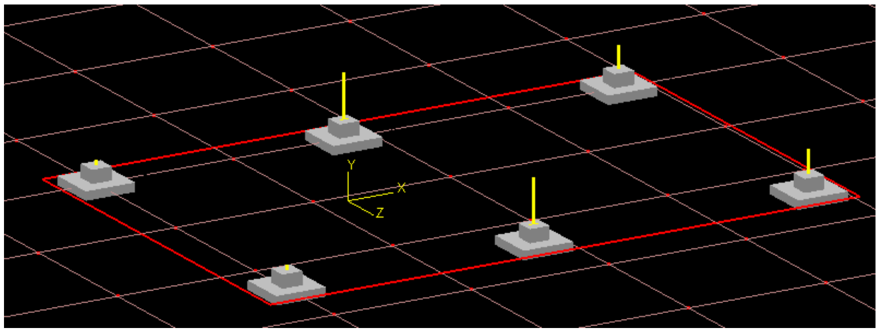
It will generate the polygon and will display in Region Preview window. Please note, the generation will be in XZ plane.

Add Region

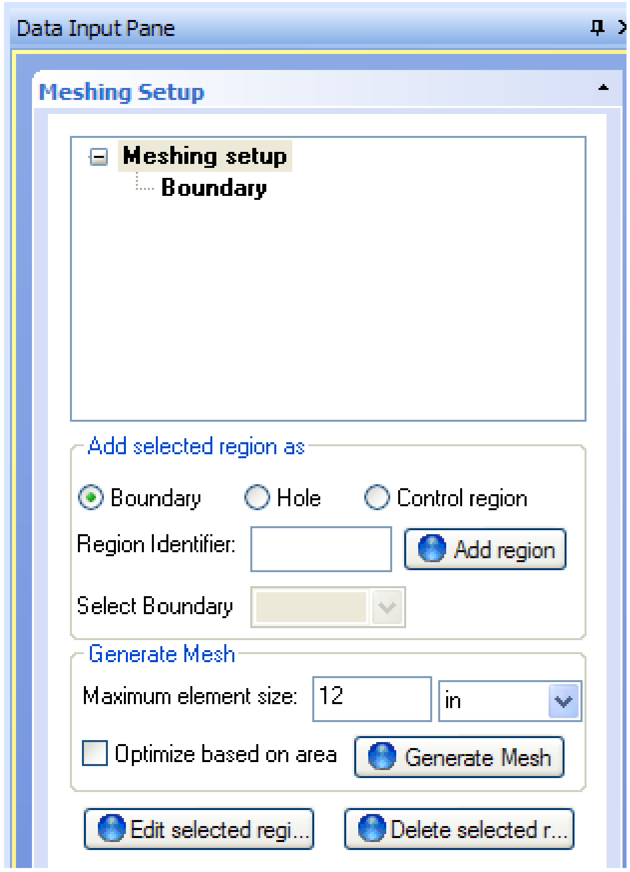
Add region will add the generated polygon to the main view area

4.6.3.4.2 Meshing Setup

Meshing setup option allows you to add created region to the current job and then to generate mesh. To add a region to the current job, first select that region in the graphics. Selected region will be highlighted.



There are three options to add a region. It can be added as a boundary or as a hole or as a control region to an already created boundary.



Boundary

This option is to add region as the main mat boundary. It is the outermost region of the mat foundation. You can have as many boundaries as needed. Boundaries can be connected or isolated.

Hole

This option is to specify a hole within a mat boundary. You can add as many holes as needed. Please note, holes must not intersect each other or the boundary or any control region.

Control Region

This option is to specify a special region within a mat boundary which might have a different slab thickness or soil property. You can add as many control regions as needed. Please note, these regions must not intersect each other or the boundary or any hole.

Region identifier

It is a unique identifier of the region to be added. Any string can be used.

Add Region

Click on this button to add the selected region in the main view to the current job

Select Boundary

This option will be active only when the region type will be selected as *Hole* or *Control Region*. Select the boundary to which the hole or control region will be added.

Maximum element size

It's the size of one side of a plate element to be created. This parameter will be used by mesh generation engine to generate plates. This option allows you to control meshing density and plate counts which in turn control analysis run time and output size.

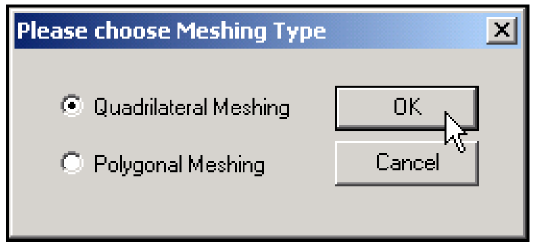
Optimize based on area

It's an optimization technique to be used only for triangular plate generation for non quad mat boundary. By default, program optimize meshing based on element size.

Generate Mesh

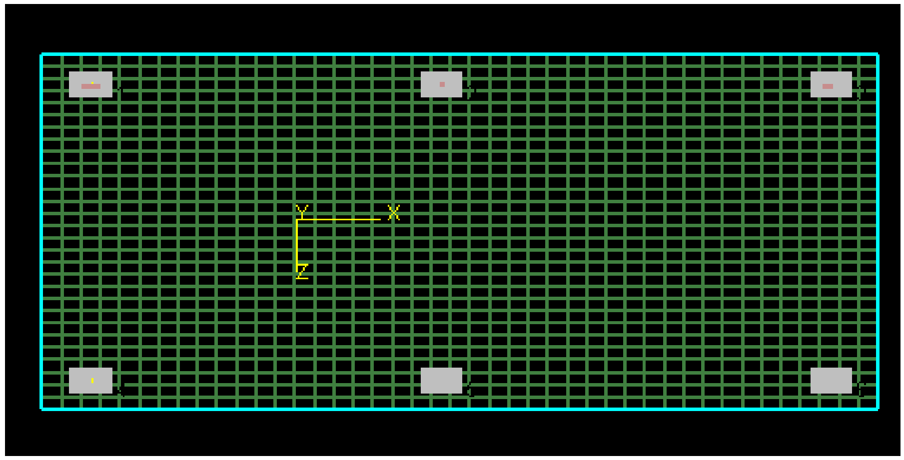
Click on this button to generate mesh of the selected bounding region. Once you click on the *Create Mesh* button, a dialog box will appear allowing you to choose the type of meshing to create. The following two types of meshing are available:

- Quadrilateral Meshing
- Polygonal Meshing.

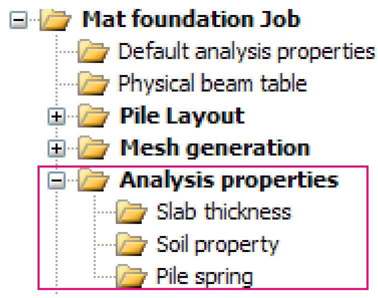


A *Quadrilateral Mesh* works well for slabs with quadrilateral boundaries and when there is no hole or control region. A *Polygonal Mesh* is the better choice for slabs with irregular shapes, like a Y-shaped slab, or slabs with round holes, irregular-shaped holes, round edges, etc.

After you have selected the desired meshing type and clicked *OK*, STAAD.*foundation* will create the mesh and display it in the *Graphics Window*.


Y
X
Z

4.6.4 Analysis properties

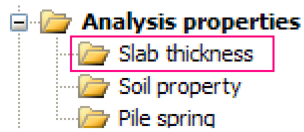


The *Analysis properties* group allows you to input slab thicknesses, soil properties and pile spring constants. This group is only active for mat foundation job types.

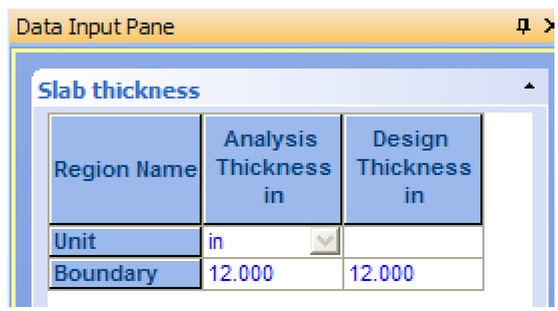
The *Analysis properties* group contains the following elements:

- Slab thickness
- Soil Property
- Pile spring

4.6.4.1 Slab Thickness



Clicking on the *Slab Thickness* leaf opens a table in the *Data Area* pane that allows you to change the element thickness for the plate elements in a mesh you are using to model a mat foundation.

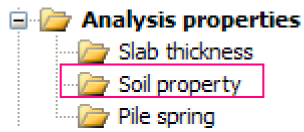


As slab is added as a physical entity in STAAD.foundation, default slab thickness property will be automatically created and assigned to each slab region.

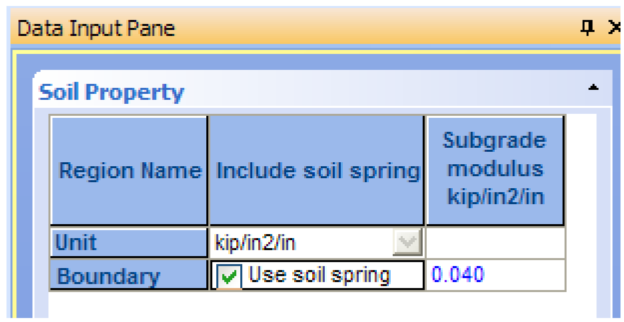
The first row of the table is to select unit for thickness. You can have only one unit for all slab thickness. Second row onwards will be list of slab thickness properties. Left most cells of each row will show the region identifier name as specified in Meshing Setup operation.

You can have different thickness for analysis and design. Analysis thickness will be used for FEM analysis of mat foundation and design thickness will be used to design the mat slab.

4.6.4.2 Soil Property



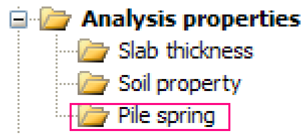
Clicking on the *Soil Property* leaf opens a table in the *Data Area* pane that allows you to change and assign soil properties for the design of mat foundations.



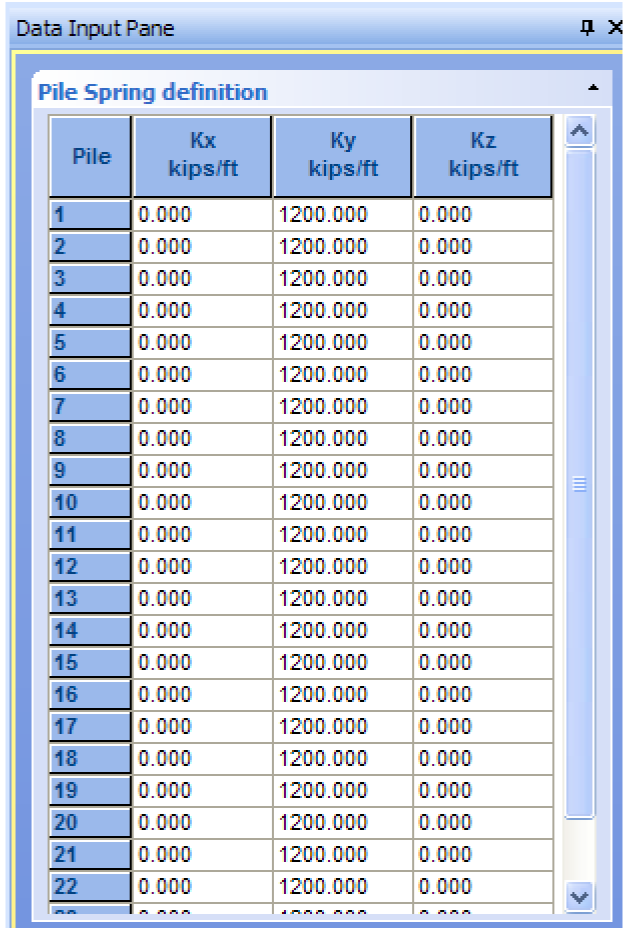
As slab is added as a physical entity in STAAD.foundation, default soil property will be automatically created for each slab region. But by default soil property will not be assigned to the region as the mat foundation could be supported on piles only.

If the soil spring is not assigned to the region, value for subgrade modules will be shown in Red color. Click on the *Include soil spring* check box to assign the soil property to the region. If Include soil spring check box is checked, value for subgrade modulus will be shown in blue color.

4.6.4.3 Pile Spring



Clicking on the *Pile Spring* leaf opens a table in the data area pane that allows you to edit the pile spring constant values for all the piles present in the current job.



Pile	Kx kips/ft	Ky kips/ft	Kz kips/ft
1	0.000	1200.000	0.000
2	0.000	1200.000	0.000
3	0.000	1200.000	0.000
4	0.000	1200.000	0.000
5	0.000	1200.000	0.000
6	0.000	1200.000	0.000
7	0.000	1200.000	0.000
8	0.000	1200.000	0.000
9	0.000	1200.000	0.000
10	0.000	1200.000	0.000
11	0.000	1200.000	0.000
12	0.000	1200.000	0.000
13	0.000	1200.000	0.000
14	0.000	1200.000	0.000
15	0.000	1200.000	0.000
16	0.000	1200.000	0.000
17	0.000	1200.000	0.000
18	0.000	1200.000	0.000
19	0.000	1200.000	0.000
20	0.000	1200.000	0.000
21	0.000	1200.000	0.000
22	0.000	1200.000	0.000

The following pile support properties are available:

- Kx
- Ky
- Kz

Kx

The *Kx* field allows you to specify a spring constant K value for the X-Direction.

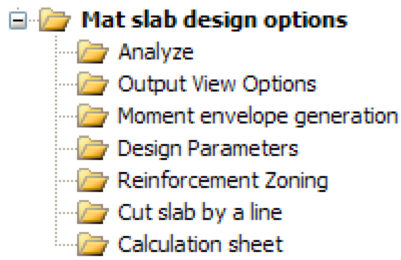
Ky

The *Ky* field allows you to specify a spring constant K value for the Y-Direction.

Kz

The *Kz* field allows you to specify a spring constant K value for the Z-Direction.

4.6.5 Mat slab design options

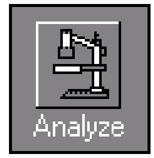


The *Mat slab design options* group allows you to perform an analysis on a mat foundation, review output results and finally design mat foundation. This group is only active for mat foundation job types.

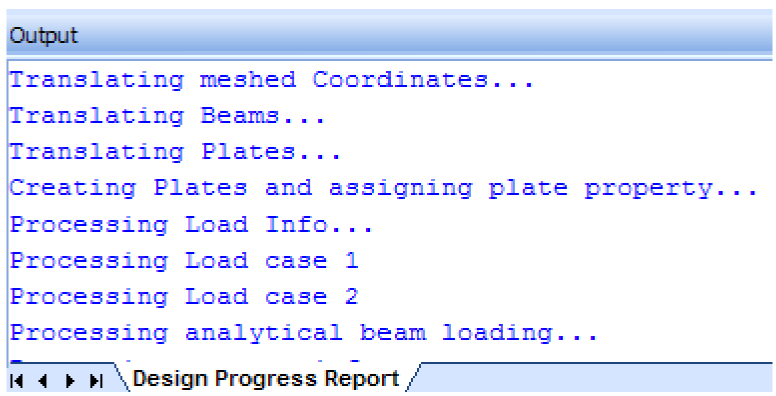
The *Mat slab design options* group contains following elements:

- Analyze
- Output View Options
- Moment envelope generation
- Design parameters
- Reinforcement Zoning
- Cut slab by a line
- Calculation sheet

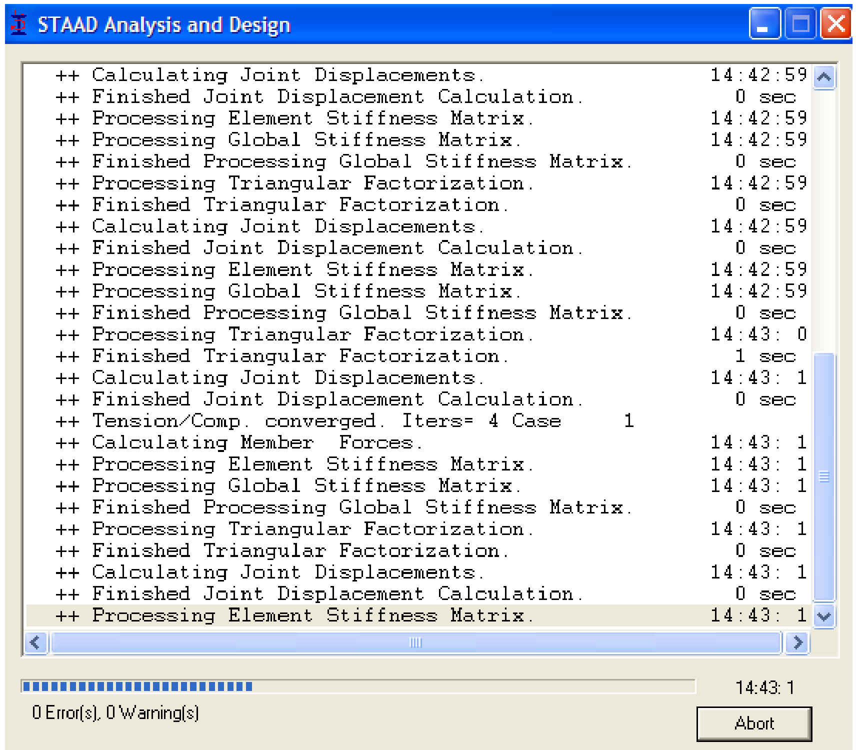
4.6.5.1 Analyze



Clicking on the *Analyze* leaf allows you to analyze a mat foundation. All data relevant to performing an analysis, including slab boundary, plate thickness and soil properties, must be entered prior to selecting this command, otherwise you will not obtain a successful analysis. After clicking on the *Analyze* leaf, output pane will display analysis progress messages where program will create an analytical model by decomposition of the foundation structure.



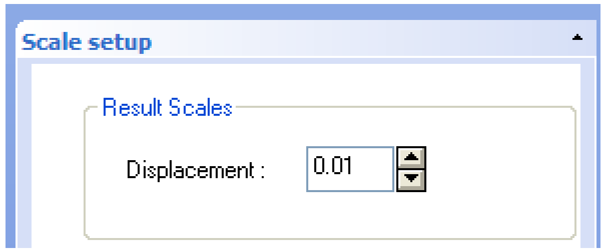
After analytical model is created, program will launch its analysis engine to analyze the structure. A separate window will be shown displaying analysis progress messages. Once the analysis is completed this window will be automatically dismissed.

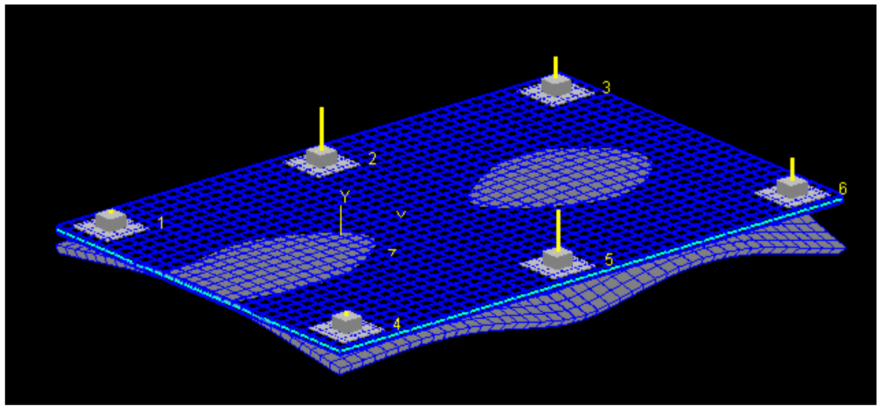


After successful analysis, program will convert analytical results to physical entity based results to allow user to review output and design slab.

By default, the deformed plates showing the node displacements will appear in the *Graphics Window*. To change the viewing scale of the displacement diagram, click on the *Scale* icon in toolbar.

It will open a form in the data area pane. Change the Displacement scale for suitable display of results diagram. Please note that increasing scale will make diagram appear smaller.

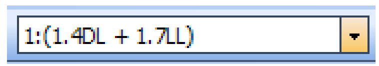




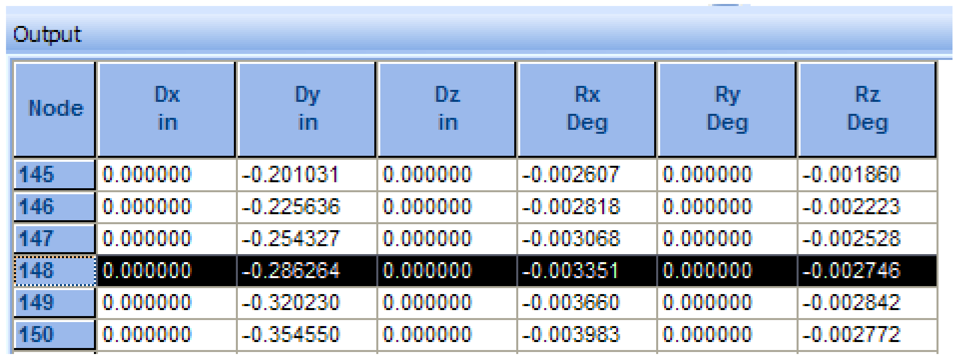
After successful analysis, the program will add several tabs in output pane to display different output results like node displacement, plate stress, support reaction etc. Tabs for beam analysis results like beam section force will be added if the current job has physical beams defined.

4.6.5.2 Displacement

Click on the *Displacement* tab to view node displacement table for all nodes for current load case as selected in *select current load case* icon in toolbar.

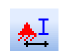


Clicking on any row of the table will highlight that node in the graphics.

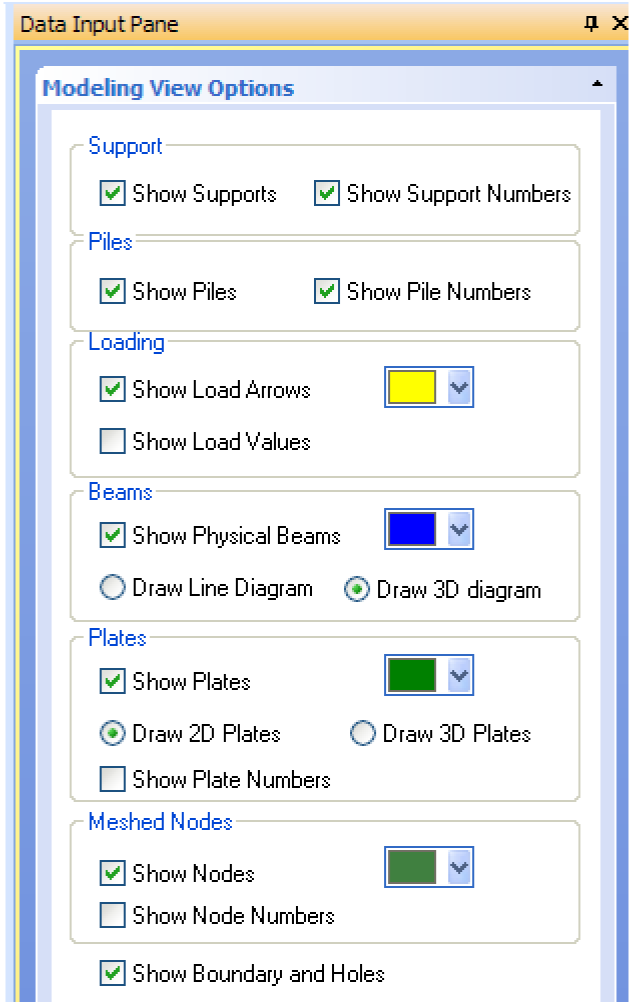

Output

Node	Dx in	Dy in	Dz in	Rx Deg	Ry Deg	Rz Deg
145	0.000000	-0.201031	0.000000	-0.002607	0.000000	-0.001860
146	0.000000	-0.225636	0.000000	-0.002818	0.000000	-0.002223
147	0.000000	-0.254327	0.000000	-0.003068	0.000000	-0.002528
148	0.000000	-0.286264	0.000000	-0.003351	0.000000	-0.002746
149	0.000000	-0.320230	0.000000	-0.003660	0.000000	-0.002842
150	0.000000	-0.354550	0.000000	-0.003983	0.000000	-0.002772

If the node is not highlighted, click on the *View Options* icon in the toolbar.



It will open a form in data area pane which will allow you to setup view options. Check on *Show Nodes* option under *Meshed Nodes* group to display meshed nodes in the graphics.


Data Input Pane
Modeling View Options
Support
Show Supports
Show Support Numbers
Piles
Show Piles
Show Pile Numbers
Loading
Show Load Arrows
Show Load Values
Beams
Show Physical Beams
Draw Line Diagram
Draw 3D diagram
Plates
Show Plates
Draw 2D Plates
Draw 3D Plates
Show Plate Numbers
Meshed Nodes
Show Nodes
Show Node Numbers
Show Boundary and Holes

4.6.5.3 Disp (Displacement) summary

Click on the *Disp Summary* tab to view node displacement summary table among all load cases.

It displays 12 rows where each row shows either maximum or minimum value for a particular degree of freedom. It also displays corresponding displacement values for other degrees of freedom on that row. The table first lists three translational degrees of freedom and then three rotational degrees of freedom. First row of each degree of freedom starts with maximum value. Please note, here minimum and maximum are algebraic signed values.

Output

	Node	Load Case	Dx in	Dy in	Dz in	Rx Deg	Ry Deg	Rz Deg
Max Dx	1	1	**0.000000**	-0.296130	0.000000	-0.002332	0.000000	0.000834
Min Dx	1	1	**0.000000**	-0.296130	0.000000	-0.002332	0.000000	0.000834
Max Dy	693	1	0.000000	**0.049418**	0.000000	0.000150	0.000000	-0.000102
Min Dy	1320	1	0.000000	**-0.792751**	0.000000	0.005146	0.000000	-0.003932
Max Dz	1	1	0.000000	-0.296130	**0.000000**	-0.002332	0.000000	0.000834
Min Dz	1	1	0.000000	-0.296130	**0.000000**	-0.002332	0.000000	0.000834
Max Rx	1144	1	0.000000	-0.523888	0.000000	**0.006002**	0.000000	-0.003256
Min Rx	220	1	0.000000	-0.519242	0.000000	**-0.005586**	0.000000	-0.003260
Max Ry	1	1	0.000000	-0.296130	0.000000	-0.002332	**0.000000**	0.000834
Min Ry	1	1	0.000000	-0.296130	0.000000	-0.002332	**0.000000**	0.000834
Max Rz	1302	1	0.000000	-0.472672	0.000000	0.004098	0.000000	**0.003573**
Min Rz	1316	1	0.000000	-0.579236	0.000000	0.004439	0.000000	**-0.004835**

4.6.5.4 (Support) Reaction(s)

Click on the *Reaction* tab to review support reaction results. This option is available only if the mat is supported on soil. In case of mat supported by soil each plate node of the mat region will have one soil spring attached to it.

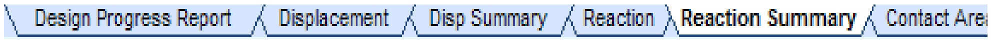
Reaction tab shows support reactions for current job for current load case only. Please select your desired load case from *Select Current Load* icon in toolbar.

The table shows reactions for all six degree of freedom for all nodes. Clicking on any row will highlight the corresponding node in graphics.

Output

Node	Fx kip	Fy kip	Fz kip	Mx kip-in	My kip-in	Mz kip-in
1	0	0.4219324	0	0	0	0
2	0	0.8148363	0	0	0	0
3	0	0.7834994	0	0	0	0
4	0	0.7491724	0	0	0	0
5	0	0.713971	0	0	0	0
6	0	0.6823493	0	0	0	0
7	0	0.6589678	0	0	0	0
8	0	0.6475495	0	0	0	0
9	0	0.6508462	0	0	0	0
10	0	0.6707619	0	0	0	0
11	0	0.7084866	0	0	0	0
12	0	0.7645538	0	0	0	0
13	0	0.838753	0	0	0	0

4.6.5.5 (Support) Reaction Summary



Click on the *Reaction Summary* tab to review support reaction summary results. This option is available only if the mat is supported on soil.

Reaction summary table displays maximum and minimum reaction forces for all directions among all load cases. Each row displays either a maximum or minimum value of a particular DOF along with node and load case number. Clicking on any row will highlight corresponding node in the graphics.

Output

	Node	Load Case	Fx kip	Fy kip	Fz kip	Mx kip-in	My kip-in	Mz kip-in
Max Fx	1192	1	**1.496520042**	1.239615082	-0.256951004	0	0.003219750	0
Min Fx	1230	1	**-1.917670011**	3.298340082	-0.284902006	0	-0.000755416	0
Max Fy	1275	1	0	**3.902504444**	0	0	0	0
Min Fy	471	1	0	**0**	0	0	0	0
Max Fz	154	1	0.134885996	2.467761278	**0.204851001**	0	-0.001639820	0
Min Fz	1230	2	-1.218780040	2.879164934	**-5.640240192**	0	0.856886982	0
Max M	1	1	0	0.421932399	0	**0**	0	0
Min Mx	1	1	0	0.421932399	0	**0**	0	0
Max M	1192	2	1.244420051	1.088517308	-2.414170026	0	**4.461840152**	0
Min My	136	1	1.068950057	1.036438465	0.183538004	0	**-0.002299840**	0
Max M	1	1	0	0.421932399	0	0	0	**0**
Min Mz	1	1	0	0.421932399	0	0	0	**0**

4.6.5.6 Contact Area

Click on *Contact Area* tab to review slab and soil contact information. The table displays area in contact and area out of contact with the soil for each load case. This option is available only for Mat slab supported by soil.

Output

Load Case	Area in Contact (in²)	% of Total Area	Area Out of Contact (in²)	% of Total Area
1	148964.9012222	83.84121906447	28710.11696625	16.15878093553
2	148110.0090714	83.36006411112	29565.00911713	16.63993588888

4.6.5.7 Plate Stresses

Click on *Plate Stress* tab to open plate stress table. It displays 8 basic stress types for current load case. The stress types are

- SQX
- SQY
- SX
- SY
- SXY
- MX
- MY
- MXY

These stresses are based on plate local coordinate system. During slab design program will automatically transform these local stresses to global axes system.

Output

Plate	SQX kip/in2	SQY kip/in2	SX kip/in2	SY kip/in2	SXY kip/in2	MX kip-in/in	MY kip-in/in	MXY kip-in/in
1	0.003575	0.006584	0.000000	0.000000	0.000000	1.022040	1.121771	-0.813954
2	-0.007066	0.014716	0.000000	0.000000	0.000000	3.014122	0.783163	-1.431519
3	-0.027232	0.014872	0.000000	0.000000	0.000000	2.750760	0.503668	-1.043669
4	-0.040070	0.008342	0.000000	0.000000	0.000000	-1.204627	0.397224	-0.580257
5	-0.040132	0.001704	0.000000	0.000000	0.000000	-6.574391	0.261886	-0.572458
6	-0.031777	-0.000615	0.000000	0.000000	0.000000	-10.978528	0.016307	-0.578476
7	-0.021131	-0.002165	0.000000	0.000000	0.000000	-14.251686	-0.229303	-0.218469

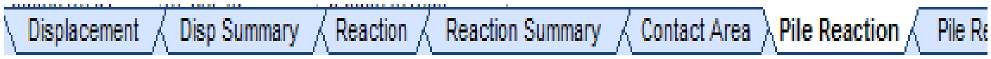
4.6.5.8 Plate Stresses Summary

Like all other summary table plate stress summary table displays minimum and maximum stress of all stress types among all load cases along with plate and load number.

Output

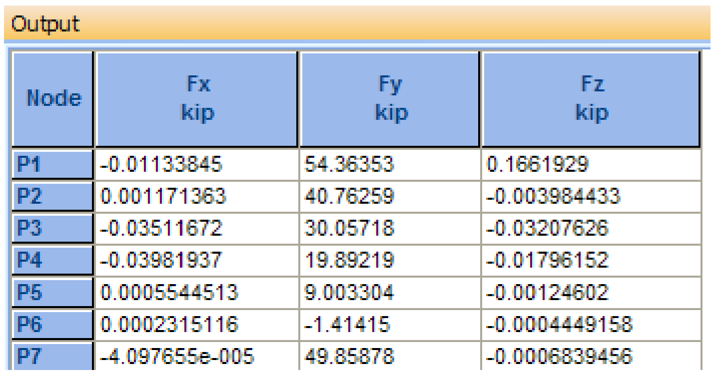
	Plate	Load	SQX kip/in2	SQY kip/in2	SX kip/in2	SY kip/in2	SXY kip/in2	MX kip-in/in	MY kip-in/in	MXY kip-in/in
Max S	106	1	**0.287948**	-0.015921	0.000000	0.000000	0.000000	49.995811	26.713346	4.157263
Min SQ	108	1	**-0.302032**	-0.014793	0.000000	0.000000	0.000000	50.613258	27.294970	-2.959754
Max S	1096	1	0.007524	**0.254060**	0.000000	0.000000	0.000000	52.232689	10.750283	-0.807425
Min SQ	150	1	0.016595	**-0.305387**	0.000000	0.000000	0.000000	55.269028	19.792627	0.649519
Max SX	1	1	0.003575	0.006584	**0.000000**	0.000000	0.000000	1.022040	1.121771	-0.813954
Min SX	1	1	0.003575	0.006584	**0.000000**	0.000000	0.000000	1.022040	1.121771	-0.813954
Max SY	1	1	0.003575	0.006584	0.000000	**0.000000**	0.000000	1.022040	1.121771	-0.813954

4.6.5.9 Pile Reaction



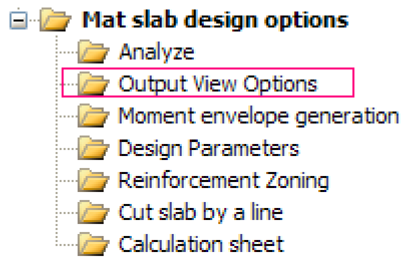
If the mat is supported by piles there will be an additional *Pile Reaction* table will be added in *Output* pane.

Pile Reaction table displays reaction forces on all piles present in current job. Piles are treated as spring support where all rotational degrees are released. So, the table displays three translational reactions for each pile.

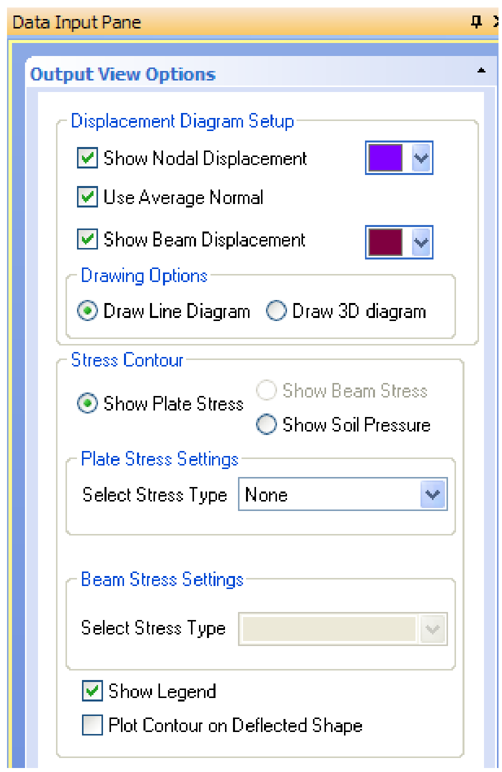

Output

Node	Fx kip	Fy kip	Fz kip
P1	-0.01133845	54.36353	0.1661929
P2	0.001171363	40.76259	-0.003984433
P3	-0.03511672	30.05718	-0.03207626
P4	-0.03981937	19.89219	-0.01796152
P5	0.0005544513	9.003304	-0.00124602
P6	0.0002315116	-1.41415	-0.0004449158
P7	-4.097655e-005	49.85878	-0.0006839456

4.6.5.10 Output View Options



Click on *Output View Options* to display different sets of output like displacement diagrams, soil pressure contour, plate stress contour etc. This option will open a form in data area pane where user will be able to select different options.



The following options are available to setup output view options

Show Nodal Displacement

Selecting this option will show displacement diagram for current load case in graphic area. The color picker control next to this check box allows user to select a suitable color to be used to draw the displacement diagram.

Use Average Normal

This option is used to draw 3D displacement diagram where lighting will be applied to the average normal direction.

Show Beam Displacement

Selecting this option will allow user to draw beam displacement diagram if present in current job. The color picker control at right side of this checkbox allows user to choose a suitable color which will be used to draw beam displacement diagram.

Drawing options

Displacement diagrams can be drawn as wireframe or as a true 3D solid diagram. Draw line diagram option will draw a wireframe diagram of the displaced shape. Draw 3D diagram will draw plates and beam displacements as 3D solid diagram.

Stress Contour

There are three types of contours available.

1. Plate Stress
2. Beam Stress
3. Soil Pressure

If you select *Show Plate Stress* contour, *Select Stress Type* drop down box will be enabled allowing you to choose stress type to display. By default program shows stress type as *None*.

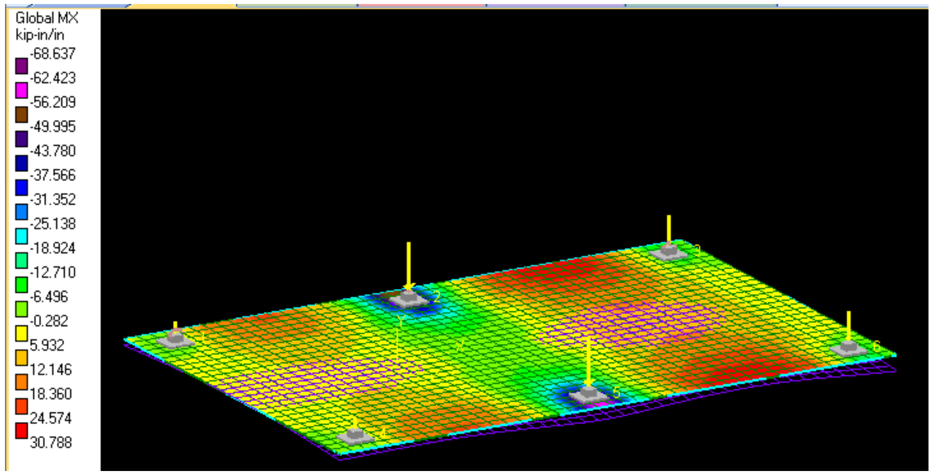
Two categories of plate stress contours are available. One set displays contour for plate local axis system and the other set shows global plate moment.

Local stresses are

- Max Absolute
- Max Top
- Max Bottom
- Max Von Mis
- Max Von Mis Top
- Max Von Mis Bottom
- SX
- SY
- SXY
- MX
- MY
- MXY
- SQX
- SQY

Global moments are available for both MX and MZ.

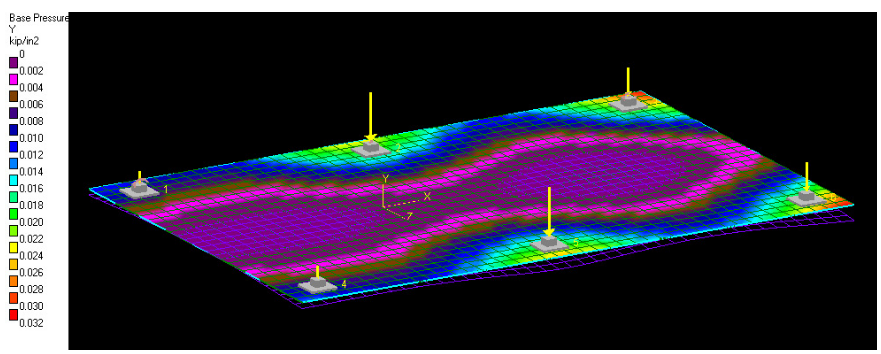
After selecting suitable stress type program will display contour in graphics window along with a legend.



Show Soil Pressure

If you choose stress contour type as Show Soil Pressure, program will display soil pressure contour for the selected load case along with a legend.

Please note, soil pressure values are directly related to soil bearing capacity. If the maximum pressure exceeds soil bearing capacity you need to increase mat dimension and run the analysis again. Base pressure for each node is calculated dividing the reaction of a plate node by the tributary area of that node.

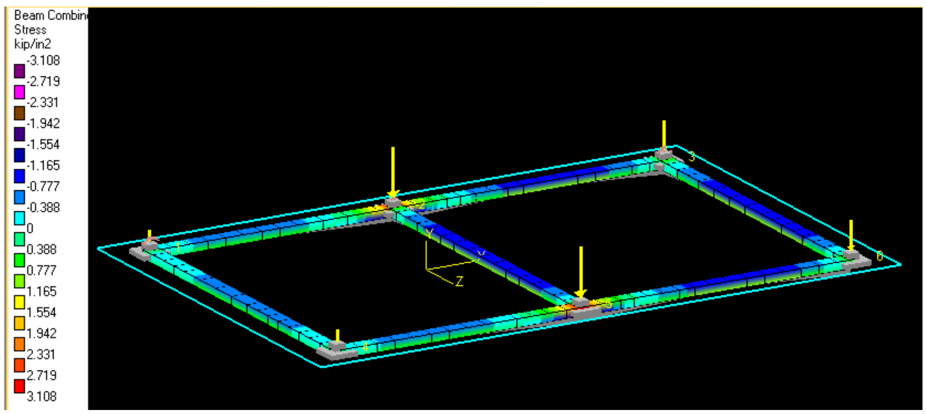


Show Beam Stress

This option is available only if the mat foundation includes physical beams. After selecting show beam stress, Select Stress Type under beam stress setup group will be enabled. Select any stress type to view the contour along with a legend.

Available beam stress types are

1. Axial stress
2. Bending Y stress
3. Bending Z stress
4. Combined stress

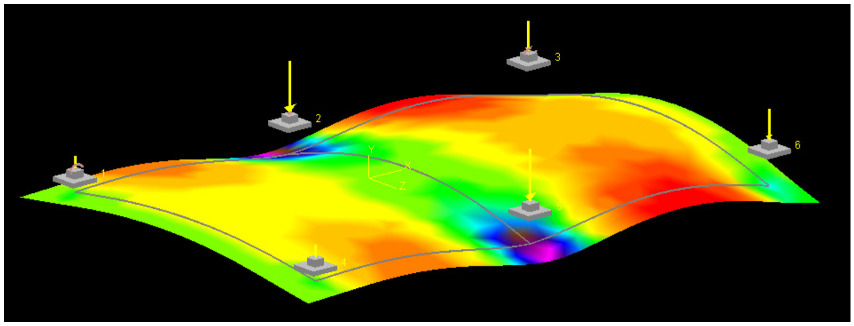


Show Legend

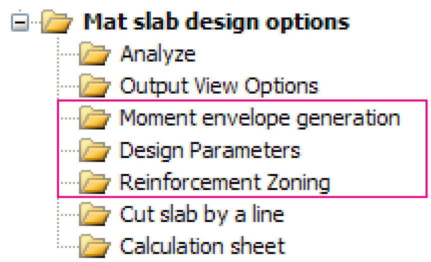
Use this option to switch on/off legend display

Plot contour on deflected shape

Select this option to draw stress contour on the deflected shape.



4.6.6 Slab Design



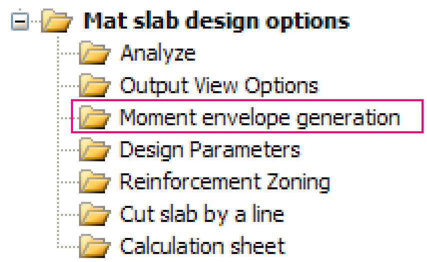
Slab Design for Mat foundation is divided into three simple steps

1) Moment envelope generation
2) Design slab
3) Create reinforcing zones and detailing

The *Slab Design* page contains the following sub-pages:

- FEM Slab Design
- Slab Detailing
- Section Design Along a Line

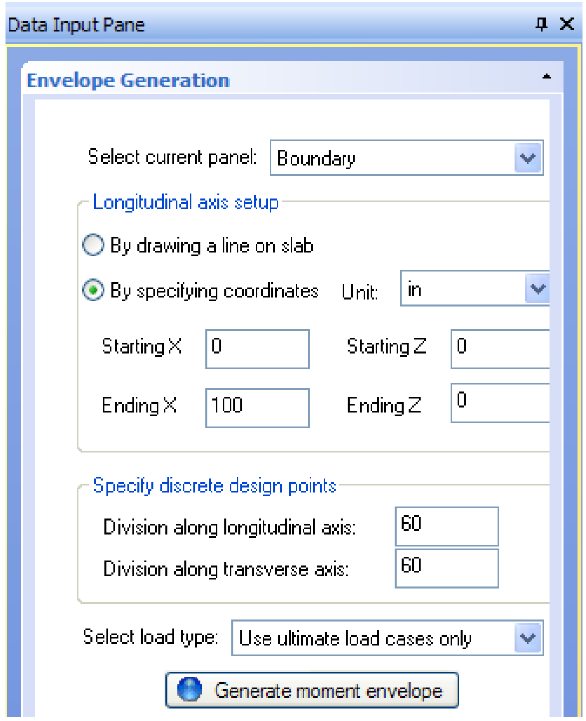
4.6.6.1 Moment Envelope Generation



Clicking on the *Moment Envelope Generation* opens a form in data area pane which allows user to choose longitudinal reinforcement directions and generate moment envelope. Please note, longitudinal axis is just a vector direction.

Mat slab is a physical entity in STAAD.foundation, so to design the slab, program uses a unique technique. It first divides the slab into finite number of discrete points and then calculates stress on those nodes to create moment envelope. Please note that program automatically transforms stresses to the specified longitudinal direction.

To generate moment envelope you first need to define longitudinal reinforcement direction. You can define X,Z coordinate to define an axis or click on any two points on the screen.



The following commands and inputs are available to generate moment envelope.

Select Current Panel

If you have multiple boundaries you need to choose current panel to be designed. By default program selects the first created boundary.

Longitudinal Axis Setup

There are two methods to define the longitudinal axis. You can setup the axis either by defining two X,Z coordinates or by clicking on two points on the screen.

By drawing a line on slab

Select this option to click on two points on the screen to define longitudinal axis. Once the first point is clicked program will draw a line from the first point to the mouse point to show the axis. After second point is clicked on the screen, program will calculate the X,Z coordinates of those points and fill up the form start and end coordinates.

By specifying coordinates

Select this option to input X,Z coordinates of the start and end points of the axis. By default program shows a global X axis as longitudinal axis.

Division along longitudinal axis

Number of slab divisions along longitudinal axis. It must be a positive number. Program uses 60 as default value.

Division along transverse axis

Number of slab divisions along transverse axis. It must be a positive number. Program uses 60 as default value.

Select load type

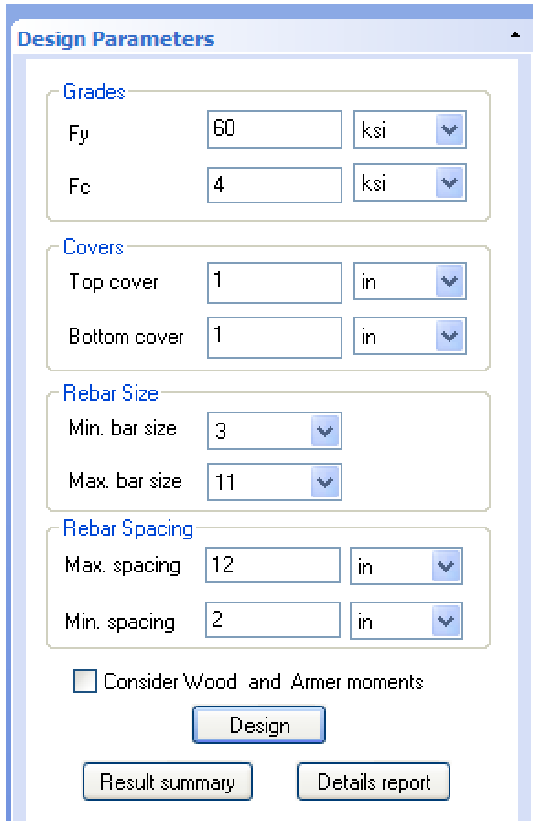
Shear and reinforcement design for foundation are done only for ultimate (factored) load combinations. User has the option here to choose only load cases defined as ultimate or all load cases assigned to the current job.

Generate Moment envelope

Click on this button to create grid and calculate moment envelope on the grid intersection points.

4.6.6.2 Design Parameters

Clicking on the Design Parameters opens a form in data area pane which allows you to input design parameters, design current panel and review design results.



Design parameters are very standard input where you need to input information on material, cover and rebar.

Fy: Allowable steel stress
Fc: crushing concrete strength
Top Cover: Cover for slab top reinforcement
Bottom Cover: Cover for slab bottom reinforcement

Min. bar size: Minimum rebar size to be used
Max. bar size: Maximum rebar size to be used
Max. spacing: Maximum rebar spacing
Min. spacing: Minimum rebar spacing

Consider Wood and Armer moments

Use this option to consider Mxy moment to design the slab. This is a method published by Wood and Armer where Mxy moment is transformed to Mx and My moment.

Design

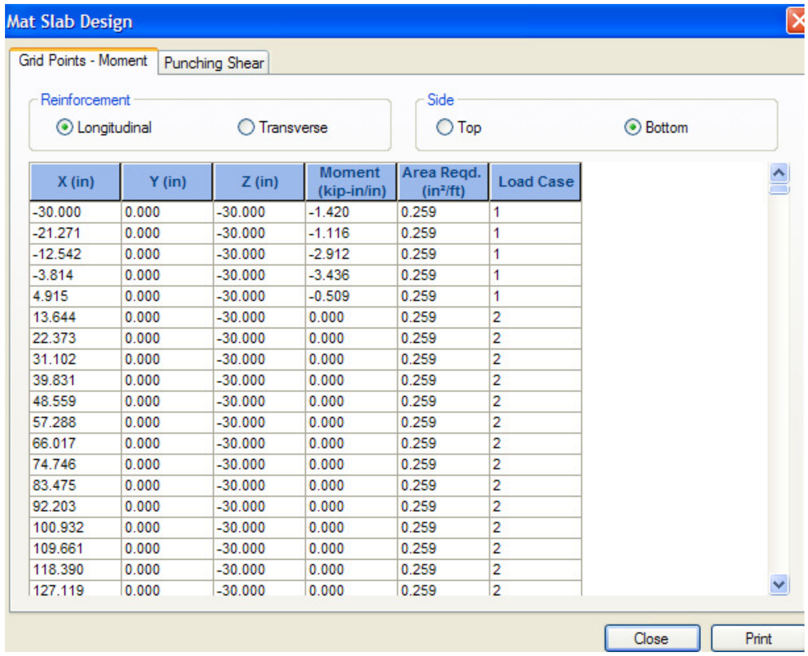
Click on this button to design the slab. When the design operation is completed, a message box will appear.

Result summary

Clicking on this button will open a table which will show maximum reinforcement requirement condition for all slab faces and direction. The table shows four rows for longitudinal top, longitudinal bottom, transverse top and transverse bottom reinforcement requirement.

Details report

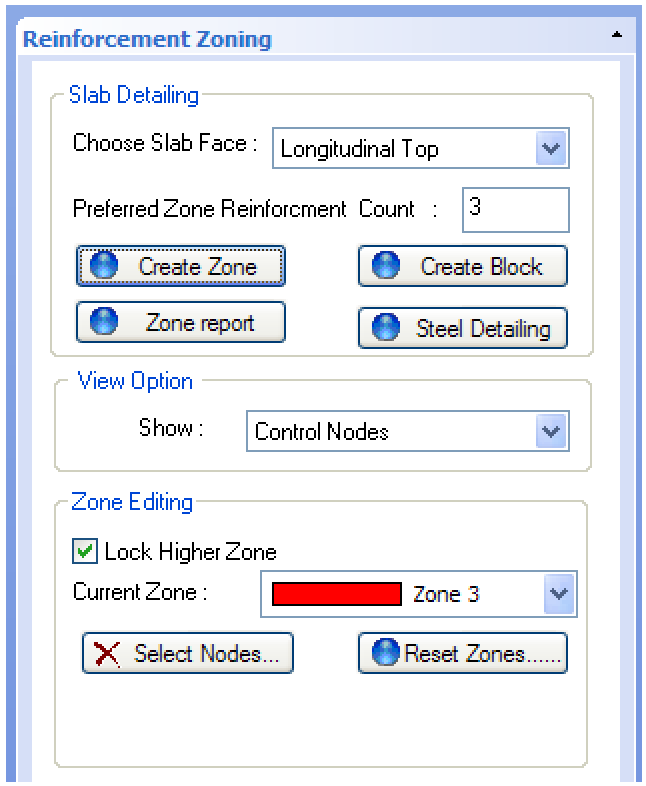
Clicking on this button will open a dialog box which will have two tabs. One is for moment design and other one for punching shear check. It lists all the grid points created to design the slab. It shows X,Y,Z coordinates for each point, moment for that face and direction and the corresponding reinforcement requirements.



X (in)	Y (in)	Z (in)	Moment (kip-in/in)	Area Reqd. (in^2/ft)	Load Case
-30.000	0.000	-30.000	-1.420	0.259	1
-21.271	0.000	-30.000	-1.116	0.259	1
-12.542	0.000	-30.000	-2.912	0.259	1
-3.814	0.000	-30.000	-3.436	0.259	1
4.915	0.000	-30.000	-0.509	0.259	1
13.644	0.000	-30.000	0.000	0.259	2
22.373	0.000	-30.000	0.000	0.259	2
31.102	0.000	-30.000	0.000	0.259	2
39.831	0.000	-30.000	0.000	0.259	2
48.559	0.000	-30.000	0.000	0.259	2
57.288	0.000	-30.000	0.000	0.259	2
66.017	0.000	-30.000	0.000	0.259	2
74.746	0.000	-30.000	0.000	0.259	2
83.475	0.000	-30.000	0.000	0.259	2
92.203	0.000	-30.000	0.000	0.259	2
100.932	0.000	-30.000	0.000	0.259	2
109.661	0.000	-30.000	0.000	0.259	2
118.390	0.000	-30.000	0.000	0.259	2
127.119	0.000	-30.000	0.000	0.259	2

4.6.6.3 Reinforcing zones

As design is performed on thousands of points it will be impossible to go through all those numbers and create a reinforcement layout. STAAD.foundation has a tool to create reinforcement zones much like reinforcement contour plot. Number of zones is user specified. By default program use three zones.



Choose Slab face

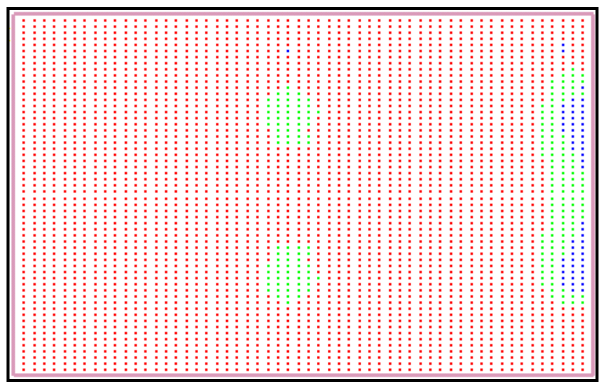
Reinforcement zoning is done for one face at a time. So, this step needs to be repeated four times to detail all faces and direction. Select the current slab face from drop down list.

Preferred Zone Reinforcement Count

Use this edit box to tell the program how many different sizes of reinforcing steel bars (rebar) you want the program to allow in the slab design. The program divides the slab into the number of zones you designate. Each zone will contain only one size of reinforcing steel.

Create Zone

Click this button to create the number of reinforcing zones specified by the value entered in the *No. of Zones* edit box. The following figure shows how the display might appear when three zones are created.

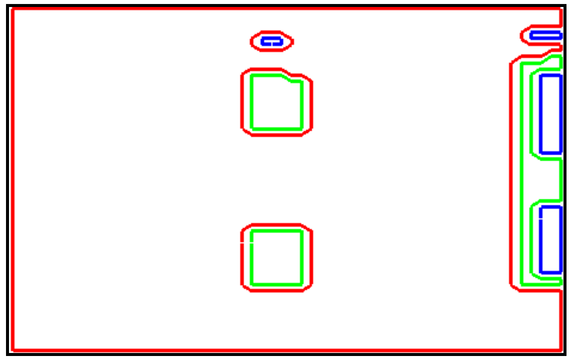


A colored dot in the center of each element of the mesh indicates the reinforcing zone that the element belongs to.

Create Block

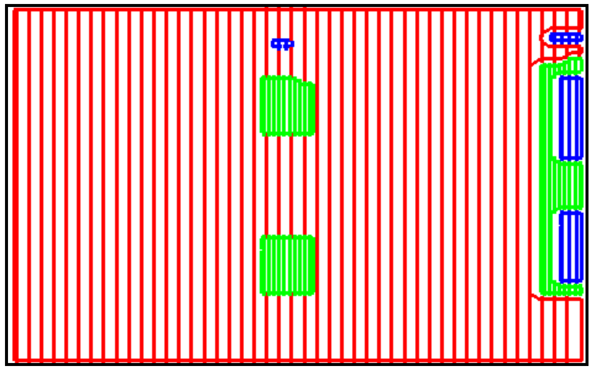
Click on this button to divide the slab into block-shaped areas, based on the reinforcement zones generated by the *Create*

Reinforcing Zones command. These rectangular areas are created to allow a practical layout of the various sizes of reinforcing steel.



Steel Detailing

Click this button to plot the location of the reinforcing steel bars on the model view.



Zone editing

Even after block generation reinforcement zones may not become regular rectangular blocks. So, the program has an option to visually adjust those zones to form rectangular regions.

Select Zone

Select the current zone to be edited. He drop down box will show color and zone number.

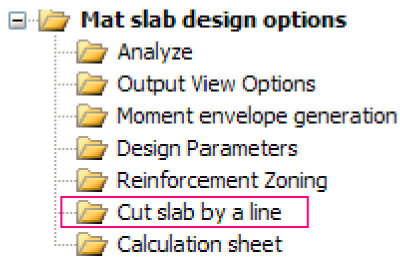
Select Nodes

Clicking on select nodes button will change the button status and will allow user to select grid points to be edited. Click on the graphics and draw a rubber band to select nodes. Nodes will be highlighted in yellow color.

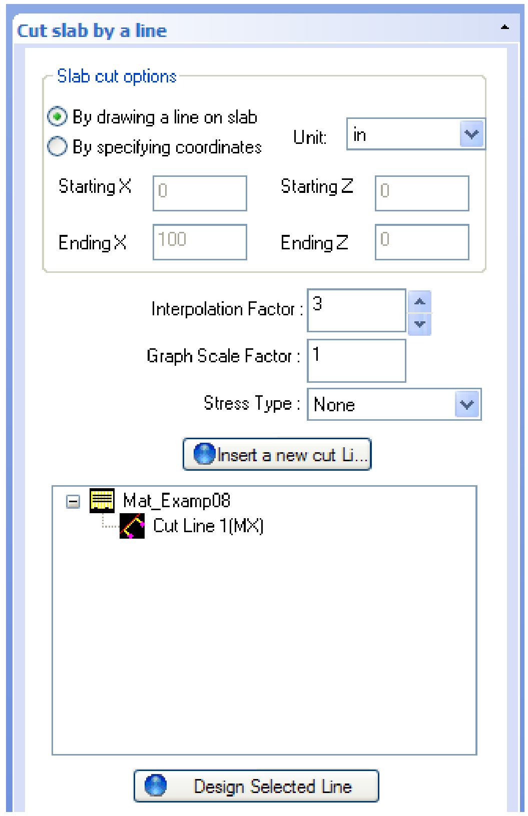
Reset zone

Click on reset zone button to edit the selected points. Current zone will be assigned to the selected node. Please note that if Lock Higher Zone option is selected, program will not overwrite higher zone with lower zone. This option is recommended to be always selected.

4.6.6.4 Section Design along a Line



Clicking on the *Cut slab by a Line* opens a form in the *Data Area* pane that allows you to draw a stress diagram along a specified section line and then design slab along that line.



The following commands and options are available in the *Data Area* pane:

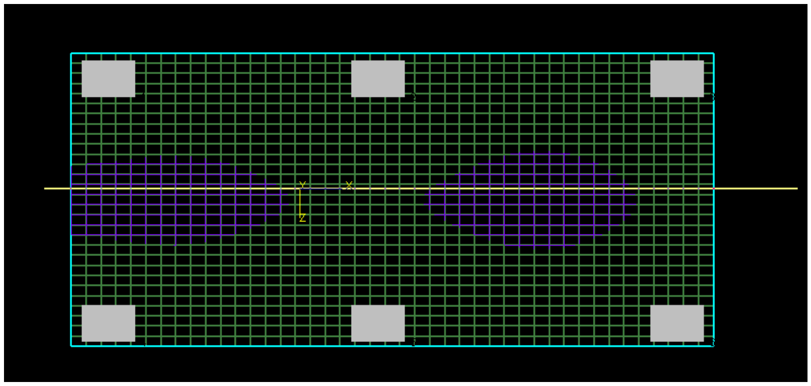
- Slab cut options
- Interpolation Factor
- Graph Scale Factor
- Insert a new Cut Line
- Stress Type
- Design Selected Line

Slab cut options

The cut line or the line on the slab can be drawn either by inputting two coordinates or by clicking at two points on the screen.

By drawing a line on slab

Using this option will allow user to draw a line in the graphics by clicking on two points. Click on first point and then stretch the line to next point and click again. It will transfer coordinates of those two points to the form under start and end points.



Graph Scale Factor

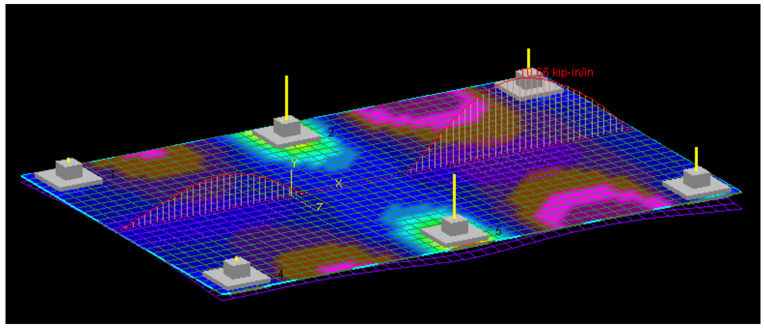
The *Graph Scale Factor* allows you to change the vertical exaggeration factor of the stress diagram in the *Graphics Window*.

Stress Type

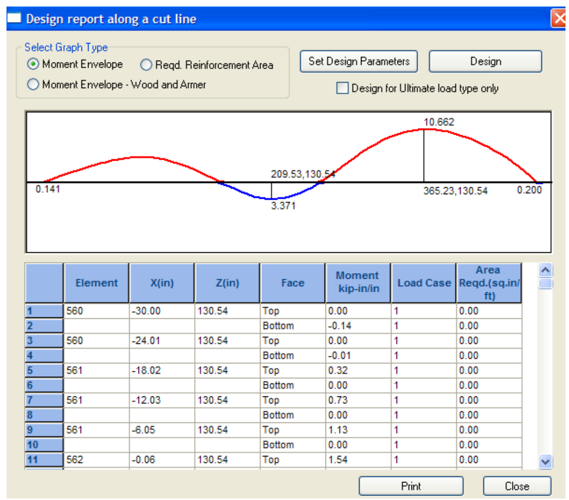
The *Stress Type* drop-down list box allows you to the type of plate stress you want to plot along the cut line.

Insert a new Cut Line

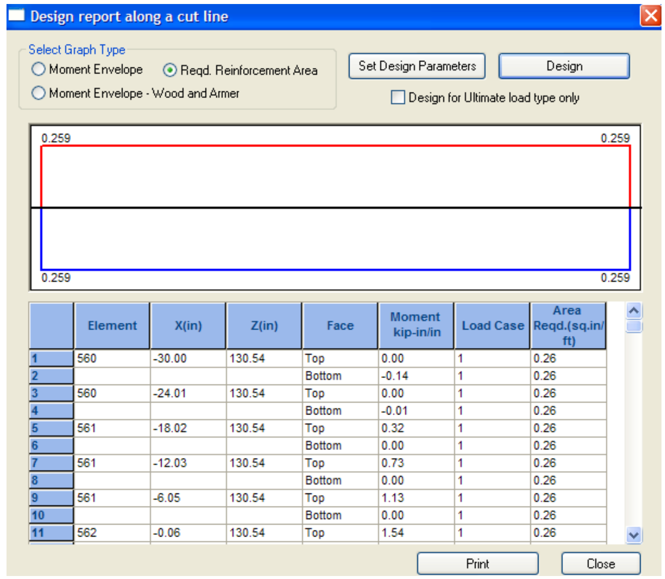
Now click on the button labeled *Insert a new Cut Line*. The following figure appears in the graphics window.



Now click the button labeled *Design Selected Line*. A dialog box labeled *Design Report Along a Selected Line* will be displayed.



Click on the Design button to calculate the required reinforcement area for each element along the cut line.


Design report along a cut line
Select Graph Type
Moment Envelope
Reqd. Reinforcement Area
Moment Envelope - Wood and Armer
Set Design Parameters
Design
Design for Ultimate load type only
0.259
0.259
0.259
0.259
Element
X(in)
Z(in)
Face
Moment kip-in/in
Load Case
Area Reqd.(sq.in/ft)
Print
Close

4.6.6.5 Calculation sheet

Click on calculation sheet to review design steps, analysis results and load values.

Panel Name	Fy (kip/in2)	Fc (kip/in2)	Top Cover (in)	Bottom Cover (in)	Min Bar Size (in)	Max Bar Size (in)	Min Spacing (in)	Max Spacing (in)	Wood and Armer Moment
Boundary	60.000	4.000	1.000	1.000	3	11	2.000	12.000	Not Used

DESIGN OUTPUT

Top of Mat Longitudinal Direction

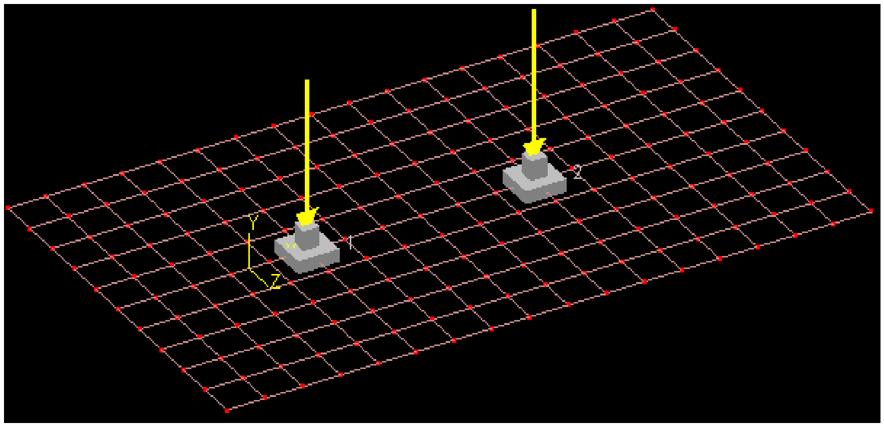
Zone:-	1
Governing Moment(M_{GOV})=	11.268(kip-in)
For F_c <4.0	$\beta = 0.85$
ρ_{min} =	0.0018
$\rho_{max} = 0.75 \times 0.85 \times \beta \times F_c \times \frac{87000}{f_y(87000+f_y)} =$	0.021
Effective Depth(deff)=	10.813(in)
Steel Percentage (P_{ct})= ρ_{reqd} = $\max\left[1-\left(\sqrt{1-\left(\frac{2 \times R_n \times m}{f_y}\right)}\right) \times \frac{1}{m}, \rho_{min}\right] =$	0.0018
Where $m = \frac{f_y}{0.85 \times f_c} =$	17.647
$R_n = \frac{M_{GOV}}{0.9 \times d_{eff}^2 \times width} =$	0.107(kip/in2)

4.7 Combined Footing

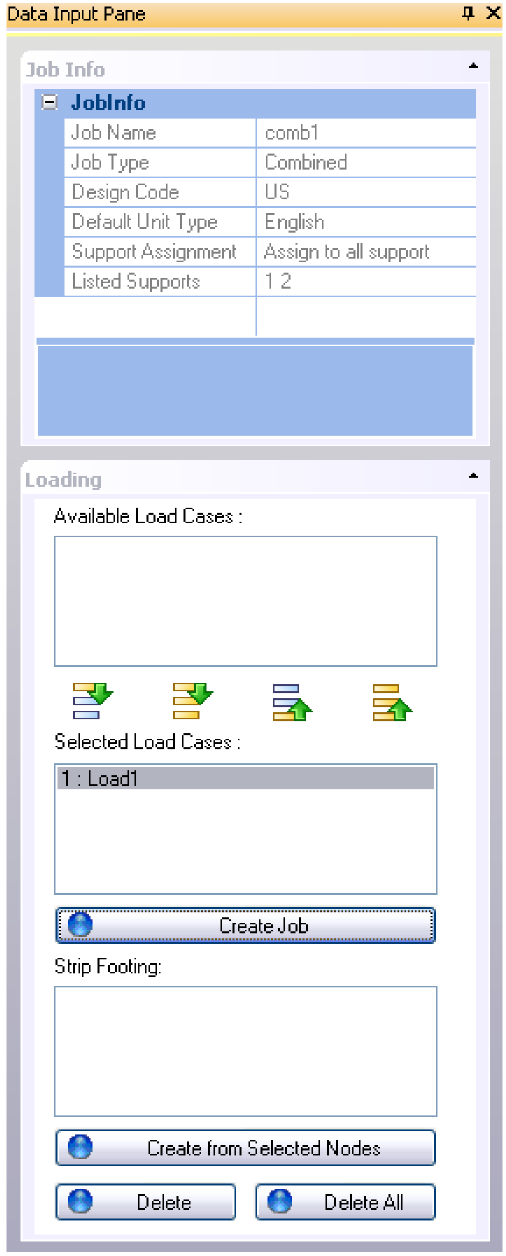
In this job setup you can create a combined footing with two supports as well as a strip footing with more than two supports.

4.7.1 Creating Combined Footing Job

Create linear supports from “Column Position” as follows and add loads as described earlier.

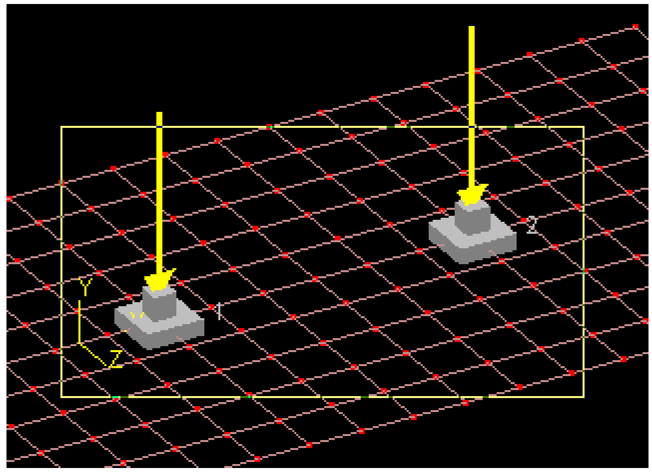


Now go to “Create New Job”. Give a suitable job name, Choose “Job Type” as “Combined”. Select and move the load from “Available Load Case” list to “Selected Load Case” list. Then click on “Create Job” button. You will see some new control came at the bottom of the “Job Info” data input pane to create a combined footing with selected supports.

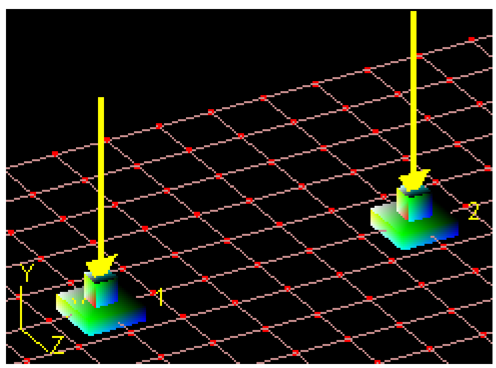

Data Input Pane
Job Info
JobInfo
Job Name
comb1
Job Type
Combined
Design Code
US
Default Unit Type
English
Support Assignment
Assign to all support
Listed Supports
1 2
Loading
Available Load Cases :
Selected Load Cases :
1 : Load1
Create Job
Strip Footing:
Create from Selected Nodes
Delete
Delete All

4.7.2 Creating the Combined Footing

Now select the support in the "Geometry" view using the mouse dragging.

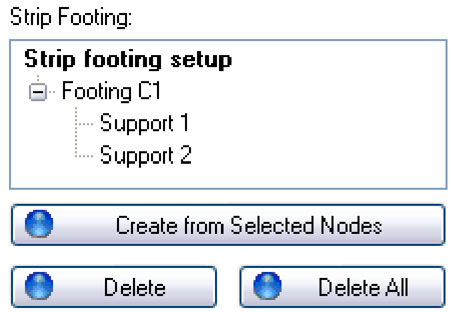


The selected support will be highlighted as below.

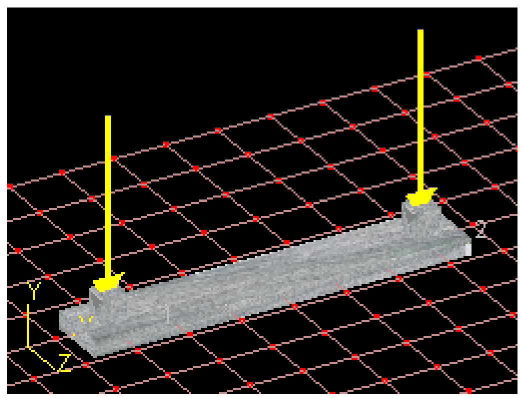


<u>Create from Selected Nodes</u>

Now click on the button "Create from Selected Nodes" which will show a tree view with the footing included with support.



Also click once on the "Geometry" view which will show the real life picture of the footing.



<u>Note:</u>

The supports have to be collinear to become a part of a combined footing. If the supports are non-collinear then it will show an error message box as follows.

Delete

To delete a footing, select the footing from the tree, click on "Delete". Deletion of support from a combined footing is not allowed. You need to recreate the combined footing to edit it. This will generate the following error message.

Delete All

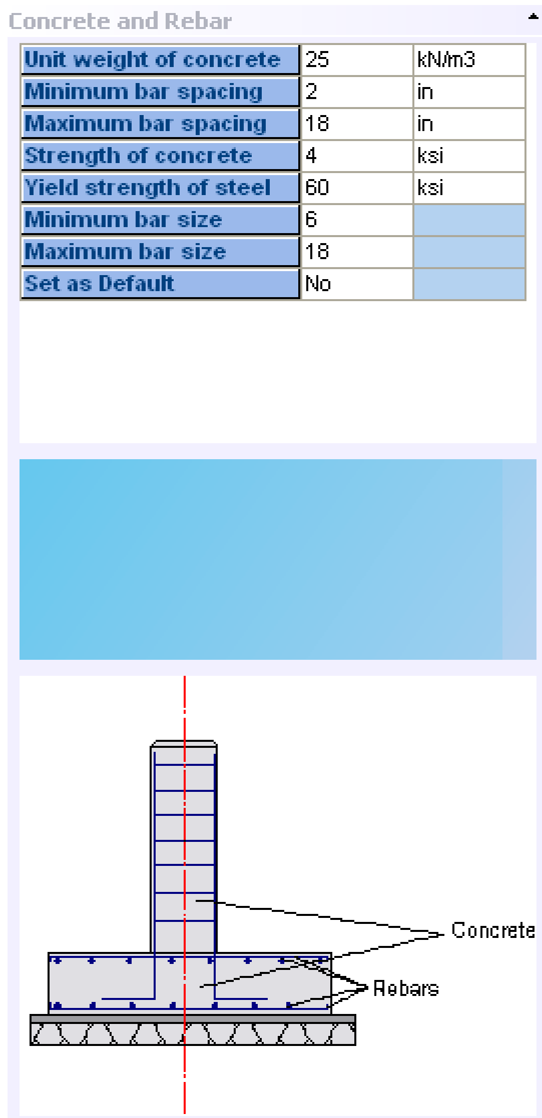
To delete all the combined footing at a click, simply click on "Delete All" button.

4.7.3 Defining the Design Parameters

Note that after you have created a combined footing job, the left side main navigator tree view is populated with the "Design Parameter" for "Combined Footing Job".

4.7.3.1 Concrete and Rebar

Here you have to give all the necessary inputs related to concrete and reinforcement bar to be used.



Unit weight of Concrete

Unit weight of concrete with proper unit.

Minimum Bar Spacing

Minimum spacing of bar to use for design.

Maximum Bar Spacing

Maximum spacing of bar to use for design.

Fc

Strength of concrete.

Fy

Strength of steel.

Minimum Bar Dia

Minimum diameter of bar to use for design.

Maximum Bar Dia

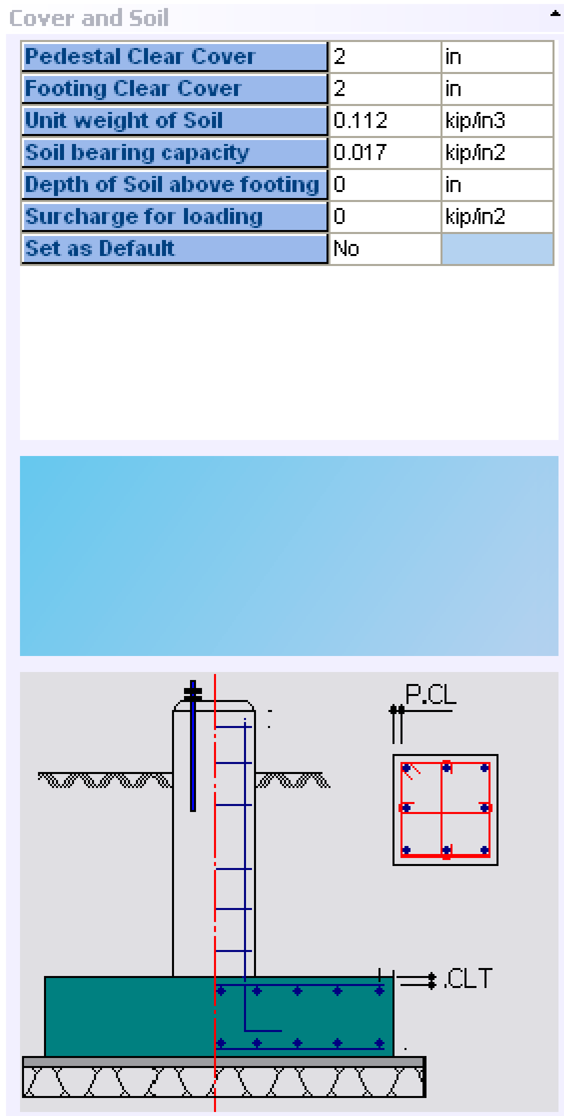
Maximum diameter of bar to use for design.

Set as Default

Set "Yes" to save the data set for the application, so that each time you create such job, these fields will be populated with this value set. Else set "No".

4.7.3.2 Cover and Soil

In this page you have to enter parameters related to clear covers and soil data.



<u>Pedestal Clear Cover</u>

Clear cover to be used for the pedestal.

<u>Footing Clear Cover</u>

Clear cover to be used for the footing.

<u>Unit Weight of Soil</u>

Unit weight of soil under consideration.

<u>Soil Bearing Capacity</u>

Allowable bearing capacity of soil.

<u>Depth of Soil above footing</u>

Soil depth above footing.

<u>Surcharge for loading</u>

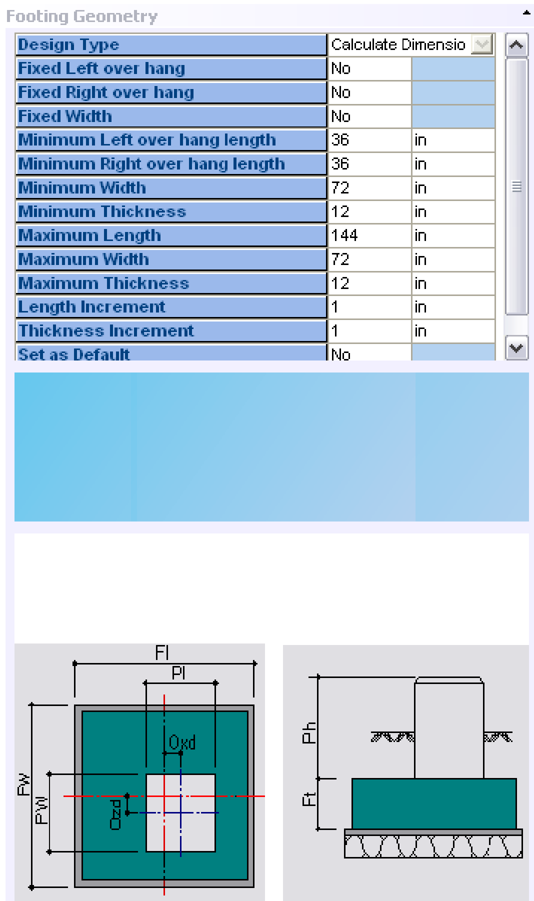
Loading surcharge of the footing.

<u>Set as Default</u>

This is explained in the earlier page.

4.7.3.3 Footing Geometry

These are the geometrical parameter used for design.



<u>Design Type</u>

Choose the way of design from here. You can use the software for an optimizing design procedure or you can check for a fixed dimension.
For the first case choose “Calculate Dimension” and for the latter use “Set Dimension”.

Fixed Left over hang

Choose "Yes" or "No" respectively for setting the dimension as fixed during check or optimize it.

Fixed Right over hang

Choose "Yes" or "No" respectively for setting the dimension fixed during check or optimize it.

Fixed Width

Choose "Yes" or "No" respectively for setting the dimension fixed during check or optimize it.

Minimum Left over hang length

Start length of left over hang for design.

Minimum Right over hang length

Start length of right over hang for design.

Minimum Width

Start width for design.

Minimum Thickness

Start thickness for design.

Maximum length

If "Calculate Dimension" is chosen then the maximum range of length to check.

Maximum Width

If “Calculate Dimension” is chosen then the maximum range of width to check.

Maximum Thickness

If “Calculate Dimension” is chosen then the maximum range of thickness to check.

Length Increment

Increment of length required for the iteration process of design if “Calculate Dimension” Is chosen.

Thickness Increment

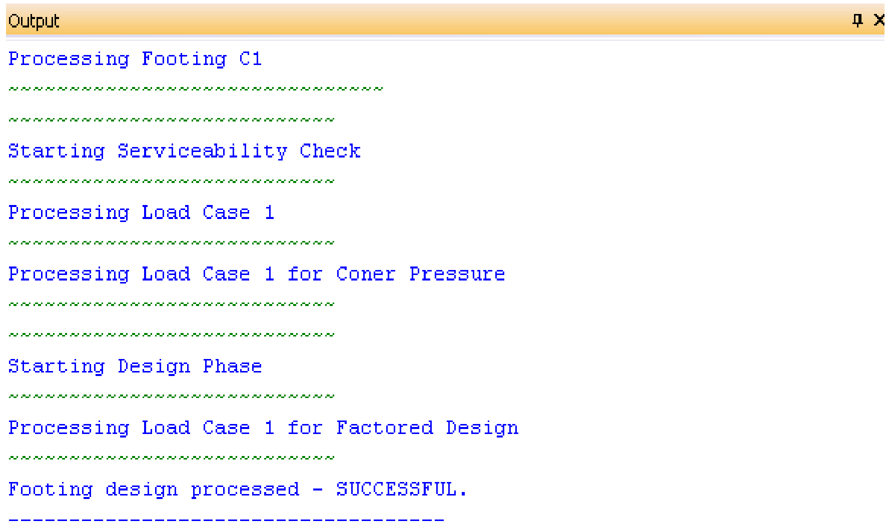
Increment of thickness required for the iteration process of design if “Calculate Dimension” Is chosen.

Set as Default

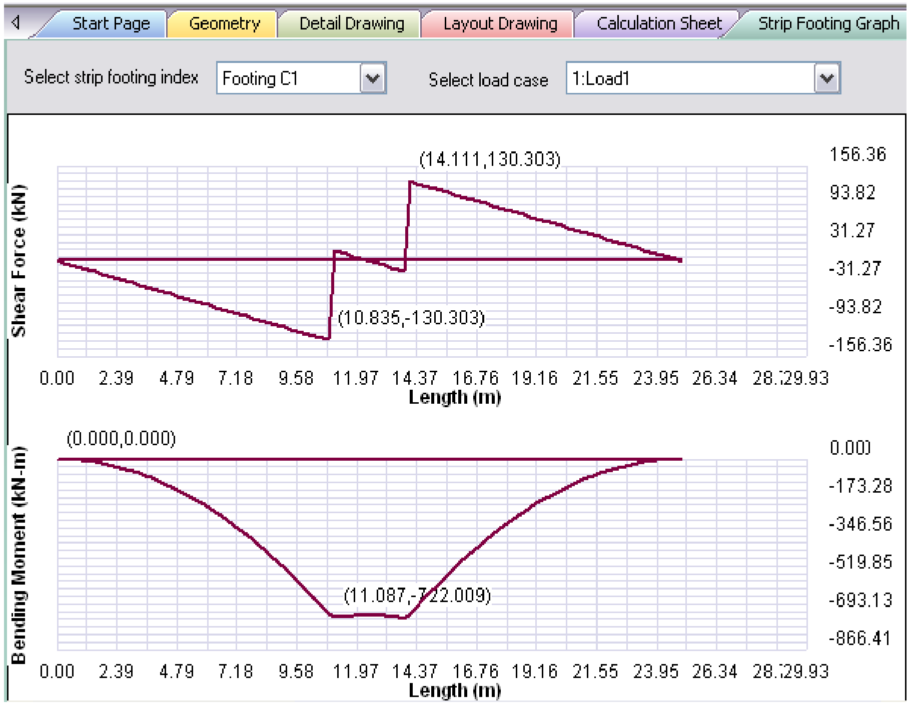
As described earlier.

4.7.3.4 Design

Next click on "Design". The design progress report will be generated in the "Design Progress Report" window.



A detailed calculation sheet will be generated in the "Calculation Sheet" tab. A graphical report of Bending Moment and Shear Force for the footing will be generated in the "Strip Footing Graph" tab as follows.



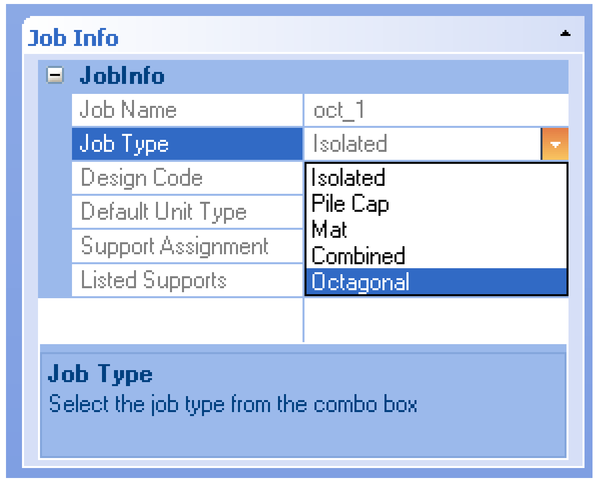
Start Page
Geometry
Detail Drawing
Layout Drawing
Calculation Sheet
Strip Footing Graph
Select strip footing index
Footing C1
Select load case
1:Load1
Shear Force (kN)
(14.111,130.303)
(10.835,-130.303)
156.36
93.82
31.27
-31.27
-93.82
-156.36
0.00 2.39 4.79 7.18 9.58 11.97 14.37 16.76 19.16 21.55 23.95 26.34 28.7 29.93
Length (m)
Bending Moment (kN-m)
(0.000,0.000)
(11.087,-722.009)
0.000
-173.28
-346.56
-519.85
-693.13
-866.41
0.00 2.39 4.79 7.18 9.58 11.97 14.37 16.76 19.16 21.55 23.95 26.34 28.7 29.93
Length (m)

4.8 Octagonal Footing

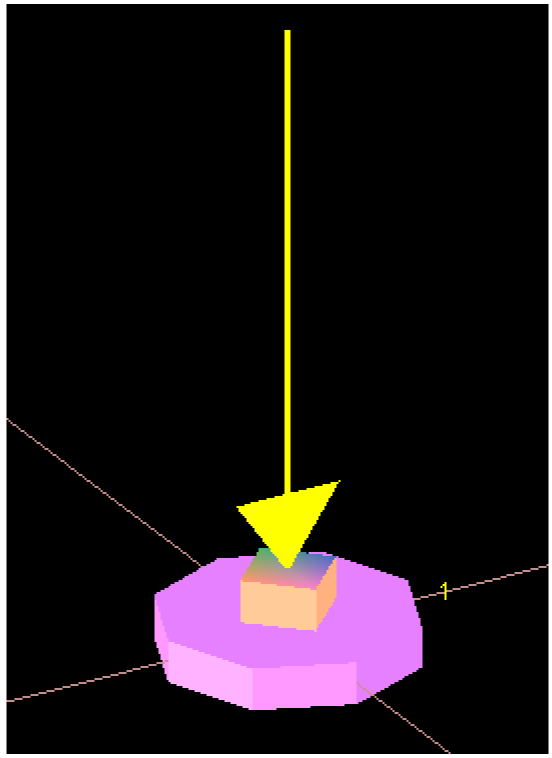
In this job setup you can design an octagonal footing.

4.8.1 Creating Octagonal Footing Job

Create support from “Column Position” as follows and add loads as described earlier. Now go to “Create New Job”. Give a suitable job name, Choose “Job Type” as “Octagonal”.



Select and move the load from “Available Load Case” list to “Selected Load Case” list. Then click on “Create Job” button. This will create the octagonal footing as you can see in the “Geometry” tab.

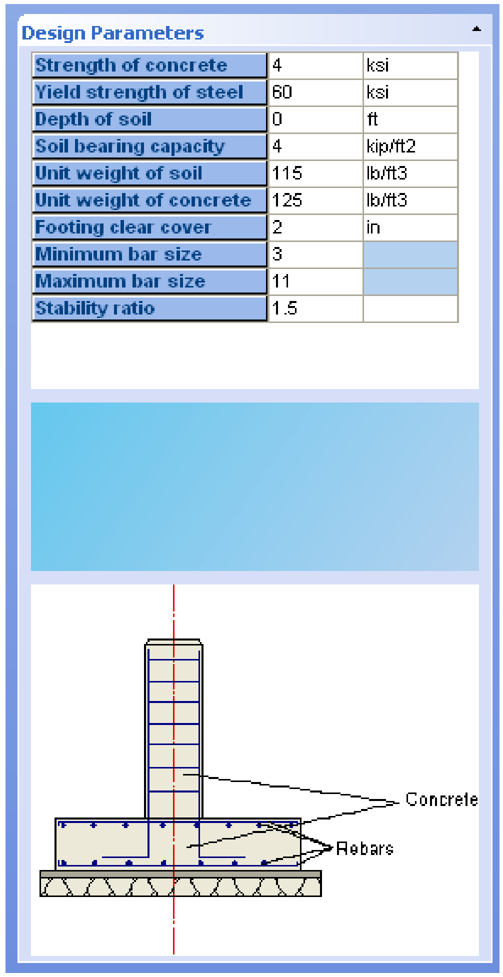

1

4.8.2 Defining the Design Parameters

Note that after you have created a octagonal footing job, the left side main navigator tree view is populated with the "Design Parameter" for "Octagonal Footing Job".

4.8.2.1 Design Parameters

Here you have to give all the necessary inputs related to concrete, reinforcement bar and other design parameters details.



Strength of concrete	4	ksi
Yield strength of steel	60	ksi
Depth of soil	0	ft
Soil bearing capacity	4	kip/ft2
Unit weight of soil	115	lb/ft3
Unit weight of concrete	125	lb/ft3
Footing clear cover	2	in
Minimum bar size	3	
Maximum bar size	11	
Stability ratio	1.5	

<u>Strength of concrete.</u>

<u>Fc</u>

Strength of steel.

Fy

Depth of Soil

Soil depth above footing.

Soil Bearing Capacity

Allowable bearing capacity of soil.

Unit Weight of Soil

Unit weight of soil under consideration.

Unit weight of Concrete

Unit weight of concrete with proper unit.

Footing Clear Cover

Clear cover to be used for the footing.

Minimum Bar Dia

Minimum diameter of bar to use for design.

Maximum Bar Dia

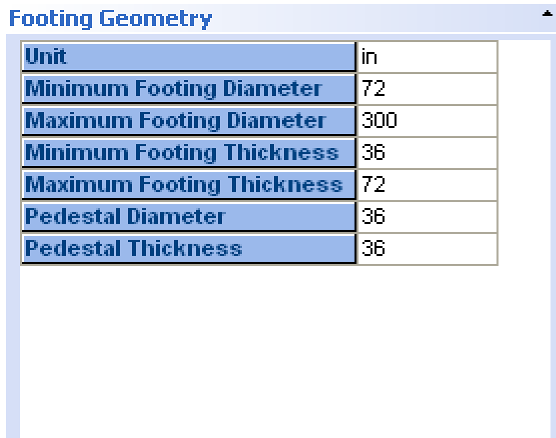
Maximum diameter of bar to use for design.

Stability Ratio

Starting Stability Ratio to be used for design.

4.8.2.2 Footing Geometry

These are the geometrical parameter used for design.



<u>Unit</u>

<u>Minimum Footing Diameter</u>

The minimum footing diameter that will be used for starting design.

<u>Maximum Footing Diameter</u>

The maximum range of footing diameter that will be used for optimizing the design.

<u>Minimum Footing Thickness</u>

The minimum footing thickness that will be used for starting design.

<u>Maximum Footing Thickness</u>

The maximum range of footing thickness that will be used for optimizing the design.

Pedestal Diameter

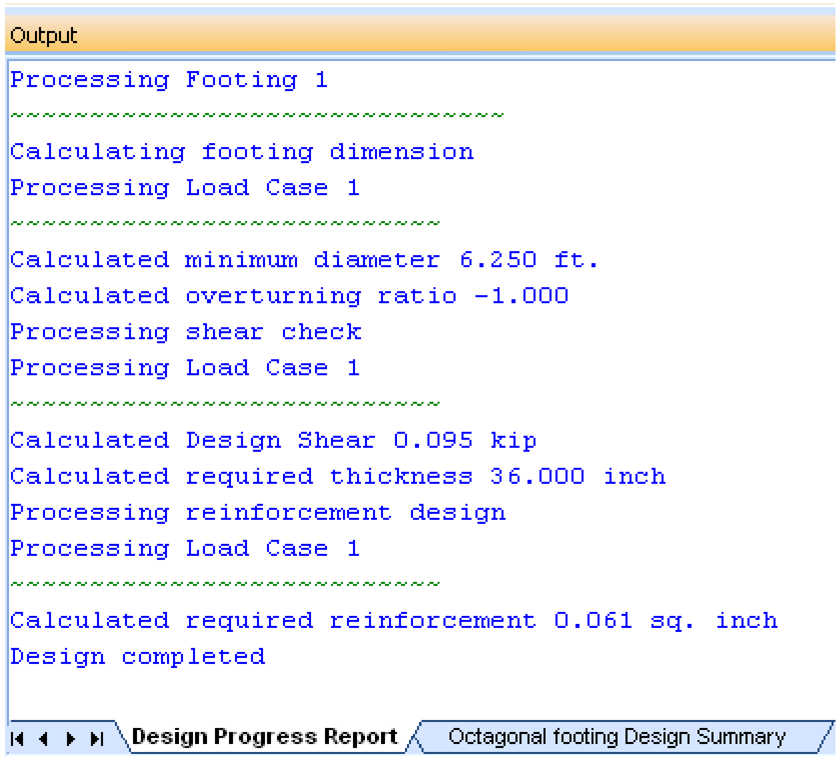
Diameter of the pedestal.

Pedestal Thickness

Thickness of the pedestal.

4.8.2.3 Design

Next click on "Design". The design progress report will be generated in the "Design Progress Report" window and a summery of design resulted will be displayed in the "Octagonal footing Design Summary" tab of the "Output" window.

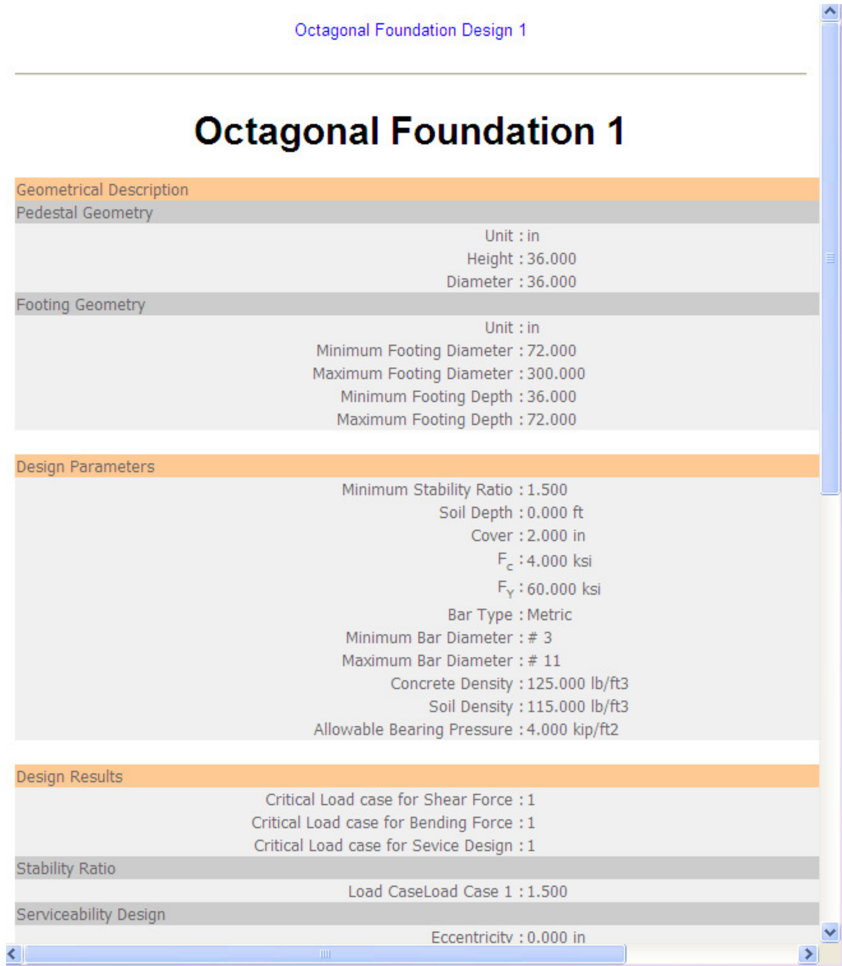



Output

FootNo	Footing Diameter m	Thickness m	Stability Ratio	Reqd. Reinforcement	
				Bar No.	Spacing
1	1.90	0.91	-1.00	#8	0.03 m

Design Progress Report | Octagonal footing Design Summary

A detailed calculation sheet will be generated in the "Calculation Sheet" tab.



Octagonal Foundation Design 1

Octagonal Foundation 1

Geometrical Description

Pedestal Geometry

Unit : in
Height : 36.000
Diameter : 36.000

Footing Geometry

Unit : in
Minimum Footing Diameter : 72.000
Maximum Footing Diameter : 300.000
Minimum Footing Depth : 36.000
Maximum Footing Depth : 72.000

Design Parameters

Minimum Stability Ratio : 1.500
Soil Depth : 0.000 ft
Cover : 2.000 in
F_c : 4.000 ksi
F_Y : 60.000 ksi
Bar Type : Metric
Minimum Bar Diameter : # 3
Maximum Bar Diameter : # 11
Concrete Density : 125.000 lb/ft3
Soil Density : 115.000 lb/ft3
Allowable Bearing Pressure : 4.000 kip/ft2

Design Results

Critical Load case for Shear Force : 1
Critical Load case for Bending Force : 1
Critical Load case for Sevice Design : 1

Stability Ratio

Load CaseLoad Case 1 : 1.500

Serviceability Design

Eccentricity : 0.000 in

4.9 The Menu Commands

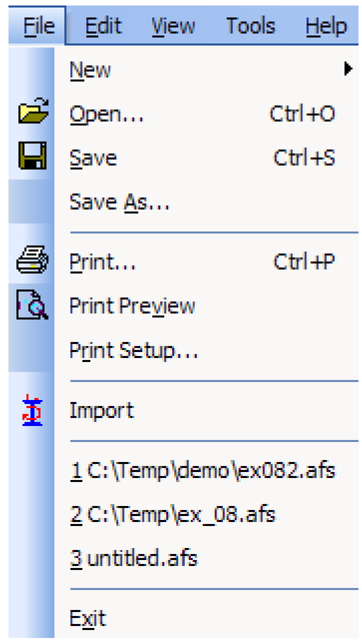
This section provides a description of the commands available from STAAD.*foundation's* pull-down *Menu Bar*.

The names of the pull-down menus, from left to right across the top of the screen, are as follows:

- File
- Edit
- View
- Tools
- Help

4.9.1 File Menu

The *File Menu* allows you to perform project file related operations such as creating a new project, opening an existing project, saving a project, etc.



The *File Menu* contains the following menu commands:

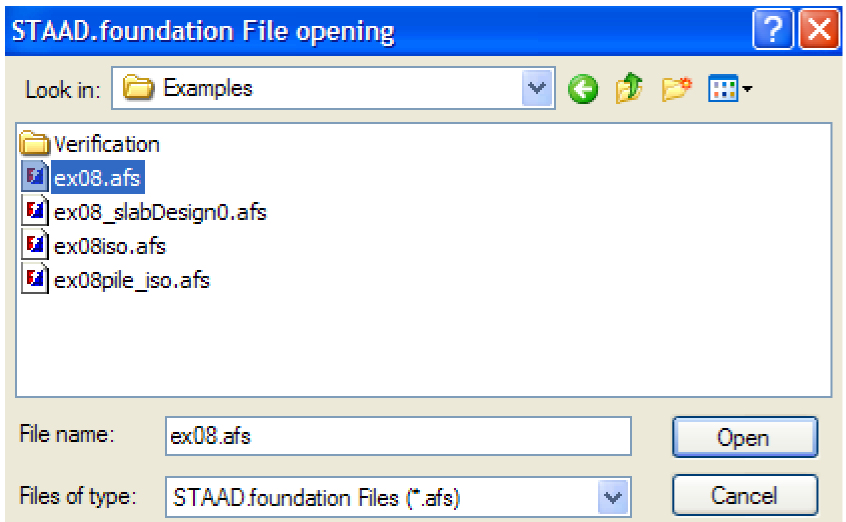
- New
- Open
- Save
- Save As
- Print
- Print Preview
- Print Setup
- Import
- Recent Project Files.

New

The *New* menu command opens and creates a new project.

Open

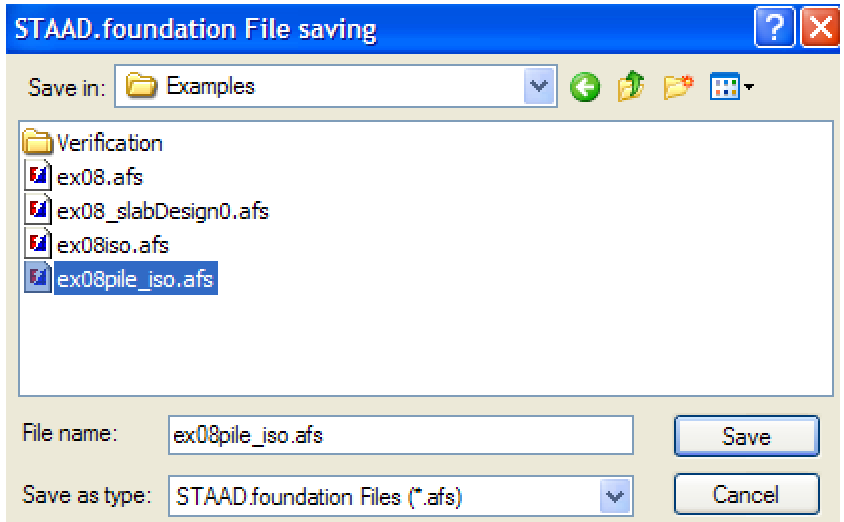
The *Open* menu command brings up the open file dialog box and allows you to open an existing STAAD.*foundation* project.



To open an existing project, navigate to the directory in which the project file is located and then select the file and click on *Open*.

Save

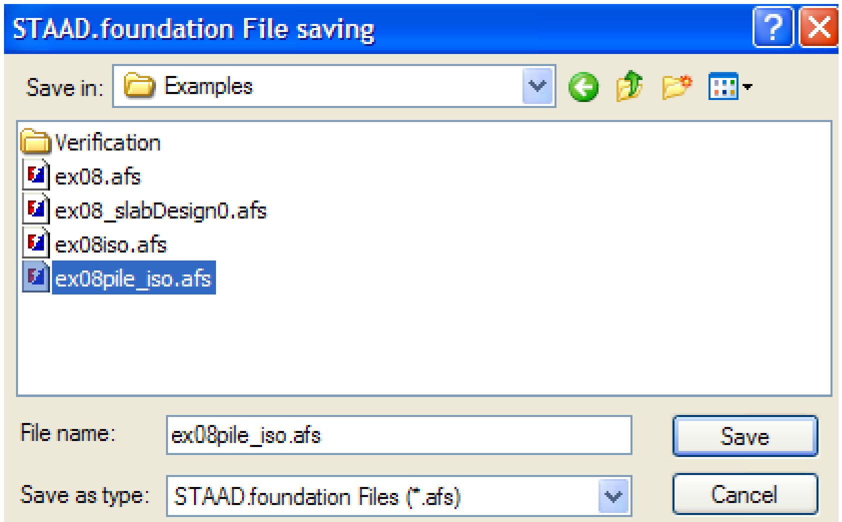
The *Save* menu command brings up the save file dialog box the first time the icon is clicked and allows you to save the active project to a file. To save a project, navigate to the directory in which you want to save the project, type in a file name for the project and then click on *Save*.



STAAD.*foundation* projects are saved with an .afs file extension. After a project has been saved to a file, clicking on the *Save* menu command again will simply save any updates made to the project to the file specified when you first saved the project.

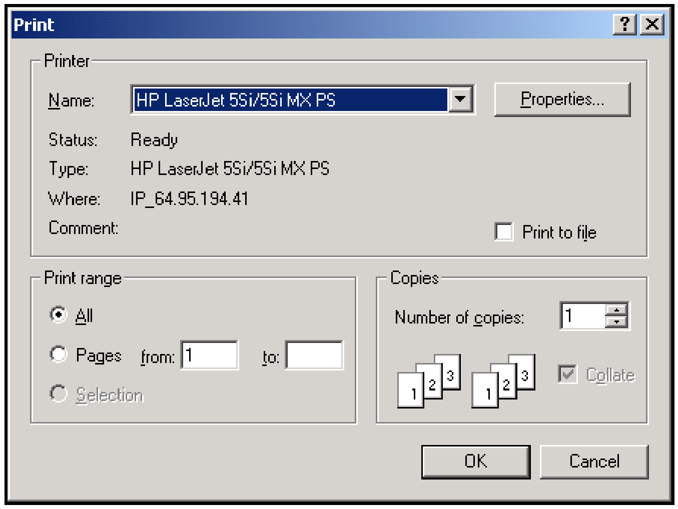
Save As

The *Save As* menu command opens the save as file dialog box and allows you to save the active project to a file. To save a project, navigate to the directory in which you want to save the project, type in a file name for the project and then click on *Save*.



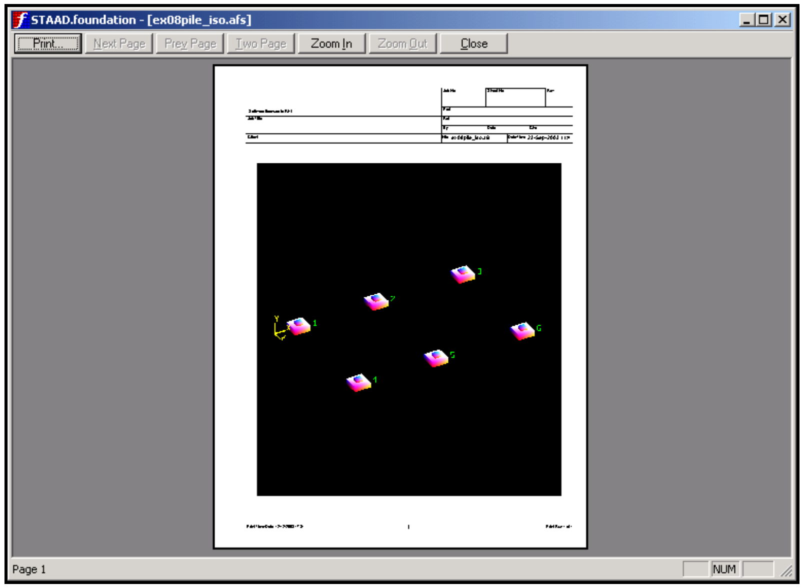
Print

The *Print* menu command opens a standard Windows print dialog box and allows you to print the active project report.



Print Preview

The *Print Preview* menu command opens a new window allowing you to view what the active report will look like when printed out.



The following command buttons are available in the *Print Preview* window:

- Print
- Next Page
- Prev. Page
- Two Page
- Zoom In
- Zoom Out
- Close.

Print

The *Print* button opens a standard Windows print dialog box and allows you to print the active project report.

Next Page

The *Next Page* button displays the next page in a report. If there is only one page in a report or you are at the last page in a report, the button will be grayed out.

Prev. Page

The *Prev. Page* button displays the previous page in a report. If you are at the first page in a report, the button will be grayed out.

Two Page

The *Two Page* button allows you to display two pages of a report on the screen at a time. Once you are in two page mode, the text on the button will change to "*One Page*." If you click on the button again, one page will be displayed on the screen at a time and the text on the button will change back to "*Two Page*."

Zoom In

The *Zoom In* button allows you to zoom in closer on a page of a report. After you click on the *Zoom In* button, your mouse cursor will change to a magnifying glass. You may then zoom in on a portion of a report by clicking on the region you want to zoom in on. Once you have zoomed in, the *Zoom In* button will become grayed out. You may return to the original viewing distance by clicking on the *Zoom Out* button.

Zoom Out

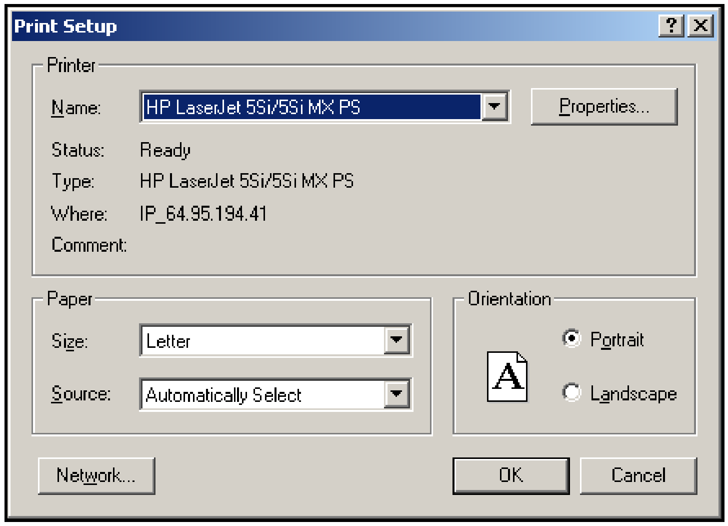
The *Zoom Out* button allows you to zoom back out after zooming in on a page of a report. The *Zoom Out* button is only active after zooming in on a page of a report.

Close

The *Close* button removes the print preview window.

Print Setup

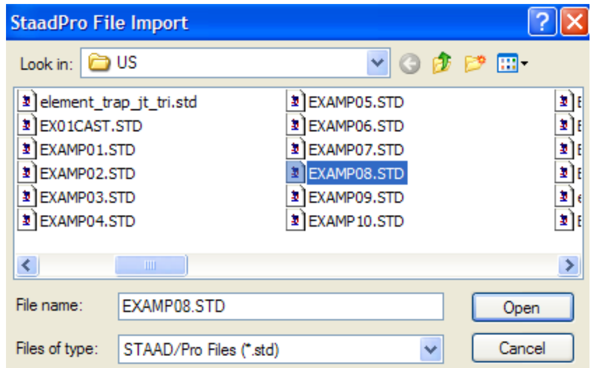
The *Print Setup* menu command opens a standard Windows print setup dialog box that allows you to configure printer settings.



Import

The *Import* menu command is used to begin a new project by importing the support geometry and support reactions from a STAAD.*Pro* analysis. The ability to import analysis data from other structural analysis software programs will be provided in a future release of STAAD.*foundation*.

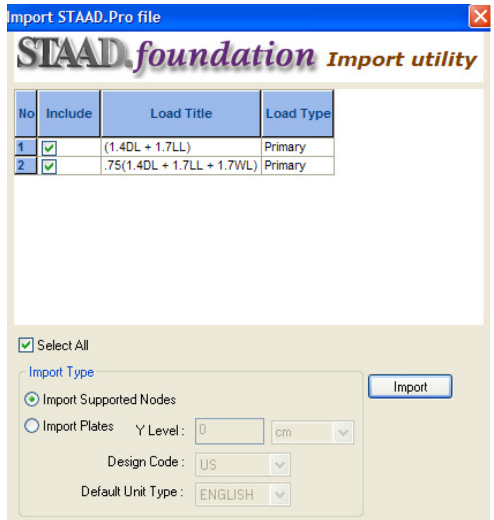
You can only import a STAAD.*Pro* model that has been successfully analyzed, because you will want to have the support reactions available for the foundation design. When the *Import* command is executed, an import dialog box will appear.



To import a STAAD.*Pro* file, navigate to the directory in which the file is located and then select the file and click on *Open*. Another dialog box will appear listing all the available load cases in the STAAD.*Pro* file. You may select any or all of the load cases by toggling on the corresponding check box, and then clicking on the *Import* button. As STAAD.Pro does not have the definition for serviceability and ultimate load type user should assign right load type attribute to each load case here. By default program defines all load cases as primary load types.

You can import either support coordinates or a slab already analyzed in STAAD.Pro to design in STAAD.foundation. To import support coordinates simply click on import button and it will import all support positions along with support reactions for all selected load cases.

To import an analyzed slab select *Import Plates* option and then enter Y level of the slab position. Program can only import a slab defined in XZ plane. Now select design code to be used to design the slab. Click on Import button and that will import the slab along with plate stresses and node displacements for selected load cases. The program will automatically create a mat foundation job.



Recent Project Files

The area below the *Print Setup* menu command displays a list of four *Recent Project Files* you have worked on. Selecting a project from the list will open the project.

4.9.2 Edit Menu

The *Edit* menu allows you to perform editing operations .

The *Edit* menu contains the following menu command:

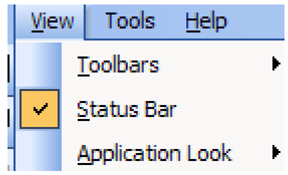
- Delete

Delete

The *Delete* menu command deletes the selected item(s). The *Delete* command is only active when a relevant item like support position, beam, pile etc. is selected.

4.9.3 View Menu

The *View* menu contains commands that turns various toolbars, status bars and menus on and off.

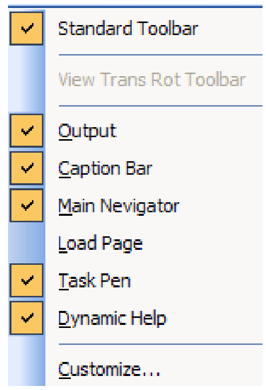


The *View Menu* contains the following menu commands:

- Toolbars
- Status Bar.
- Application Look

Toolbars

The *Toolbars* menu command displays the option to switch on/off different toolbars and controls. If you click on customize button, the program opens a dialog box allowing you to customize the toolbars.



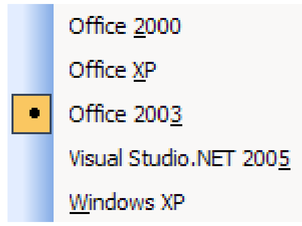
Status Bar

The *Status Bar* command toggles the display of the status bar on and off. The *Status Bar* is positioned at the bottom of the STAAD.*foundation* screen and displays a variety of helpful information, depending on which part of the program you are using, and which command is currently active.

When you hold your mouse cursor over a toolbar button, the left side of the *Status Bar* displays an explanation on how to use the command associated with that particular button. When you hold your mouse cursor over a menu command, the left side of the *Status Bar* displays an explanation of what that menu command does.

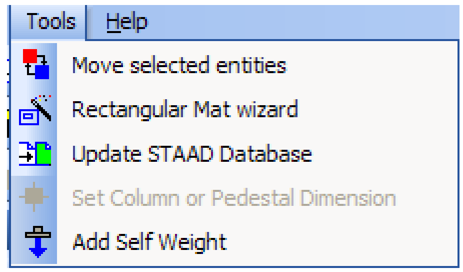
Application Look

STAAD.foundation offers 5 different application look to choose from. Program uses *Office 2003* look by default.



4.9.4 Tools Menu

The *Tools* menu contains commands for manipulating the structure geometry, managing jobs, and adding self weight.

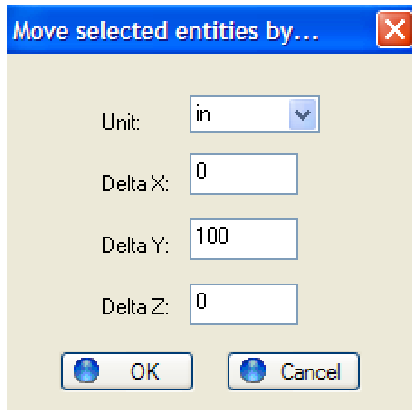


The *Tools* menu contains the following menu commands:

- Move Selected Entities
- Rectangular Mat wizard
- Update STAAD Database
- Set Column or Pedestal Dimension
- Add Self weight

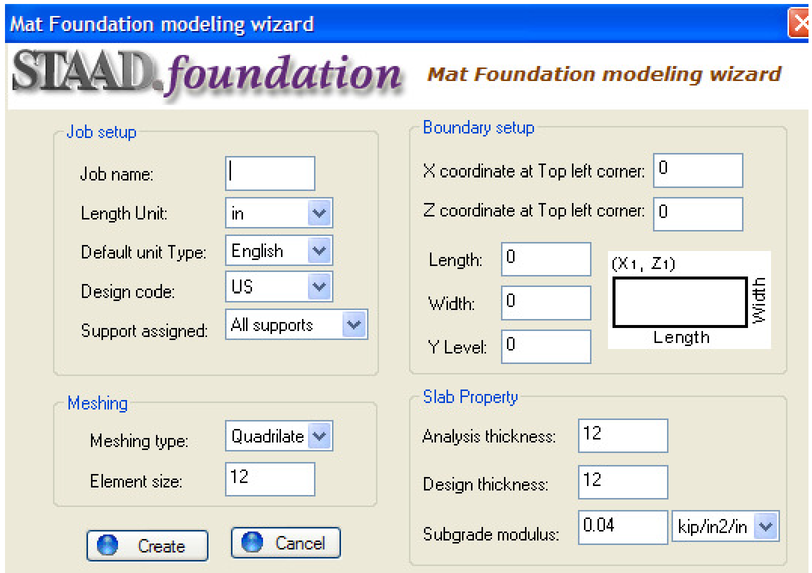
Move Selected Entities

The *Move Selected Entities* menu command allows you to move selected entities like support positions, beams and piles. After selecting the entities to be moved click on the *Move Selected Entities* button, program will bring a dialog box where you need to input incremental X,Y,Z distance.



Rectangular Mat Wizard

This option provides a simple wizard to create mat foundation job. Using this option you can create a job, define boundary, mesh it and define analysis properties.



The following commands are available to create rectangular mat foundation.

Job name

It's an identifier to assign each job a unique name.

Length Unit

Length unit will be used to define mat boundary and to assign slab thickness.

Default Unit Type

Default unit type to be used to setup design parameters.

Design Code

Concrete design code. Program currently supports ACI, BS 8110 and IS 456 codes.

Support assignment

Select the supports to be assigned to the new mat job. You have the option to include all supports or selected supports only.

Boundary Setup

As it's a rectangular mat foundation we just need to define top left corner and then specify Length and Width values of the boundary. The slab will always be in XZ plane. Specify Y level of the slab too.

Meshing

Select mesh generation technique to be used. Program can generate either quadrilateral shaped plate element or triangular shaped plate element. As it is a rectangular mat quadrilateral shape will be more commonly used. However, if the mat is supported on pile or the mat has line loads triangular meshing is recommended.

Slab Property

Define both analysis and design thickness of the slab. Define soil subgrade modulus. Please note, the wizard will create the soil property of the slab but will not assign automatically.

Create

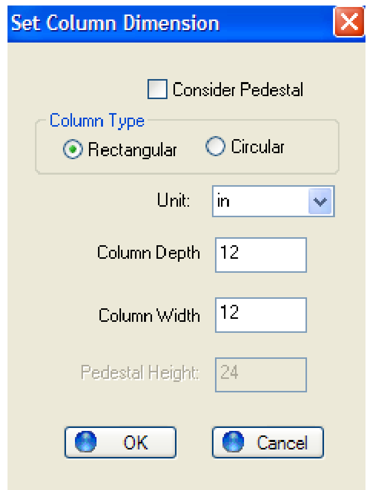
Click on create button to generate a rectangular mat foundation job.

Update STAAD Database

This utility is only useful if the global geometry of the foundation project was originally imported from STAAD.Pro. If the STAAD.Pro results get changed after the foundation project is created, this utility allows user to update current project's input database with the changed STAAD.Pro output. Clicking on this menu item will open a file open dialog box where you need to choose the original file which was imported.

Set Column or Pedestal Dimension

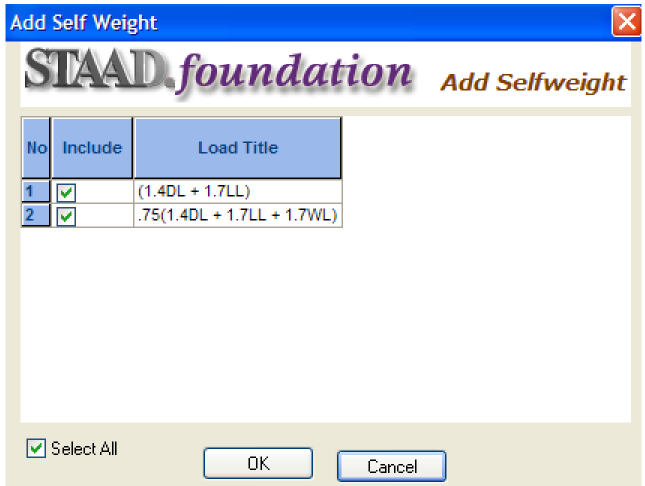
This option will only be enabled if any support position is selected. Clicking on the menu item will open a dialog box which will allow user to set column or pedestal dimension.



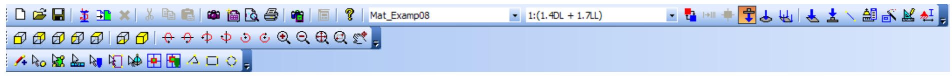
You can specify whether the column is rectangular or circular and input column dimension. If you want to add a pedestal, click on the option called “Consider Pedestal”. It will change the input from column type to pedestal type. For pedestal in addition to the plan dimension you can specify pedestal height.

Add Self Weight

This option allows user to add self weight to the selected or all load cases. This self weight definition is only applicable for Mat foundation as program will not add self weight of the mat slab by default. For other types of footing like isolated or combined footing program automatically adds self weight for all service load cases. Clicking on this button will open a dialog box where all load cases in the project will be listed. Check *Include* check box to include self weight to a load case. At the bottom of the dialog box there is a control to check on/off all load cases. Click on OK to assign/unassign self weight.



4.10 The Toolbars



STAAD.*foundation* offers a set of "dockable" and "floating" toolbars for quick access to frequently used commands. By default, the toolbar icons appear at the top of the STAAD.*foundation* screen immediately below the menu bar. You may, however, drag each toolbar and place it at any position on the screen (hence the term "floating"). In addition, if you drag a floating toolbar close to the edge of the screen, the toolbar gets embedded at the side the screen (hence the term "dockable"). The title of a "docked" toolbar is not displayed. However, if you drag the toolbar and leave it "floating" on the screen, a title is displayed at the top of the toolbar.

Each toolbar icon offers tooltip help. If you are not sure what a toolbar icon does, place your mouse cursor over the toolbar icon for a moment and a floating help message appears to identify what the toolbar icon does.

STAAD.*foundation* offers several toolbars, each of which contains several toolbar icons. The following toolbars are available:

- Standard Toolbar
- Rotate Toolbar
- Zoom Toolbar
- Select Toolbar

4.10.1 Standard Toolbar

Standard toolbar has several icons in it and can be categorized as followings

- File Toolbar
- Print Toolbar
- Import Toolbar
- Save Picture
- Change Job
- Change Current Load Case
- Tools Toolbar
- Loading Toolbar
- View Options Toolbar
- Unit setup toolbar
- Scale setup toolbar

4.10.1.1 File Toolbar

The *File Toolbar* allows you to perform project file related operations such as creating a new project, opening an existing project and saving a project.

The *File Toolbar* contains the following toolbar icons:

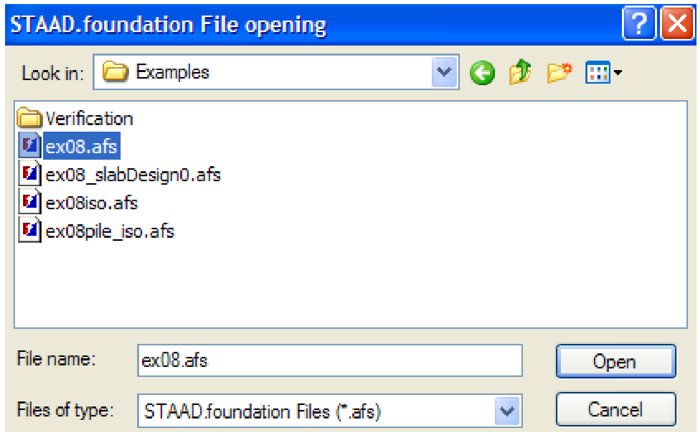
- New
- Open
- Save

New

The *New* icon creates a new STAAD.*foundation* project.

Open

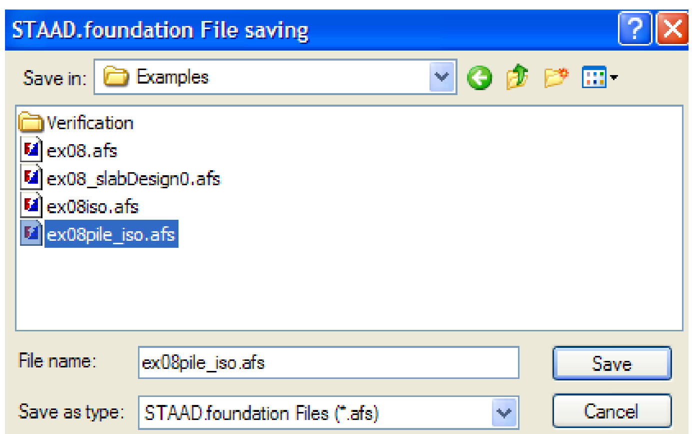
The *Open* icon brings up the open file dialog box and allows you to open an existing STAAD.*foundation* project.



To open an existing project, navigate to the directory in which the project file is located and then select the file and click on *Open*.

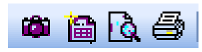
Save

The *Save* icon brings up the save file dialog box the first time the icon is clicked and allows you to save the active project to a file.



STAAD.*foundation* projects are saved with an .afs file extension. After a project has been saved to a file, clicking on the *Save* icon again will simply save any updates made to the project to the file specified when you first saved the project.

4.10.1.2 Print Toolbar



The *Print Toolbar* allows you to perform print related operations for project reports.

The *Print Toolbar* contains the following toolbar icons:

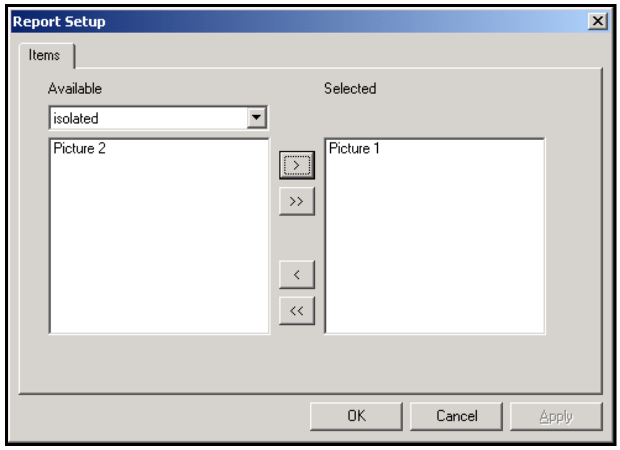
- Take Picture
- Report Setup for Printing
- Print Preview
- Print

Take Picture

The *Take Picture* icon takes a snapshot of the *Graphics Window*. Pictures taken will then be selectable items when creating reports via the *Report Setup for Print* dialog box. Pictures are grouped together with the job they are created in.

Report Setup for Printing

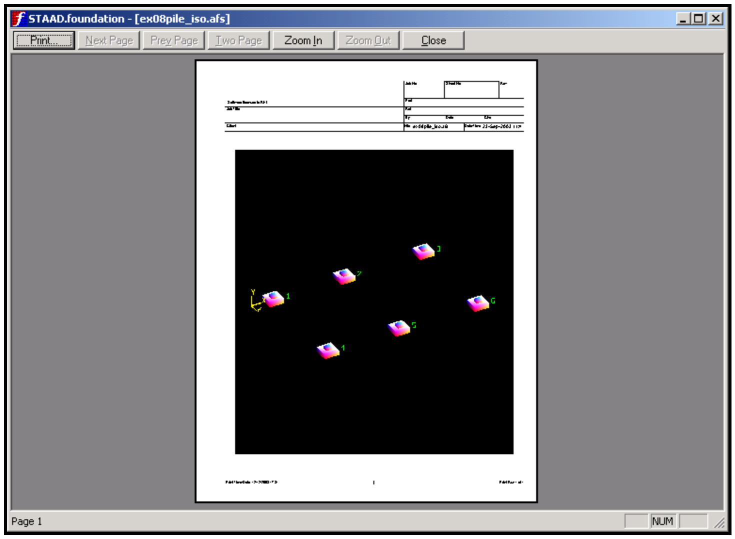
The *Report Setup for Print* icon opens a dialog box allowing you to select what items will appear in the active project report.



The drop-down list box under the heading *Available* allows you to choose the job to select items from. Once a job is selected, the list box under *Available* will contain the items existing for that particular job. You can then use the > button to transfer selected items to a report and the >> button to transfer all items to a report. To remove items from a report, use the < button to remove selected items and the << button to remove all items.

Print Preview

The *Print Preview* icon opens a new window allowing you to view what the active report will look like when printed out.



The following command buttons are available in the *Print Preview* window:

- Print
- Next Page
- Prev. Page
- Two Page
- Zoom In,
- Zoom Out
- Close

Print

The *Print* button opens a standard Windows print dialog box and allows you to print the active project report.

Next Page

The *Next Page* button displays the next page in a report. If there is only one page in a report or you are at the last page in a report, the button will be grayed out.

Prev. Page

The *Prev. Page* button displays the previous page in a report. If you are at the first page in a report, the button will be grayed out.

Two Page

The *Two Page* button allows you to display two pages of a report on the screen at a time. Once you are in two page mode, the text on the button will change to "*One Page*." If you click on the button again, one page will be displayed on the screen at a time and the text on the button will change back to "*Two Page*."

Zoom In

The *Zoom In* button allows you to zoom in closer on a page of a report. After you click on the *Zoom In* button, your mouse cursor will change to a magnifying glass. You may then zoom in on a portion of a report by clicking on the region you want to zoom in on. Once you have zoomed in, the *Zoom In* button will become grayed out. You may return to the original viewing distance by clicking on the *Zoom Out* button.

Zoom Out

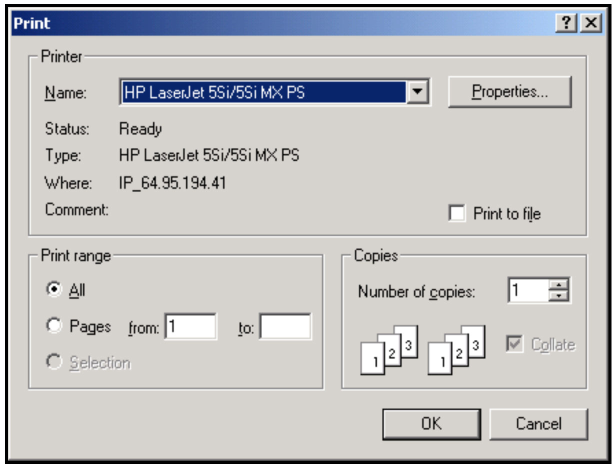
The *Zoom Out* button allows you to zoom back out after zooming in on a page of a report. The *Zoom Out* button is only active after zooming in on a page of a report.

Close

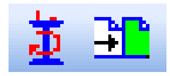
The *Close* button removes the print preview window.

Print

The *Print* icon opens a standard Windows print dialog box and allows you to print the active project report.


Print
Printer
Name:
HP LaserJet 5Si/5Si MX PS
Properties...
Status:
Ready
Type:
HP LaserJet 5Si/5Si MX PS
Where:
IP_64.95.194.41
Comment:
Print to file
Print range
All
Pages
from:
1
to:
Selection
Copies
Number of copies:
1
Collate
OK
Cancel

4.10.1.3 Import Toolbar



Import toolbar allows you to import any analyzed STAAD.Pro file and update foundation input database if the STAAD.Pro file gets changed.

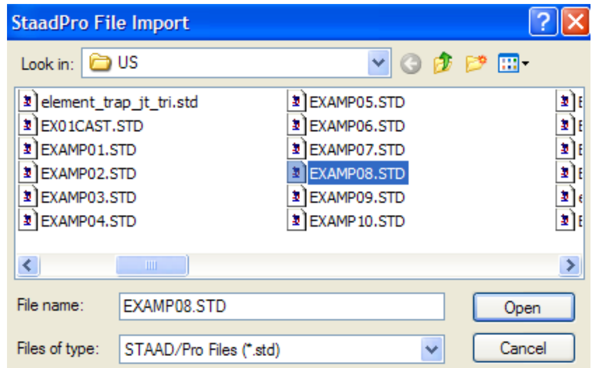
Import toolbar has following icons

- Import
- Update database

Import

The *Import* menu command is used to begin a new project by importing the support geometry and support reactions from a STAAD.*Pro* analysis. The ability to import analysis data from other structural analysis software programs will be provided in a future release of STAAD.*foundation*.

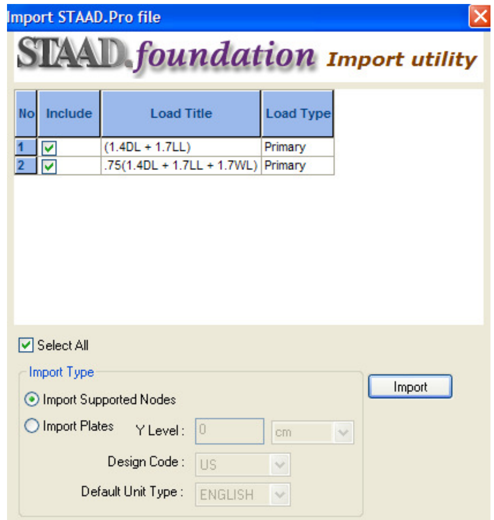
You can only import a STAAD.*Pro* model that has been successfully analyzed, because you will want to have the support reactions available for the foundation design. When the *Import* command is executed, an import dialog box will appear.



To import a STAAD.*Pro* file, navigate to the directory in which the file is located and then select the file and click on *Open*. Another dialog box will appear listing all the available load cases in the STAAD.*Pro* file. You may select any or all of the load cases by toggling on the corresponding check box, and then clicking on the *Import* button. As STAAD.Pro does not have the definition for serviceability and ultimate load type, the user should assign the right load type attribute to each load case here. By default program defines all load cases as primary load types.

You can import either support coordinates or a slab already analyzed in STAAD.Pro to design in STAAD.foundation. To import support coordinates simply click on import button and it will import all support positions along with support reactions for all selected load cases.

To import an analyzed slab select *Import Plates* option and then enter Y level of the slab position. Program can only import a slab defined in XZ plane. Now select design code to be used to design the slab. Click on Import button and that will import the slab along with plate stresses and node displacements for selected load cases. The program will automatically create a mat foundation job.

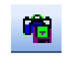


The *Jobs Toolbar* allows you to create, select, and edit jobs. Jobs allow you to assign a set of constraints for STAAD.*foundation* to use in performing a foundation design. Each project may contain multiple jobs so that you can evaluate different design scenarios for a given set of physical conditions.

Update STAAD Database

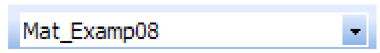
This utility is only useful if the global geometry of the foundation project was originally imported from STAAD.Pro. If the STAAD.Pro results get changed after the foundation project is created, this utility allows user to update current project's input database with the changed STAAD.Pro output. Clicking on this menu item will open a file open dialog box where you need to choose the original file which was imported.

4.10.1.4 Save Picture Toolbar



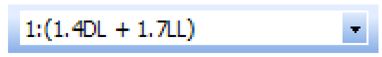
This icon allows you to save current screen to bitmap picture file.

4.10.1.5 Change Job Toolbar



The *Change Job* drop-down list box allows you to select a job from a list of jobs you have created for the active project. To change jobs, simply select the job you wish to change to from the drop-down list box. If no jobs have been created for a project, the drop-down list box will be empty.

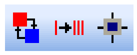
4.10.1.6 Change Current Load Case Toolbar



The *Change Load* drop-down list box allows you to change load cases by selecting from a list of load cases available in the active project. To change load cases, simply select the load case you wish to change to from the drop-down list box. If no load cases have been created for a project, the drop-down list box will be empty.

4.10.1.7 Tools Toolbar

Tools toolbar allows you to move and generate geometry and set column/pedestal dimensions.

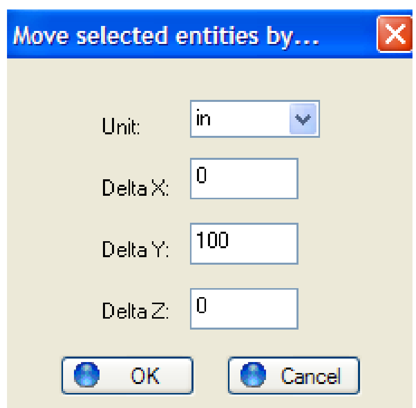


Tools toolbar has following icons

- Move Selected Entities
- Translational Repeat
- Set Column/Pedestal dimension
- Mat foundation wizard

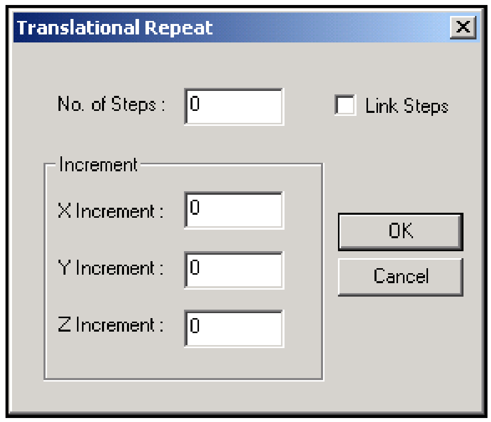
Move Selected Entities

The *Move Selected Entities* command allows you to move selected entities like support positions, beams and piles. After selecting the entities to be moved click on the *Move Selected Entities* button, program will bring a dialog box where you need to input incremental X,Y,Z distance.



Translational Repeat

The *Translational Repeat* icon opens a dialog box that allows you to duplicate objects in a model. The command works similar to a copy command, except multiple copies of an object can be made at a time. In addition, objects may be linked to together with transverse members. In order to use *Translational Repeat*, you must first select at least one object before the command will become available.



The *Translational Repeat* dialog box contains the following fields and options:

- No. of Steps
- Increment
- Link Steps

No. of Steps

The *No. of Steps* field allows you to specify the number of copies to make.

Increment

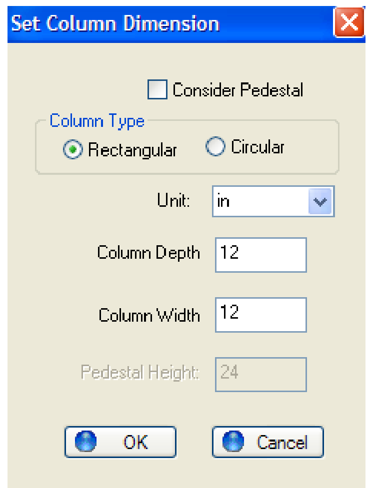
The *Increment* group box allows you to specify the distance from the original object(s) to the copied object(s). When *No. of Steps* is greater than 1, the increment values will also specify the distance between the multiple copied objects. The increment unit used is specified in the *Change Length Unit* drop-down list box in the *Tools* toolbar.

Link Steps

The *Link Steps* check box allows you to specify whether the copied objects should be linked together with transverse members.

Set Column or Pedestal Dimension

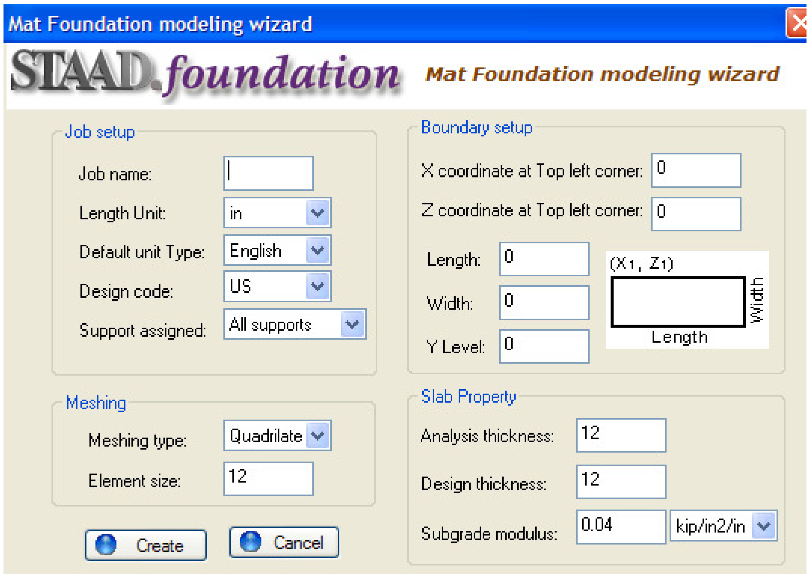
This option will only be enabled if any support position is selected. Clicking on the menu item will open a dialog box which will allow user to set column or pedestal dimension.



You can specify whether the column is rectangular or circular and input column dimension. If you want to add a pedestal, click on the option called “Consider Pedestal”. It will change the input from column type to pedestal type. For pedestal in addition to the plan dimension you can specify pedestal height.

Rectangular Mat Wizard

This option provides a simple wizard to create mat foundation job. Using this option you can create a job, define boundary, mesh it and define analysis properties.



The following commands are available to create rectangular mat foundation.

Job name

It's an identifier to assign each job a unique name.

Length Unit

Length unit will be used to define mat boundary and to assign slab thickness.

Default Unit Type

Default unit type to be used to setup design parameters.

Design Code

Concrete design code. Program currently supports ACI, BS 8110 and IS 456 codes.

Support assignment

Select the supports to be assigned to the new mat job. You have the option to include all supports or selected supports only.

Boundary Setup

As it's a rectangular mat foundation we just need to define top left corner and then specify Length and Width values of the boundary. The slab will always be in XZ plane. Specify Y level of the slab too.

Meshing

Select mesh generation technique to be used. Program can generate either quadrilateral shaped plate element or triangular shaped plate element. As it is a rectangular mat quadrilateral shape will be more commonly used. However, if the mat is supported on pile or the mat has line loads, then triangular meshing is recommended.

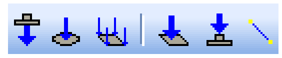
Slab Property

Define both analysis and design thickness of the slab. Define soil subgrade modulus. Please note, the wizard will create the soil property of the slab but will not assign automatically.

Create

Click on create button to generate a rectangular mat foundation job.

4.10.1.8 Loading Toolbar



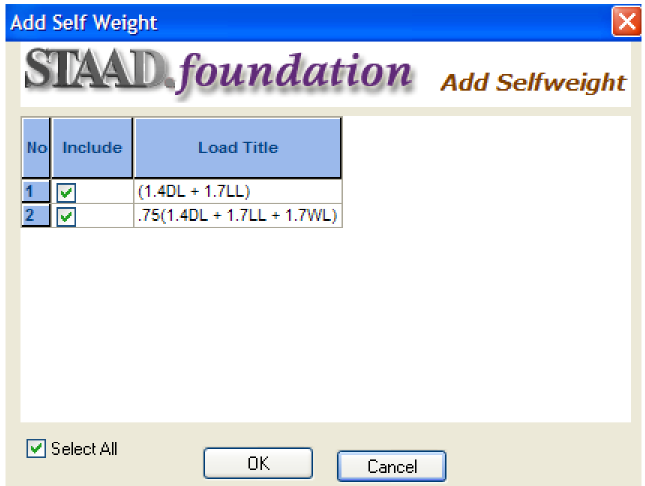
Loading toolbar allows you to add selfweight and other physical load items like circular load, quadrilateral load etc. to the current load case

Loading toolbar has following icons

- Add Self weight
- Add Circular Pressure Load
- Add Quadrilateral Pressure Load
- Add Point load on space
- Add column reaction load
- Add line load

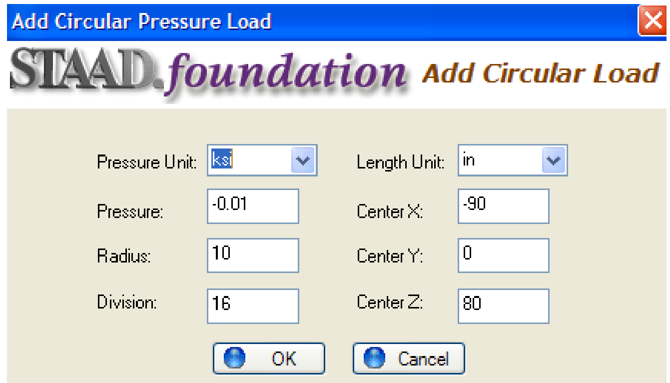
Add Self Weight

This option allows user to add self weight to the selected or all load cases. This self weight definition is only applicable for Mat foundation as program will not add self weight of the mat slab by default. For other types of footing like isolated or combined footing program automatically adds self weight for all service load cases. Clicking on this button will open a dialog box where all load cases in the project will be listed. Check *Include* check box to include self weight to a load case. At the bottom of the dialog box there is a control to check on/off all load cases. Click on OK to assign/unassign self weight.



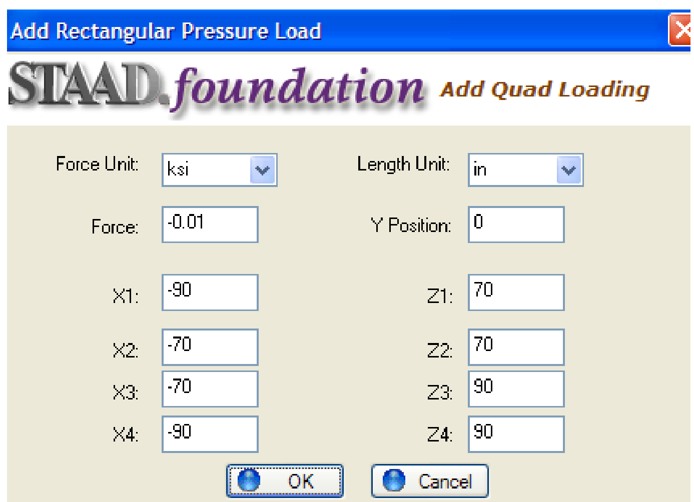
Add Circular Pressure Load

This option allows user to add a circular pressure load to the current load case. Please note, circular pressure is applicable only to mat foundation. To create the circular pressure load click on any grid intersection point and that will become the center of the circle and then drag the mouse to the desired circular radius and release the mouse on a grid intersection node. The distance between the first point and second point is the radius of the circle. If the mouse cursor is not released on a grid intersection point load will not be created. After successful mouse release on a grid intersection point a dialog box will appear where you can input pressure and modify center and radius values.



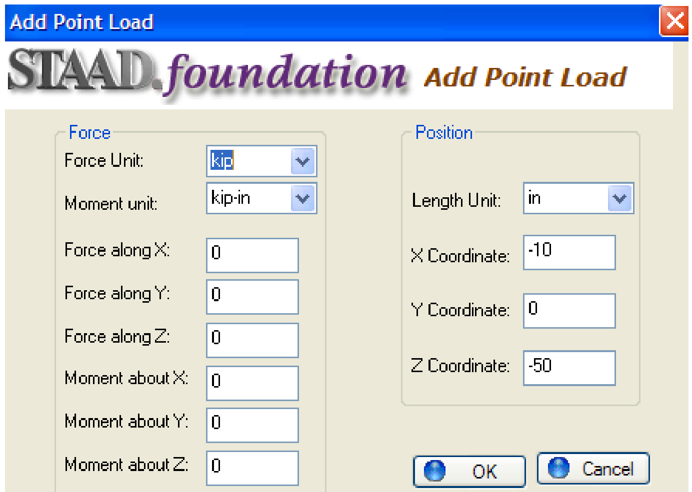
Add Quadrilateral Pressure Load

This option allows user to add a quadrilateral pressure load to the current load case. Please note, quadrilateral pressure is applicable only to mat foundation. To create the quadrilateral pressure load click on any grid intersection point and that will become the top left corner of the rectangle. Draw the rubber band and release the mouse cursor on a grid intersection point which will be bottom right corner of the rectangle. A dialog box will appear which will allow you to input pressure and modify coordinates.



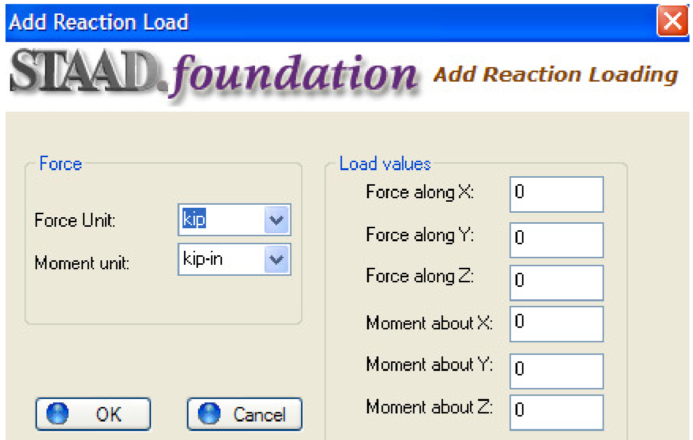
Add Point Load on Space

This option allows user to create a point load on space. This option is available only for mat foundation. Click on any grid intersection point and the program will add a point load at that point. After clicking on a grid intersection point a dialog box appears which will allow user to input load values for all six degrees of freedom.



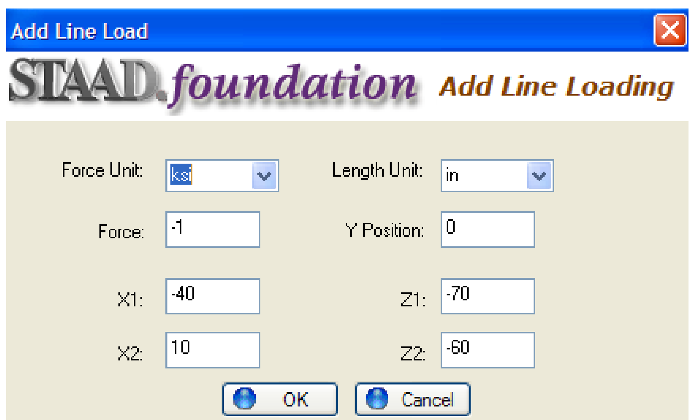
Add Column reaction load

This option allows user to add a reaction load to a support position. After clicking on this icon click on any support node and a dialog box will appear allowing you to input load values. Input load values and click on OK program will create the reaction load and assign to the selected support.



Add Line load

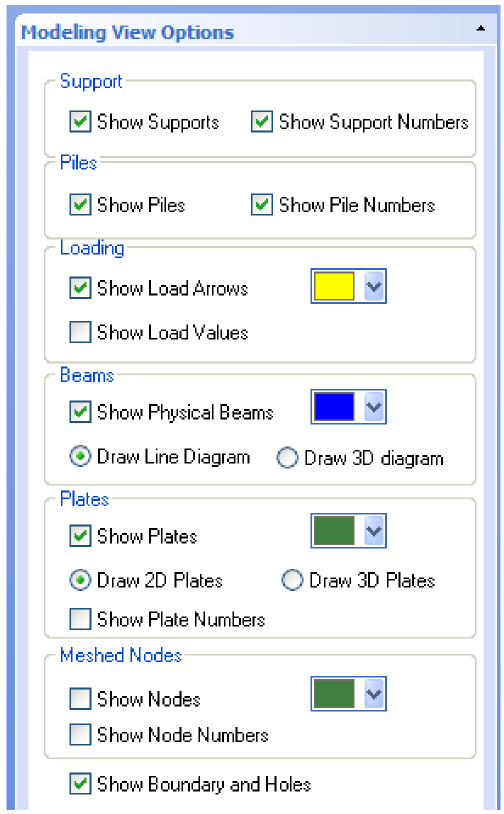
This option allows user to add a line load on mat slab. This option is only useful for mat foundation. Click on any grid intersection point and that point will become the first point of the line. Drag mouse cursor and click on second grid intersection point a dialog box will appear allowing you to input load value and modify coordinates.



4.10.1.9 View options Toolbar

This option allows you to control graphics display by switching on/ off certain options. It also has options to change color of certain entities.

The following commands are available



Show Support

Switch on to view supports in the graphics

Show Support Numbers

This option is used to display support numbers. Support numbers will not be displayed if *Show Supports* option is switched off.

Show Piles

This option is used to switch on/off display of piles in graphics area.

Show Pile Numbers

This option is used to display pile numbers. Pile numbers will not be displayed if *Show Piles* option is switched off.

Show Load Arrows

This option is used to display load arrows. The color picker control next to it allows user to select a suitable color to draw load arrows.

Show Load Values

This option is used to display load values next to the load arrows.

Show Physical Beams

This option is used to display physical beams if present in the project. The color picker control next to it allows user to select a suitable color to draw physical beams.

Draw Line/3D diagram

Select this option to draw physical beam as a line or as a solid surface. Beam property will be used to draw the rectangular beam shape.

Show Plates

This option is used to display plate elements if present in current job. The color picker control next to it allows user to select a suitable color to draw meshed plates.

Draw 2D/3D Plates

This option gives user a choice to display plates as 2d surface or a solid 3d diagram.

Show Plate Numbers

This option is used to display plate numbers at the center of each plate. This option won't display plate numbers if *Show Plates* is switched off.

Show Nodes

This option is used to display plate nodes as blobs. This option is switched off by default. The color picker control next to it allows user to choose a suitable color to draw plate nodes.

Show Node Numbers

This option is used to display node numbers next to the plates nodes. This option won't display node numbers if *Show Nodes* is switched off.

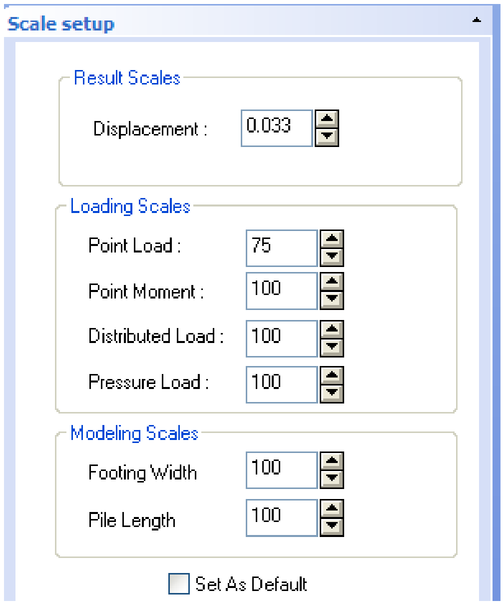
Show Boundary and Holes

This option is used to display boundary and holes created for mat foundation.

4.10.1.10 Scale Setup Toolbar

The *Scales* page allows you to control the scale at which displacements, loads, and drawing entities like footings and piles are displayed on the model. If the structure's loads or deformed shape are not clearly visible in the *Graphics Window* when the options to display them are turned on, you may need to change the scaling values.

The following commands are available



- Results Scales
- Loading Scales
- Modeling Scales
- Set As Default

Results Scales

This group allows you to change the displacement scale of a mat foundation. Displacement diagram is only available for mat foundation after a successful analysis.

Note: You should decrease the scaling value to increase the amount of deflection or loading shown on the diagram. Why do you decrease the parameter value to increase the apparent size? The values in the dialog box represent the actual displacement or loading per unit distance on the graphic diagram. Therefore, if you reduce the amount of actual structural deflection required to display a unit distance of deflection on the diagram, you will see a larger apparent displacement on the diagram.

Loading Scales

This group allows you to change the display of load arrows. Concentrated force and moment for a point load has different scaling options. Distributed load scale is applicable to line load on mat and beam loads. Pressure load scale is applicable to quadrilateral and circular pressure load.

Modeling Scales

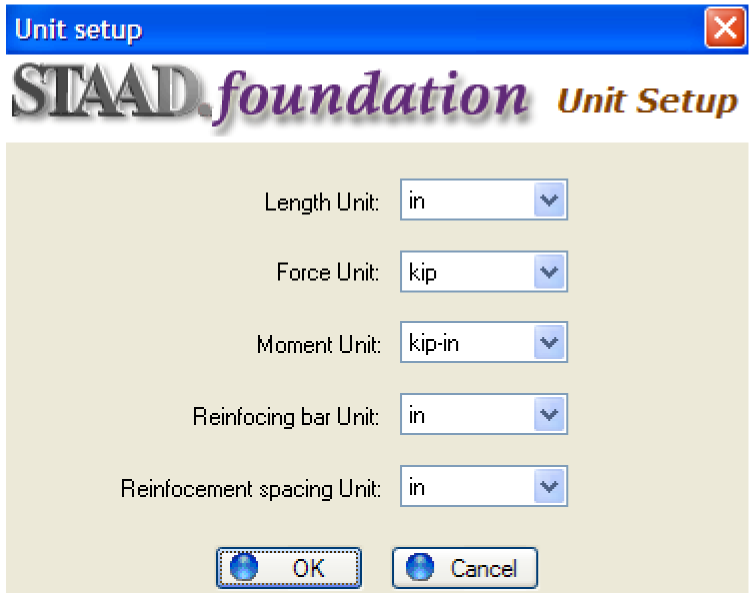
This group allows you to change the display size of supports and piles. Drawings of footing size are not scaled as the sizes are not known, so sometimes those entities may seem too big or small. Changing the scale user can control the sizes of those drawings.

Set As Default

This option allows user to save the current scale setup for later use, so user don't need to change the scale each time a project is opened.

4.10.1.11 Unit Setup Toolbar

The *Set Output Unit* command opens a dialog box that allows you to set output units.



The following output units can be set:

- Length Unit
- Force Unit
- Moment Unit
- Reinforcing Bar Diameter
- Reinforcing Bar Spacing

Length Unit

Length unit is used in all tables and pages as appropriate. Column position, column dimension, pile position, slab thickness are some of the pages where length units are used. Forms for loading etc., output tables and calculation sheet also uses length unit.

Force Unit

Force unit is used for all input and output related force. Pressure unit is determined combining length and force unit.

Moment unit

Moment unit is primarily used for loading input and output.

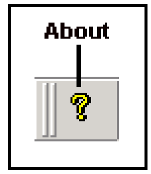
Reinforcing Bar Diameter

This unit is used in output table and calculation sheet to report reinforcing bar diameter.

Reinforcing Bar Spacing

This unit is used in output table and calculation sheet to report required and provided reinforcing bar spacing.

4.10.2 Help Toolbar



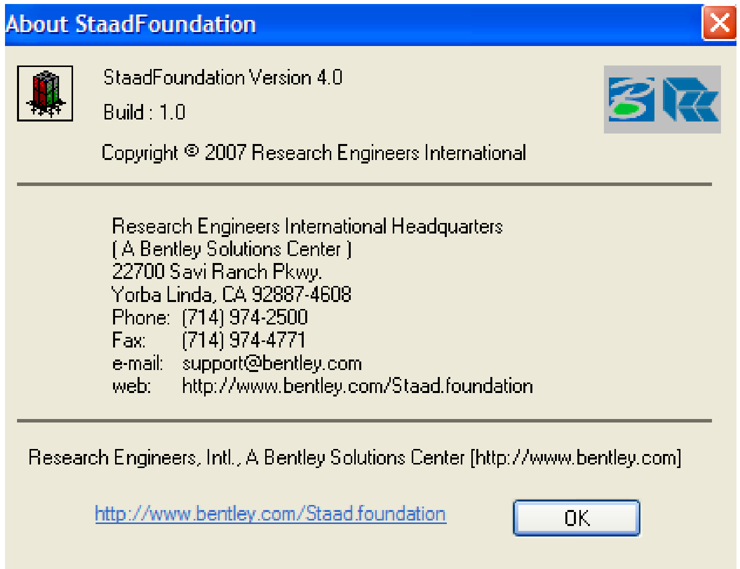
The *Help Toolbar* allows you to obtain information about STAAD.*foundation*.

The *Help Toolbar* contains the following toolbar icon:

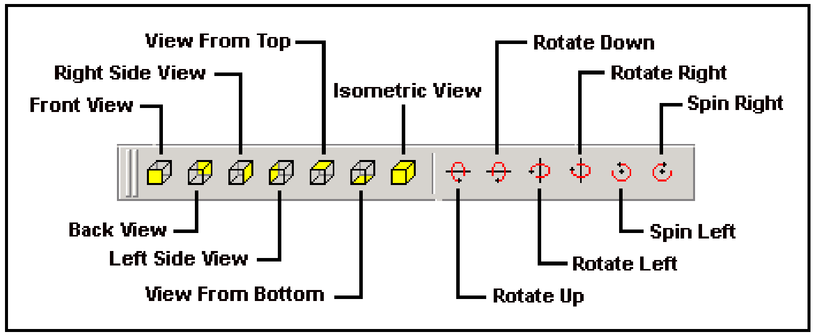
- About

About

The *About* icon opens a dialog box containing information on the version of STAAD.*foundation* you're currently running. The information in the dialog box includes product name, release number and build number. In addition, the physical address, web address and phone numbers for the Research Engineers International is displayed.


About StaadFoundation
StaadFoundation Version 4.0
Build : 1.0
Copyright © 2007 Research Engineers International
Research Engineers International Headquarters
(A Bentley Solutions Center)
22700 Savi Ranch Pkwy.
Yorba Linda, CA 92887-4608
Phone: (714) 974-2500
Fax: (714) 974-4771
e-mail: support@bentley.com
web: http://www.bentley.com/Staad.foundation
Research Engineers, Intl., A Bentley Solutions Center [http://www.bentley.com]
http://www.bentley.com/Staad.foundation
OK

4.10.3 Rotate Toolbar



The *Rotate Toolbar* contains two sets of icons: view and rotate. The view icons allow you to change the viewing angle in the main view pane with respect to the global axis system. The rotate icons allow you to rotate the foundation about the origin.

The *Rotate Toolbar* contains the following toolbar icons:

- Front View
- Back View
- Right Side View
- Left Side View
- View From Top
- View from Bottom
- Isometric View
- Rotate Up
- Rotate Down
- Rotate Left
- Rotate Right
- Spin Left
- Spin Right

Front View

The *Front View* icon allows you to display a foundation as seen from the front. When the global Y-axis is vertical, this is the elevation view, as looking towards the negative direction of the Z-axis.

Back View

The *Back View* icon allows you to display a foundation as seen from the back. When the global Y-axis is vertical, this is the elevation view, as seen looking towards the positive direction of the Z-axis.

Right Side View

The *Right Side View* icon allows you to display a foundation as seen from the right side. When the global Y-axis is vertical, this is the side elevation view, as seen looking towards the negative direction of the X-axis.

Left Side View

The *Left Side View* icon allows you to display a foundation as seen from the left side. When the global Y-axis is vertical, this is the side elevation view, as seen looking towards the positive direction of the X-axis.

View From Top

The *View From Top* icon allows you to display a foundation as seen from the top looking down. When the global Y-axis is vertical, this is the plan view, as seen from the sky looking down.

View From Bottom

The *View From Bottom* icon allows you to display a foundation as seen from the bottom looking up towards the sky.

Isometric

The *Isometric* icon allows you to display a foundation in an isometric view. The angle that defines the isometric view is generally X = 30, Y = 30, and Z = 0.

Rotate Up

The *Rotate Up* icon allows you to rotate the view of a structure about the global X-axis, in the direction indicated by the circular arrow in the icon.

Rotate Down

The *Rotate Down* icon allows you to rotate the view of a structure about the global X-axis, in the direction indicated by the circular arrow in the icon.

Rotate Left

The *Rotate Left* icon allows you to rotate the view of a structure about the global Y-axis, in the direction indicated by the circular arrow in the icon.

Rotate Right

The *Rotate Right* icon allows you to rotate the view of a structure about the global Y-axis, in the direction indicated by the circular arrow in the icon.

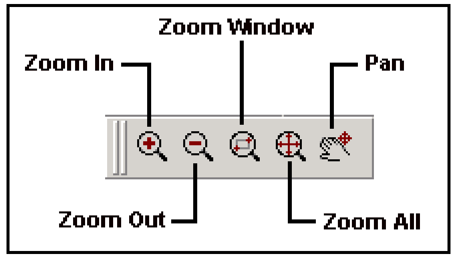
Spin Left

The *Spin Left* icon allows you to spin the view of a structure about the global Z-axis, in the direction indicated by the circular arrow in the icon.

Spin Right

The *Spin Right* icon allows you to spin the view of a structure about the global Z-axis, in the direction indicated by the circular arrow in the icon.

4.10.4 Zoom Toolbar



The *Zoom Toolbar* allows you to alter the viewing distance of the objects in the *Graphics Window*.

The *Zoom Toolbar* contains the following toolbar icons:

- Zoom In
- Zoom Out
- Zoom Window
- Zoom All
- Pan

Zoom In

The *Zoom In* icon allows you to move in closer to the objects in the *Graphics Window*.

Zoom Out

The *Zoom Out* icon allows you to move farther away from the objects in the *Graphics Window*.

Zoom Window

The *Zoom Window* icon allows you to create a selection around an area in the *Graphics Window* that you would like zoom in on. The area that is selected will occupy the entire *Graphics Window*.

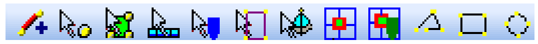
Zoom All

The *Zoom All* icon allows you to return to the viewing distance in which all objects in the *Graphics Window* are visible.

Pan

The *Pan* icon allows you to move the objects in the *Graphics Window* up, down, left or right with your mouse cursor.

4.10.5 Select Toolbar



The *Select Toolbar* has several different cursors that allow you to only select certain objects in a model with your mouse cursor.

The *Select Toolbar* contains the following toolbar icons:

- Add Beam
- Select Meshed Nodes Cursor
- Select Plates Cursor
- Select Physical Member Cursor
- Select Pile Cursor
- Select Mat Boundary cursor
- Create a Column Position Clicking On Grid Intersection Point
- Create a Pile Position Clicking On Grid Intersection Point

Add Beam

The *Add Beam Cursor* icon allows you to add physical beams graphically. Select this cursor and then click on two support nodes to create a beam between those two nodes.

Select Meshed Nodes Cursor

The *Select Meshed Nodes Cursor* icon allows you to select only mesh nodes with your mouse cursor, causing all other objects to be ignored.

Select Plates Cursor

The *Select Plates Cursor* icon allows you to select only plates with your mouse cursor, causing all other objects to be ignored.

Select Physical Member Cursor

The *Select Physical Member Cursor* icon allows you to select only physical members with your mouse cursor, causing all other objects to be ignored.

Select Pile Cursor

The *Select Pile Cursor* icon allows you to select only piles with your mouse cursor, causing all other objects to be ignored.

Select Mat Boundary Cursor

The *Select Mat Boundary Cursor* icon allows you to select only mat boundary with your mouse cursor, causing all other objects to be ignored.

Create a Column Position Clicking On Grid Intersection Point

The *Create a Column Position Clicking on Grid Intersection Point* icon allows you to place column positions on grid intersection points using your mouse cursor. A grid is created using the *Grid Setup* page under the *Foundation Plan* group.

Create a Pile Position Clicking On Grid Intersection Point

The *Create a Pile Position Clicking on Grid Intersection Point* icon allows you to place pile positions on grid intersection points using your mouse cursor. A grid is created using the *Grid Setup* page under the *Foundation Plan* group.

N
o
t
e
s

Notes

Plant Foundations

Section 5

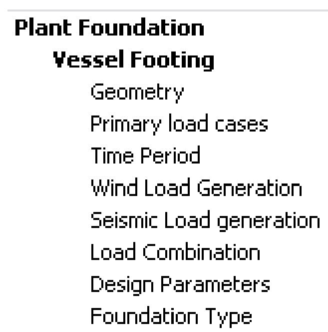
This section includes discussion on the following topics:

- Introduction
- Vertical Vessel Foundation
- Heat Exchanger Foundation

5.1 Introduction

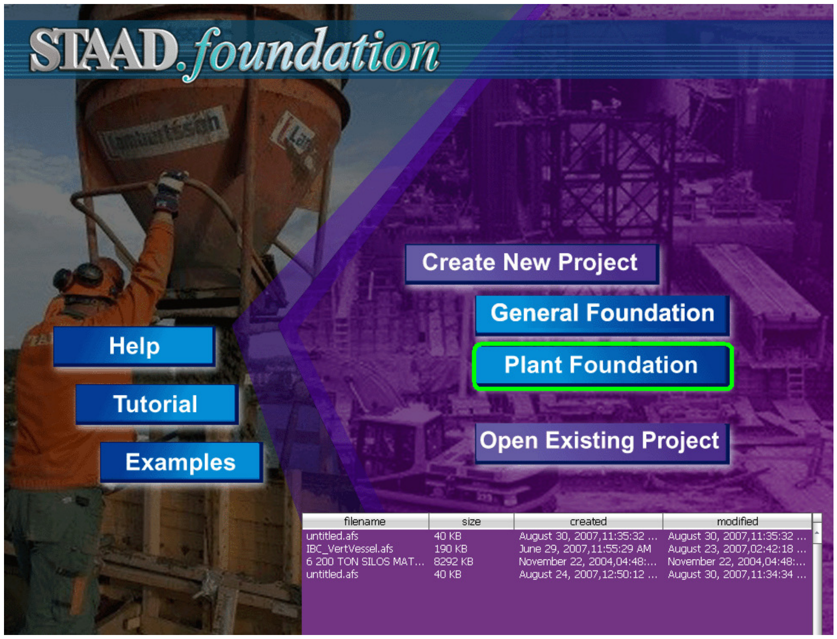
This section provides an overview of two new modules added in STAAD.*foundation 4.0*. These modules are added keeping in mind the necessities of the fast growing plant industry. Theses two modules are *Vertical Vessel Foundation* and *Heat Exchanger Foundation.*

These are completely wizard driven modules. Wizard will guide the user to easily create the foundation step by step. You will finish up a job just simply clicking on "*Next*" button from the wizard and putting some input values. However you can directly jump to any page using a tree control on the left side of the wizard pane. Following picture shows the tree control.

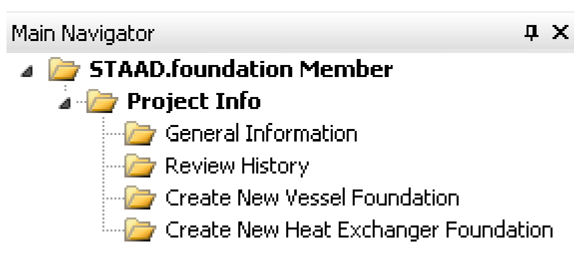


5.1.1 Creating a New Plant Setup Job

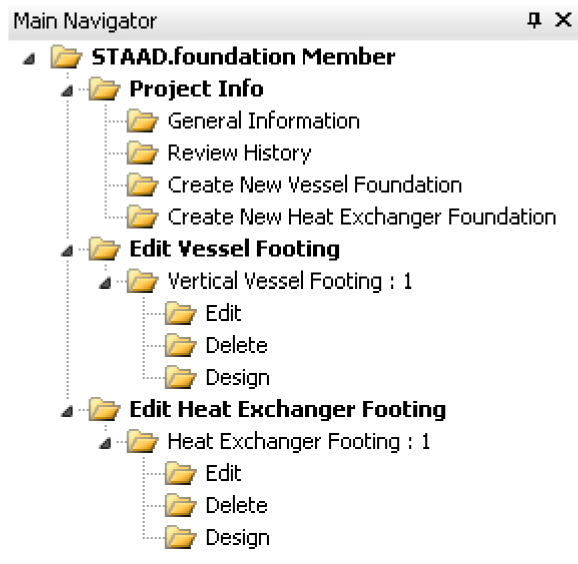
Go to the *Start Page* of STAAD.*foundation 4.0*. Choose "*Plant Foundation*".



This will crate the GUI for plant setup. In the left side of the window there will be a tree control in the *"Main Navigator"* pane as the following figure shows.



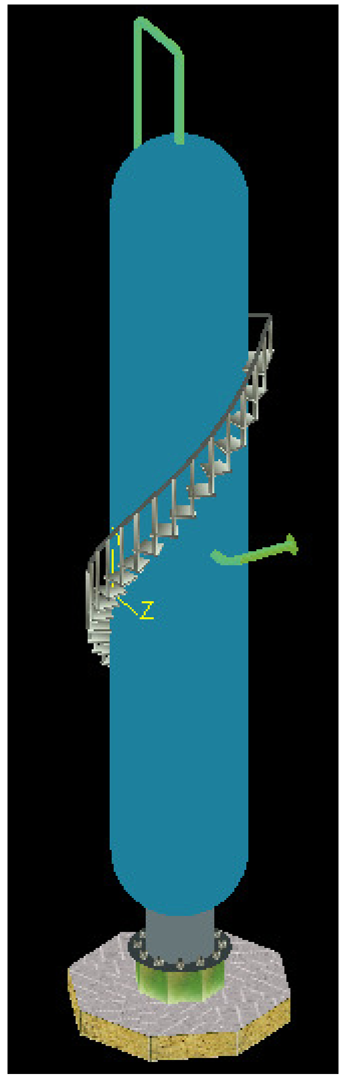
Here you will see two leafs for *"Vertical Vessel Foundation"* and *"Heat Exchanger Foundation"*. Click on one which you want to use. You can create many vessels as well as heat exchangers as you wish. The created jobs will be listed in the tree view as follows.



Later you can edit or delete any of them just clicking on *"Edit"* & *"Delete"* in the tree leaf. And on clicking on *"Design"* it will design the corresponding foundation.

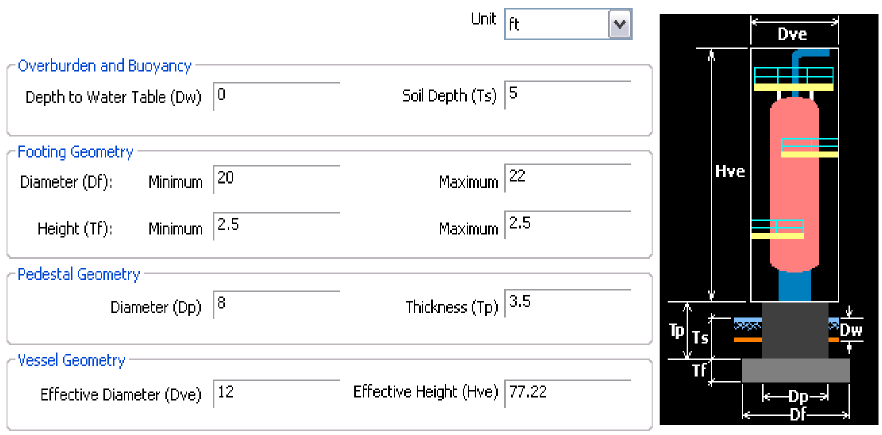
5.2 Vertical Vessel Foundation

Three types of foundation are allowed to design for vertical vessel. They are octagonal footing on soil, square pile cap and octagonal pile cap. The following articles describe all the pages of the wizard for vertical vessel.



5.2.1 Geometry Page

This is the first page of the wizard where you have to input all the relevant geometrical data. Following picture shows the corresponding page.



As you can see that a picture placed right to the page shows the diagrammatic view of the corresponding dimensions.

<u>Unit</u>

Unit of length for all the input in this page only.

Overburden and Buoyancy

<u>Depth of Water Table(Dw)</u>

Depth of the water tables measured from the ground level.

<u>Soil Depth(Ts)</u>

Depth of soil above the foundation measured from the top face of the footing base.

Footing Geometry

Diameter (Df)

Enter the minimum diameter which will be used in starting the design and will be checked up to the maximum value until the design reaches the safety limit.

Height (Tf)

Enter the minimum height which will be used in starting the design and will be checked up to the maximum value until the design reaches the safety limit.

Pedestal Geometry

Diameter (Dp)

Diameter of the pedestal.

Thickness (Tp)

Thickness of the pedestal.

Vessel Geometry

Effective Diameter (Dve)

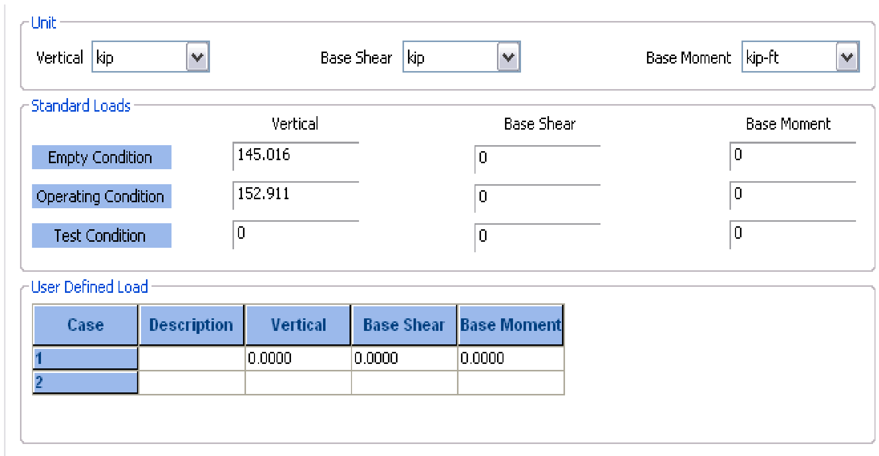
The effective diameter is the diameter that will be used to calculate the wind pressure on the vessel.

Effective Height (Hve)

The effective height is the effective height of the vessel that will be used to calculate the wind pressure and the seismic effect on the vessel.

5.2.2 Primary Load Page

Here you have to input the primary loads other than wind load and seismic load.



<u>Unit</u>

Select the input unit for vertical force, base shear and base moment for this page only.

Standard Loads

For vertical vessel the standard loaded conditions are *Empty condition, Operating condition* & *Test condition*. You may input all the three kind of forces for three kinds of conditions. They are vertical force, base shear and base moment.

User Defined Load

You can also input those kinds of load as *User Defined Load*.

5.2.3 Time Period Page

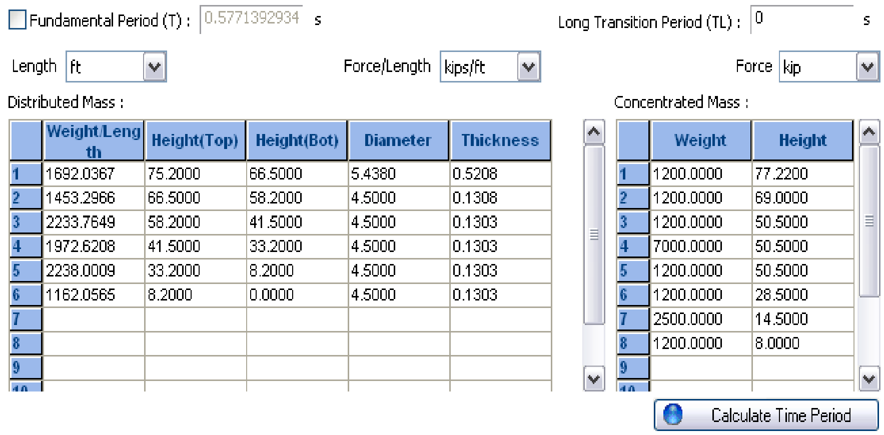
Inputs in this page are basically required for wind load and seismic load calculation according to ASCE 7-05. You can manually enter the value of the *Fundamental time period* of the vessel or you can use the software to calculate it for you.

The time period is calculated using Von Miss Theorem as described below,

$$T = \left(\frac{H}{100}\right)^2 \sqrt{\frac{\sum w_i \Delta\alpha_i + \frac{1}{H}\sum W_i \beta_i}{\sum E D_i^3 t_i \Delta\gamma_i}}$$

Where,
H = Overall height in feet,
D = Diameter of each section in feet,
w = Distributed weight per foot of each section,
W = Weight of each concentrated mass,
t = Shell thickness of each section in inches,
E = Modulus of elasticity for each section in millions of psi,
α,β and γ = Are coefficient for a given level depending on h_i/H(the ratio of height of the level above grade to the overall height).
$\Delta\alpha$ and $\Delta\gamma$ are the difference in the values of α and γ,from the top to the bottom of each section of uniform weight, diameter and thickness. β is determined for each concentrated mass.

Now let us describe the input required in the *Time Period Page*. The following picture shows the corresponding page.



Fundamental Period (T) : 0.5771392934 s Long Transition Period (TL) : 0 s

Length ft Force/Length kips/ft Force kip

Distributed Mass :

	Weight/Length	Height(Top)	Height(Bot)	Diameter	Thickness
1	1692.0367	75.2000	66.5000	5.4380	0.5208
2	1453.2966	66.5000	58.2000	4.5000	0.1308
3	2233.7649	58.2000	41.5000	4.5000	0.1303
4	1972.6208	41.5000	33.2000	4.5000	0.1303
5	2238.0009	33.2000	8.2000	4.5000	0.1303
6	1162.0565	8.2000	0.0000	4.5000	0.1303
7					
8					
9					

Concentrated Mass :

	Weight	Height
1	1200.0000	77.2200
2	1200.0000	69.0000
3	1200.0000	50.5000
4	7000.0000	50.5000
5	1200.0000	50.5000
6	1200.0000	28.5000
7	2500.0000	14.5000
8	1200.0000	8.0000
9		

Calculate Time Period

<u>Fundamental Period (T)</u>

If you want to input the value of the *Fundamental Period* manually then check the check box. The edit box right next to it will be active and you can enter the value. If you want a calculated value provided by the software then leave it unchecked and the edit box will be transformed into a read-only state. Now after giving all the necessary value if you click on *"Calculate Time Period"* button then the calculated value will be shown in the edit box.

<u>Long Transition Period (TL)</u>

Long-period transition period (s) determined in Section 11.4.5

<u>Unit</u>

Units for length, uniformly distributed load and force.

<u>Distributed Mass</u>

In this table put the distributed mass properties of the vessel. Inputs for diameter and thickness is not required if the fundamental period is entered manually. The other inputs are required for calculation of seismic load.

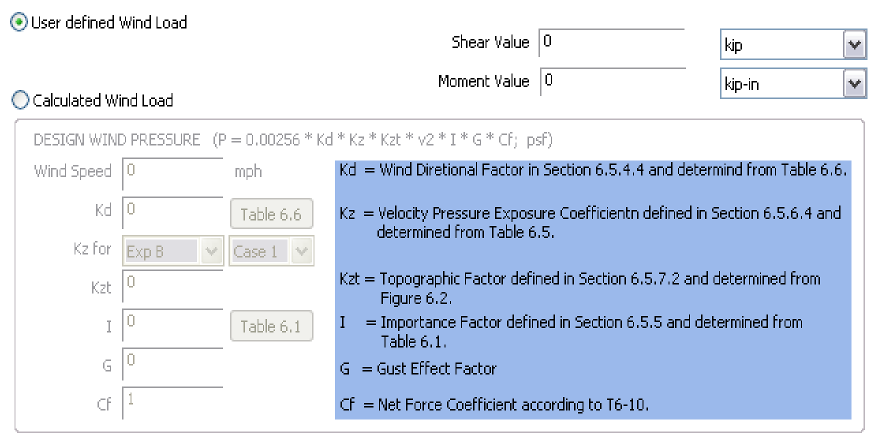
Concentrated Mass

These are the lumped masses attached to the vessel such as ladder, platform etc.

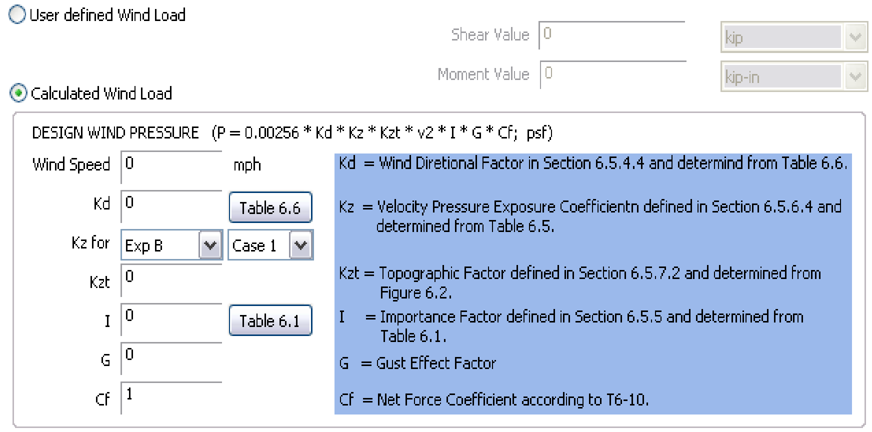
5.2.4 Wind Load Generation Page

Inputs for wind load can be given in two ways. You can directly input the shear force & moment values with choosing the proper unit or you can use the software to calculate those values using ASCE 7-2005.

To input the load directly choose *"User defined Wind Load"* radio button in the wind load page as below and give the value of shear force with choosing units from the combo box right to it.



Otherwise choose *"Calculated Wind Load"* radio button which will activate the required input fields as shown below. All the inputs are described with mentioning the section and table number of the code.



<u>Wind Speed</u>

You need to input the wind speed provided in the code in miles per hour unit.

<u>(Wind Directional Factor (Kd)</u>

Click on the button *"Table 6.6"*. This will show a table as below,

Tanks	
Square	0.90
Hexagonal	0.95
Round	0.95

Choose any value of them and click *"OK"* to use it. Else you can give your own value except those values.

<u>Velocity Pressure Exposure Coefficient (Kz)</u>

This is described in section 6.5.6.4 & Table 6.5. Choose the required combination of combo boxes for them.

<u>Topographic Factor (Kzt)</u>

This is defined in section 6.5.7.2 & determined from figure 6.2.

<u>Importance Factor (I)</u>

Importance is defined in section 6.5.5 & determined from figure 6.1. You can choose the value from a table like *"Kd"* or input your own value.

Catagory	V = 85-100 mph	V > 100 mph
I	0.87	0.77
II	1.00	1.00
II	1.15	1.15
IV	1.15	1.15

OK Cancel

<u>Gust Effect Factor (G)</u>

This is the Gust Effect Factor and it is user defined.

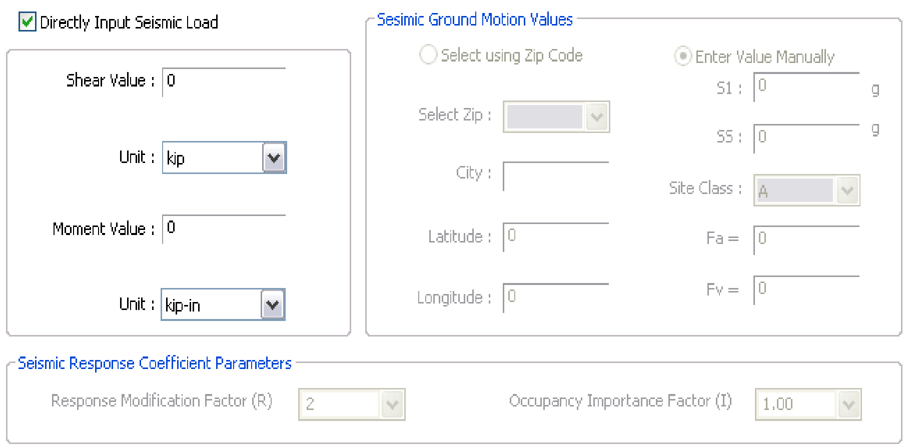
<u>Net Force Coefficient (Cf)</u>

Value of "*Cf*" according to table T6-10.

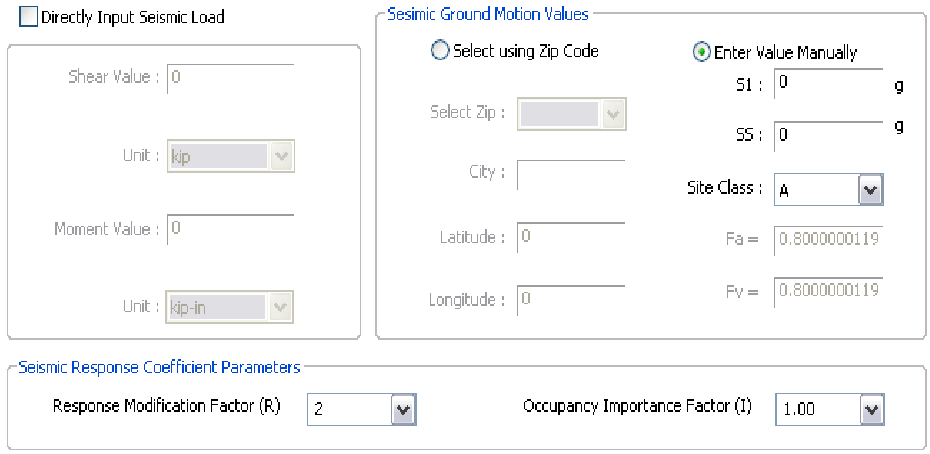
5.2.5 Seismic Load Generation Page

Inputs for seismic load can be given in two ways, same as for wind load. You can directly input the shear force & moment values with choosing the proper unit or you can use the software to calculate those values using ASCE 7-2005.

Check the *"Directly Input Seismic Load"* check box to use your own calculated value for shear force and moment.

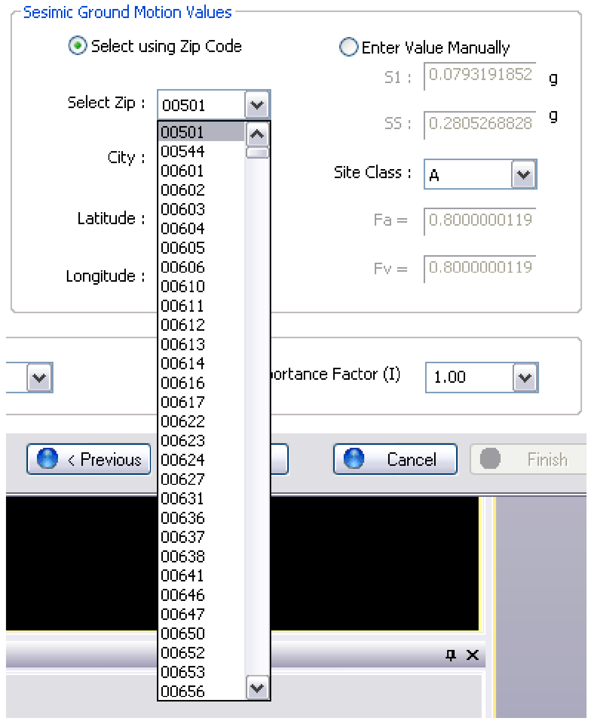


Else uncheck the above said check box to use the software to calculate the values for you. Again that can be done in two ways. You can select US Zip code to get the parametric values or else you can provide their values from your own knowledge.



<u>Select using Zip Code</u>

Choose *"Select using Zip Code"*. This will populate the *"Select Zip"* combo box and then choosing any one of them will fill up the other input boxes. Only you need to choose the *Site Class* from the *Site Class* combo box. It will also show the corresponding *City*, *Latitude* and *Longitude* for that *Zip Code*



Enter Value Manually

Choose *"Enter Value Manually"* to enter the value of *S1* and *Ss* with your own choice.

Site Class

Value of *Fa* & *Fv* depends on choice of *Site Class*. But you can use your own value for them on choosing *Site Class* as *'F'*.

Response Modification Factor (R)

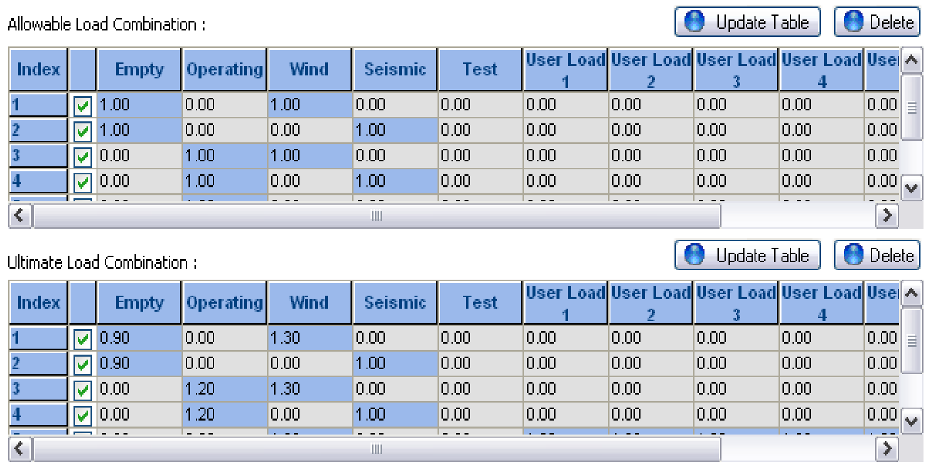
The default value for this field is 2 or 3. But the input control for this value is not a read-only combo box, so use can enter your own desired value.

Occupancy Importance Factor (I)

The default value for this field is 1, 1.25 and 1.5. But the input control for this value is not a read-only combo box, so use can enter your own desired value.

5.2.6 Load Combination Page

Two types of load combinations are used here. They are *"Allowable Load Combination"* and *"Ultimate Load Combination"*. You can create any number of load combination and can save it using an INI file. This file exits in the program installation directory as "ACILOAD.INI". This saved load combination will be application specific i.e. they are independent of file saving. Following figure shows the load combination page.


Allowable Load Combination : Update Table Delete

Index		Empty	Operating	Wind	Seismic	Test	User Load 1	User Load 2	User Load 3	User Load 4	Use
1	☑	1.00	0.00	1.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
2	☑	1.00	0.00	0.00	1.00	0.00	0.00	0.00	0.00	0.00	0.00
3	☑	0.00	1.00	1.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
4	☑	0.00	1.00	0.00	1.00	0.00	0.00	0.00	0.00	0.00	0.00

Ultimate Load Combination : Update Table Delete

Index		Empty	Operating	Wind	Seismic	Test	User Load 1	User Load 2	User Load 3	User Load 4	Use
1	☑	0.90	0.00	1.30	0.00	0.00	0.00	0.00	0.00	0.00	0.00
2	☑	0.90	0.00	0.00	1.00	0.00	0.00	0.00	0.00	0.00	0.00
3	☑	0.00	1.20	1.30	0.00	0.00	0.00	0.00	0.00	0.00	0.00
4	☑	0.00	1.20	0.00	1.00	0.00	0.00	0.00	0.00	0.00	0.00

The first column indicates the index of the load combination. The second row has a check. Check on the check boxes of the combination which you want to use. The cell with zero values appears in gray color where as with values other than zero it appears in sky color.

<u>Update Table</u>

Initially the page shows all the load combination saved in the INI file. You can add new load combination simply by adding factors in the last row. Check on the check box in the second row to use the load combination. If you save a file with those load combination then the load combination will be only saved to that file but not in

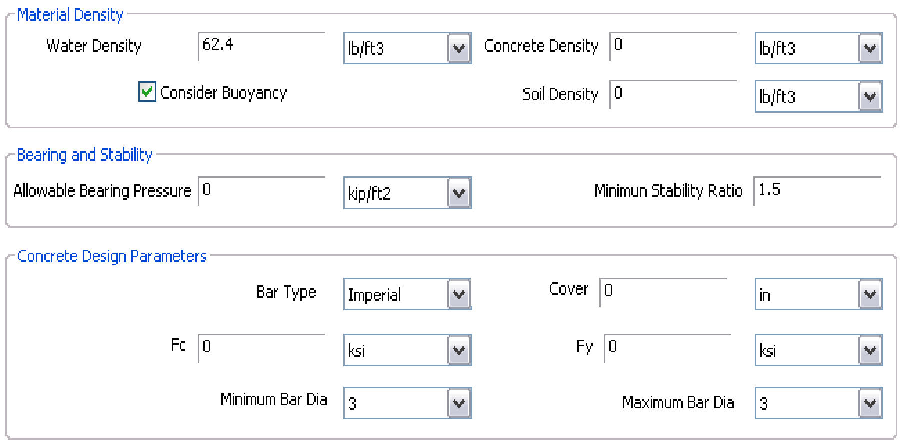
INI file. To save the load combination in the INI file you need to click on *"Update Table"*. You can also manually change the INI file.

Delete

To delete a load combination, select a row and then click on *"Delete"* to delete any particular load combination from the list. But to delete any combination from the INI file you need to click on *"Update Table"* after clicking on *"Delete"*.

5.2.7 Design Parameter Page

Design parameter is grouped under three categories. They are *Material Density, Bearing and Stability* and *Concrete Design Parameters*



Water Density

Density of water with unit to use for *Buoyancy* Check. To check for buoyancy you need to check on the check box *"Consider Buoyancy"*.

Concrete Density

Density of concrete with proper unit.

Soil Density

Density of soil with proper unit.

Allowable Bearing Pressure

Value of allowable bearing pressure used for design.

Minimum Stability ratio

Value of minimum stability ratio used for design.

Bar Type

Types of bar used for the design e.g. *Imperial* or *Metric*.

Cover

Value of clear cover with proper unit.

Fc

Strength of concrete.

Fy

Strength of steel.

Minimum Bar Dia

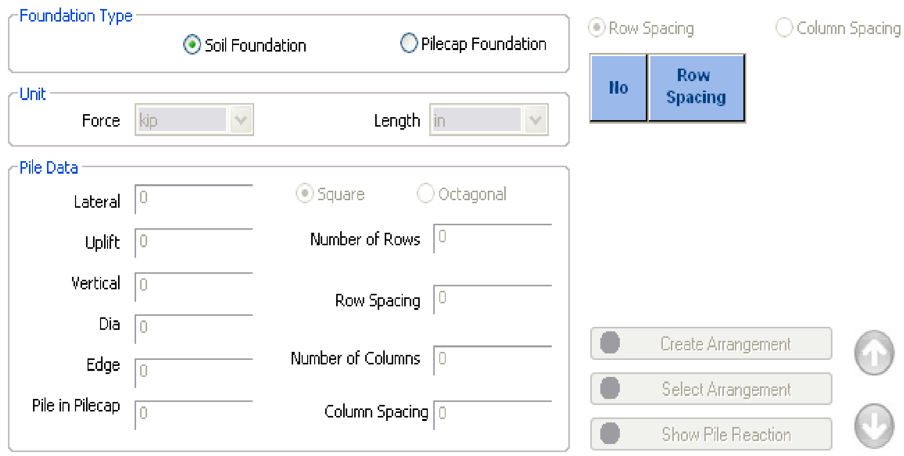
Minimum diameter of bar to use for design.

Maximum Bar Dia

Maximum diameter of bar to use for design.

5.2.8 Foundation Type Page

As discussed earlier you have three options for foundation type. They are *octagonal footing on soil, square pile cap* and *octagonal pile cap*. For the first one i.e. *octagonal footing on soil* you have to use soil foundation as shown below.

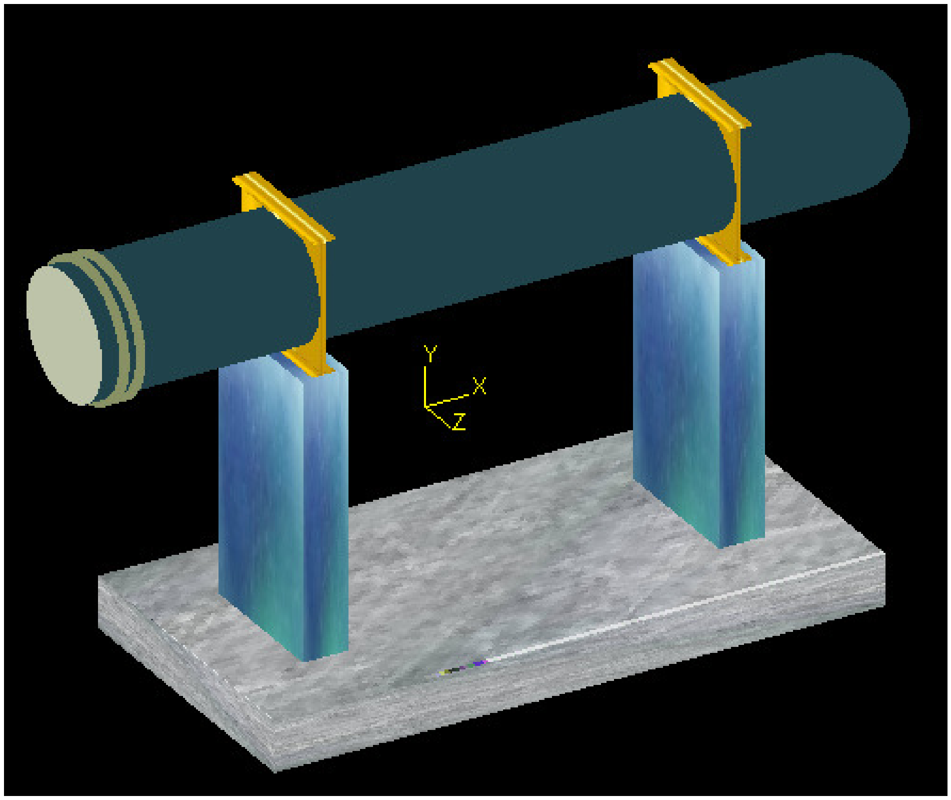


5.2.9 Finish and Design

Now click on "Finish". You will see a Vessel job is added to the tree view in the left side "Main Navigator" pane. Now click on "Design". The design progress can be seen in the "Output Window" situated below. And a detailed calculation sheet will come in the "Calculation Sheet" tab.

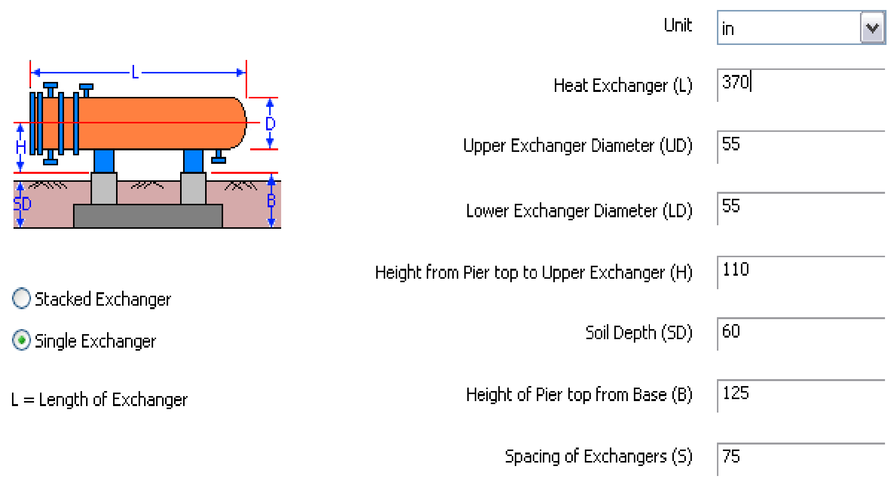
5.3 Heat Exchanger Foundation

Two types of Heat Exchanger are allowed to design. They are *Stacked Exchanger* and *Single Exchanger*.



5.3.1 Exchanger Geometry Page

This is the first page of the wizard where you have to input all the relevant geometrical data for the Heat Exchanger. Following picture shows the corresponding page. Clicking on any input fields creates a description of the corresponding field below the diagram.



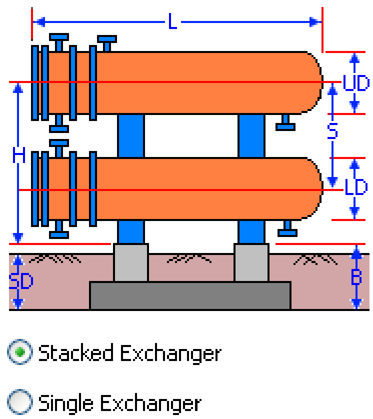
As you can see that a picture placed left to the page shows the diagrammatic view of the corresponding dimensions.

<u>Unit</u>

Unit of length for all the input in this page only.

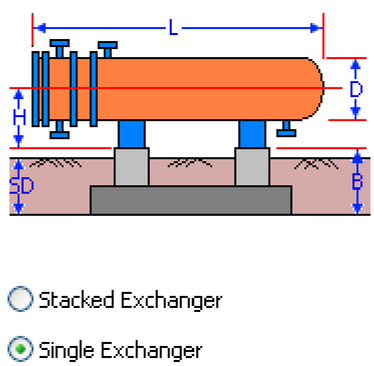
<u>Stacked Exchanger</u>

Click the *Stacked Exchanger* radio button to use one. The picture above shows it.



Single Exchanger

Click the *Stacked Exchanger* radio button to use one. The picture above shows it.



Heat Exchanger Length (L)

Length of the heat exchanger.

Upper Exchanger Diameter (UD)

Diameter of the upper exchanger in case of stacked exchanger.

Upper Exchanger Diameter (UD)

Diameter of the upper exchanger in case of stacked exchanger. This will be used if single exchanger is chosen.

Height from Pier Top to Upper Exchanger (H)

Height from the top of the pier to the center line of the upper exchanger.

Soil Depth (SD)

Depth of soil.

Height of Pier Top from Base (B)

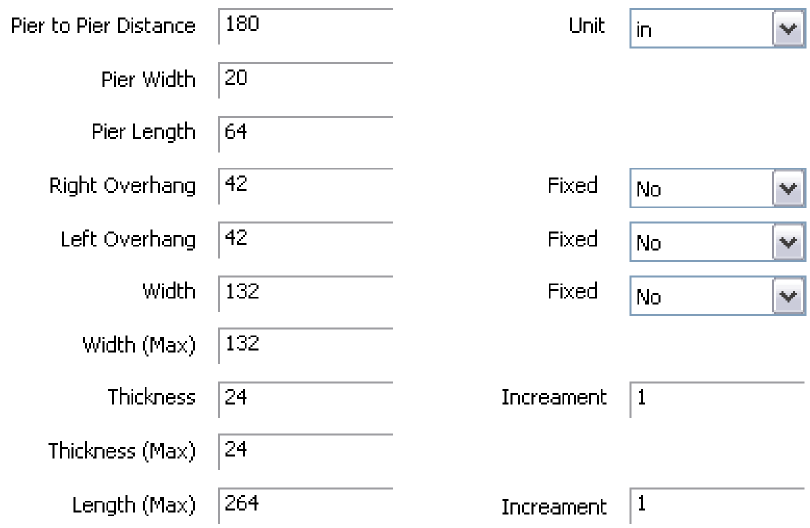
Height from the top of the pier to the base of the foundation.

Spacing of Exchanger (S)

Spacing of the central line of the exchanger in case of stacked exchanger.

5.3.2 Footing Geometry Page

Here you have to input the geometrical data relate to the footing.



<u>Unit</u>

Choose the length dimension unit.

<u>Pier to Pier Distance</u>

Distance of the central lines of the pier.

<u>Pier Width</u>

Width of the pier.

<u>Pier length</u>

Breadth of the pier.

<u>Right Overhang</u>

Length of the right overhang from the central line of the right pier. Choose "Yes" from the combo box right next to it if you wish to make it fix else "No" if you wish to allow it to increase by the design engine.

Left Overhang

Length of the left overhang from the central line of the left pier. Choose "Yes" from the combo box right next to it if you wish to make it fix else "No" if you wish to allow it to increase by the design engine.

Width

Minimum width of the footing. Choose "Yes" from the combo box right next to it if you wish to make it fix else "No" if you wish to allow it to increase by the design engine.

Width (Max)

The maximum width allowed up to which it will be incremented by the design engine.

Thickness

Minimum thickness of the footing.

Thickness (Max)

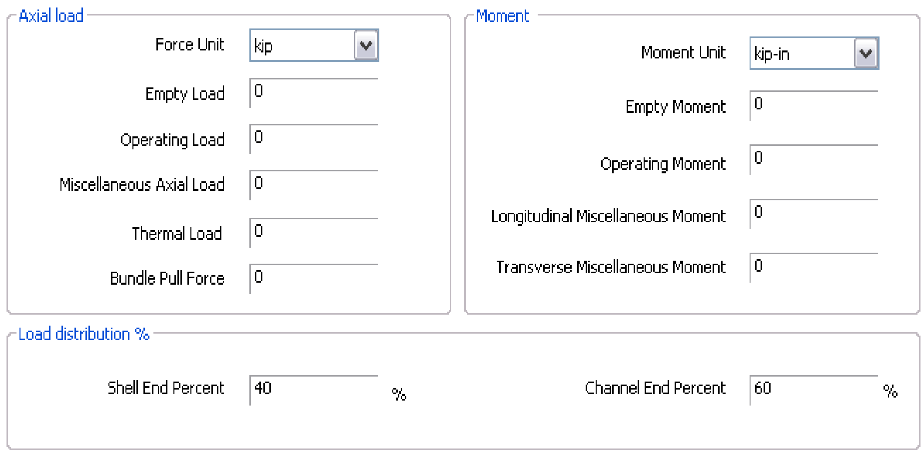
The maximum thickness allowed up to which it will be incremented by the design engine. The rate increment will have to be given on the right "Increment" input field.

Length (Max)

The maximum total length allowed up to which it will be incremented by the design engine. The rate increment will have to be given on the right “Increment” input field.

5.3.3 Primary Load Page

Here you have to input the primary loads other than wind load and seismic load.



Here the loads are grouped into two types, Axial Load and Moment.

<u>Axial Load</u>

Select the unit for Axial load from the combo box. Five types of axial forces are used for input. They are Empty Load, Operating Load, Miscellaneous Axial Force, Thermal Load and Bundle Pull Force.

<u>Moment</u>

Select the unit for Moment from the combo box. Four types of moments are used for input. They are Empty Moment, Operating Moment, Longitudinal Miscellaneous Moment and Transverse Miscellaneous Moment.

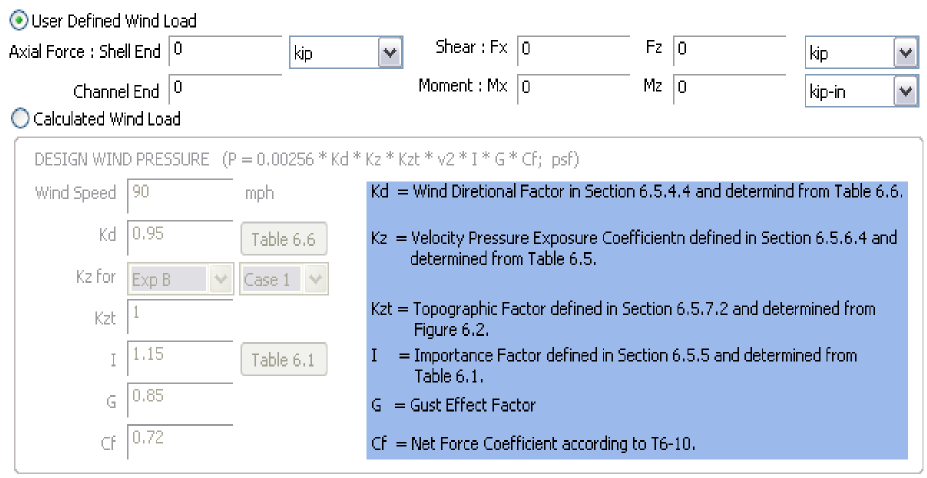
<u>Load Distribution</u>

Give the load distribution percentage for "Shell End" and "Channel End".

5.3.4 Wind Load Generation Page

Inputs for wind load can be given in two ways. You can directly input the shear force & moment values with choosing the proper unit or you can use the software to calculate those values using ASCE 7-2005.

To input the load directly choose *"User defined Wind Load"* radio button in the wind load page as below and give the value of shear force with choosing units from the combo box right next to it.



Axial Force

Give Axial Force value for both "Shell End" and "Channel End" with proper unit.

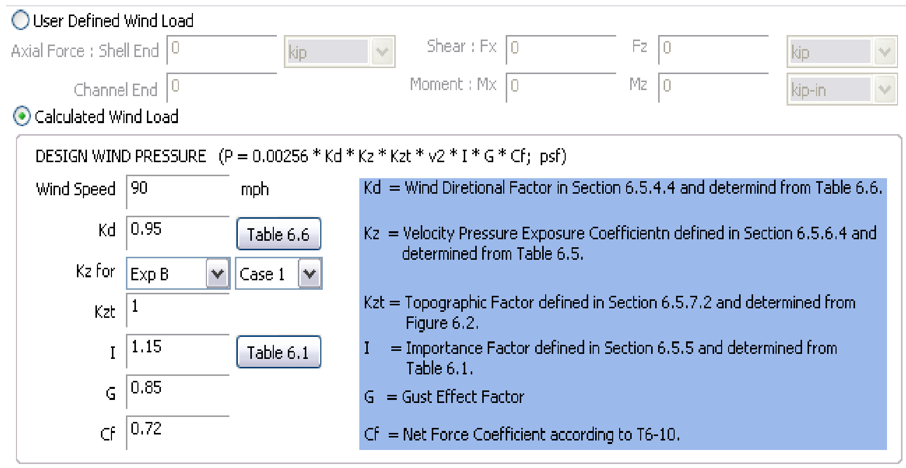
Shear

Give the value of base shear in "X" and "Z" direction with choosing the unit from the unit setup combo box.

Moment

Give the value of moment in "X" and "Z" direction with choosing the unit from the unit setup combo box.

Otherwise choose *"Calculated Wind Load"* radio button which will activate the required input fields as shown below. All the inputs are described with mentioning the section and table number of the code.



<u>Wind Speed</u>

You need to input the wind speed provided in the code in miles per hour units.

<u>(Wind Directional Factor (Kd)</u>

Click on the button *"Table 6.6"*. This will show a table as below,

Tanks	
Square	0.90
Hexagonal	0.95
Round	0.95

OK Cancel

Choose any value of them and click *"OK"* to use it. Else you can give your own value except those values.

Velocity Pressure Exposure Coefficient (Kz)

This is described in section 6.5.6.4 & Table 6.5. Choose the required combination of combo boxes for them.

Topographic Factor (Kzt)

This is defined in section 6.5.7.2 & determined from figure 6.2.

Importance Factor (I)

Importance is defined in section 6.5.5 & determined from figure 6.1. You can choose the value from a table like *"Kd"* or input your own value.

Catagory	V = 85-100 mph	V > 100 mph
I	0.87	0.77
II	1.00	1.00
II	1.15	1.15
IV	1.15	1.15

OK Cancel

Gust Effect Factor (G)

This is the Gust Effect Factor and it is user defined.

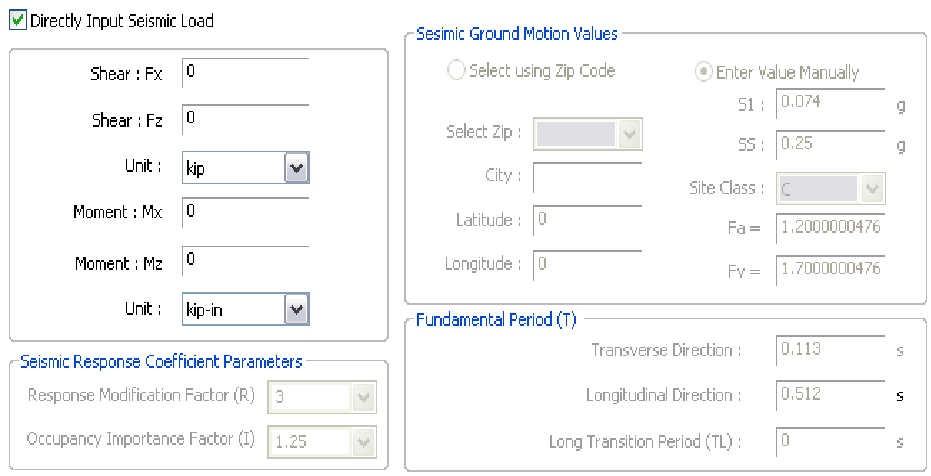
Net Force Coefficient (Cf)

Value of "*Cf*" according to the table T6-10.

5.3.5 Seismic Load Generation Page

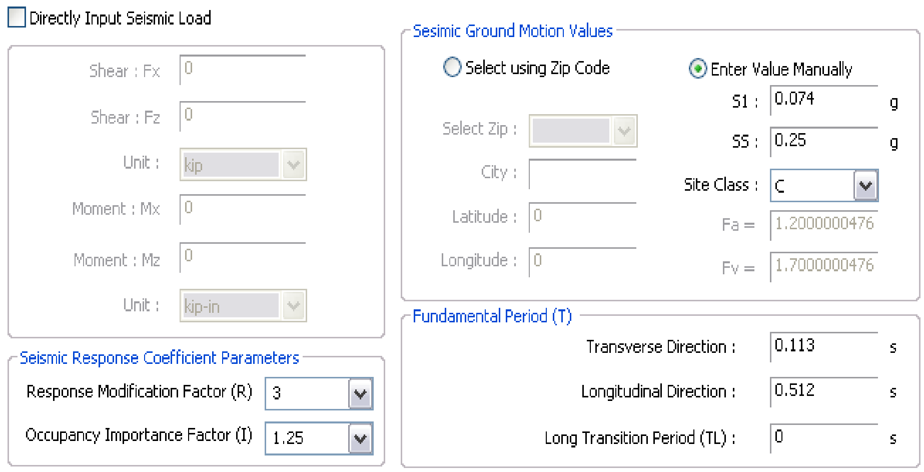
Inputs for seismic load can be given in two ways, same as for wind load. You can directly input the shear force & moment values with choosing the proper unit or you can use the software to calculate those values using ASCE 7-2005.

Check the *"Directly Input Seismic Load"* check box to use your own calculated value for shear force and moment.



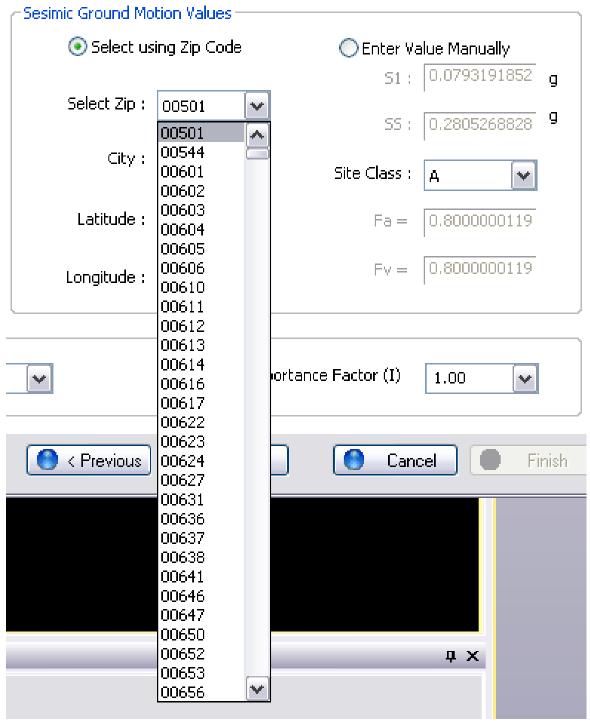
For direct input give the calculated value of shear and moment in both directions with proper choice of unit.

Else uncheck the above said check box to use the software to calculate the values for you. Again that can be done in two ways. You can select US Zip code to get the parametric values or else you can provide their values from your own knowledge.



<u>Select using Zip Code</u>

Choose *"Select using Zip Code"*. This will populate the *"Select Zip"* combo box and then choosing any one of them will fill up the other input boxes. Only you need to choose the *Site Class* from the *Site Class* combo box. It will also show the corresponding *City*, *Latitude* and *Longitude* for that *Zip Code*



Enter Value Manually

Choose *"Enter Value Manually"* to enter the value of *S1* and *Ss* with your own choice.

Site Class

Value of *Fa* & *Fv* depends on choice of *Site Class*. But you can use your own value for them on choosing *Site Class* as *'F'*.

Response Modification Factor (R)

The default value for this field is 2 or 3. But the input control for this value is not a read-only combo box, so you can enter your own desired value.

Occupancy Importance Factor (I)

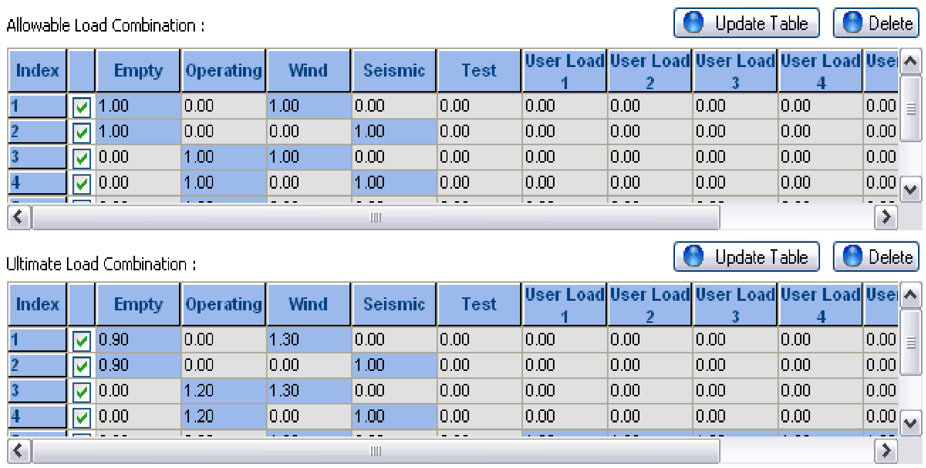
The default value for this field is 1, 1.25 and 1.5. But the input control for this value is not a read-only combo box, so you can enter your own desired value.

Fundamental Period (T)

Fundamental period in both the direction (longitudinal and transverse) have to be provided with the Period for Long Transition defined in section 11.4.5.

5.3.6 Load Combination Page

Two types of load combinations are used here. They are *"Allowable Load Combination"* and *"Ultimate Load Combination"*. You can create any number of load combination and can save it using an INI file. This file exits in the program installation directory as "ACILOAD.INI". This saved load combination will be application specific i.e. they are independent of file saving. Following figure shows the load combination page.



Index		Empty	Operating	Wind	Seismic	Test	User Load 1	User Load 2	User Load 3	User Load 4	Use
1	☑	1.00	0.00	1.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
2	☑	1.00	0.00	0.00	1.00	0.00	0.00	0.00	0.00	0.00	0.00
3	☑	0.00	1.00	1.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
4	☑	0.00	1.00	0.00	1.00	0.00	0.00	0.00	0.00	0.00	0.00

Index		Empty	Operating	Wind	Seismic	Test	User Load 1	User Load 2	User Load 3	User Load 4	Use
1	☑	0.90	0.00	1.30	0.00	0.00	0.00	0.00	0.00	0.00	0.00
2	☑	0.90	0.00	0.00	1.00	0.00	0.00	0.00	0.00	0.00	0.00
3	☑	0.00	1.20	1.30	0.00	0.00	0.00	0.00	0.00	0.00	0.00
4	☑	0.00	1.20	0.00	1.00	0.00	0.00	0.00	0.00	0.00	0.00

The first column indicates the index of the load combination. The second row has a check. Check on the check boxes of the combination which you want to use. The cell with zero values appears in gray color where as with values other than zero it appears in sky color.

Update Table

Initially the page shows all the load combination saved in the INI file. You can add new load combination simply by adding factors in the last row. Check on the check box in the second row to use the load combination. If you save a file with those load combination then the load combination will be only saved to that file but not in

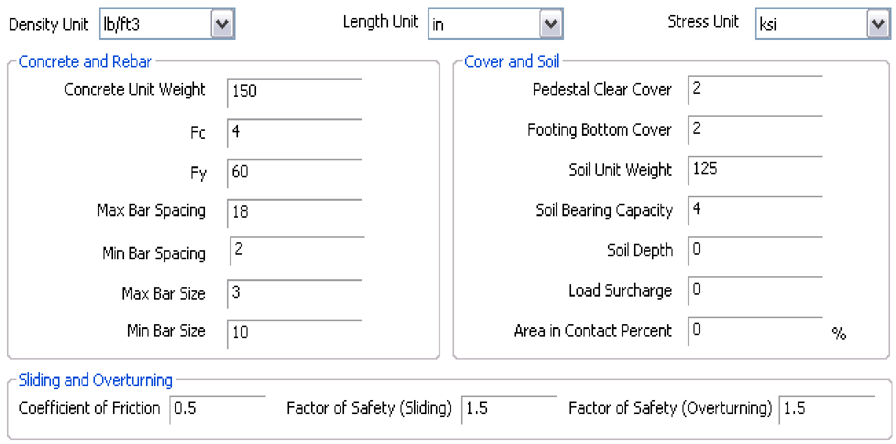
INI file. To save the load combination in the INI file you need to click on *"Update Table"*. You can also manually change the INI file.

Delete

To delete a load combination, select a row and then click on *"Delete"* to delete any particular load combination from the list. But to delete any combination from the INI file you need to click on *"Update Table"* after clicking on *"Delete"*.

5.3.7 Design Parameter Page

Design parameter is grouped under three categories. They are *Concrete and Rebar, Cover and Soil* and *Sliding and Overturning*.



Unit

First give the units for three types of dimensions, density, length and stress.

Concrete and Rebar

Concrete Unit Weight

Unit weight of concrete.

Fc

Strength of concrete.

Fy

Strength of steel.

Minimum Bar Spacing

Minimum spacing of bar to use for design.

Maximum Bar Spacing

Maximum spacing of bar to use for design.

Minimum Bar Size

Minimum diameter of bar to use for design.

Maximum Bar Size

Maximum diameter of bar to use for design.

Cover and Soil

Pedestal Clear Cover

Clear cover for pedestal.

Footing Bottom Cover

Bottom clear cover for the footing.

Soil Unit Weight

Unit weight of soil.

Soil Bearing Capacity

Allowable Bearing Capacity of soil.

Soil Depth

Depth of soil.

Load Surcharge

The surcharge load.

Area in Contact Percent

Percentage of area of contact between footing and soil.

Sliding and Overturning

Coefficient of Friction

Frictional coefficient used for design against sliding.

Factor of Safety

Factor Safety against sliding and overturning.

5.3.8 Finish and Design

Now click on “Finish”. You will see a Vessel job is added to the tree view in the left side “Main Navigator” pane. Now click on “Design”. The design progress can be seen in the “Output Window” situated below. And a detailed calculation sheet will come in the “Calculation Sheet” tab.

Indian Verification Problems

Section 6

This section includes discussion on the following topics:

- Indian Verification Problem 1
- Indian Verification Problem 2
- Indian Verification Problem 3
- Indian Verification Problem 4
- Indian Verification Problem 5

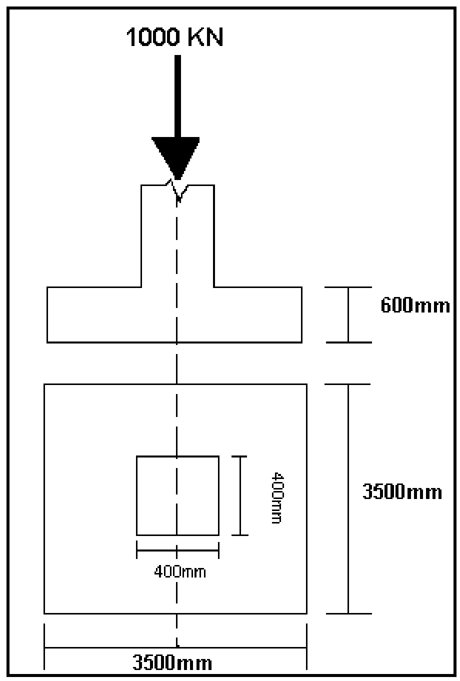
6.1 Indian Verification Problem 1

Reference

'Reinforced Concrete' by A.K. Jain, Page 539, Example 18.2.

Problem

Design an isolated footing with the given data: Load Fy = 1000 KN, fc = 15 MPa, fy = 415 MPa, Column Dimension = 400 mm X 400 mm, Bearing Capacity of Soil = 100 KN/m^2, and Load Factor = 1.5.



Solution

Approximate area of footing required = $\frac{1000}{100}$ m^2 = 10 m^2

Assuming 3.5 m x 3.5 m x 0.6 m footing dimension ($I = 12.5\ m^4$)

Weight of footing = 3.5 x 3.5 x 0.6 x 25 KN = 183.75 KN

Therefore, total load on the footing = (1000 +183.75) KN
= 1183.75 KN

Maximum pressure = $\frac{1183.75}{3.5 \times 3.5}$ KN/ m^2
= 96.633 KN/m^2 <100 KN/m^2 (Hence safe)

Ultimate pressure = $\frac{1000 \times 1.5}{3.5 \times 3.5}$ KN/m^2 = 122.45 KN/m^2

Bending moment at critical section,
M_u = 122.45 x 3.5 x $\frac{1.55 \times 1.55}{2}$ = 514.826 KN-m

Assuming 35 mm clear cover and 10 mm bar, effective depth
d_e = (600-35-0.5 x 10) mm = 560 mm

$$K_{u,max} = \frac{700}{1100 + 0.87 fy} = 0.479$$

$R_{u,max}$ = 0.36 x fc x $K_{u,max}$ x (1-0.42 $K_{u,max}$) = 2.066

Mu_{lim} = $R_{u,max}$ x B x d_e^2 = 2267.642 x 10^6 N-mm
= 2267.642 KN-m> M_u (Hence safe)

Area of Steel Required

Area of steel required along length,

Ast = 0.5 x $\frac{fc}{fy} \times \left[1 - \sqrt{1 - \frac{4.6Mu}{fc \times B \times de \times de}}\right]$ x B x d_e

= 2646.4 mm^2

Minimum area of steel Ast_{min} = 0.0012 x B x D = 2520 mm^2

Check for One-Way Shear

Percentage of steel p_t = $\frac{100Ast}{B \times de}$ = 0.135

Corresponding allowable τ_c = 0.28 N/mm^2

Developed shear stress τ_c = $\frac{Vu\max}{B \times de}$

V_{umax} = 122.45 x 3.5 x $\left(\frac{3.5 - 0.4}{2} - 0.56\right)$ = 424.289 KN

Developed shear stress τ_c = $\frac{424.289 \times 1000}{3500 \times 560}$

= 0.2165 N/mm^2 < $\tau_{c,all}$ (Hence safe)

Check for Two-Way Shear

V_{umax} = 1500 KN

Developed shear stress τ_c = $\frac{1500 \times 1000}{4 \times 960 \times 560}$ = 0.698 N/mm^2

K_s = min (0.5+1,1) = 1

Allowable shear stress = K_s x τ_c = 1 x $0.25\sqrt{fc}$ = 0.968 N/mm^2

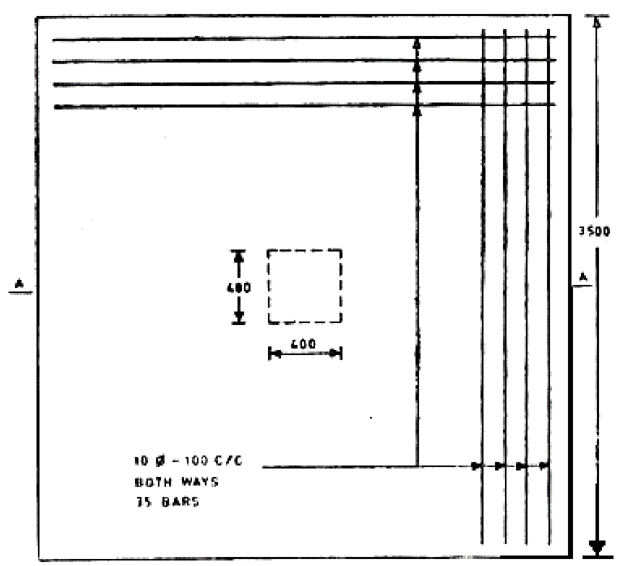
Note: We are not deducting the upward force underneath the area enclosed by the critical perimeter, because in this way we are in the conservative side.

Spacing

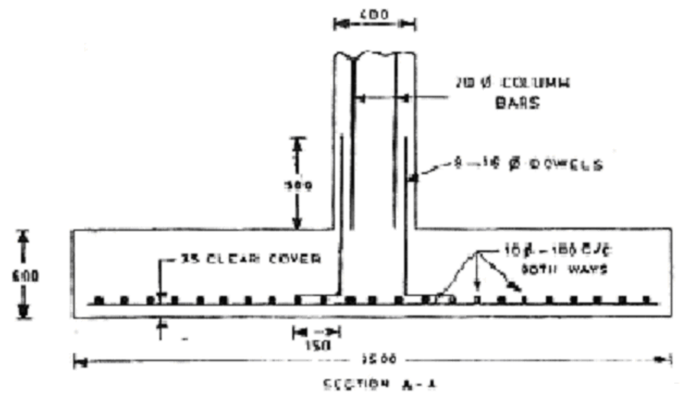
No. of 10 mm bar = $\dfrac{2646.4 \times 4}{\pi \times 10 \times 10}$ = 33.69 (34)

Spacing = $\dfrac{3500 - 50 \times 2 - 10}{34 - 1}$ = 102.73 mm

Spacing for 10 mm bar = 102.73 mm



PLAN OF REINFORCEMENT



Sectional elevation showing reinforcement

Comparison

Value Of	Reference Result	STAAD.*foundation* Result	Difference in Percent
Effective Depth	560 mm	560 mm	None
Governing Moment	514.826 KN-m	514.821 KN-m	Negligible
Area of Steal	2646.40 mm^2	2645.01 mm^2	0.05
Shear Stress (One-Way)	0.216 N/mm^2	0.216 N/mm^2	None
Shear Stress (Two-Way)	0.698 N/mm^2	0.700 N/mm^2	0.286

Table 6.1

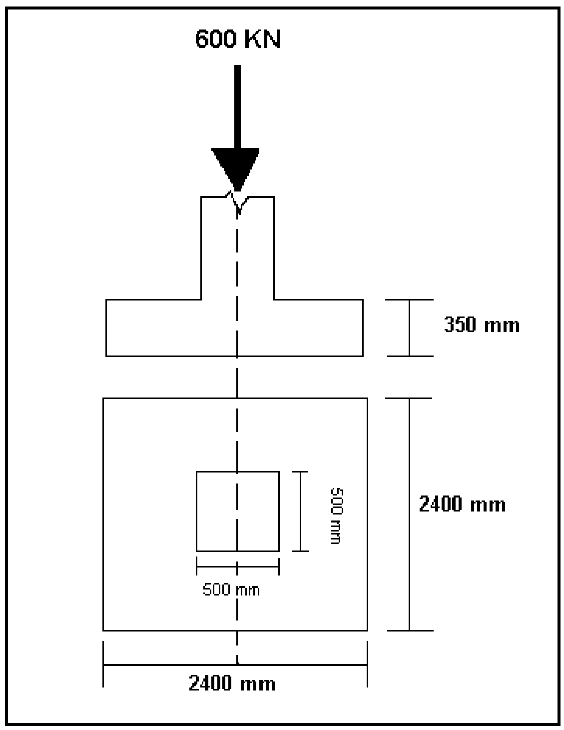
6.2 Indian Verification Problem 2

Reference

'Reinforced Concrete Structure' by Punmia-Jain-Jain, Example 25.1.

Problem

Design an isolated footing with the given data: Load Fy = 600 KN, fc = 15 MPa, fy = 250 MPa, Column Dimension = 500 mm x 500 mm, and Bearing Capacity of Soil = 120 KN/m^2.



Solution

Approximate area of footing required = $\frac{600}{120}$ m^2 = 5 m^2

Assuming 2.4 m x 2.4 m x 0.35 m footing dimension,
Weight of footing = 2.4 x 2.4 x 0.35 x 25 KN = 50.4 KN

Therefore, total load on the footing = (600+50.4) KN = 650.4 KN

Maximum pressure = $\frac{650.4}{2.4\times 2.4}$ KN/ m^2
= 112.92 KN/m^2 <120 KN/m^2 (Hence safe)

Ultimate pressure = $\frac{600\times 1.5}{2.4\times 2.4}$ KN/m^2 = 156.25 KN/m^2

Bending moment at critical section, M_u = 56.25 x 2.4 x $\frac{0.95\times 0.95}{2}$
= 169.21875 KN-m

Assuming 50 mm clear cover and 12 mm bar, effective depth
d_e = (350-50-0.5 x 12) mm = 294 mm

$K_{u,max} = \frac{700}{1100+0.87\,fy} = 0.53$

$R_{u,max}$ = 0.36 x fc x $K_{u,max}$ x (1- 0.42 $K_{u,max}$) = 2.225

Mu_{lim} = $R_{u,max}$ x B x d_e^2 = 461.568 x 10^6 N-mm
= 461.568 KN-m > M_u (Hence safe)

Area of Steel Required

Area of steel required along length,

Ast = 0.5 x $\frac{fc}{fy}\times\left[1-\sqrt{1-\frac{4.6MU}{fc\times B\times de\times de}}\right]$ x B x d_e

= 2837.87 mm^2

Minimum area of steel Ast_{min} = 0.0015 x B x D = 1260 mm^2

Check for One-Way Shear

Percentage of steel p_t = $\frac{100Ast}{B\times de}$ = 0.4022

Corresponding allowable τ_c = 0.42 N/mm^2

Developed shear stress τ_c = $\frac{Vu\max}{B\times de}$

V_{umax} = 156.25 x 2.4 x $\left(\frac{2.4-0.5}{2}-0.294\right)$ = 246 KN

Developed shear stress τ_c = $\frac{246\times1000}{2400\times294}$

= 0.3486N/mm2 < $\tau_{c,all}$ (Hence safe)

Check for Two-Way Shear

V_{umax} = 900 KN

Developed shear stress τ_c = $\frac{900\times1000}{4\times794\times294}$ = 0.96 N/mm^2

K_s = min (0.5+1, 1) = 1

Allowable shear stress = K_s x τ_c = 1 x 0.25 $\sqrt{fc}$
= 0.968 N/mm^2 (Hence safe)

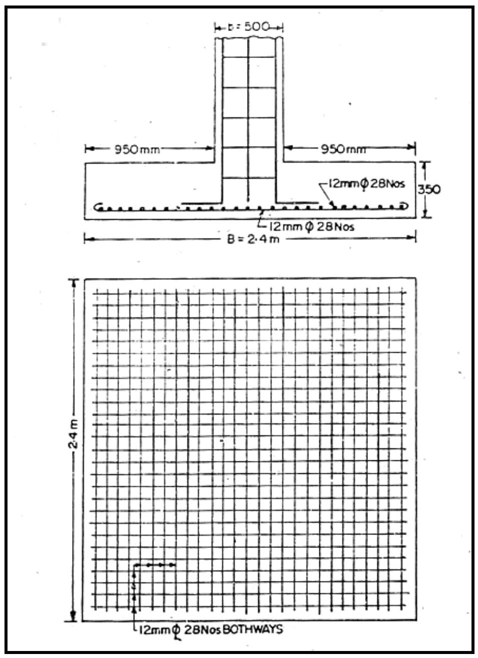
Note: We are not deducting the upward force underneath the area enclosed by the critical perimeter, because in this way we are in the conservative side.

Spacing

No. of 12 mm bar = $\dfrac{2837.87 \times 4}{\pi \times 12 \times 12}$ = 25.09 (26)

Spacing = $\dfrac{2400 - 50 \times 2 - 12}{26 - 1}$ = 91.52 mm

Spacing for 12 mm bar = 91.52 mm



Comparison

Value Of	Rereference Result	STAAD.*foundation* Result	Difference in Percent
Effective Depth	294 mm	294 mm	None
Governing Moment	169.2187 KN-m	169.2187 KN-m	None
Area of Steel	2837.87 mm^2	2836.34 mm^2	0.05
Shear Stress (One-Way)	0.3486 N/mm^2	0.3486 N/mm^2	None
Shear Stress (Two-Way)	0.96 N/mm^2	0.96 N/mm^2	None

Table 6.2

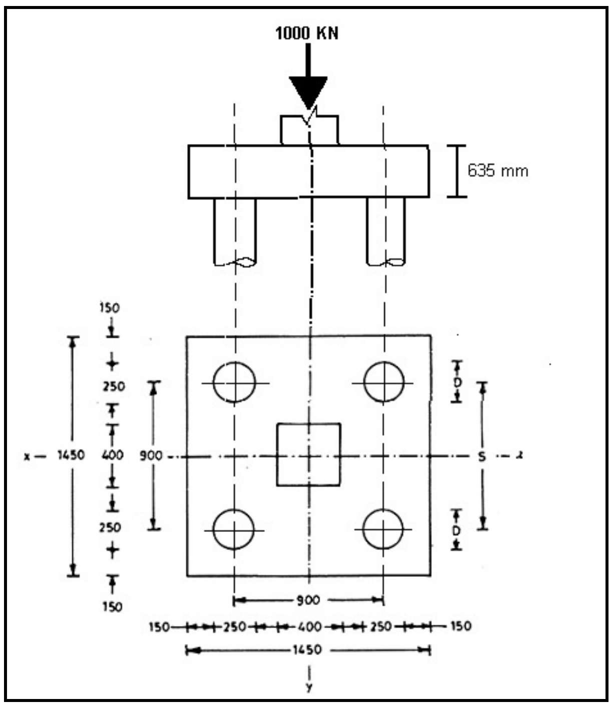
6.3 Indian Verification Problem 3

Reference

'Reinforced Concrete Design' by S.N.Sinha, Problem 11-12.

Problem

Design a pile cap with the given data: Load Fy = 1000 KN, Spacing = 900 mm, Pile in Pile Cap = 75 mm, Bottom Cover = 100 mm, Edge Distance = 275 mm, No. of Pile = 4, Dia. of Pile = 250 mm, fc = 15 MPa, fy = 415 MPa, Column Dimension = 400mm x 400mm, Load Factor = 1.5.



Solution

Ultimate load = 1.5 x 1000 KN = 1500 KN

Pile reaction = (Total load / no. of pile) = $\frac{1500}{4}$ = 375 KN

Bending moment at critical section (at column face),
M_u = 2 x 375 x 0.25 = 187.5 KN-m

Taking Effective depth d_e = 454 mm

$K_{u,max} = \frac{700}{1100 + 0.87 fy} = 0.479$

$R_{u,max}$ = 0.36 x fc x $K_{u,max}$ x (1- 0.42 $K_{u,max}$) = 2.066

Mu_{lim} = $R_{u,max}$ x B x d_e^2 = 617.462 KN-m > M_u (Hence safe)

Area of Steel Required

Area of steel required along length,

Ast = 0.5 x $\frac{fc}{fy} \times \left[1 - \sqrt{1 - \frac{4.6MU}{fc \times B \times de \times de}} \right]$ x B x d_e = 1205.524 mm^2

Minimum area of steel Ast_{min} = 0.0012 x B x D = 1104.90 mm^2

Check for One-Way Shear

Percentage of steel $p_t = \frac{100Ast}{B \times de}$ = 0.183 %

Corresponding allowable τ_c = 0.303 N/mm^2

Developed shear stress $\tau_c = \dfrac{V_{u\,max}}{B \times d_e}$

$V_{u,max}$ = 0.00 KN

Developed shear stress τ_c = 0.00 < $\tau_{c,all}$ (Hence safe)

Check for Two-Way Shear

$V_{u,max}$ = 1500 KN

Developed shear stress $\tau_c = \dfrac{1500 \times 1000}{4 \times 854 \times 454}$ N/mm^2

= 967.203 KN/m^2

K_s = min (0.5+1, 1)

Allowable shear stress = K_s x τ_c = 1 x 0.25 $\sqrt{fc}$

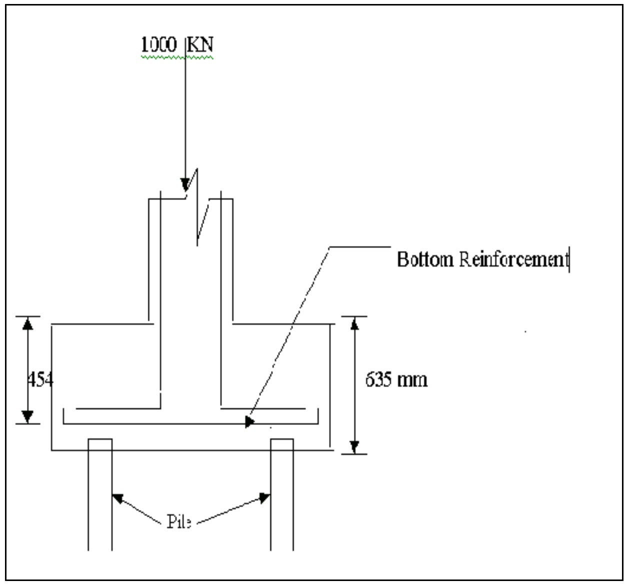
= 968.246 KN/m^2 > developed τ_c (hence safe)

Spacing

No. of 12 mm bar = $\dfrac{1205.524 \times 4}{\pi \times 12 \times 12}$ = 10.66 (11)

Spacing = $\dfrac{1450 - 100 \times 2 - 12}{11 - 1}$ = 123.8 mm

Spacing for 12 mm bar = 123.8 mm



Comparison

Value Of	Reference Result	STAAD.*foundation* Result	Difference in Percent
Effective Depth	454 mm	454 mm	None
Governing Moment	187.5 KN-m	187.5 KN-m	None
Area of Steal	1205.524 mm^2	1204.886 mm^2	0.058
Shear Stress (Two-Way)	0.967 N/mm^2	0.967 N/mm^2	None

Table 6.3

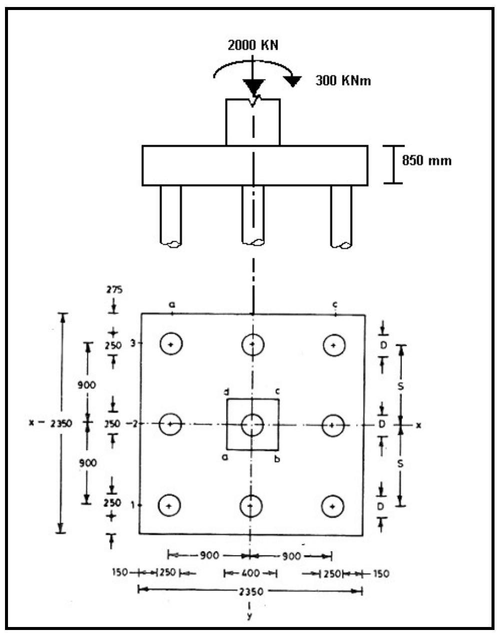
6.4 Indian Verification Problem 4

Reference

'Reinforced Concrete Design' by S.N.Sinha, Problem 11-13.

Problem

Design a pile cap with the given data: Load Fy = 2000KN, MZ = 300 KN-m, Spacing = 900 mm, Pile in Pile Cap = 75 mm, Bottom Cover = 100 mm, Edge Distance = 275 mm, No. of Pile = 9, fc = 15 MPa, fy = 415 MPa, Column Dimension = 500mm x 500mm, Ultimate Load Factor = 1.5.



Solution

Pile reaction:

$$P1 = \frac{1.5\times 2000}{9}+\frac{1.5\times 300\times 0.9}{6\times(0.9\times 0.9)} = 416.667 \text{ KN}$$

$$P2 = \frac{1.5\times 2000}{9}+\frac{1.5\times 300\times 0.9}{6\times(0.9\times 0.9)} = 416.667 \text{ KN}$$

$$P3 = \frac{1.5\times 2000}{9}+\frac{1.5\times 300\times 0.9}{6\times(0.9\times 0.9)} = 416.667 \text{ KN}$$

$$P4 = \frac{1.5\times 2000}{9} = 333.333 \text{ KN}$$

$$P5 = \frac{1.5\times 2000}{9} = 333.333 \text{ KN}$$

$$P6 = \frac{1.5\times 2000}{9} = 333.333 \text{ KN}$$

$$P7 = \frac{1.5\times 2000}{9}-\frac{1.5\times 300\times 0.9}{6\times(0.9\times 0.9)} = 250 \text{ KN}$$

$$P8 = \frac{1.5\times 2000}{9}-\frac{1.5\times 300\times 0.9}{6\times(0.9\times 0.9)} = 250 \text{ KN}$$

$$P9 = \frac{1.5\times 2000}{9}-\frac{1.5\times 300\times 0.9}{6\times(0.9\times 0.9)} = 250 \text{ KN}$$

Bending moment at critical section (at column face),
M_u (along length) = 3 x 416.667 x 0.65 = 812.5 KN-m
M_u (along width) = (416.667+333.333+250) x 0.65 KN-m
= 650 KN-m

Assuming 850 mm overall depth and 12 mm bar,
Effective depth d_e = 850-(100+75+6) = 669 mm

$K_{umax} = \dfrac{700}{1100 + 0.87 fy} = 0.479$

R_{umax} = 0.36 x fc x K_{umax} x (1- 0.42 K_{umax}) = 2.066

Mu_{lim}= R_{umax} x B x d_e^2 = 2172.953 KN-m > M_u (Hence safe)

Area of Steel Required

Area of steel required along length,

Ast = 0.5 x $\dfrac{fc}{fy} \times \left[1 - \sqrt{1 - \dfrac{4.6MU}{fc \times B \times de \times de}}\right]$ x B x d_e = 3592.61 mm^2

Minimum area of steel Ast_{min} = 0.0012 x B x D = 2397 mm^2

Area of steel required along width,

= 0.5 x $\dfrac{fc}{fy} \times \left[1 - \sqrt{1 - \dfrac{4.6MU}{fc \times B \times de \times de}}\right]$ x B x d_e = 2833.69 mm^2

Minimum area of steel Ast_{min} = 0.0012 x B x D = 2397 mm^2

Check for One-Way Shear (Along Length)

Percentage of steel $p_t = \dfrac{100Ast}{B \times de}$ = 0.2285 %

Corresponding allowable τ_c = 0.338 N/mm^2

Developed shear stress $\tau_c = \dfrac{Vu \max}{B \times de}$

$V_{umax} = \dfrac{3 \times 416.667 \times 106}{250}$ KN = 530 KN

Developed shear stress $\tau_c = \dfrac{530 \times 1000}{2350 \times 669}$

= 0.337 N/mm^2 < $\tau_{c,all}$ (Hence safe)

Check for Two-Way Shear

V_{umax} = (3000-333.33) KN = 2666.67 KN

Developed shear stress $\tau_c = \dfrac{2666.67 \times 1000}{4 \times 1169 \times 669}$ = 0.852 N/mm^2

K_s = min(0.5+1 , 1) = 1

Allowable shear stress = K_s x τ_c = 1 x 0.25 $\sqrt{fc}$

= 0.968 N/mm^2 > developed τ_c (hence safe)

Spacing

Along length:

No. of 12 mm bar = $\dfrac{3592.61 \times 4}{\pi \times 12 \times 12}$ = 31.76 (32)

Spacing = $\dfrac{2350 - 100 \times 2 - 12}{32 - 1}$ = 68.967 mm

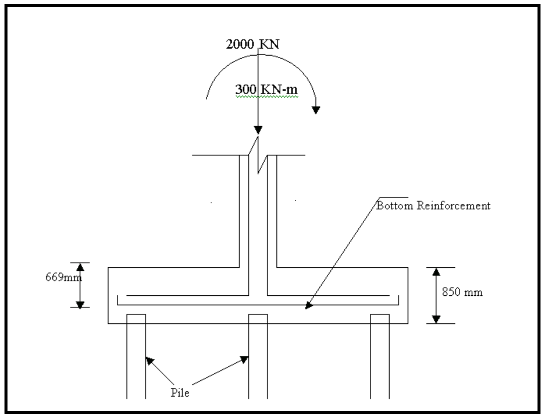
Spacing along length for 12 mm bar = 68.967 mm

Along width:

No. of 12 mm bar = $\dfrac{2833.69 \times 4}{\pi \times 12 \times 12}$ = 25.06 (26)

$$\text{Spacing} = \frac{2350 - 100 \times 2 - 12}{26 - 1} = 85.52 \text{ mm}$$

Spacing along width for 12 mm bar = 85.52 mm



Comparison

Value Of	Reference Result	STAAD.*foundation* Result	Difference in Percent
Effective Depth	669 mm	669 mm	None
Governing Moment (length)	812.5 KN-m	812.503 KN-m	Negligible
One-Way Shear Stress (Length)	0.337 N/mm^2	0.337 N/mm^2	None
Two-Way Shear Stress	0.852 N/mm^2	0.852 N/mm^2	None

Table 6.4

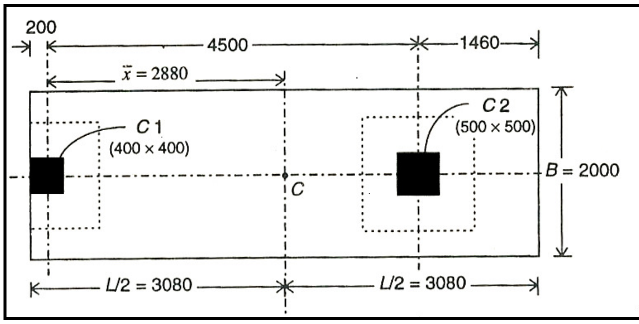
6.5 Indian Verification Problem 5

Reference

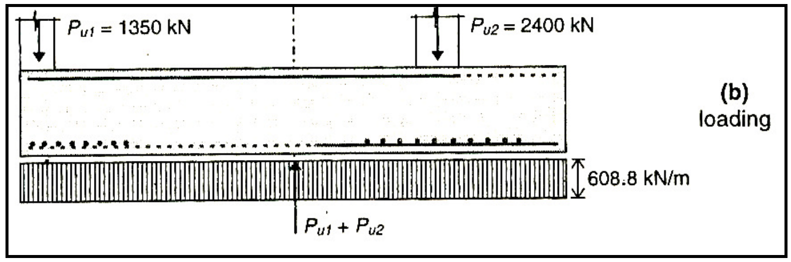
'Reinforced Concrete Design' by Pillai & Menon, Page 652, Example 14.7.

Problem

Design a combined footing for two columns with the given data: C1 (400 mm x 400 mm) with 4-25 Ø bars and C2 (500 mm x 500mm) with 4-28 Ø bars supporting axial loads P1 = 900 KN and P2 = 1600 KN respectively (under service dead and live loads). The column C1 is an exterior column whose exterior face is flush with the property line. The center-to-centre distance between C1 and C2 is 4.5 meters. The allowable soil pressure at the base of the footing, 1.5 m below ground level, is 240 KN/m^2. Assume a steel of grade Fe 415 in the columns as well as the footing, and a concrete grade of M 20 in the footing.



(a) footing plan



Solution

Dimension of Mat (Based on the bearing Capacity given):
Length = 6.16 m
Width = 2 m
Depth = 0.95 m

Calculation for base-pressure

Self-weight of mat = 6.16 x 2 x 0.95 x 25 KN = 292.6 KN

Total load on the mat = (1600+900+200.2) KN = 2792.6 KN

Base pressure = $\dfrac{2792.6}{6.16 \times 2}$ KN/m^2
= 226.67 KN/m^2 < 240 KN/m^2 (Hence Safe)

Ultimate load for $C_1 = P_{u1}$ = 1.5 x 900 = 1350 KN

Ultimate load for $C_2 = P_{u2}$ = 1.5 x 1600 = 2400 KN

Then uniformly distributed upward load = $(P_{u1}+P_{u2})/6.16$ KN/m
= 608.8 KN/m

Calculation for maximum bending moment

Positive bending moment:

The maximum positive bending moment is at the face of the support C_2 = $M_u(+)$ = 608.8 x $(1.46-0.25)^2/2$ = 446 KN-m

Negative bending moment:

The maximum negative bending moment occurs at the location of zero shear. So we first find the location of zero shear:

Location of zero shear from the left side x = (1350/608.8)
= 2.2175 m.

Therefore, $M_u(-)$ = 608.8 x $(2.2175)^2/2$ – 1350 x (2.2175 - 0.2)
= -1227 KN-m.

Calculation for punching shear

Assuming 75 mm clear cover and 20 Ø bars,
Effective depth (d_e) = (950 – 75 –20/2) = 865 mm

Upward pressure = 608.8/2 KN/m^2 = 304.4 KN/m^2

Allowable shear stress = K_s x τ_c

K_s = min (1+β, 1)

Where β = 1, K_s = 1 and $\tau_c = 0.25\sqrt{f_c}$ = 1.18 N/mm^2

Therefore allowable shear stress = 1 x .118 N/mm^2 = 1.118 N/mm^2

Maximum shear for C_1 = 1350 KN

Developed shear stress,

$$\tau_v = \frac{1350 \times 1000}{\{(2 \times 400 + 865) + (400 + 865)\} \times 865}$$

= 0.533 N/mm^2 < $\tau_{c,allowable}$ (Hence safe)

Maximum shear for C_2 = 2400 KN

Developed shear stress,

$$\tau_v = \frac{2400 \times 1000}{4 \times (500 + 865) \times 865}$$

= 0.508 N/mm^2 < $\tau_{c,allowable}$ (Hence safe)

Note: We are not deducting the upward force underneath the area enclosed by the critical perimeter, because in this way we are in the conservative side.

Calculation of reinforcement

Maximum negative moment M_u(-) = 1227 KN-m
Maximum negative moment/width = 1227/2 KN-m/m
= 613.5 KN-m/m

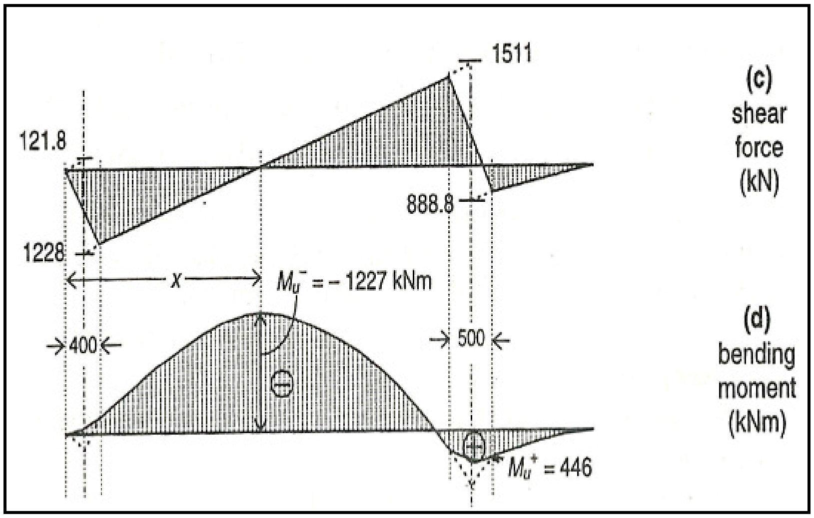
Area of steel required on top face along length,

$$A_{st} = 0.5 \text{ x } \frac{fc}{fy} \times \left[1 - \sqrt{1 - \frac{4.6MU}{fc \times B \times de \times de}}\right] \text{ x B x } d_e$$

B = 1000 mm
d_e = 865 mm
M_u= 613.5 x 106 N-mm

A_{st} = 2067.97 mm^2/m

$A_{st,min}$ = 0.0012 x B x D = 1140mm^2/m



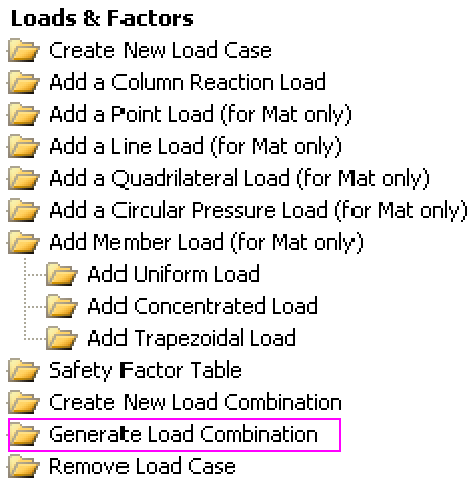
Comparison

Value Of	Reference Result	STAAD.*foundation* Result	Difference in Percent
Max Bending Moment(-)	603.201 KN-m/m	613.5 KN-m/m	1.68
Max Bending Moment(+)	219.687 KN-m/m	223 KN-m/m	1.48
Area of Steal Required	2014.835 mm^2/m	2067.97 mm^2/m	2.56
Base Pressure	227 KN/m^2	226.67 KN/m^2	Negligible

Table 6.5

Load Combination Generation

Clicking on "Generate Load Combination" leaf under "Loads and Factors" group will bring a load combination dialog, allowing you to generate the load combination automatically.



Two types of load combinations are used here. They are *"Allowable Load Combination"* and *"Ultimate Load Combination"*.

Allowable Load Combination:

Allowable load combinations are load combinations used to check soil pressure and optimize footing plan dimensions.

Ultimate Load Combination:

Ultimate load combinations are load combinations used to check for shear and design for reinforcement.

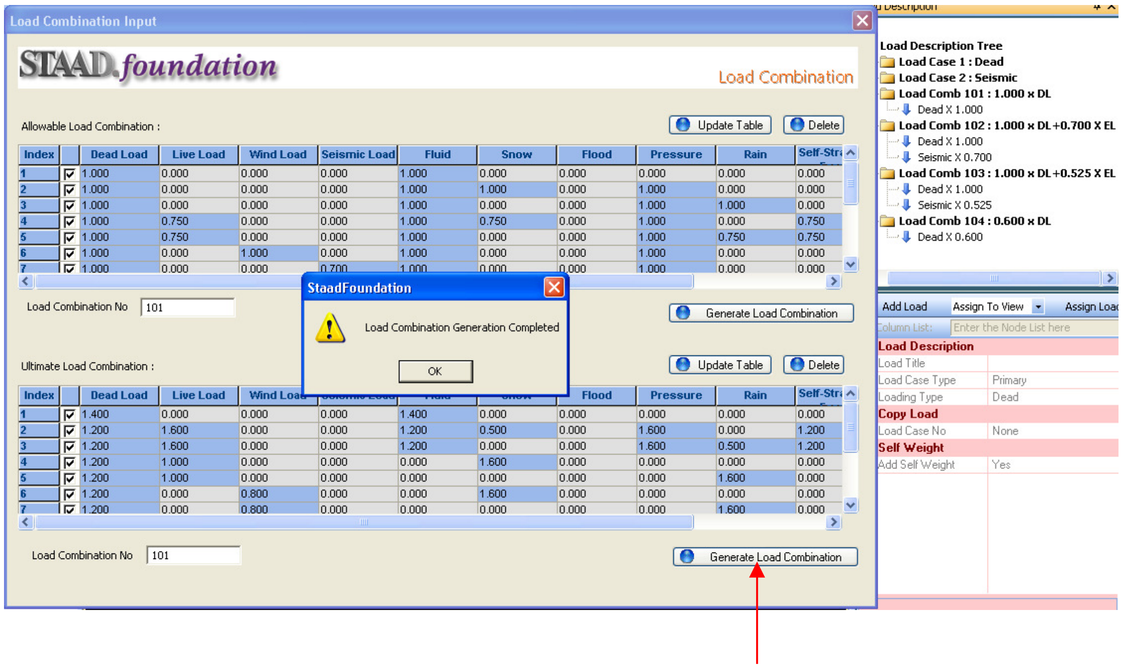
The two tables represent these two load combinations i.e. Allowable Load Combination and Ultimate Load Combination. The cells represent the factors to be added with the primary load cases, depending upon the rules of the US Standard.

After adding load cases, clicking on the Generate Load Combination Button for the specific table, the loads will be generated with the factors, taken from the table, and will be added in the Load Tree. The child node of each load combination node will represent the Load Case and the factors multiplied with it.

The load combination number starts from 101 and you can also give load combination number of your own choice. If the number exists, the load combination number is automatically incremented with each new load combination as “Load Comb” and the number.

You can create any number of load combination and can save it using an INI file. This file exits in the program installation directory as “ACILOAD.INI”. This saved load combination will be application specific i.e. they are independent of file saving.

Following figure shows the load combination page after clicking generate load combination button.



Generate Load Combination Button

The first column indicates the index of the load combination. The second column has a check. Check on the check boxes of the combination which you want to use. The cell with zero values appears in gray color where as with values other than zero it appears in sky color. On clicking the Generate Load Combination Button as shown in the above figure, the load combination for the added load cases will be generated.

Update Table

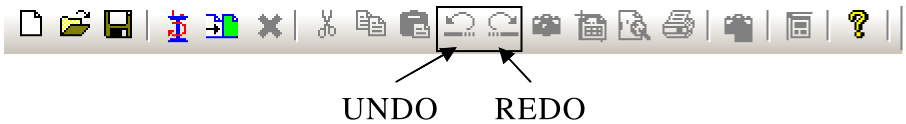
Initially the page shows all the load combination saved in the INI file. You can add new load combination simply by adding factors in the table. Check on the check box in the second row to use the load combination. If you save a file with those load combination then the load combination will be only saved to that file but not in INI file. To save the load combination in the INI file you need to click on *“Update Table”*. You can also manually change the INI file.

Delete

To delete a load combination, select a row and then click on *“Delete”* to delete any particular load combination from the list. But to delete any combination from the INI file you need to click on *“Update Table”* after clicking on *“Delete”*.

Undo-Redo

Undo stands for cancel out the recent operation and *Redo* stands for insert which is just deleted / canceled. Only by clicking anyone can rectify the work.



The above figure is to give a clear idea about the position of "*Undo-Redo"* in toolbar. The rectangular area indicates "Undo-Redo". One can do *Undo* by pressing (Ctrl + z) and *Redo* by pressing (Ctrl +y).

Area of Implementation

There are some specific operations on which we have presently implemented the feature mentioned below. They are:

1. Addition of Footing :

Whenever we add any wrong footing or unnecessary footing we can remove it by clicking the left one i.e. *Undo* in rectangle.

2. Deletion of Footing :

Suppose a footing has deleted but it is required to be there in its' original position. In this situation one can implement the feature *Redo* i.e. the right one in the mentioned recta

3. Addition of Load Case :

Just added new load cases can be removed by clicking on *Undo* if it becomes unnecessary.

4. Deletion of Load Case :

If any deleted load case become necessary to be inserted which is just deleted, one can solve that problem by click on *Redo*.

5. Addition of Load Item :

Just added load items can be easily deleted by clicking on *Undo* if it is not needed.

6 .Deletion of Load Item :

Deleted load item can be needed sometimes, in that moment we can resolve the problem on clicking on *Redo*.

7. Create job :

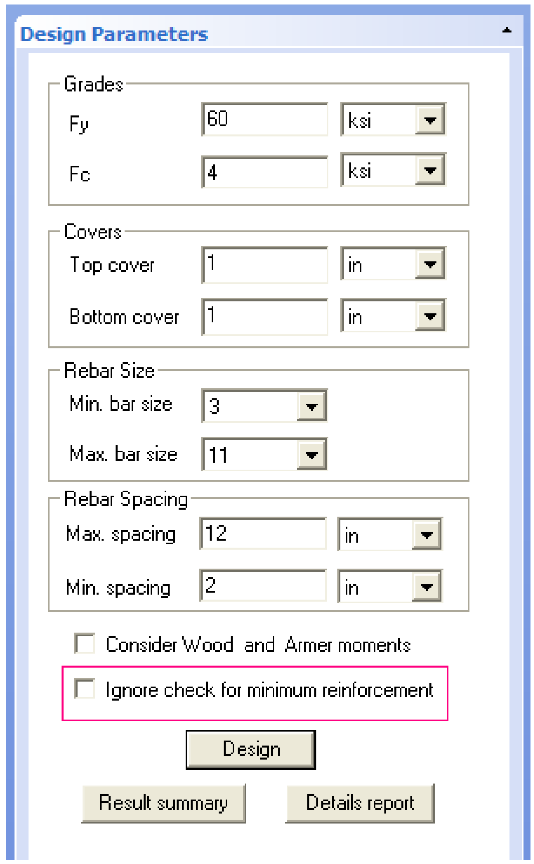
Using this option we can create a new job and the new job can be deleted by clicking on *Undo*.

8. Delete job :

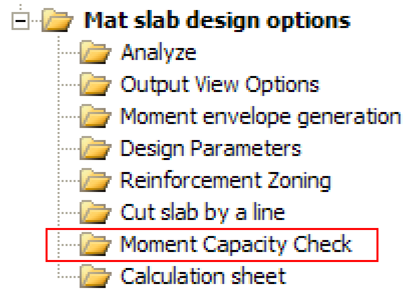
We have implemented the *Redo* feature over here to regain the deleted job that is required to be there.

Mat Slab Capacity Check:

Design parameters form for mat slab design has a new input now which will instruct program to design slab without considering check for minimum reinforcement.

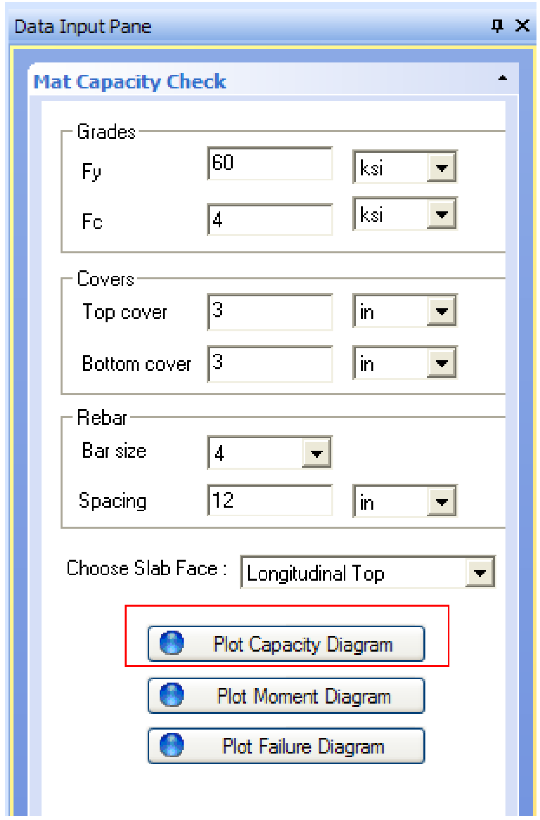


Switching on/off minimum reinforcement check

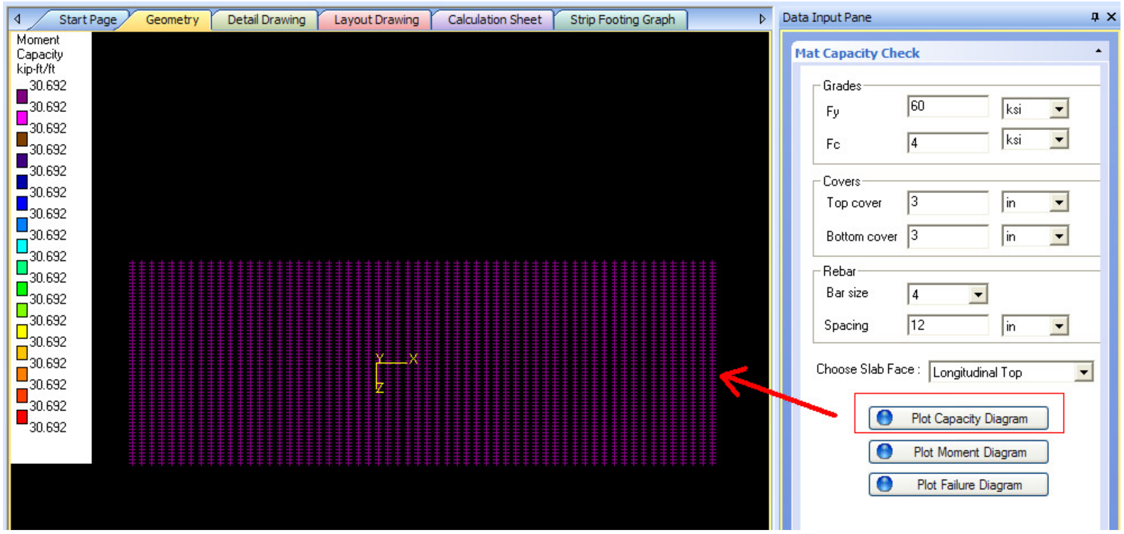


“Moment Capacity Check”

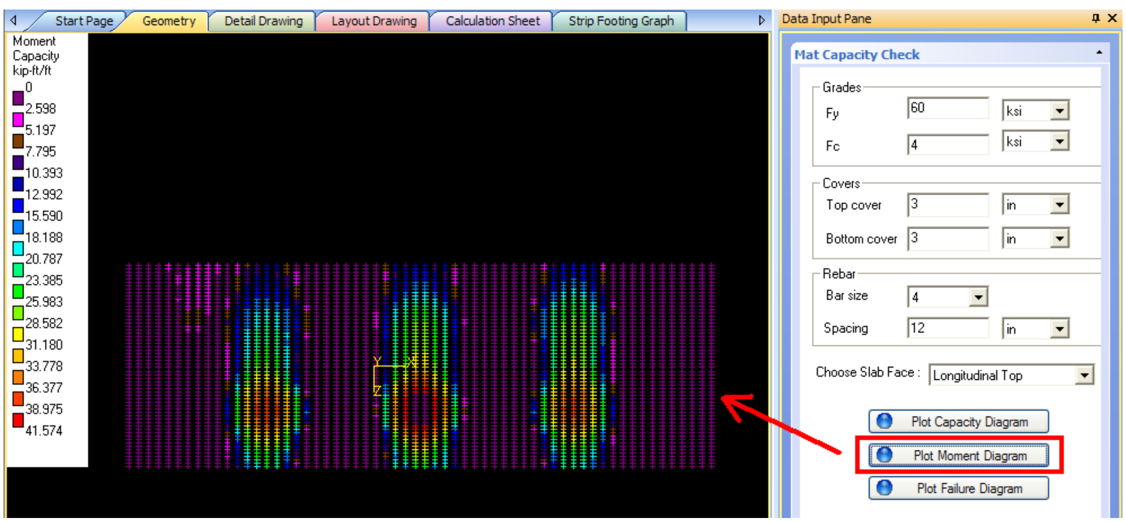
A new tree leaf added in Mat slab design options category to check the capacity of existing mat slab. Here program allows the user to define reinforcement layout and program calculates moment capacity of the slab based on slab thickness, covers, reinforcement layout etc. User can plot capacity diagram, actual moment diagram and then compare those two diagrams and plot failure (or unity check) diagram. If at any portion of the slab, actual moment is more than the moment capacity, program will identify that portion with red color and plot failure diagram as shown below. Check has to be performed for one slab face at a time. So, for all four faces the check should be performed for 4 times.



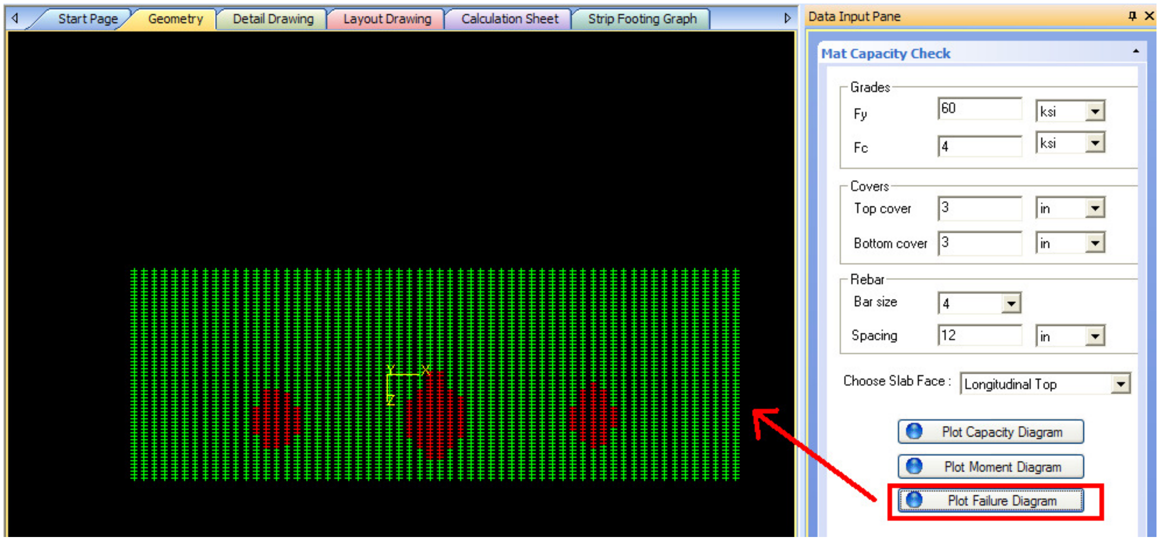
Mat capacity check form



Moment Capacity diagram plot for Longitudinal Top



Actual moment diagram plot for Longitudinal Top



Unity check diagram where green indicates pass and red indicates failure