ANALYSIS AND DESIGN USING PLAXIS

BY

JITENDER THAKUR - 081614

ANKIT CHOPRA - 081627

RISHAV SHARMA - 081629

AKSHAY CHAUDHARY - 081631

GUIDED BY

Prof. ASHOK K. GUPTA

DEPARTMENT OF CIVIL ENGINEERING

JAYPEE UNIVERSITY OF INFORMATION TECHNOLOGY

WAKNAGHAT, DISTT. SOLAN

(HIMACHAL PRADESH)

CERTIFICATE

This is to certify that the project report entitled "**Analysis and Design using Plaxis.**", submitted by **Jitender Thakur , Ankit Chopra , Rishav Sharma , Akshay Chaudhary** in partial fulfilment for the award of degree of Bachelor of Technology in Civil Engineering to **Jaypee University of Information Technology**, Waknaghat, Solan (H.P) has been carried out under my supervision.

Date: 28/5/12

(Dr. Ashok K. Gupta)
Professor and Head

Certified that this work has not been submitted partially or fully to any other University or Institute for the award of this or any other degree or diploma

Thakur (Jitender Thakur)
Ankit Chopra (Ankit Chopra)
Rishav (Rishav Sharma)
Akshay (Akshay Chaudhary)

1. **Jitender Thakur (081614)**
2. **Ankit Chopra (081627)**
3. **Rishav Sharma (081629)**
4. **Akshay Chaudhary (081631)**

ACKNOWLEDGEMENT

It has been a wonderful and intellectually stimulating experience working on **Analysis And Design Using Plaxis** which is in itself a new and innovative idea in the field of Soil Mechanics .

We gratefully acknowledge the Management and Administration of Jaypee University of Information Technology for providing us the opportunity and hence the environment To initiate and complete our project.

For providing with the finest suggestions for the project, we are greatly thankful to our Project guide **Prof. Ashok K. Gupta**. He provided us the way to get the job done, not providing the exact way to do it, but the concept behind the complexities so that we can make better use of existing knowledge and build up higher skills to meet the industry needs. His methodology of making the system strong from inside has taught us that output is not end of project .

Date: 28/5/12

Jthakur (Jitender Thakur)
Ankit chopra (Ankit Chopra)
Rishav (Rishav Sharma)
Akshay (Akshay Chaudhary)

Jitender Thakur
Ankit Chopra
Rishav Sharma
Akshay Chaudhary

CONTENTS

CHAPTER: 3

UNIT: 2

CHAPTER: 4

LIST OF FIGURES

LIST OF TABLES

OBJECTIVES

- **Analysis of Soil Profile Using Plaxis, Performing tests Using Plaxis** – The Soil test option is a quick and convenient procedure to simulate basic soil test on the basis of a single point algorithm, i.e. without the need to create a complete finite element model. This option can be used to compare the behaviour as defined by the soil model and the parameters of a soil data set with the results of laboratory test data obtained from a site investigation.
- **Comparing Soil Profile as obtained by SPT with Soil Test performed on Plaxis.**
- **Analysis of Foundations Using Soil Profile** - Foundations of engineering structures transfer and distribute their loading to the underlying soil and rock. Foundations are mostly supported by multi-layered soil profiles. It is widely known that the spatial variability in soil properties has significant effects on foundation performance (i.e. bearing capacity and settlement).Due to the variety of soil types and multi-layered soil profiles that exist in nature, establishing a probabilistic analysis experimentally without the aid of modern high-speed computers would be tedious, if not impossible.

ABSTRACT

PLAXIS 2D is a two-dimensional finite element program, developed for the analysis of deformation, stability and groundwater flow in geotechnical engineering. The development of PLAXIS began in 1987 at Delft University of Technology as an initiative of the Dutch Ministry of Public Works and Water Management. Because of continuously growing activities, the PLAXIS company (Plaxis Bv) was formed in 1993.In 1998, the first PLAXIS 2D for Windows was released. The PLAXIS 3D program was released in 2010.

The program uses a convenient graphical user interface that enables users to quickly generate a geometry model and finite element mesh based on a representative vertical cross-section of the situation at hand. Users need to be familiar with the Windows environment.

Our Project Titled "ANALYSIS AND DESIGN USING PLAXIS" mainly focussed on learning of Plaxis tool by performing various soil tests, and analysis of soil profile. This Project helps us to analyse soil behaviour for various buildings and foundations.

INTRODUCTION

The PLAXIS 2D program is a special purpose two-dimensional finite element program Used to perform deformation and stability analysis for various types of geotechnical Applications. Real situations may be modelled either by a plane strain or an axisymmetric model.

The generation of a two-dimensional (2D) finite element model in the PLAXIS 2D Program is based on the creation of a geometry model. This geometry model is created in the x -y -plane of the global coordinate system, whereas the z -direction is the out-of-plane direction. In the global coordinate system the positive z -direction is pointing towards the user.

Although PLAXIS 2D is a 2D program, stresses are based on the 3D Cartesian.

GENRAL INFORMATION

UNITS AND SIGN CONVENTIONS

It is important in any analysis to adopt a consistent system of units. At the start of the input of geometry, a suitable set of basic units should be selected. The basic units comprise a unit for length, force and time. These basic units are defined in the Project properties window of the Input program. The default units are meters [m] for length, kilo Newton [kN] for force and day [day] for time. Table Below gives an overview of all available units, the [default] settings and conversion factors to the default units.

Length	**Conversion**	**Force**	**Conversion**
Mm	= 0.001 m	N	= 0.001 kN
[m]	= 1 m	[kN]	= 1 kN
in (inch)	= 0.0254 m	MN	= 1000 kN
ft (feet)	= 0.3048 m	lbf (pounds force)	= 0.0044482 kN
		kip (kilo pound)	= 4.4482 kN

Time	**Conversion**
s (sec)	= 1/86400 day
min	= 1/1440 day
[h]	= 1/24 day
[day]	= 1 day

For convenience, the units of commonly used quantities in two different sets of units are Listed below:

		Int. system (SI)	Imperial system
Basic units:	Length	[m]	[in]
	Force	[kN]	[lbf]
	Time	[day]	[sec]
Geometry:	Coordinates	[m]	[in]
	Displacements	[m]	[in]
Material properties:	Young's modulus	[kN/m^2]=[kPa]	[psi]=[lbf/in^2]
	Cohesion	[kN/m^2]	[psi]
	Friction angle	[deg.]	[deg.]
	Dilatancy angle	[deg.]	[deg.]
	Unit weight	[kN/m^3]	[lbf/cu in]
	Permeability	[m/day]	[in/sec]
Forces & stresses:	Point loads	[kN]	[lbf]
	Line loads	[kN/m]	[lbf/in]
	Distributed loads	[kN/m^2]	[psi]
	Stresses	[kN/m^2]	[psi]

Sign convention

The generation of a two-dimensional (2D) finite element model in the PLAXIS 2D program is based on the creation of a geometry model. This geometry model is created in the x -y -plane of the global coordinate system (Figure below), whereas the z -direction is the out-of-plane direction. In the global coordinate system the positive z -direction is pointing towards the user. In all of the output data, compressive stresses and forces, including pore pressures, are taken to be negative, whereas tensile stresses and forces are taken to be positive. Figure below shows the positive stress directions.

Although PLAXIS 2D is a 2D program, stresses are based on the 3D Cartesian Coordinate system shown in Figure below. In a plane strain analysis σ_{zz} is the out-of-plane stress. In an axisymmetric analysis, x represents the radial coordinate, y represents the axial coordinate and z represents the tangential direction. In this case, σ_{xx} represents the radial stress and σ_{zz} represents the hoop stress.

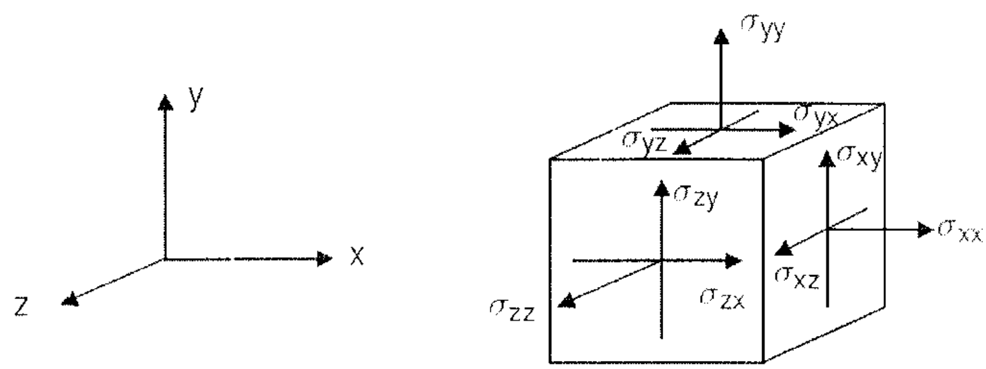


Figure: Coordinate system and indication of positive stress components

CHAPTER: 1

1. SOIL TEST

The Soil test option is a quick and convenient procedure to simulate basic soil test on the basis of a single point algorithm, i.e. without the need to create a complete finite element model. This option can be used to compare the behaviour as defined by the soil model and the parameters of a soil data set with the results of laboratory test data obtained from a site investigation.

The Soil test option is available from the Material sets window if a soil data set is selected (see Figure 1.1). Alternatively, the Soil test option can be reached from the Soil dialog.

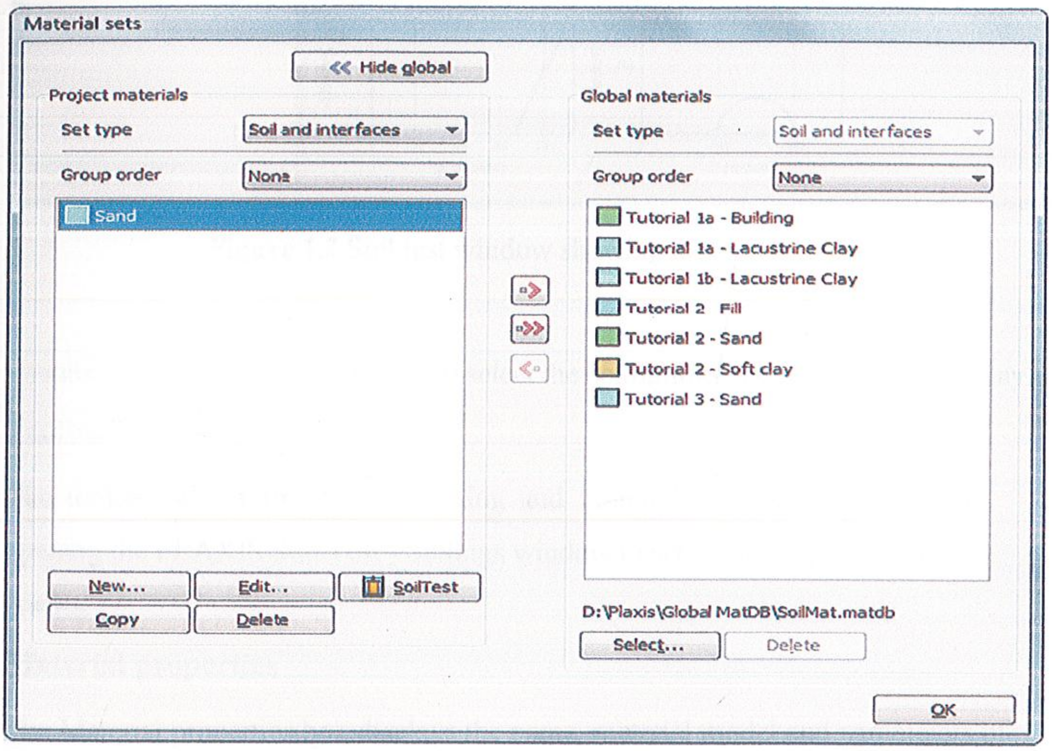


Figure 1.1. Material sets window showing the project and the global database

Once the Soil test option has been selected, a separate window will open (Figure 1.2). This window contains a menu, a toolbar and several smaller sections. The various items are described in more detail below.

Main menu

The menus available in the menu bar are:

File To open, save and close a soil test data file (*.vlt).

Test To select the test that will be simulated. The options available

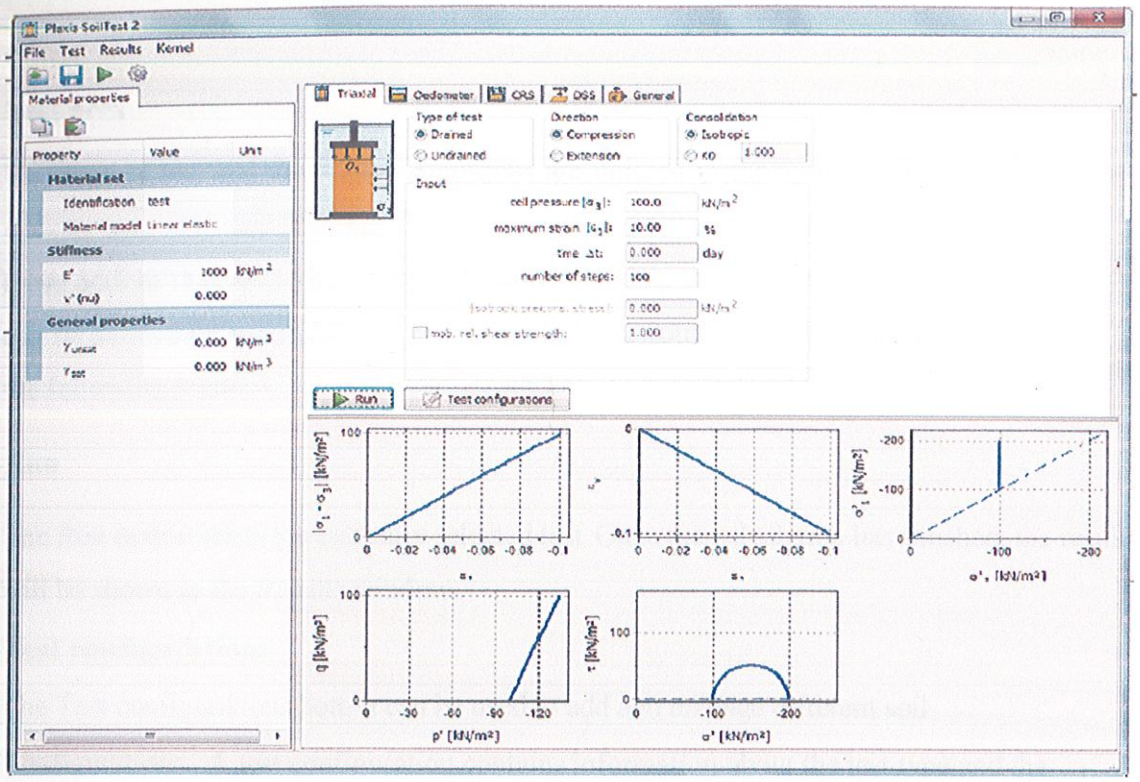


Figure 1.2 Soil test window showing drained triaxial test input

Results	To select the configuration of diagrams to display.

Toolbar

The toolbar allows for loading, saving and running of soil test results and opening the PLAXIS Soil Test - Settings window to set the configuration of the results.

Material properties

The Material properties box displays the name, material model and parameters of the currently selected data set. The parameters can be edited to optimise the mesh to real soil lab test data. Transferring of material parameters to and from the material database is possible. To copy the modified parameters to the material database:

Click the Copy material button in the Material properties box.

• In the program open the Material sets window and either select the corresponding Material set or clicks New.

In the Soil window click Paste material button. The parameters will be copied in the material database. In the same way it is also possible to copy material from material database to soil test.

Test area

The type of test and the testing conditions are defined in the test area. The test options available are Triaxial, Oedometer, CRS, DSS and General. As one of these options is selected by clicking the corresponding tab, the testing conditions can be defined inthe tab sheet. A more detailed description of the tests is given in the following sections.

Run

The Run button starts the currently selected test .Once the calculation has finished, the results will be shown in the Results window.

Test configurations

The Test configurations button can be used to add and manage different soil Configurations . A test configuration contains information about the test type and the Values of test input parameters. To save a test configuration select the Save option in the menu displayed as the Test configuration button is clicked. The Manage option can be used to manage the test configurations available. When the Manage option is selected, the Manage configuration window pops up. Note that the name of the window indicates the test to which the configuration belongs (Figure 1.3).

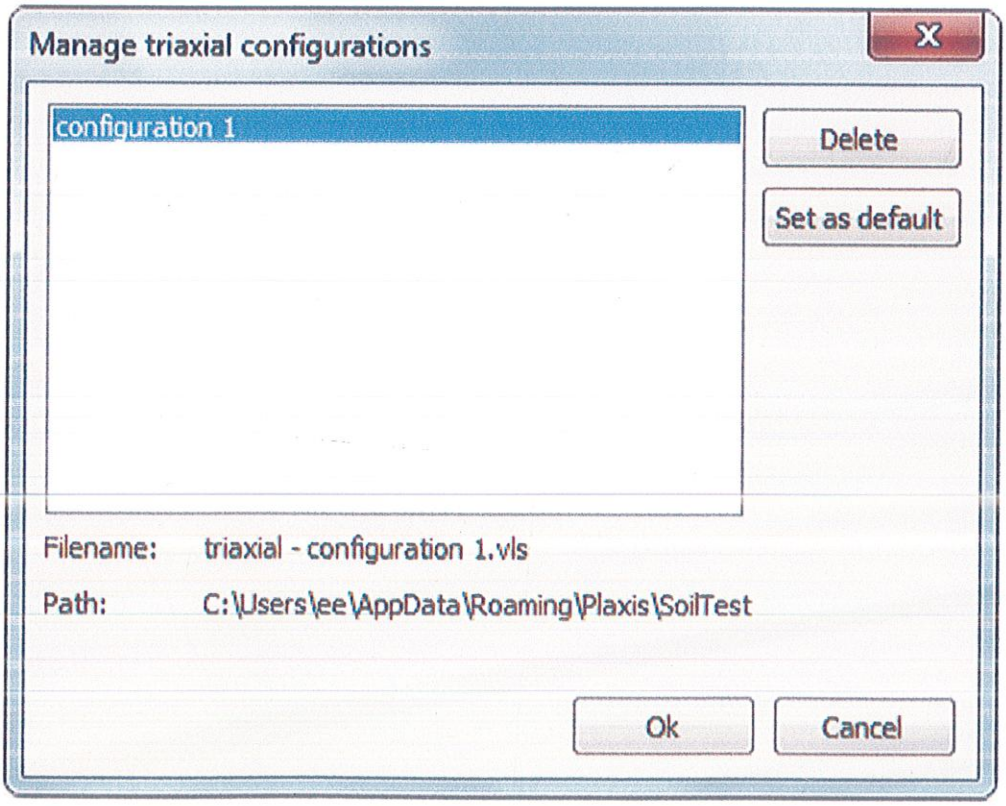


Figure 1.3 Manage configurations window for triaxial tests

The name and the location of the configuration file is indicated in the Filename and Path respectively in the Manage configurations window.

Set as default

The Set as default button saves the current input parameters as the default parameters. These will be initialised as such the next time the Soil test window is opened.

Loaded tests

When previously saved tests of the current type have been opened from the File menu, the loaded tests window lists all these tests within each tabsheet. The results of all loaded tests are shown together with the results of the current test. The Delete button can be used to remove the selected test from the list of loaded tests. It does not remove the soil test file (*.vlt) from disk.

Results

The results of the test are displayed in the predefined

1.1 TRIAXIAL TEST

The Triaxial tab sheet contains facilities to define different types of triaxial tests. Before specifying the test conditions, a selection can be made between different triaxial tests options.

Triaxial test - Options

Drained / undrained triaxial test

In the latter case, undrained soil conditions and zero drainage are assumed (similar as when the Drainage type has been set to Undrained (A) or Undrained (B)), irrespective of the drainage type setting in the material data set.

Triaxial compression / triaxial extension test

In the former case the axial load is increased; in the latter case the axial load is decreased.

Isotropically consolidated / K_0 -consolidated test

In the latter case the K_0 -value (ratio of lateral stress over axial stress) can be specified to set the initial stress state.

Triaxial test - Conditions

The following test conditions can be defined:

Cell pressure $|\sigma_3|$

the absolute value of the isotropic cell pressure at which the Sample is consolidated, entered in units of stress. This sets the initial stress state. In the case of a K_0 -consolidated test, this value represents the initial lateral stress, σ_3 ; the initial vertical stress, σ_1 , is defined as σ_3 /K_0 .

Maximum strain $|\varepsilon_1|$

the absolute value of the axial strain that will be reached in the last calculation step.

Time Δt Time increment (only relevant for time-dependent model Consolidation is not considered).

Number of steps the number of steps that will be used in the calculation.

Isotropic pre-consolidation stress

The isotropic pre-consolidation pressure to which the soil has Been subjected. If the soil is normally consolidated this value should be set equal to the Cell pressure or kept zero. This option is only available for the advanced soil models.

Mobilized relative shear strength

This option is only available for the Hardening Soil model and HS small model to set the initial shear hardening contour. This value Must be between 0 (= isotropic stress state) and 1 (= failure state).

1.2 OEDOMETER

The Oedometer tabsheet contains facilities to define a one-dimensional compression (oedometer) test. The following settings can be defined:

Isotropic pre-consolidation stress

The isotropic pre-consolidation pressure to which the soil has been subjected. If the soil is normally consolidated this value should be set equal to the initial stress state, i.e. zero. This option is only available for the advanced soil models.

Mobilized relative shear strength

This option is only available for the Hardening Soil model and HS small model to set the initial shear hardening contour. This value must be between 0 (= isotropic stress state) and 1 (=failure state).

Phases

Lists the different phases of the oedometer test. Each phase is defined by a Duration (in units of time), a vertical Stress increment (in units of stress) and a Number of steps. The initial state is always assumed to be stress free. The given stress increment will be reached at the end of the given duration in the given number of steps. The input values can be changed by clicking in the table. A negative stress increment implies additional compression, whereas a positive stress increment implies unloading or tension. If a period of constant load is desired, enter the desired duration with a zero stress increment.

Add — Adds a new phase to the end of the Phases list.

Insert — Inserts a new phase before the currently selected phase.

Remove — Removes the currently selected phase from the Phases list.

1.3 CONSTANT RATE OF STRAIN (CRS)-TEST

The CRS tabsheet contains facilities to define a constant rate-of-strain compression test. The following settings can be defined:

Type of test — The type of the test, whether Drained or Undrained can be Specified.

Isotropic pre-consolidation stress

The isotropic pre-consolidation pressure to which the soil has been subjected. If the soil is normally consolidated this value should be set equal to the initial stress state, i.e. zero. This option is only available for the advanced soil models.

Mobilized relative shear strength

This option is only available for the Hardening Soil model and HS small model models to set the initial shear hardening contour. This value must be between 0 (= isotropic stress state) and 1 (=failure state).

Phases

Lists the different phases of the CRS test. Each phase is definedby a Duration (in units of time), a vertical Strain increment (in %)and a Number of steps. The initial state is always assumed to be stress free. The given strain increment will be reached at the end of the given duration in the given number of steps. The input values can be changed by clicking in the table. A negative strain increment implies additional compression, whereas a positive strain increment implies unloading or tension. If a period of zero strain is desired, enter the desired duration with a zero strain increment.

Add — Adds a new phase to the end of the Phases list.

Insert — Inserts a new phase before the currently selected phase.

Remove — Removes the currently selected phase from the Phases

1.4 DIRECT SHEAR STRENGHT (DSS)-TEST

The DSS tabsheet contains facilities to define a direct simple-shear test. Before specifying the test conditions, a selection can be made between different test options.

DSS - Options

Drained / undrained DSS test

In the latter case, undrained soil conditions and zero drainage are assumed (similar as when the Drainage type has been set to Undrained (A) or Undrained (B), see Section 4.2), irrespective of the drainage type setting in the material data set.

Isotropically consolidated / K_0 -consolidated test

In the latter case the K_0 -value (ratio of lateral stress over axial stress) can be specified to set the initial stress state.

DSS - Conditions

The following settings can be defined:

Isotropic pre-consolidation stress

The isotropic pre-consolidation pressure to which the soilhas been subjected. If the soil is normally consolidated thisvalue should be set equal to the initial stress state or kept zero. This option is only available for the advanced soil models.

Mobilized relative shear strength

This option is only available for the Hardening Soil model and HS small model to set the initial shear hardening contour. This value must be between 0 (= isotropic stress state) and 1 (=failure state).

Initial stress $|\sigma_{yy}|$

The absolute value of the initial vertical stress at which the sample is consolidated, entered in units of stress. In the case of an isotropically consolidated test, the initial lateral stress is equal to the initial vertical stress. In the case of a K_0 -consolidated test, the initial lateral stress is equal to $K_0\ \sigma_{yy}$.

Time Δt — Time increment (only relevant for time-dependent models; Consolidation is not considered).

Number of steps — The number of steps that will be used in the calculation.

Maximum shear strain — The maximum value of shear strain (entered in %) that will Will be reached in the last calculation step. $|\gamma_{xy}|$

1.5 GENERAL

The General tabsheet contains facilities to define arbitrary stress and strain conditions. The following settings can be defined:

Isotropic pre-consolidation stress

The isotropic pre-consolidation pressure to which the soil has been subjected. If the soil is normally consolidated thisvalue should be set equal to the initial stress state or kept zero. This option is only available for the advanced soil models.

Mobilized relative shear strength

This option is only available for the Hardening Soil model and HS small model to set the initial shear hardening contour. This value must be between 0 (= isotropic stress state) and 1 (=failure state).

Phases

Lists the initial stress conditions and the stress/strain conditionsin the subsequent phases of the test. In the initial phase it should be indicated for each direction whether a stress increment or a strain increment is defined for that direction (applies to all phases). Each phase is defined by a Duration (in units of time) and a Number of steps, followed by the applied stress or strain increments. The given stress or strain increment will be reached at the end of the given duration in the given number of steps.

Add — Adds a new phase to the end of the Phases list.

Insert — Inserts a new phase before the currently selected phase.

Remove — Removes the currently selected phase from the Phases list.

1.6 RESULTS

The Results window shows several predefined typical diagrams to display the results of the current test. Double-clicking one of the graphs opens the selected diagram in alarger window (Figure 1.4). This window shows thes electeddiagram on the Graphic tab sheet. The Data tabsheet in this window lists the data points that are used to plot this diagram. Both the diagram and the data can be copied to the clip board using the Copy button on the toolbar.

The diagram can be zoomed in or out using the mouse by first clicking and holdingthe left mouse button in the diagram area and then moving the mouse to a second location and releasing the mouse button. Moving the mouse from the left upper corner to the right lower corner zooms the diagram to the selected area, whereas moving the mouse from the right lower corner to the left upper corner resets the view. The zoom action can also be undone using the Zoom out option on the toolbar.

The wheel button of the mouse can be used for panning: click and hold the mouse wheel down and move the diagram to the desired position.

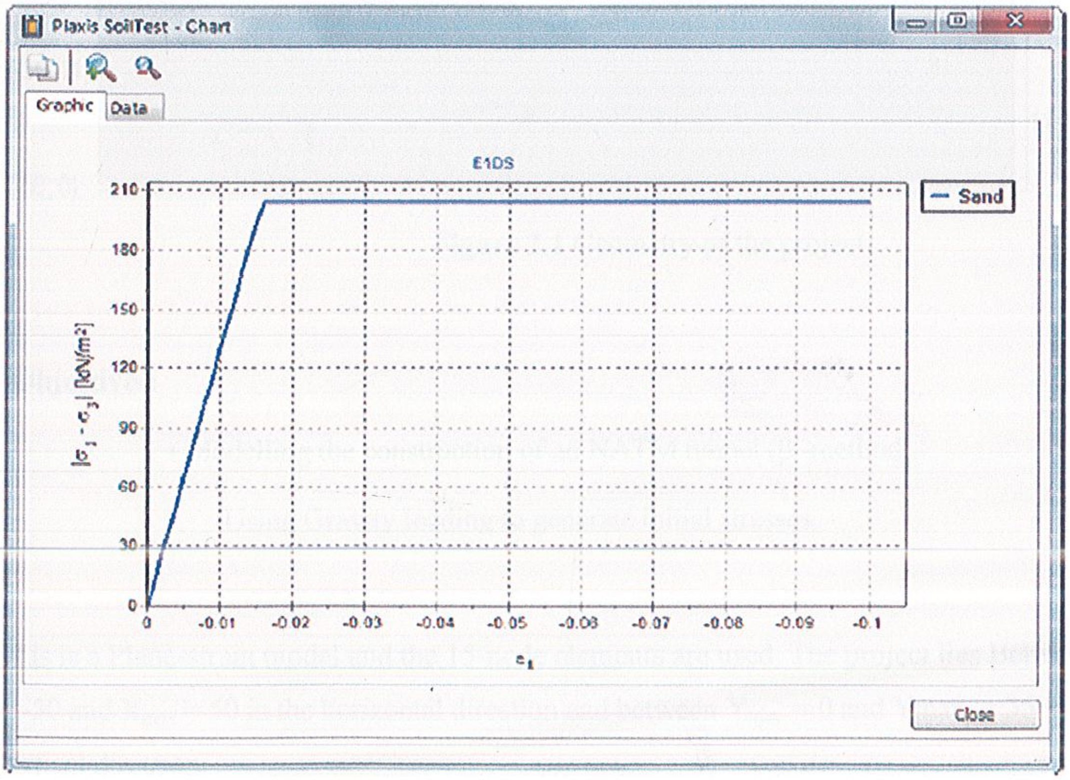


Figure1. 4 Results diagram

CHAPTER : 2 2 .EXCAVATION OF A NATM TUNNEL

This lesson illustrates the use of PLAXIS for the analysis of the construction of a NATM tunnel. The NATM is a technique in which ground exposed by excavation is stabilized with shotcrete to form a temporary lining. Rapid and consistent support of freshly excavated ground, easier construction of complex intersections and lower capital cost of major equipment are some of the advantages of NATM. Some of the limitations of this method are that it is slow compared to shield tunnelling in uniform soils, dealing with water ingress can be difficult, and it demands skilled man power.

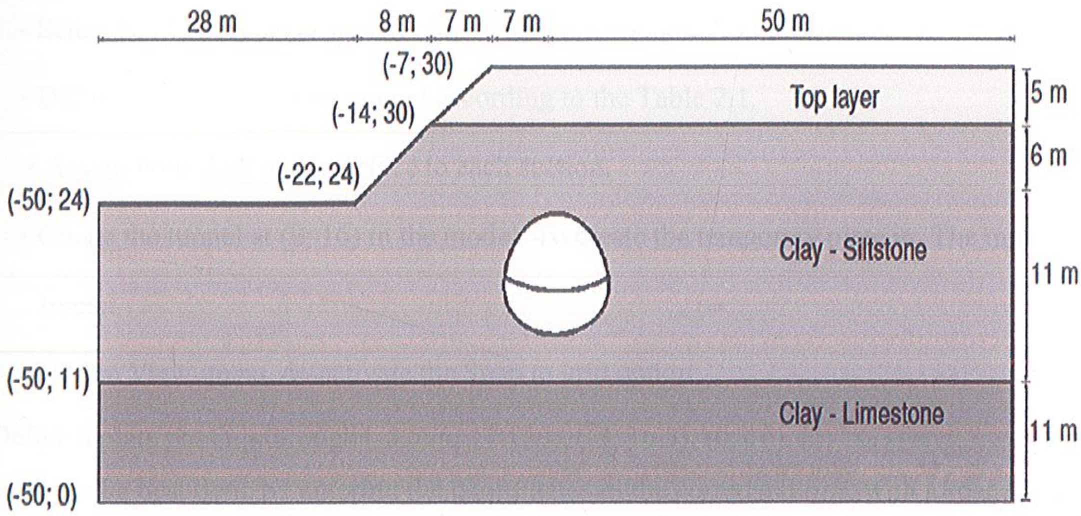


Figure 2.1 Geometry of the project

Objectives:

- Modelling the construction of an NATM tunnel (β -method).
- Using Gravity loading to generate initial stresses.

This is a Plane-strain model and the 15-node elements are used. The project lies Between X_{min} = -50 and X_{max} = 50 in the horizontal direction and between Y_{min} = 0 and Ymax = 35 in the vertical direction.

2.1 INPUT

The basic geometry including the three soil layers, as shown in Figure 2.1 (excluding the tunnel), can be created using the geometry line option. In the model 11 m of the Clay-limestone layer is considered. The bottom of this layer is considered as reference in y direction ($y_{min} = 0$). After generating the basic geometry, follow these steps to design the tunnel:

Click the Tunnel button in the toolbar. The Tunnel designer window appears, with a number of options in its toolbar for creating tunnel shapes. By default the Whole tunnel button is active. This is valid in this project.

- The Symmetric option is selected for the shape of the tunnel by default. This option is valid in this example
- Select NATM tunnel as tunnel type in the corresponding drop-down menu.
- Define the sections of the tunnel according to the Table 2.1.
- Assign both shell and interface to each section.
- Create the tunnel at (0; 16) in the model. To create the temporary plate in The middle of the tunnel
- In the View menu, de-activate the Snap to grid option.

Define a plate passing through (-5.596; 17.120), (-3, 16.3), (0; 16), (3; 16.3) and(5.596;17.120). This can be best done by entering the coordinates in the command line, with a space between the x- and y-coordinate.

Table 2.1 Section properties of the tunnel

Section	Type	Centre (y)	Radius	Angle
1	Arc	7.0	10.40	22.0
2	Arc	n/a	2.40	47.0
3	Arc	n/a	5.60	25.0
4	Arc	n/a	5.89	43.0
5	Arc	n/a	5.48	43.0

Boundary conditions

Click the Standard fixities button to apply the appropriate boundary conditions.

Material sets and meshes generation

A single material data set will be defined for the lining of the tunnel and the temporary plate. Create the material dataset according to Table 2.2.

Table 2.2 Material properties of the plates

Parameter	Name	Lining	Unit
Material type	Type	Elastic	kN/m
Normal stiffness	EA	$6.0 \cdot 10^6$	kN/m
Flexural rigidity	EI	$2.0 \cdot 10^4$	kNm^2 /m
Weight	w	5.0	kN/m/m
Poisson's ratio	v	0.15	-

The properties of the different soil types are given in Table 2.3. The soil materials are Drained. As a result definition of flow parameters is not necessary. The initial parameters in the Initial tab sheet are not used, since the initial stresses will be generated by means of Gravity loading. Assign the data to the corresponding clusters in the geometry model and the clusters inside the tunnel.

To generate the mesh:

- Select the Global coarseness option from the Mesh menu and set the element size Distribution to Fine. Selected in the Mesh menu.
- Refine the soil clusters inside the tunnel.

Table 2.3 Material properties of the soil layers

Parameter	Name	Top layer	Clay-Siltstone	Clay-limestone	Unit
General					
Material model	Model	Hardening soil	Hoek-Brown	Hoek-Brown	-
Type of material behaviour.	Type	Drained	Drained	Drained	-
Soil unit weight above - -phreatic level .	γunsat	20	25	24	kN/m^3
Soil unit weight below- -phreatic level.	γsat	22	25	24	kN/m^3
Initial void ratio	einit	0.5	0.5	0.5	-
Parameters					
Secant stiffness in- -standard drained - triaxial test.	50	40000	-	-	kN/m^2
Tangent stiffness for- -primary oedometer- -loading.	oed	40000	-	-	kN/m^2
Unloading / reloading- -stiffness	ur	120000	-	-	kN/m^2
Power for stress-level dependency of stiffness	m	0.5	-	-	-
Young's modulus	E'	-	$1.0 \cdot 10^6$	$2.5 \cdot 10^6$	kN/m^2

Poisson's ratio	vur'	0.2	0.25	0.25	-
Uniaxial compressive strength	σci	-	25000	50000	kN/m^2
Material constant for the intact rock	mi	-	4.0	10.0	-
Geological Strength Index	GSI	-	40.0	55.0	-
Disturbance factor	D	-	0.2	0.0	-
Cohesion	cref '	10	-	-	kN/m2
Friction angle	ϕ'	30	-	-	°
Dilatancy parameter	ψmax	-	30.0	35.0	°Dilatancy
parameter	σ_ψ	-	400	1000	kN/m^2
Interfaces					
Interface strength	–	Rigid	Manual	Rigid	-
Strength reduction factor	Rinter	1.0	0.5	1.0	-

2.2 CALCULATIONS

To simulate the construction of the tunnel it is clear that a staged construction calculation is needed after defining the initial conditions. The calculation process is modelled and executed in the Classical mode.

Initial phase: Initial conditions

Note that the soil layers are not horizontal. As a result, the K0 procedure can not be used to generate the initial effective stresses in this example. Instead Gravity loading should be used. This option is available in the General tabsheet of the Calculations program for the Initial phase. Water will not be considered in this example. The general phreatic level should remain at the model base. Make sure that the tunnel is inactive.

Simulation of the construction of the tunnel

A staged construction calculation is needed in which the tunnel lining is activated and the soil clusters inside the tunnel are deactivated. Deactivating the soil inside the tunnel only affects the soil stiffness and strength and the effective stresses. The calculation phases are Plastic analyses, Staged construction. The three-dimensional arching effect is emulated by using the so-called β -method. The idea is that the initial stresses p_k acting around the location where the tunnel is to be constructed are divided into a part (1−β) pk that is applied to the unsupported tunnel and a part β pk that is applied to the supported tunnel. To apply this in PLAXIS one can use the staged construction option with a reduced ultimate level of ΣMstage.

To define the calculation process follow these steps:

Phase 1

- Click the Next button to introduce a next calculation phase.
- In the Loading input box available in the Parameters tabsheet click Advanced.
- Define a value of 0.6 for ΣMstage. This corresponds to a β -value of 1-ΣMstage =0.4.
- In the Staged construction mode deactivate the upper cluster in the tunnel. Do not Activate the tunnel lining.

Phase 2

- Click the Next button to introduce a next calculation phase.
- In the Staged construction mode activate the lining and interfaces of the part of the tunnel excavated in the previous phase (top heading).
- Note that the value of ΣMstage is automatically reset to 1.0.

Phase 3

- Click the Next button to introduce a next calculation phase.
- In the Loading input box available in the Parameters tabsheet click Advanced. The Advanced parameters window pops up.
- Define a value of 0.6 for ΣMstage ($\beta = 0.4$).
- In the Staged construction mode deactivate the lower cluster (invert) and the temporary Lining in the middle of the tunnel.

Phase 4

- Click the Next button to introduce a next calculation phase.
- In the Staged construction mode activate the remaining lining and interfaces. All the plates and interfaces around full tunnel are active.
- Note that the value of ΣMstage is automatically reset to 1.0.
- Select node at slope crest point and the tunnel crest. These points might be of interest to evaluate the deformation during the construction phases.
- Start the calculations.

RESULTS

After the calculation, select the last calculation phase and click the View calculation results button. The Output program is started, showing the deformed mesh at the end of the calculation phases (Figure 2.2).

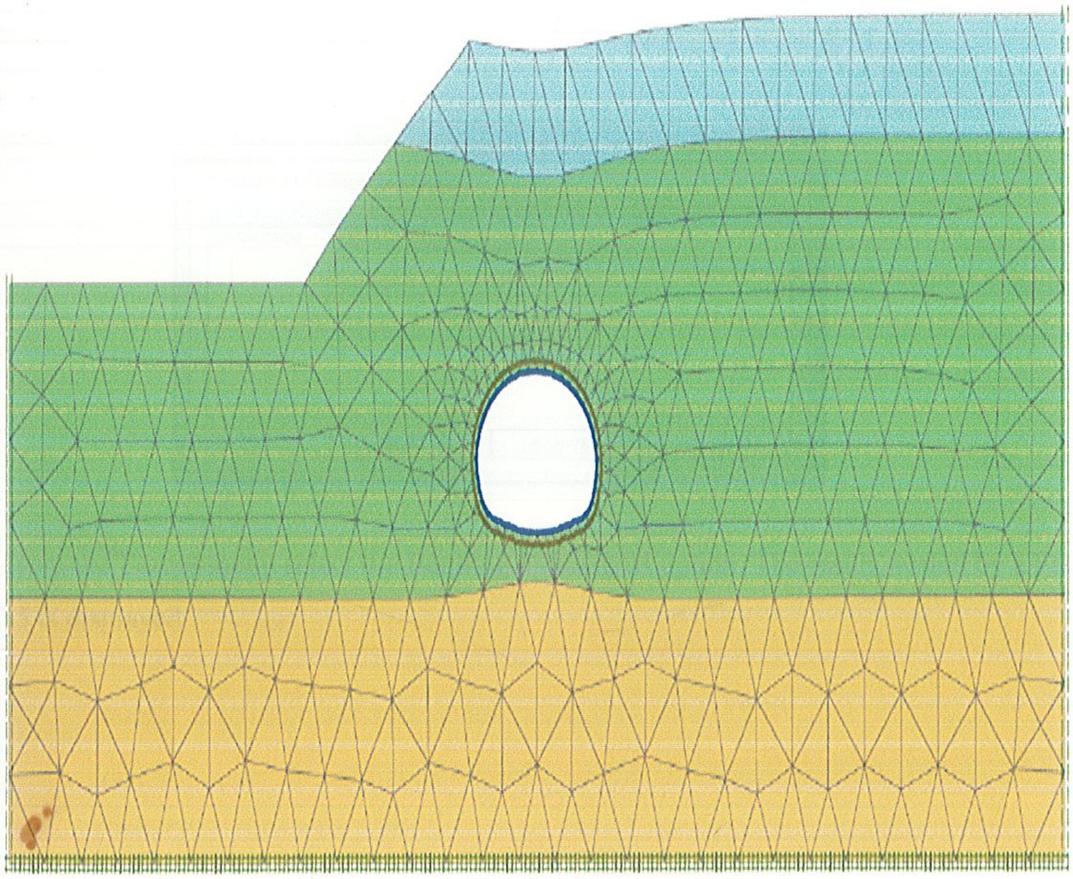
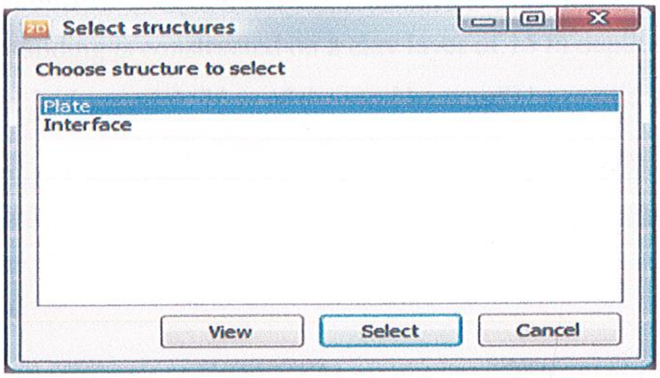


Figure 2.2 The deformed mesh at the end of the final calculation phase

To display the bending moments resulting in the tunnel:

To select the lining of all the tunnel sections, click the corresponding button in the side toolbar and drag the mouse to define a rectangle where all the tunnel sections are included. Select the Plate option in the appearing window (Figure 2.3) and press View. Note that the tunnel lining is displayed in the Structures view.



From the Forces menu select the Bending moment M option. The result, scaled by a Factor or 0.5 is displayed.

CHAPTER: 3

3. FREE VIBRATION AND EARTHQUAKE ANALYSIS OF A BUILDING

This example demonstrates the natural frequency of a five-storey building when subjected to free vibration and earthquake loading.

The building consists of 5 floors and a basement. It is 10 m wide and 17 m high including basement. The total height from the ground level is 5 x 3 m = 15 m and the basement is 2 m deep. A value of 5 kN/m^2 is taken as the weight of the floors and the walls. The building is constructed on a clay layer of 15 m depth underlayed by a deep sand layer. In the model, 25 m of the sand layer will be considered.

3.1 INPUT

General settings

- start the Input program and select Start a new project from the Quick select dialog box.
- In the Project tabsheet of the Project properties window, enter an appropriate title and make sure that Model is set to Plane strain and that Elements is set to 15-node.
- Keep the default units and set the model dimensions to X_{min} = −80, X_{max} = 80, Y_{min} = −40 and Y_{max} = 15. Keep the default values for the grid spacing (Spacing = 1 m; Number of intervals = 1).

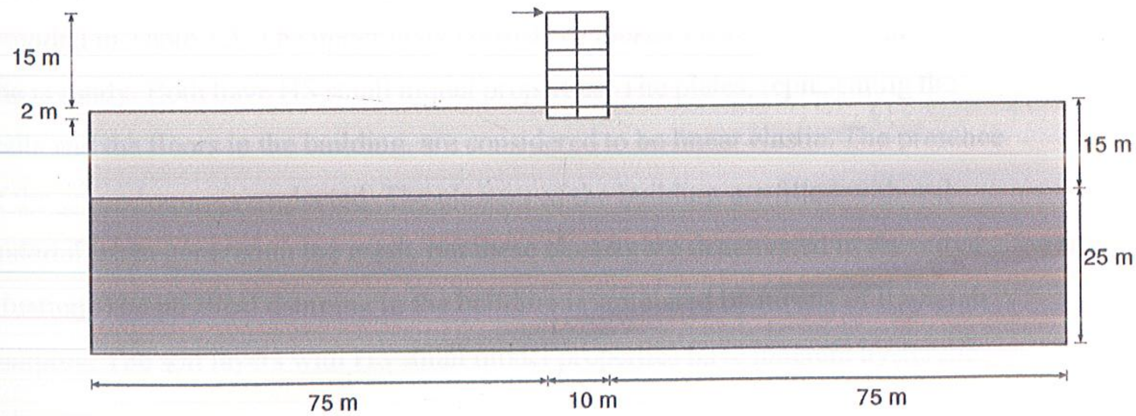


FIG 3.1 GEOMETRY OF THE MODEL

3.1.1 GEOMETRY MODEL

Use the Geometry line to define the soil cluster

Define the building by using the Plate button.

Define a node to node anchor passing through (0; 15) and (0; -2).

Define an interface to model the interaction between soil and building Around the basement floor.

Define a static lateral force of 1 kN/m at the top left corner of the building by Clicking the point loads-Load system A Button in the toolbar and by clicking On the location of the load.

The option of Standard fixities is used to assign full restraints on the movements in horizontal direction on the vertical boundaries on two sides and full restraints in both the horizontal and vertical directions along the lowermost horizontal geometry line.

3.1.2 MATERIAL PROPERTIES

The properties of the subsoil are given in Table 3.1 and the plate properties are provided in Table 3.3. The upper layer consists of mostly clayey soil and the lower one is sandy. Both have HS small model properties. The plates, representing the walls and the floors in the building, are considered to be linear elastic. The presence of the groundwater is neglected. The clusters of the building are filled with soil material when generating the mesh, but these clusters are deactivated in the initial situation. The physical damping in the building is simulated by means of Rayleigh damping. The soil layers with HS small model properties have inherent hysteretic damping.

Parameter	Name	Upper clayey layer	Lower sandy layer	Unit
General				
Material model	Model	HS small	HS small	-
Type of material behaviour	Type	Drained	Drained	-
Soil unit weight above phreatic level	γunsat	16	20	kN/m^3
Soil unit weight above phreatic level	γsat	20	20	kN/m^3
Parameters				
Secant stiffness in standard drained triaxial test	50	$2.0 \cdot 10^4$	$3.0 \cdot 10^4$	kN/m^2
Tangent stiffness for primary oedometer loading	oed	$2.561 \cdot 10^4$	$3.601 \cdot 10^4$	kN/m^2
Unloading / reloading stiffness	ur	$9.484 \cdot 10^4$	$1.108 \cdot 10^5$	kN/m^2
Power for stress-level dependency of stiffness	m	0.5	0.5	-
Cohesion	c'_{ref}	10	5	kN/m^2
Friction angle	ϕ'	18	28	°
Dilatancy angle	ψ	0	0	°
Shear strain at which $Gs=0.722G_0$	$\gamma_{0.7}$	$1.2 \cdot 10^{-4}$	$1.5 \cdot 10^{-4}$	-
Shear modulus at very small strains	G^{0ef}	$2.7 \cdot 10^5$	$1.0 \cdot 10^5$	kN/m^2
Poisson's ratio	ν'	0.2	0.2	

TABLE 3.1 MATERIAL PROPERTIES OF SUB LAYER

When subjected to cyclic shear loading, the HS small model will show typical hysteretic behaviour. Starting from the small-strain shear stiffness, G^{0ef} , the actual stiffness will decrease with increasing shear. Figure 14.2 displays the Modulus reduction curves, i.e. the decay of the shear modulus with strain.

In the HS small model, the tangent shear modulus is bounded by a lower limit ,G_{ur} . $G_{ur} = E^{ur} / 2(1 + \nu_{ur})$.

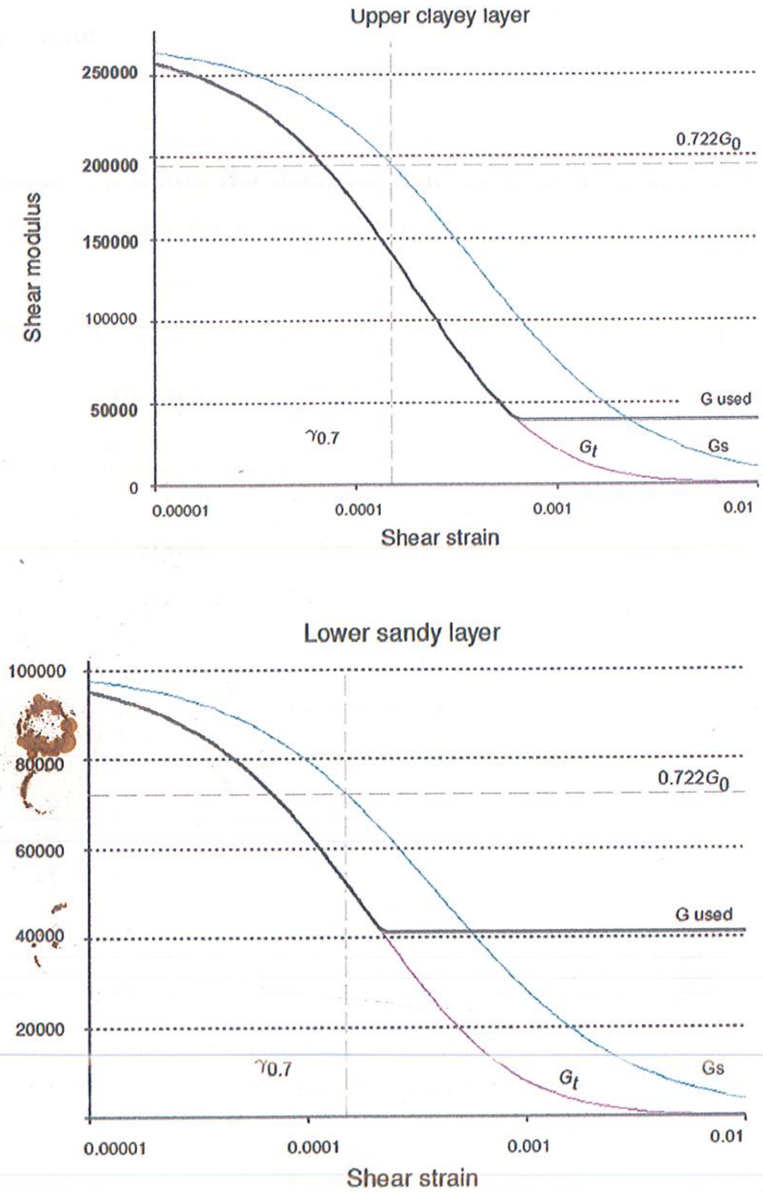


FIG 3.2 MODULUS REDUCTION CURVES

The values of Gur for the Upper clayey layer and Lower sandy layer and the ratio to G0 are shown in Table 3.2. This ration determines the maximum damping ratio that can be obtained.

TABLE 3.2 Gur values and ratio to G^{0ef}

Parameter	Unit	Upper clayey Layer	Lower sandy layer
Gur	kN/m^2	39517	41167
G^{0ef}/Gur	-	6.75	2.5

Figure 3.3 shows the damping ratio as a function of the shear strain for the material used in the model. For a more detailed description and elaboration from the modulus reduction curve to the damping curve can be found in the literature . Define the material dataset for the for the plates representing the structure according to Table 3.3.

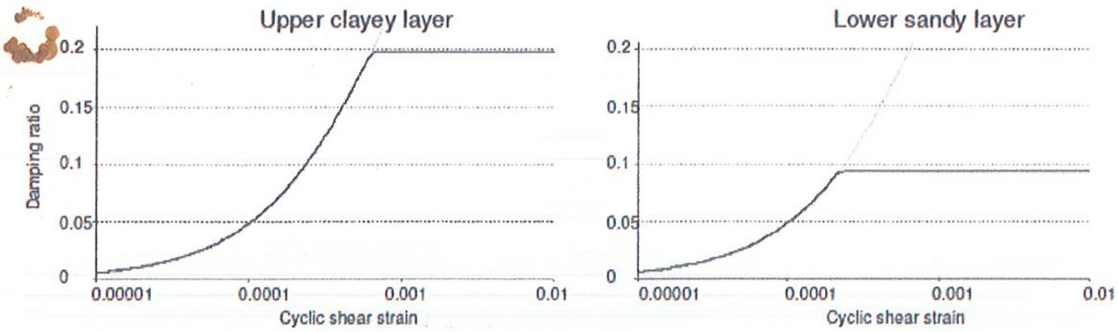


FIG 3.3 DAMPING CURVES

TABLE 3.3 Material properties of the building (plate properties)

Parameter	Name	Rest of building	Basemen	Unit
Materal type	Type	Elastic	Elastic	-
Normal stiffness	EA	$9.0 \cdot 10^6$	$1.2 \cdot 10^7$	kN/m
Flexural Rigidity	EI	$6.75 \cdot 10^4$	$1.6 \cdot 10^5$	kNm^2 /m
Weight	w	10	20	kN/m
Poissons ratio	ν	0.0	0.0	-
Rayleigh Damping	α	0.2320	0.2320	-
	β	$8.0 \cdot 10^{-3}$	$8.0 \cdot 10^{-3}$	-

TABLE 3.4 Material properties of the node-to-node anchor

Parameter	Name	Column	Unit
Normal Stiffness	EA	$2.5 \cdot 10^6$	kn
Material Type	Type	Elastic	-
Spacing Out Of Lane	Lspacing	3.0	m

3.13 MESH GENERTAION

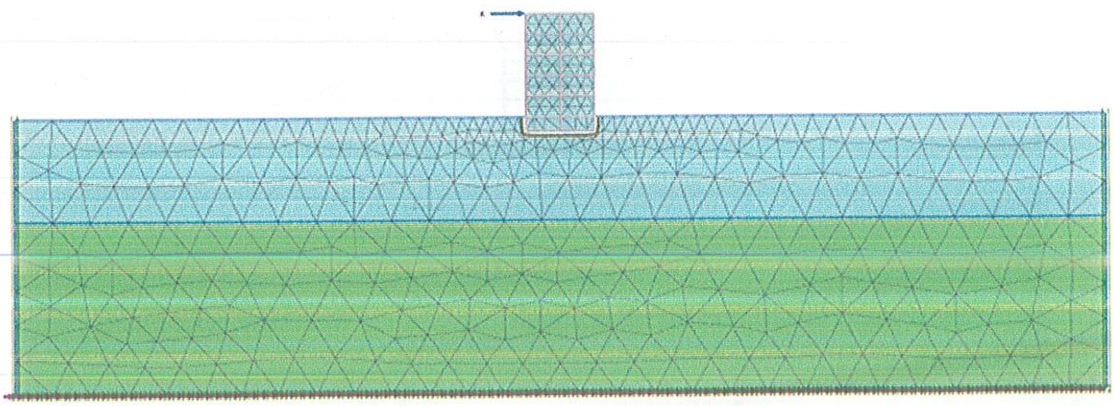


FIG 3.4 Mesh of Soil - Building System.

3.2 CALCULATIONS

The calculation process consists of the initial conditions phase, simulation of the construction of the building, loading, free vibration analysis and earthquake analysis.

Initial phase

- In the General tabsheet the K0 procedure option is automatically selected as calculation Type.
- In the Parameters window accept the default values and click Define.
- In the Staged construction mode check that the building and load areinactive.
- In the Water conditions mode define a phreatic level at y = -15.
- Click Update to proceed to the Calculations program.

Phase 1

- Click Next to add a new phase.
- In the General tabsheet the Plastic option is automatically selected as calculation type.
- In the Parameters window accept the default values and click Define.
- In the Staged construction mode construct the building (activate all the plates and the Anchor) and deactivate the basement volume.
- Click Update to proceed to the Calculations program.

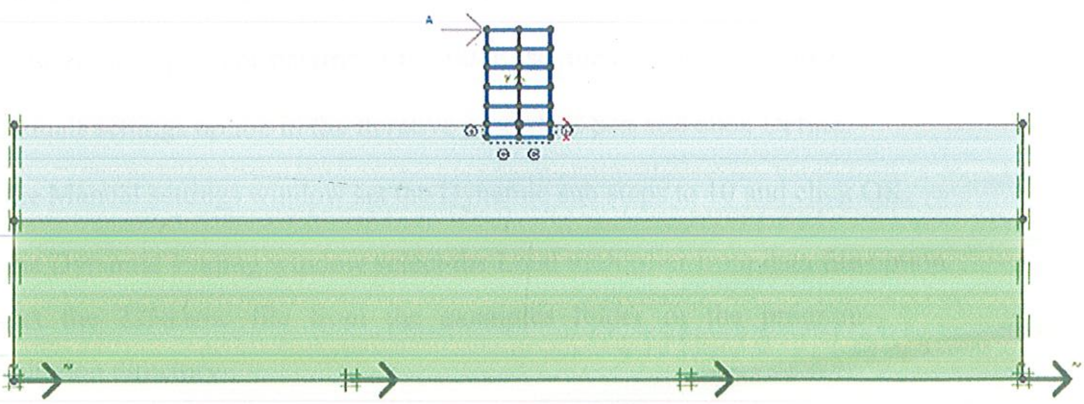


FIG 3.5 Construction of the building

Phase 2

- Click Next to add a new phase.
- In the General tabsheet the Plastic option is automatically selected as calculation type.
- In the Parameters window select the Reset displacement to zero, accept the default values And click define.
- In the Staged construction mode activate the load and assign a value of 10 kN/m to it.
- Click Update to proceed to the Calculations program.

Phase 3

- Click Next to add a new phase.
- In the General tabsheet select the Free vibration option as calculation type.
- In the Parameters tabsheet, set the Time interval to 5 sec.
- The Additional steps parameter is automatically set to 100. Select the Manuals settings Option in the iterative procedure box and click define .
- In the Manual settings window set the Dynamic sub steps to 10 and click OK.

Phase 4

- Click Next to add a new phase.
- In the General tabsheet set the Start from phase option to Phase 1 (construction of building).
- Select the Dynamic option as calculation type.
- In the Parameters tabsheet, set the Time interval to 20 sec.
- Set the Additional steps parameter to 200. Reset the displacements to zero. Select the Manuals settings option in the Iterative procedure box and click Define.
- In the Manual settings window set the Dynamic sub steps to 10 and click OK.
- In the Dynamic loading window select the Load multiplier from data file option. Select the 225a.smc file from the examples folder of the program installation directory.

In the Multipliers tabsheet click the button next to $\Sigma - MdispX$.

Select interesting points for curves (e.g. top of the bulding, basement) and calculate the project.

3 .3 RESULTS

Figure 3.6 shows the deformed structure at the end of the Phase 2 (application of horizontal load).

Figure 3.7 shows the time history of displacements of the selected points A (0; 15) for the free vibration phase. It may be seen from the figure that the vibration slowly decays with time due to damping in the soil and in the building.

In the Chart tabsheet of the Settings window select the Use frequency representation (spectrum) and Use standard frequency (Hz) options in the Dynamics box. The plot is shown in Figure 3.8. From this figure it can be evaluated that the dominant building frequency is around 1 Hz.

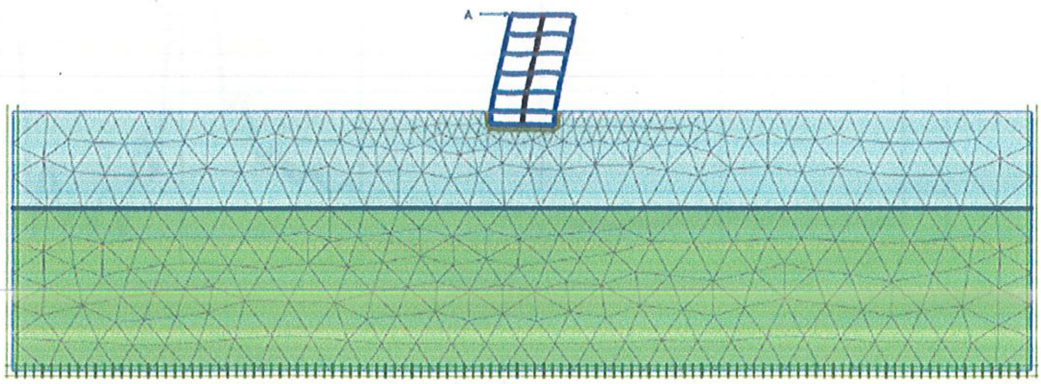


FIG 3.6 Deformed mesh of the system

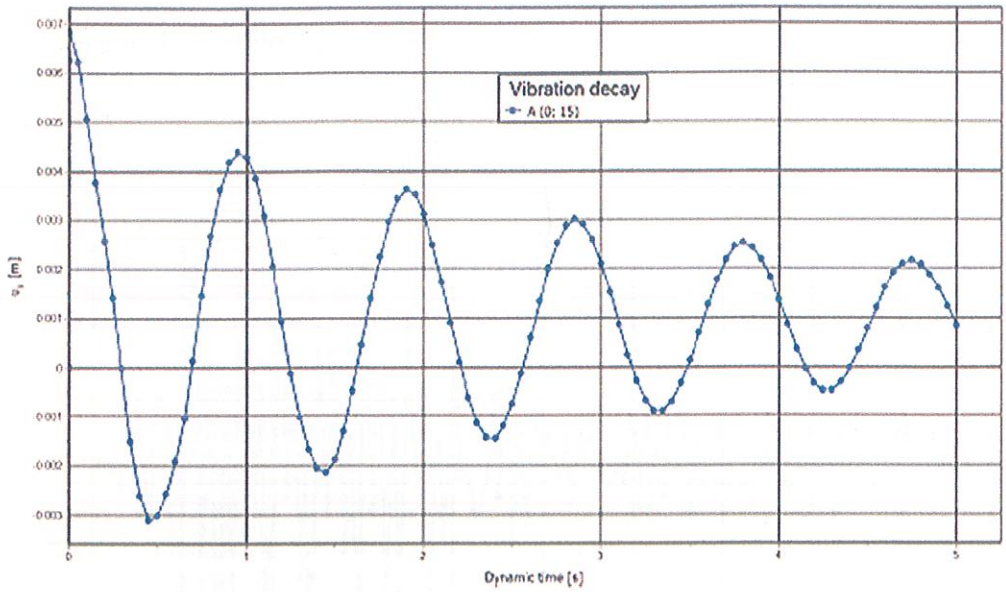


FIG 3.7 Time history of displacements at selected points

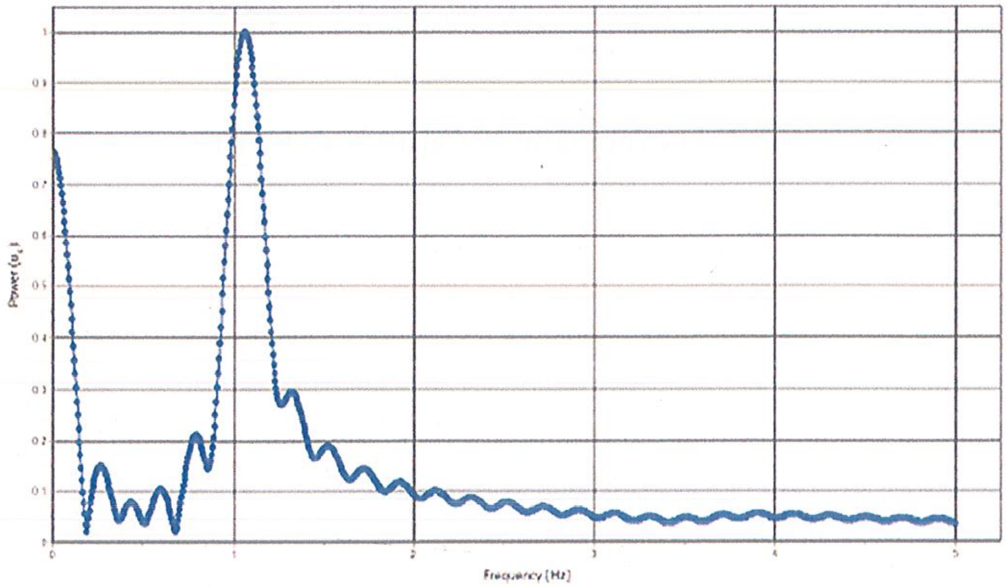


FIG 3.8 Frequency representations (spectrum)

Figure 3.9 shows the time history of the lateral acceleration of the selected points A (0; 15) for the earthquake phase (dynamic analysis). For a better visualisation of the results animations of the free vibration and earthquake can be created.

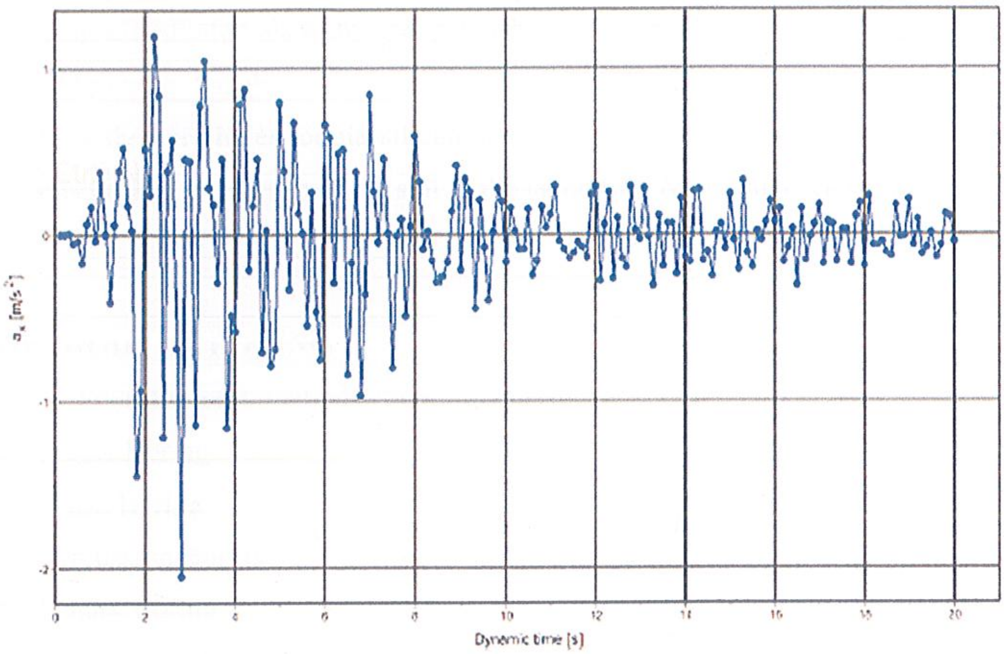


FIG 3.9 Variation of acceleration in dynamic time

CHAPTER: 4 ANALYSIS OF SOIL PROFILE

4.1 GENERAL

The soil investigation work was done at the prorposed site. A bore hole, 35.0 meter deep , was executed at the proposed site . Besides Boring, Standard Penetration Tests at specified intervals were also performed as per B.I.S. specifications and in addition to the above Test , undisturbed as well as auger samples were collected from the bore-holes for classification of soil, shear strength tests and determination of mechanical properties of the soil in the laboratory & all other relevant tests .

4.2 METHODS OF BORING

Four different methods commonly used :

1. Auger Boring
2. Wash Boring
3. Percussion Boring
4. Rotary Boring

4.3 METHODS OF SAMPLING

- Disturbed samples used for determining index properties (ex. gsd, consistency, etc.)
- Undisturbed samples used for determining engineering properties (ex. Density, water content, shear strength parameters, etc.)
- Wash samples obtained from wash boring water or mud.
- Representative sample retains all constituents of the soil, but is disturbed from natural state and structure. (ex. Split spoon sampler)
- Block samples are carved out form sides or bottoms of excavations, sealed in a box and taken to lab.
- Open drive samplers consist of thin walled tubes which are driven or pushed into the soil at the bottom of the hole. (ex. Shelby Tube sampler)

4.4 STANDARD PENETRATION TEST (SPT)

- IS:2131 – 1981 Method for Standard Penetration Test for Soils.
- Most important and most commonly used field test
- Typical equipment: drill rig, split spoon sampler, hammering equipment, casing pipe.

SPT PROCEDURE

- Drive a section of casing pipe.
- Complete wash boring and clean the hole
- Replace driving bit by split spoon sampler at the bottom end of the driving rod
- Drive the sampler by dropping a hammer of 63.5kg weight through a height of 75 cm
- The number of blows required to penetrate three successive lengths of 15 cm are noted.
- The first 15 cm drive is considered as seating load and is ignored. The total number of blows required to penetrate the remaining 30 cm is called the blow count or penetration number N.
- Raise the sampler to the surface, open it and extract the sample.
- Drive the next length of casing and repeat the process until required depth is reached.
- At the end of the borehole, note the gwt after it stabilizes and then backfill the borehole.

CORRECTION TO 'N' VALUES

- Overburden Correction applied to N value based on chart by Peck, et. al. (varies from 0.45 to 2)
  - ❑ $N' = C_n * N$ where C_n is correction factor
- Correction due to dilatancy for fine sand and silt below gwt having N' > 15
  - ❑ $N'' = 15 + 0.5*(N'-15)$

CORRELATIONS BETWEEN N VALUES AND SOIL PROPERTIES

Table: 3.1

N	Compactness	Relative Density (%)	f^o
0 to 4	Very Loose	0-15	< 28
4 to 10	Loose	15 - 35	28 -30
10 to 30	Medium Dense	25 - 65	30 -36
> 50	Very Dense	> 85	> 41

Table: 3.2

Consistency	N	qu (kPa)
Very Soft	0 to 2	< 25
Soft	2 to 4	25 to 50
Medium Stiff	4 to 8	50 to 100
Stiff	8 to 12	100 to 200
Very Stiff	15 to 30	200 to 400
Hard	> 30	> 400

UNDISTURBED SOIL SAMPLES:

Undisturbed soil sample collected in field have been tested in laboratory and preparation of sample for the under mentioned tests have been done in accordance with I.S.2720-(Part-I)-1983.

1. Sieve analysis test as per I S. Specification No. 2720 --(Part-IV).
2. Atterburg limit test (L.L. & P.I.) As per I.S. Specification No. 2720 -- (Part-II).
3. Natural moisture content as per I.S.Specification No.2720 – (Part-IV).
4. Particle size analysis test as per I.S.Specification No. 2720-(Part-VI).
5. Wet density test as per I.S.Specification No 2720- (Part-VI).
6. Dry density test as per I.S.Specification No. 2720- (Part-VI)
7. Specific Gravity test as per I.S.Specification No-2720-(Part-III)-Sec.2.
8. Triaxial compression test and determination of shear parameter (C & Ø as per I.S. XII) & I.S. 2720 – (part – XIII).
9. Consolidation test conducted as per I.S Specification No. 2720 (Part-XV).

DISTURBED SOIL SAMPLES:

Disturbed Soil samples have been prepared in accordance with I.S. Specification No. 2720- (Part-I)- 1983 and tested as follows:-

1. Sieve analysis test as per I.S.Specification No. 2720- (Part- IV).
2. Atterburg limit test (L.L.. & P.I..) as per I.S.Specification No. 2720 --(Part-II).
3. Particle size analysis test as per I.S.Specification No. 2720-(Part-VI).

Calculation of bearing capacity is governed generally by I.S. Specification No . 8009-)Part-I)- 1976, I.S.No.2720- (Part – II)- 1980, I.S. No 6403-1981, I.S. 1904-1978 and I.S. 1080-1985 and other relevant I.S. Codes as well as based on assessment and latest developments.

SOIL CLASSIFICATION & GENERAL NATURE OF THE SOIL STRATA:

The soil classification has been done with the help of soil properties obtained by laboratory tests of soil.

Depth (m)	Soil Classification	Description
NSL - 10.5	GW-SW	Compact Sand. + Boulders/Gravels.
10.5 - 12.0	SM-SP	Compact Sand. + Boulders/Gravels
12.0 - 16.0	CL	Firm to stiff clay
16.0 - 20.0	SM-SP	Compact Sand.
20.0 - 26.0	SP	Compact Sand.
26.0 - 28.0	M(NP)	Compact Silt.
28.0- 32.5	SM-SP	Compact Sand.
32.5- 35.0	SM	Compact Sand.

CORRECTION OF N' VALUES:

Depth of foundation	Over burden Pressure kg/cm^2	Correction factor	N - Values	N – Corrected
1.5	0.150	1.70	50	Refusal

3.0	0.300	1.45	50	Refusal
4.5	0.450	1.25	50	Refusal
6.0	0.600	1.15	50	Refusal
7.5	0.750	1.085	50	Refusal
9.0	0.900	1.05	50	Refusal
10.5	1.050	0.905	44	39.8
12.0	1.200	0.91	28	25.5
13.5	1.350	0.905	18	16.3
15.0	1.500	0.85	25	21.2
16.5	1.650	0.83	28	23.2
18.0	1.800	0.81	30	24.3
20.0	2.000	0.79	32	25.3
21.5	2.150	0.76	35	26.6
23.0	2.300	0.73	37	27.0
24.5	2.450	0.715	30	21.4
26.0	2.600	0.70	35	24.5
27.5	2.750	0.675	35	23.6
30.0	3.000	0.65	37	24.0
31.5	3.150	0.635	40	25.4
33.0	3.300	0.62	47	29.1
35.0	3.500	0.60	42	25.2

Table: 3.3

ALLOWABLE BEARING CAPACITY CRITERIA

A foundation can fail by two modes i.e.

i) Shear failure.

ii) Excessive settlement.

- Shear failure being catastrophic, an adequate factor of safety is applied To ultimate bearing capacity that can initiate this type of failure.
- BIS recommends a value of FOS = 2.5 to obtain the net safe bearing capacity qns by using the physical characteristics of the foundation and relevant shear strength parameters of soil.
- Through Settlement analysis a net loading intensity qn is obtained by using the physical characteristics of the foundation and the relevant compressibility characteristics of the Underlying soil.
- The value so obtained ensures that the foundation shall not settle more than that which is permissible as per BIS recommendations. The permissible settlement depends upon the type of superstructure and the nature of supporting strata.

The lesser of these computed values i.e. qns or qn is adopted as the allowable bearing capacity for proportioning the foundation of superstructures.

NOTATIONS

Gw	:	Well graded gravels
SW	:	Well graded gravels
SP	:	Poorly graded gravels
M(NP)	:	Non plastic silt
CL	:	Clay of low plasticity
LL	:	Liquid Limit
PI	:	Plasticity Index

CONCLUSION

- Plaxis is a structured and efficient way of modelling since you can store a coherent set of partial factors (a design approach) for loads and materials in one location. This DA can be easily applied during the definition of phrases to make an ULS calculation in addition to Serviceability Limit State (SLS) calculation.
- PLAXIS 2D allows orthotropic material behaviour in geogrids and plates, introducing an additional axial stiffness in the out of plane direction.
- PLAXIS 2D provides an optional Updated Mesh' analysis to perform basic types of calculations (Plastic calculation, Consolidation analysis, Phi-c reduction).
- The results of soil tests help us to understand soil behaviour for various building construction and foundations.
- PLAXIS2D provides option to the users to select mesh density in the range from very coarse to very fine.
- Loosening the soil would allow water to flow quickly because there is more room between the grains. If soil is tightly packed water is given less space to flow, making the drainage time slower.
- To prevent Earthquake damage buildings should not be made too heavy & adequate building materials must be used.
- Strong column and weak beams , making beams more ductile strategies must be used for designing Earthquake resistant Buildings.
- All the Irregularities i.e Plan Irregularities and Vertical Irregularities should be taken care off while designing Earthquake resistant Buildings.

REFERENCES

1. Agarwal P. "Experimental study of seismic strengthening and Retrofitting measures in Masonry Buildings" , Ph.D. Thesis IIT ROORKEE .
2. Bathe, K.J. (1996). Finite Element Procedures. Prentice Hall.
3. Das, B.M. (1995). Fundamentals of soil dynamics
4. Galavi, V. (2010). Groundwater flow, fully coupled flow deformation and undrained analyses in PLAXIS 2D and 3D. Technical report, Plaxis BV
5. Gulhati & Datta (2005) .Geotechnical Engineering
6. IS: 13935-1993 Bureau of Indian standards.
7. IS:2131 – 1981 Bureau Of Indian Standards.
8. IS: 2720-1983 (1) Bureau of Indian Standards.
9. PLAXIS 2D MANUALS-2010
10. Pankaj Agarwal and Manish Shrikhande (2006) – Earthquake Resistant design of structures . Prentice Hall of India.