PLAXIS Version 8
Dynamics Manual

TABLE OF CONTENTS

1 INTRODUCTION

Soil and structures are often subjected not only to static loads due to constructions in and on the ground surface but also to dynamic loads. If the loads are powerful, as in earthquakes, they may cause severe damages. With the PLAXIS Dynamic analysis module you can analyse the effects of vibrations in the soil.

Vibrations may occur either manmade or natural. In urban areas, vibrations can be generated due to pile driving, vehicle movement, heavy machinery and/or train travel. A natural source of vibrations in the subsoil is Earthquakes.

The effects of vibrations have to be calculated with a dynamic analysis when the frequency of the dynamic load is in the order or higher than the natural frequency of the medium. Low frequency vibrations can be calculated with a pseudo-static analysis.

In modelling the dynamic response of a soil structure, the inertia of the subsoil and the time dependence of the load are considered. Also, damping due to material and/or geometry is taken into account. Initially the linear elastic model can be utilised for the simulation of the dynamic effects, but in principle any of the available soil models in PLAXIS can be used.

Excess pore pressures can be included in the analysis if undrained soil behaviour is assumed. Liquefaction, however, is not considered in Version 8. Future versions may incorporate a model that is able to simulate this phenomenon.

Even though vibrations often have 3D-characteristics, in PLAXIS Professional Version 8, the dynamic model is limited to plane strain and axisymmetric conditions.

The dynamic calculation program was developed in cooperation with the University of Joseph Fourier in Grenoble. This cooperation is gratefully acknowledged.

1.1 ABOUT THIS MANUAL

This manual will help the user to understand and work with the PLAXIS Dynamics module. New users of PLAXIS are referred to the Tutorial section in the complete manual (PLAXIS Version 8).

Tutorial Chapters

The Dynamics manual starts as a tutorial. The user is advised to work through the exercises. In the first exercise the influence of a vibrating source over its surrounding soil is studied. The second exercise deals with the effects of pile driving. The third exercise analyse the effect of an earthquake on a five-storey building.

Reference Chapters

The second part of the Dynamics manual consists of four chapters. These chapters describe the four parts of the PLAXIS Program (input, calculation, output and curves) in view of the functionality of the Dynamics module.

Validation/Verification Chapters

The third part of the manual describes some of the test cases that were used to validate the accuracy and performance of the dynamics module.

Theory Chapters

In the fourth part of the manual you will find a brief review of the theoretical aspects of the dynamic model as used and implemented in PLAXIS.

1.2   DYNAMIC LOADING FEATURES OF VERSION 8

The way dynamic loads in PLAXIS Version 8 are applied during calculations is similar to that in Version 7. The creation and generation of dynamic loads is summarized below:

1.   In the Input program:

   - create loads such as load system A or B and/or prescribed displacements.

   - set the appropriate load (load system A, B and/or prescribed displacements) as a dynamic load using the *Loads* menu

2.   In the Calculation program:

   - activate dynamic loads using the dynamic load Multipliers input window in the

     Multipliers tab sheet. An active button will appear for each load.

Unlike the way static loads are defined in Version 8 (using Staged construction) dynamic loads are defined by means of the dynamic Multipliers. These Multipliers operate as scaling factors on the inputted values of the dynamic loads (as entered in the Input program) to produce the actual load magnitudes. If a particular load system is set as a dynamic load, the load is initially kept active, but the corresponding load multiplier is set to zero in the Input program. In the Calculation program, it is specified how the (dynamic) load multiplier changes with time rather the input value of the load. The time dependent variation of the load multiplier acts on all loads in the corresponding load system.

2 TUTORIAL

This tutorial is intended to help users to become familiar with the features of the PLAXIS dynamics module. New users of PLAXIS are referred to the Tutorial section in the complete manual (PLAXIS Version 8). The lessons in this part of the manual deal with three specific dynamic applications of the program.

Generator on elastic foundation

- an axisymmetric model for single source vibrations
- dynamic soil-structure interaction
- standard absorbent boundaries

Pile driving

- plastic behaviour
- influence of water

Building subjected to an Earthquake

- a plane strain analysis for earthquake problems
- SMC file used for acceleration input
- standard earthquake boundaries

2.1 DYNAMIC ANALYSIS OF A GENERATOR ON AN ELASTIC FOUNDATION

Using PLAXIS, it is possible to simulate soil-structure interaction. Here the influence of a vibrating source over its surrounding soil is studied.

Due to the three dimensional nature of the problem, an axisymmetric model is used. The physical damping due to the viscous effects is taken into consideration via the Rayleigh damping. Also, due to axisymmetry 'geometric damping' can be significant in attenuating the vibration.

The modelling of the boundaries is one of the key points. In order to avoid spurious wave reflections at the model boundaries (which do not exist in reality), special conditions have to be applied in order to absorb waves reaching the boundaries.

2.1.1 INPUT

The vibrating source is a generator founded on a 0.2 m thick concrete footing of 1 m in diameter, Figure 2.1. Oscillations caused by the generator are transmitted through the footing into the subsoil. These oscillations are simulated as a uniform harmonic loading,

with a frequency of 10 Hz and amplitude of 10 kN/m^2. In addition to the weight of the footing, the weight of the generator is assumed 8 kN/m^2, modelled as a uniformly distributed load.
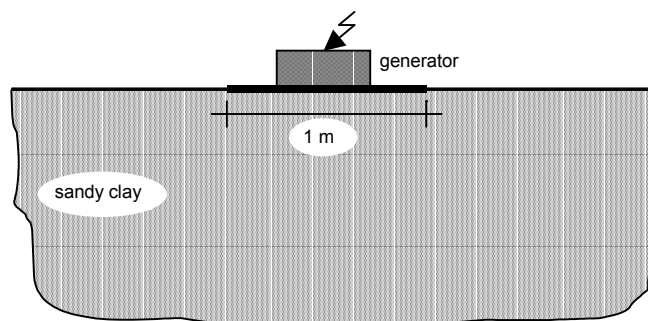
Figure 2.1 Generator founded on elastic subsoil.

Geometry model

The problem is simulated using an axisymmetric model with 15-noded elements. The geometry model is shown in Figure 2.2. Use [s] (seconds) as the unit of time, since dynamic effects are usually in the order of seconds rather than days.

The model boundaries should be sufficiently far from the region of interest, to avoid disturbances due to possible reflections. Although special measures are adopted in order to avoid spurious reflections (absorbent boundaries), there is always a small influence and it is still a good habit to put boundaries far away. In a dynamic analysis, model boundaries are generally taken further away than in a static analysis.

To set up the problem geometry, the following steps are necessary:

- Enter the geometry model as shown in Figure 2.2.
- Use plate elements to model the footing.
- Use Standard fixities.
- Apply a distributed load (system A) on the footing to model the weight of the generator.
- Apply a distributed load (system B) on the footing to model the dynamic load.
- In the *Loads* menu, set the *Dynamic load system* to load system B.

Absorbent boundaries

Special boundary conditions have to be defined to account for the fact that in reality the soil is a semi-infinite medium. Without these special boundary conditions the waves would be reflected on the model boundaries, causing perturbations. To avoid these

spurious reflections, absorbent boundaries are specified at the bottom and right hand side boundary.

To add the absorbent boundaries you can use the *Standard absorbent boundaries* option in the *Loads* menu. If necessary, the absorbent boundaries can be generated manually as:

1. Select the menu option *Absorbent boundaries* in the *Loads* menu.

2. Click on the lower left point of the geometry,

3. Proceed to the lower right point and click again,

4. Proceed to the upper right point and click again.

Only the right and bottom boundaries are absorbent boundaries. The left boundary is an axis of symmetry and the upper boundary is a free surface.
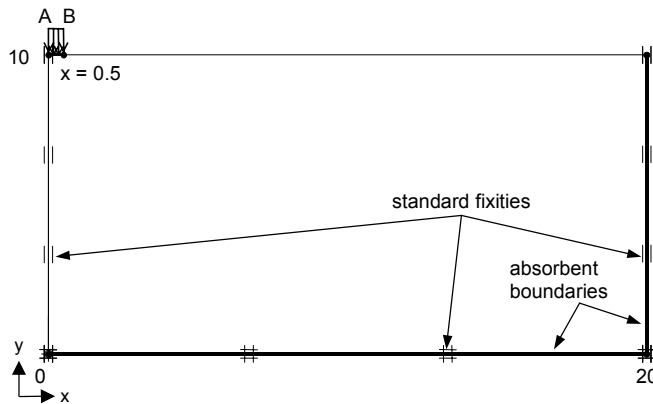
Figure 2.2 Generator model with absorbent boundaries

Material properties

The properties of the subsoil are given in Table 2.1. It consists of sandy clay, which is assumed to be elastic. The Young's modulus in Table 2.1 seems relatively high. This is because the dynamic stiffness of the ground is generally considerably larger than the static stiffness, since dynamic loadings are usually fast and cause very small strains. The unit weight suggests that the soil is saturated; however the presence of the groundwater is neglected.

> **Hint:**   When using Mohr-Coulomb or linear elastic models the wave velocities V_p and V_s are calculated from the elastic parameters and the soil weight. V_p and V_s can also be entered as input; the elastic parameters are then calculated automatically. See also Elastic parameters in Section 3.1.5 and the Wave Velocity relationships in Section 5.2.1.

Table 2.1 Material properties of the subsoil

Parameter	Name	Value	Unit
Material model	Model	Elastic	-
Type of material behaviour	Type	Drained	-
Unit weight	γ	20.0	kN/m^3
Young's modulus (constant)	E_{ref}	50000	kN/m^2
Poisson's ratio	ν	0.3	-

The footing has a weight of 5 kN/m^2 and is also assumed to be elastic. The properties are listed in Table 2.2.

Table 2.2 Material properties of the footing

Parameter	Name	Value	Unit
Normal stiffness	EA	$7.6 \cdot 10^6$	kN/m
Flexural rigidity	EI	24000	kNm^2/m
Weight	W	5.0	kN/m/m
Poisson's ratio	ν	0.0	-

Mesh generation

Because of the expected high concentration of stresses in the area below the footing, a local refinement is proposed there. The mesh is generated with the global coarseness set to 'coarse' and then the line of the footing is refined two times. The result is plotted in Figure 2.3.
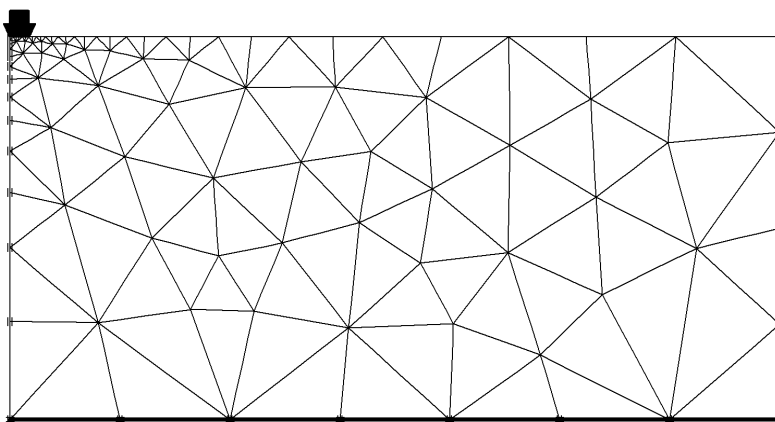

Figure 2.3 Geometry and mesh

2.1.2   INITIAL CONDITIONS

Water pressures:
Since water is not considered in this example, the generation of water pressures can be skipped.

Initial stresses:
The initial stresses are generated by means of the K_0 procedure, using a K_0 value of 0.5. In the initial situation, the footing and the static load do not exist and therefore they are deactivated. The dynamic load seems active but the corresponding multiplier is automatically set to zero.

2.1.3   CALCULATIONS

There are three calculation phases. In the first phase the footing is built and the static load (weight of the generator) is applied. The second phase is the situation when the generator is running. In the third phase the generator is turned off and the soil is let to vibrate freely. The last two phases involve dynamic calculations.

Phase 1:
1. Select *Plastic calculation* in the General tab sheet.

2. Select *Staged construction* in the Parameter tab sheet and click on the Define button

3. Click on the plate element and select all objects from the *Select items* window. By using the Change option, the static load (system A) is set to 8 kN/m^2. Note that this value can also be set in the Input program (see the main Reference manual).

Phase 2:
In this phase, a vertical harmonic load, with a frequency of 10 Hz and amplitude of 10 kN/m^2, is applied to simulate the vibrations transmitted by the generator. Five cycles with a time interval of 0.5 sec are considered.

1. Select *Dynamic analysis* in the General tab sheet.

2. Use 100 for the number of *Additional steps*.

3. Reset displacement to zero.

4. Set *Time interval* to 0.5 s.

5. Select Total multipliers and click on the Define button.

6. Click on next to *ΣMloadB* in the Multipliers tab sheet to proceed with the definition of the dynamic load.

7. Click the option *Harmonic load multiplier* in the window *Dynamic loading-Load System B*.

8. Set the *Amplitude multiplier* to 10, frequency to 10 Hz and *Initial phase angle* to 0 (See Figure 2.4).
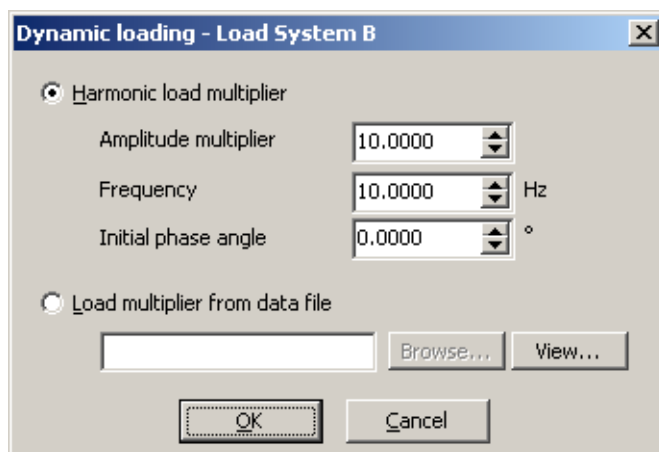
Figure 2.4 Harmonic load

Phase 3:

In this phase, the generator is turned off. The soil is vibrating freely after the initial excitation.

1. Select *Dynamic analysis* in the General tab sheet.

2. Use 100 for the number of *Additional steps*.

3. Set *Time interval* to 0.5 s. The estimated end time is 1 sec.

4. Select Total multipliers and click on the Define button.

5. Click on next to Σ*MloadB* in the Multipliers tab sheet. Set all parameters to zero in the Dynamic loading window.

Before running the calculation, select points at the surface at about 1.40m, 1.9m and 3.6m. They will be used by the Curves program to visualise the deformation as a function of time. You can now start the calculation.

Additional calculation with damping:

In a second calculation, material damping is introduced by means of Rayleigh damping. Rayleigh damping can be entered in the material data set.

The following steps are necessary:

• Start the Input program and select the generator project.

• Save the project under another name.

• Open the material data set of the soil. In the General tab sheet click on the *Advanced* button.

• Change the Rayleigh damping parameters α and β to 0.001 and 0.01 respectively, Fig. 2.5.

• Close the data base, proceed to Initial conditions and save the project.

• In the Calculations program, check whether the phases are still properly defined (according to the information given before).

Figure 2.5 Input of Rayleigh damping

2.1.4   OUTPUT

The Curves program is particularly useful for dynamic analysis. You can easily display the actual loading versus time (input) and also displacements, velocities and accelerations of the pre-selected points versus time. Figure 2.6 shows the evolution of the applied load with time, as defined by the calculation phases 2 and 3.

Load system B
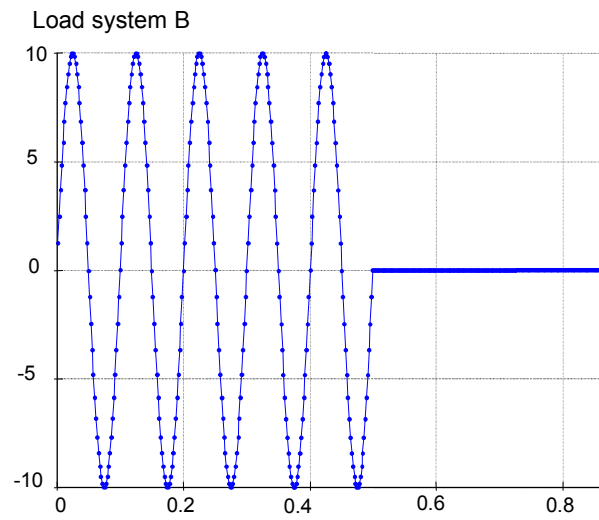
Figure 2.6 Load – time curve

Figure 2.7 shows the response of the pre-selected points at the surface of the structure. It can be seen that even with no damping, the waves are dissipated which can be attributed to the geometric damping.
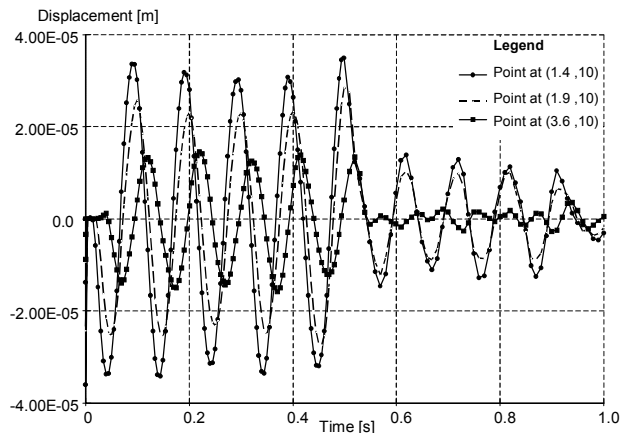
Figure 2.7   Displ.- time on the surface at different distances to the vibrating source. Without damping (Rayleigh $\alpha = 0$; $\beta = 0$).

The presence of damping is clear in Figure 2.8. It can be seen that the vibration is totally seized after the removal of the force (after t = 0.5 s). Also, the displacement amplitudes are lower. Compare Figure 2.7 (without damping) with Figure 2.8 (with damping).
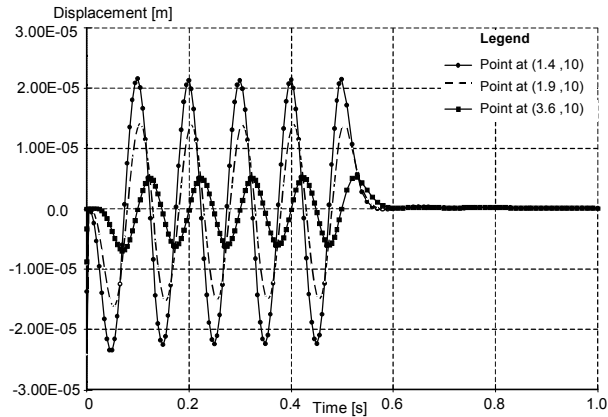
Figure 2.8 Displ.- time. With damping (Rayleigh $\alpha = 0.001$; $\beta = 0.01$).

It is possible in the Output program to display displacements, velocities and accelerations at a particular time, by choosing the appropriate option in the *Deformations* menu. Figure 2.9 shows the total accelerations in the soil at the end of phase 2 ($t = 0.5$ s).
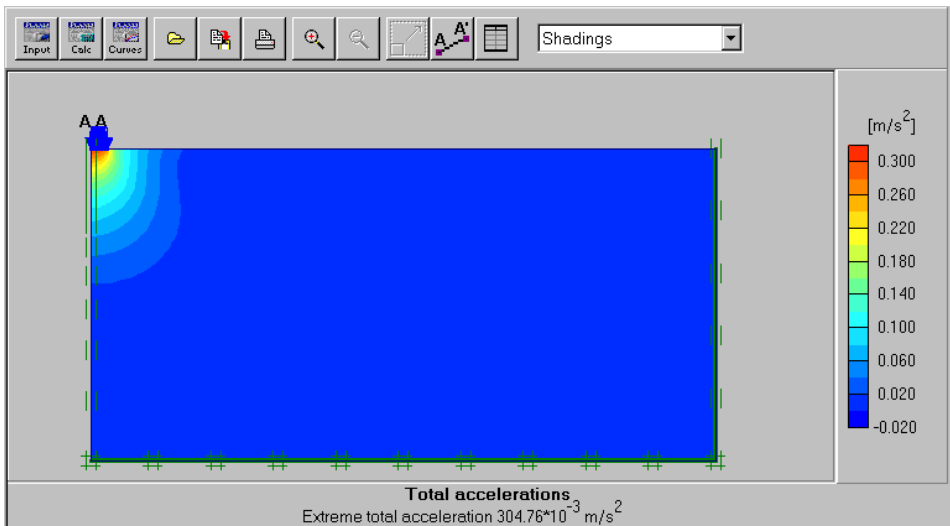
Figure 2.9 Total accelerations in the soil at the end of phase 3.

2.2   PILE DRIVING

This example involves driving a concrete pile through an 11 m thick clay layer into a sand layer, Figure 2.10. The pile has a diameter of 0.4 m. Pile driving is a dynamic process that causes vibrations in the surrounding soil. Moreover, excess pore pressures are generated due to the quick stress increase around the pile.

In this example focus is placed on the irreversible deformations below the pile. In order to simulate this process most realistically, the behaviour of the sand layer is modelled by means of the Hardening Soil model.
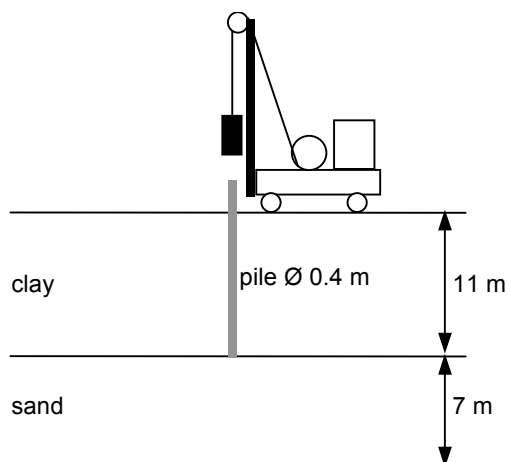
Figure 2.10 Pile driving situation

Geometry model

The geometry is simulated by means of an axisymmetric model in which the pile is positioned along the axis of symmetry (see Figure 2.11). In the general settings, the standard gravity acceleration is used (9.8 m/s^2). The unit of time should be set to seconds [s].

Both the soil and the pile are modelled with 15-noded elements. The subsoil is divided into an 11 m thick clay layer and a 7 m thick sand layer. Interface elements are placed around the pile to model the interaction between the pile and the soil. The interface should be extended to about half a meter into the sand layer (see Figure 2.12). A proper modelling of the pile-soil interaction is important to include the material damping caused by the sliding of the soil along the pile during penetration and to allow for sufficient flexibility around the pile tip. Use the zoom option to create the pile and the interface.
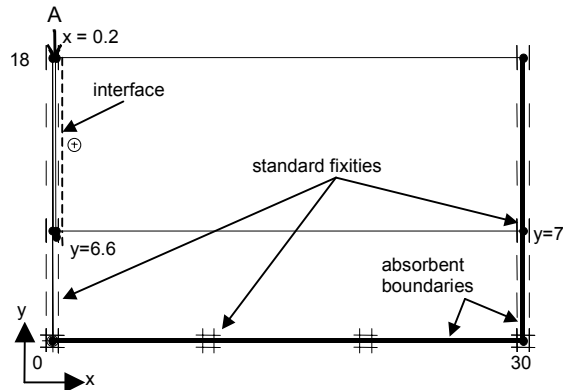
Figure 2.11 Geometry model of pile driving problem

The boundaries of the model are taken sufficiently far away to avoid direct influence of the boundary conditions. Standard absorbent boundaries are used at the bottom and at the right hand boundary to avoid spurious reflections.

In order to model the driving force, a distributed unit load (system A) is created on top of the pile.
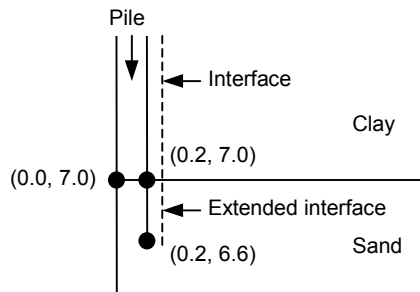
Figure 2.12 Extended Interface

Material properties

The clay layer is modelled with the simple Mohr-Coulomb model. The behaviour is considered to be undrained. An interface strength reduction factor is used to simulate the reduced friction along the pile shaft.

In order to model the non-linear deformations below the tip of the pile in a right way, the sand layer is modelled by means of the Hardening Soil model. Because of the fast loading process, the sand layer is also considered to behave undrained. The short interface in the sand layer does not represent soil-structure interaction. As a result, the interface strength reduction factor should be taken equal to unity (rigid).

The pile is made of concrete, which is modelled by means of the linear elastic model considering non-porous behaviour. In the beginning, the pile is not present, so initially the clay properties are also assigned to the pile cluster. The parameters of the two layers and the concrete pile are listed in Table 2.3.

Table 2.3 Material properties of the subsoil and pile

Parameter	Symbol	Clay	Sand	Pile	Unit
Material model	Model	Mohr-C.	Hardening-S	Linear elast.	-
Type of behaviour	Type	Undrained	Undrained	Non-porous	-
Unit weight above phreatic line	γ_{unsat}	16	17	24	kN/m^3
Unit weight below phreatic line	γ_{sat}	18	20	-	kN/m^3
Young's modulus	E_{ref}	15000	50000	$3 \cdot 10^7$	kN/m^2
Oedometer modulus	E_{oed}	-	50000	-	kN/m^2
Power	m	-	0.5	-	-
Unloading modulus	E_{ur}	-	150000	-	kN/m^2
Poisson's ratio	ν	0.3	0.2	0.1	-
Reference stress	P_{ref}	-	100	-	kN/m^2
Cohesion	c	2	1	-	kN/m^2
Friction angle	φ	24	31	-	$^\circ$
Dilatancy angle	ψ	0	0	-	$^\circ$
Interface strength reduction	R_{inter}	0.5	1.0 (rigid)	1.0 (rigid)	-

- It should be noted that there is a remarkable difference in wave velocities between the clay layer and the concrete pile due to the large stiffness difference. This may lead to small time increments (many sub steps) in the automatic time stepping procedure. This causes the calculation process to be very time consuming. Many sub steps may also be caused by a very small (local) element size. In such situations it is not always vital to follow the automatic time stepping criterion. You can reduce the number of sub steps in the *Manual setting* of the Iterative procedure (Section 3.2.3).

- When the Hardening Soil model is used wave velocities are not shown because they vary due to the stress-dependent stiffness.

Mesh generation

The mesh is generated with a global coarseness set to *coarse* (default). A local refinement is made in the pile cluster. The result of the mesh generation is plotted in Figure 2.13.
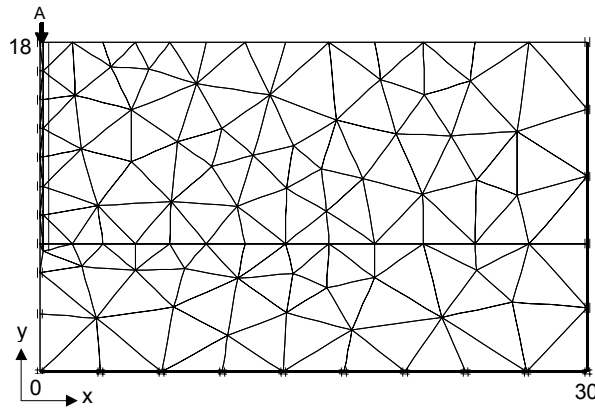
Figure 2.13 Finite element mesh for pile driving problem.

2.2.1   INITIAL CONDITIONS

Water pressures:

The phreatic level is assumed to be at the ground surface. Hydrostatic pore pressures are generated in the whole geometry according to this phreatic line.

Initial stresses:

Initial effective stresses are generated by the K_0 procedure, using the default values. Note that in the initial situation the pile does not exist and that the clay properties should be assigned to the corresponding clusters.

2.2.2   CALCULATIONS

The analysis consists of three calculation phases. In the first phase the pile is created. In the second phase the pile is subjected to a single stroke, which is simulated by activating half a harmonic cycle of load system A. In the third phase the load is kept zero and the dynamic response of the pile and soil is analysed in time. The last two phases involve dynamic calculations.

Phase 1:

1.   Select *Plastic calculation* in the general tab sheet.

2.   Select *Staged construction* in the parameter tab sheet.

3.   Assign the pile properties to the pile cluster.

Phase 2:

1. Select *Dynamic analysis* in the general tab sheet.

2. Use standard *Additional steps* (250).

3. Reset displacements to zero.

4. Enter 0.01 s for the *Time interval*.

5. Select *Manual setting* for the iterative procedure and click *Define*. The initial number of *Dynamic sub steps* is relatively large, due to the large difference in wave speeds and the small element sizes (see earlier remark on material properties). Set the number of *Dynamic sub steps* to 1. All other settings remain at their default.

6. Click ∿ next to *Load system A* in the multiplier tab sheet to apply the dynamic loading. Enter the values as indicated in Figure 2.14.
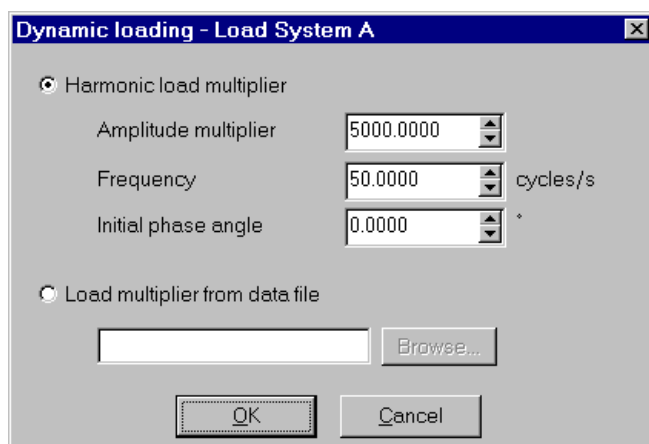
Figure 2.14 Dynamic loading parameters

The result of this phase is half a harmonic cycle of the external load in system A. At the end of this phase, the load is back to zero.

Phase 3:

1. Select *Dynamic analysis* in the general tab sheet.

2. Use standard Additional steps (250).

3. Enter a *Time interval* of 0.19 s.

4. Select the *Manual setting* for the *Iterative procedure* and click *Define*. Set the number of *Dynamic sub steps* to 1. This results in equal time steps in phase 2 and 3.

5. In the Multiplier tab sheet, all multipliers remain at their default values.

6. Click ⎍ next to Load system A and set all parameters in the *Dynamic Loading* window to zero.

7. Select a node at the top of the pile for load displacement curves.

2.2.3   OUTPUT

Figure 2.15 shows the settlement of the pile (top point) versus time. From this figure the following observations can be made:

- The maximum vertical settlement of the pile top due to this single stroke is about 24 mm. However, the final settlement is almost 22 mm.

- Most of the settlement occurs in phase 3 after the stroke has ended. This is due to the fact that the compression wave is still propagating downwards in the pile, causing additional settlements.

- Despite the absence of Rayleigh damping, the vibration of the pile is damped due to soil plasticity and the fact that wave energy is absorbed at the model boundaries.
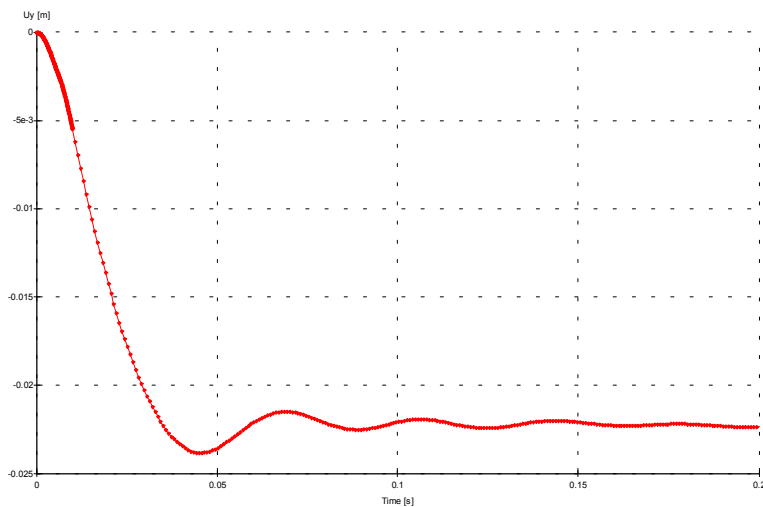
Figure 2.15 Pile settlement vs. time.

When looking at the output of the second calculation phase ($t = 0.01$ s, i.e. just after the stroke), it can be seen that large excess pore pressures occur very locally around the pile tip. This reduces the shear strength of the soil and contributes to the penetration of the pile into the sand layer. The excess pore pressures remain also in the third phase since consolidation is not considered.

Figure 2.16 shows the shear stresses in the interface elements at $t = 0.01$ s. This plot is obtained by using a manual scaling factor of 10 and zooming into the small area of stresses along the pile. The plot shows that the maximum shear stress is reached all along the pile, which indicates that the soil is sliding along the pile.
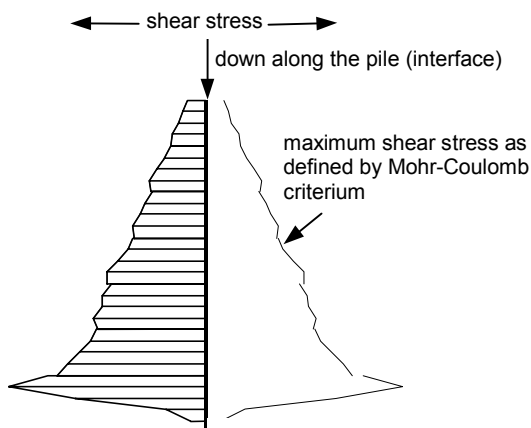
Figure 2.16 Shear stresses in the interface at t = 0.01 s.

When looking at the deformed mesh of the last calculation phase (t = 0.2 s), it can also be seen that the final settlement of the pile is about 22 mm. In order to see the whole dynamic process it is suggested to use the option *Create Animation* to view a 'movie' of the deformed mesh in time. You may notice that the first part of the animation is slower than the second part.

2.3   BUILDING SUBJECTED TO AN EARTHQUAKE

This example demonstrates the behaviour of a four-storey building when subjected to an earthquake. A real accelerogram of an earthquake recorded by USGS in 1989 is used for the analysis. It is contained in the standard SMC format (Strong Motion CD-ROM), which can be read and interpreted by PLAXIS (Section 3.2.7).

Input
The building consists of 4 floors and a basement. It is 6 m wide and 25 m high. The total height from the ground level is 4 x 3 m = 12 m and the basement is 2 m deep. The dead load and a percentage of the live load acting on each floor are added up, mounting up to 5 kN/m^2. This value is taken as the weight of the floors and the walls.

Geometry model

The length of the building is much larger than its width. The earthquake is also supposed to have a dominant effect across the width of the building. Hence, a plane strain analysis can be performed. 15-noded elements are used to simulate the situation. The unit of time is set to seconds [s]. The other dimensions are left to their defaults (length: [m], force: [kN]).

The subsoil consists of a relatively soft soil layer of 20 m thickness, overlaying a rock formation. The latter is not included in the model. The building itself is composed of 5-noded plate elements. The clusters of the building are filled with soil material when generating the mesh, but these clusters are deactivated in the initial situation.

The vertical boundaries are taken relatively far away from the building. Physical damping in the building and the subsoil is simulated by means of Rayleigh damping (Section 3.1.5).

The earthquake is modelled by imposing a prescribed displacement at the bottom boundary. In contrast to the standard unit of length used in PLAXIS [m], the displacement unit in the SMC format is [cm]. Therefore the input value of the horizontal prescribed displacements is set to 0.01 m. (When using [ft] as unit of length, this value should be 0.0328 ft). The vertical component of the prescribed displacement is kept zero (u_x=0.01m and u_y=0.00m). At the far vertical boundaries, absorbent boundary conditions are applied to absorb outgoing waves. PLAXIS has a convenient default setting to generate standard boundary conditions for earthquake loading using SMC-files (*Standard earthquake boundaries*). This option can be selected from the *Loads* menu. In this way the boundary conditions as described above are automatically generated (see Figure 2.17).
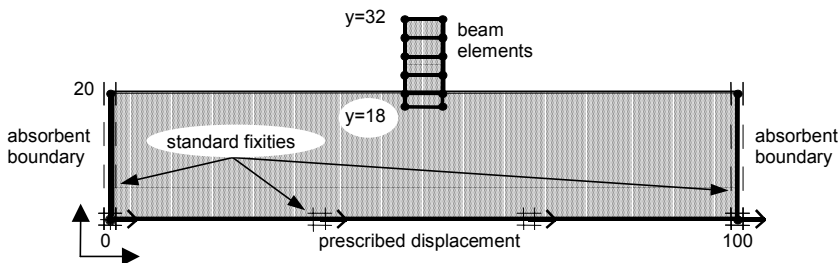
Figure 2.17 Geometry model with standard earthquake boundary conditions.

Material properties

The properties of the subsoil are given in Table 2.4. The soil consists of loam, which is assumed to be linear elastic. The stiffness is higher than one would use in a static analysis, since dynamic loadings are usually fast and cause very small strains. The presence of the groundwater is neglected. The building is also considered to be linear

elastic. The walls and the floors have similar plate properties, which are listed in Table 2.5.

Table 2.4 Material properties of the subsoil

Parameter	Name	Value	Unit
Material model	Model	Elastic	-
Type of material behaviour	Type	Drained	-
Unit soil weight	γ	17.0	kN/m^3
Young's modulus (constant)	E_{ref}	30000	kN/m^2
Poisson's ratio	ν	0.2	-
Rayleigh damping	α and β	0.01	

Considering a gravity acceleration of 9.8 m/s^2, the above properties are equivalent to a shear wave velocity of about 85 m/s and a compression wave velocity of about 140 m/s.

Table 2.5 Material properties of the building (plate properties).

Parameter	Name	Floors / Walls	Unit
Material model	Model	Elastic	-
Normal stiffness	EA	$5 \cdot 10^6$	kN/m
Flexural rigidity	EI	9000	kNm^2/m
Weight	w	5.0	kN/m/m
Poisson's ratio	ν	0.0	-
Rayleigh damping	α and β	0.01	

Mesh generation
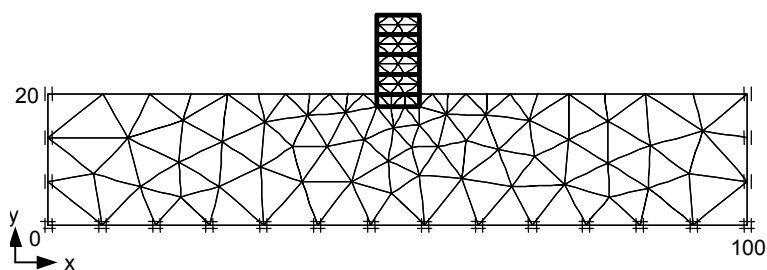
Figure 2.18 Finite element mesh.

For the mesh generation, the global coarseness is set to 'coarse' and the clusters inside the building are refined once. This is because of the high concentration of stresses that can be expected just in and under the building elements (See Figure 2.18).

2.3.1 INITIAL CONDITIONS

Water pressures:

The generation of water pressures can be skipped, since pore pressures are not considered in this example.

Initial stresses:

In the initial situation the building is not considered; therefore the plates and the clusters above the ground surface must be deactivated. The initial state of stresses is generated by the K_0-procedure, with a K_0-value of 0.5 for all active clusters.

2.3.2 CALCULATIONS

The calculation involves two phases. The first one is a normal plastic calculation in which the building is constructed. The second is a dynamic analysis in which the earthquake is simulated. To analyse the effects of the earthquake in detail the displacements are reset to zero at the beginning of this phase.

Phase 1:

1. Select *Plastic calculation* in the general tab sheet.

2. Select *Staged construction* as the loading input in the parameter tab-sheet and click *Define*.

3. Activate the plates of the building and de-activate the soil cluster in the basement. All clusters inside the building should now be inactive.

Figure 2.19 Construction of the building

Phase 2:

1. Select *Dynamic analysis* for the calculation type in the General tab sheet.

2. Set the number of *Additional steps* to 250 in the Parameters tab-sheet.

3. Select *Reset the displacements to zero*.

4. Set the *Time interval* to 10 sec in the loading input box.

5. Select *Manual setting* for the Iterative procedure. Set *Dynamic sub step* to 1.

6. Click *Define*.

7. Select the Multiplier tab sheet.

8. Click 〰️ next to the Σ-*Mdisp* multiplier.

9. Select the option *Load multiplier from data file*.
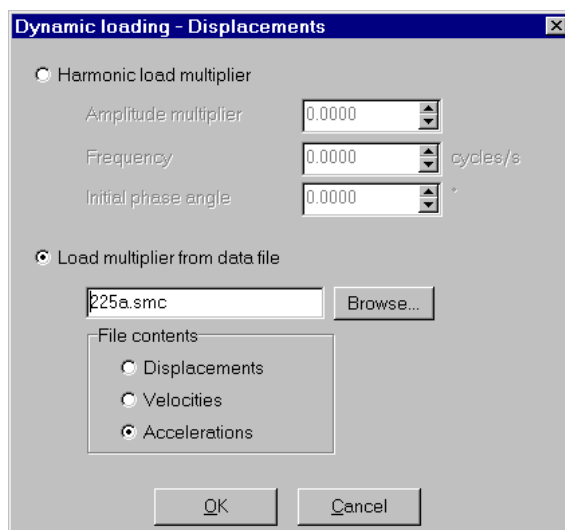
Figure 2.20 Selection of SMC file.

10. Select the appropriate SMC file (225A.smc). This file can be found in the PLAXIS program directory. The data provided in this file is an accelerogram, thus the option *Acceleration* has to be selected from the *File contents* box (See Figure 2.20).

11. Click OK.

12. Select points for load displacement curves at the top of the building, at the bottom of the basement and at the bottom of the mesh. You may now start the calculation.

Hint:	Plaxis assumes the data file is located in the current project directory when no directory is specified in the *Dynamic loading* window.
>	In the SMC files, data is given for each 0.005 s (200 values per second). The calculation step size does not correspond with the data given in the file, but the calculation program will interpolate a proper value for the actual time of each step.

2.3.3   OUTPUT

The maximum horizontal displacement at the top of the building is 75 mm and occurs at t=4.8 s. Figure 2.21 shows the deformed mesh at that time. The output program also provides data on velocities and accelerations. The maximum horizontal acceleration at the top of the building is 3.44 m/s^2 (0.34 G) and occurs at t=2.88 s (see Figure 2.22).
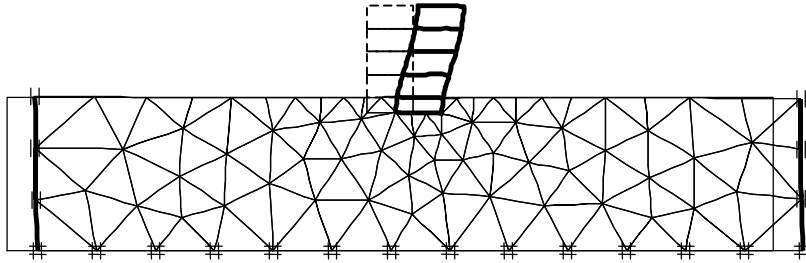
Figure 2.21 Deformed mesh at at t = 4.80 s (step 121). Maximum horizontal deformation at the top: u_x = 75 mm
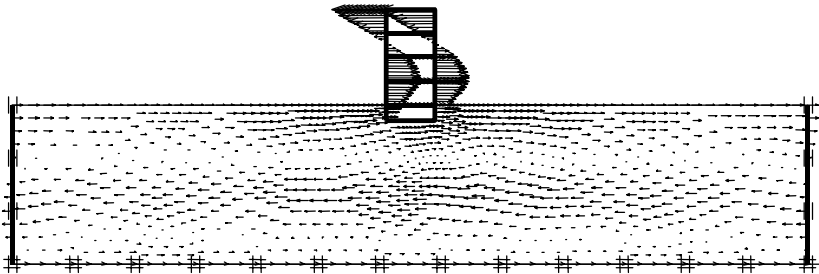
Figure 2.22 Horizontal accelerations at t = 2.88 s (step 73). Maximum value at the top: a_x = 3.44 m/s^2

Figure 2.23 and Figure 2.24 show, respectively, time-displacement curves and time-acceleration curves at the bottom of the mesh, the basement and the top of the building. From Figure 2.24 it can be seen that the maximum accelerations at the top of the building are much larger than the accelerations from the earthquake itself.
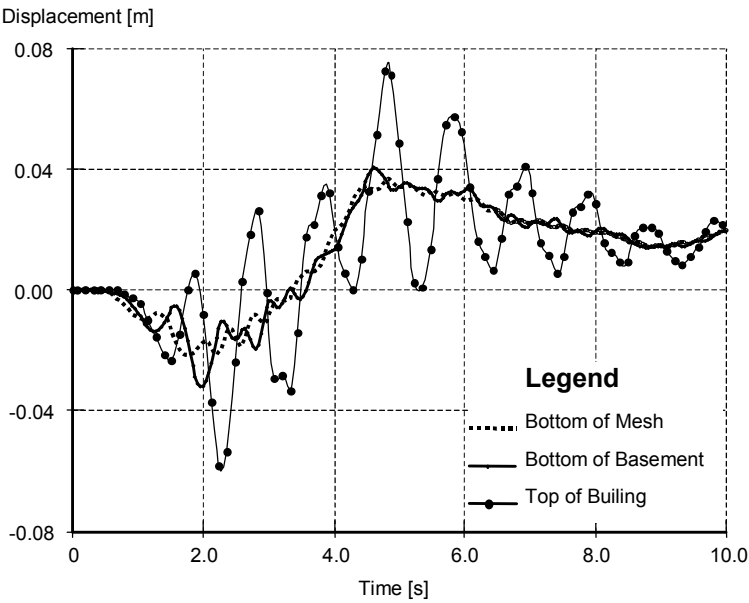
Displacement [m]

Figure 2.23 Time-displacement curve for the mesh bottom, basement and top of building
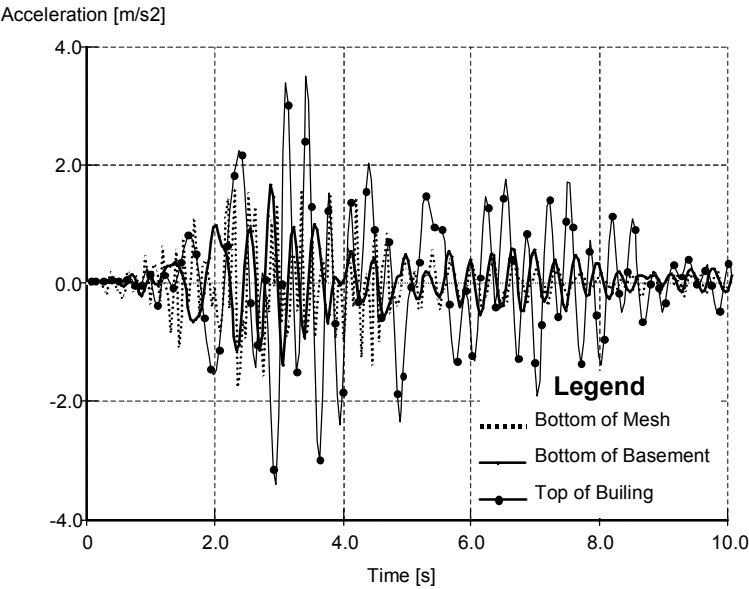
Acceleration [m/s2]

Figure 2.24 Time-acceleration curve for the mesh bottom, basement and top of building

3   REFERENCE

The procedure to perform a dynamic analysis with PLAXIS is somehow similar to that for a static analysis. This entails creation of a geometry model, mesh generation, initial stress generation, defining and executing calculation, and evaluation of results. For the general explanation of the basic functionality of PLAXIS you are advised to read the tutorial and reference sections in the complete manual (PLAXIS manual Version 8).

In this part of the manual the functionality of the Dynamics module is highlighted. The working order for a dynamic analysis in PLAXIS is followed in the explanation.

Input:
- General settings
- Loads and Boundary conditions
- Prescribed displacements
- Setting dynamic load system
- Absorbent boundaries
- Elastic parameters
- Material damping

Calculations:
- Selecting dynamic analysis
- Parameters
- Multipliers

Output:
- Animations
- Velocities
- Accelerations

Curves:
- Velocities
- Accelerations
- Input and response spectra

3.1 INPUT

The following topics will be discussed in this chapter:

- General settings

- Loads and Boundary conditions

- Prescribed displacements

- Absorbent boundaries

- Elastic parameters

3.1.1 GENERAL SETTINGS

In the general settings of a new project you can define the basic conditions for the dynamic analyses you want to perform. Dynamic analyses in PLAXIS can mainly be divided into two types of problems:

- Single-source vibrations

- Earthquake problems

Single-source vibrations

Single-source vibration problems are often modelled with axisymmetric models, even if one would normally use a plane strain model for static deformations. This is because waves in an axisymmetric system radiate in a manner similar to that in a three dimensional system. In this the energy disperses leading to wave attenuations with distance. Such effect can be attributed to the geometric damping, which is by definition included in the axisymmetric model. In single-source problems, geometric damping is generally the most important contribution to the damping of the system. Hence, for problems of single source vibrations it is essential to use an axisymmetric model.

Earthquake problems

In earthquake problems the dynamic loading source is usually applied along the bottom of the model resulting to shear waves that propagate upwards. This type of problems is generally simulated using a plane strain model. Note that a plane strain model does not include the geometric damping. Hence it may be necessary to include material damping to obtain realistic results. (Section 3.1.5)

Gravity acceleration

By default, the gravity acceleration, *g,* is set to 9.8 m/s^2. This value is used to calculate the material density, ρ [kg/m^3], from the unit weight, γ ($\rho = \gamma / g$).

Units

In a dynamic analysis the time is usually set to [seconds] rather than the default unit [days]. Hence, for dynamic analysis the unit of time should generally be changed in the general settings window. However, this is not strictly necessary since in PLAXIS *Time* and *Dynamic Time* are different parameters. The time interval in a dynamic analysis is always the dynamic time and PLAXIS always uses seconds [s] as the unit of *Dynamic Time*. In the case where a dynamic analysis and a consolidation analysis are involved, the unit of *Time* can be left as [days] whereas the *Dynamic Time* is in seconds [s].

3.1.2   LOADS AND BOUNDARY CONDITIONS

After creating the geometry model for the dynamic analysis, the loads and boundary conditions have to be applied. The *Loads* menu contains options to introduce special conditions for a dynamic analysis:

- Absorbent boundaries

- Prescribed displacements

- External loads (Distributed and Point loads)

- Set dynamic load system

- Standard boundary conditions

The standard fixities, as used for a static problem, can also be used for dynamic calculations.

3.1.3   ABSORBENT BOUNDARIES

An absorbent boundary is aimed to absorb the increments of stresses on the boundaries caused by dynamic loading, that otherwise would be reflected inside the soil body.

In PLAXIS you can generate absorbent boundaries simply by selecting the *Standard Absorbent boundaries* from *Loads* menu. For manual setting, however, the input of an absorbent boundary is similar to the input of fixities (See Section 3.4.2 of the general PLAXIS Reference Manual Version 8).

Standard absorbent boundaries

For single-source vibrations, PLAXIS has a default setting for generating appropriate absorbent boundaries. This option can be selected from the *Loads* menu. For plane strain models, the standard absorbent boundaries are generated at the left-hand, the right-hand and the bottom boundary. For axisymmetric models, the standard absorbent boundaries are only generated at the bottom and the right-hand boundaries.

Standard earthquake boundaries

PLAXIS has a convenient default setting to generate standard boundary conditions for earthquake loading. This option can be selected from the *Loads* menu. On selecting *Standard earthquake boundaries*, the program will automatically generate absorbent boundaries at the left-hand and right-hand vertical boundaries and prescribed displacements with u_x = 0.01 m and u_y = 0.00 m at the bottom boundary (see also below).

3.1.4   EXTERNAL LOADS AND PRESCRIBED DISPLACEMENTS

In PLAXIS Version 8 the input of a dynamic load is similar to that of a static load (Section 3.4 of the general Reference Manual). Here, the standard external load options (point loads and distributed loads in system A and B and prescribed displacements) can be used.

In the Input program, the user must specify which of the load system(s) will be used as a dynamic load. This can be done using the *Set dynamic load system* option in the *Loads* menu. Load systems that are set as dynamic, cannot be used for static loading. Load systems that are not set as dynamic, are considered to be static.

In the Calculation program, the dynamic loads are treated in a different way than for the static loads. The input value of a dynamic load is usually set to a unit value, whereas dynamic multipliers in the Calculation program are used to scale the loads to their actual magnitudes (Section 3.2.5). The applied load is the product of the input value and the corresponding load multiplier. This principle is valid for both static and dynamic loads. However, static loads are applied in Staged Construction by activating the load or changing the input value (whilst the corresponding load multiplier is usually equal to 1), whereas dynamic loads are applied in the Dynamic Load Multiplier input window by specifying the variation of the corresponding load multiplier with time (whilst the input value of the load is a unit value and it is active).

The procedure to apply dynamic loads is summarised below:

1. Create loads in the Input program (point loads, distributed loads in load system A or B, and/or prescribed displacements).

2. Set the appropriate load system as a dynamic load system in the *Loads* menu of the Input program.

3. Activate the dynamic loads by entering the dynamic load multipliers in the Dynamic load multipliers input window of the Calculation program.

Earthquakes

A special method for introducing dynamic loads in a model is by means of prescribed displacements (see section 3.4.1 of the general Reference manual). Earthquakes are usually modelled by means of prescribed horizontal displacements. When the *Standard earthquake boundaries* in the *Loads* menu is selected the horizontal displacement component is defined automatically by PLAXIS.

To define the prescribed displacement for an earthquake manually:

1. Enter a prescribed displacement in the geometry model (usually at the bottom).

2. Double click on the prescribed displacement.

3. Select *Prescribed displacement* in the window that appears.

4. Change the *x*-values of both geometry points to 1 (or 0.01 when working with standard SMC files with unit of length in meters, or 0.0328 with unit of length in feet) and the *y*-values to 0. Now the given displacement is one unit in horizontal direction.

5. In the *Loads* menu, set the dynamic load system to Prescribed displacements.

Scaling a displacement/load:

PLAXIS allows the use of earthquake records in SMC-format as input data for earthquake loading. The SMC-files use centimetres as unit of length. If you want to use these files, you have to have corresponding input values in PLAXIS.

In general, to be able to use SMC files in combination with a certain unit of length [length] in your PLAXIS project, the input value of the horizontal prescribed displacement should be scaled by *1 / [unit of length used in PLAXIS in cm]*

*   When the unit of length in PLAXIS is set to meters [m], you have to scale the prescribed displacement by changing $u_x = 1$ into $u_x = 0.01$.

*   When the unit of length is feet [ft] you have to scale the input value by $1/30.48 = 0.0328$.

3.1.5   MODEL PARAMETERS

A dynamic analysis does not, in principle, require additional model parameters. However, alternative and/or additional parameters may be used to define wave velocities and to include material damping.

Wave velocities V_p and V_s

The material parameters are defined in the Parameter tab sheet of the *Material properties* window. When entering the elastic parameters E and v, the corresponding wave velocities V_p and V_s are automatically generated, provided that a proper unit weight has been specified. However, for Mohr-Coulomb and linear elastic models you can enter the wave velocities V_p en V_s as an alternative for the parameters E and v. The corresponding values for E and v will then be generated by PLAXIS. (See the relationships of wave velocities in Section 5.2.1.)

Figure 3.1 Elastic parameters in Mohr-Coulomb and linear elastic models

Rayleigh alpha and beta

Material damping in a soil is generally caused by its viscous properties, friction and the development of plasticity. However, in PLAXIS the soil models do not include viscosity as such. Instead, a damping term is assumed, which is proportional to the mass and stiffness of the system (Rayleigh damping), such that:

$$C = \alpha M + \beta K$$

where C represents the damping, M the mass, K the stiffness and α (alpha) and β (beta) are the Rayleigh coefficients. In contrast to PLAXIS Version 7, where α and β were global parameters, in Version 8 the Rayleigh damping is considered to be object-dependent and therefore α and β are included in the material data sets. Hence, distinction can be made in different soil layers as well as structural objects composed of volume clusters or plates.

The standard settings in PLAXIS assume no Rayleigh damping (Rayleigh alpha and Rayleigh beta equal to 0.0) (see Figure 3.2a). However, damping can be introduced in the Material data sets for soils and interfaces. In the General tab sheet of the material window, click on Advanced. Then in the Advanced General Properties window set values for Rayleigh alpha and/or Rayleigh beta. Similarly, Rayleigh damping can be included in the material data sets of plates (see Figure 3.2b).

In single-source type of problems using an axisymmetric model, it may not be necessary to include Rayleigh damping, since most of the damping is caused by the radial

spreading of the waves (geometric damping). However, in plane strain models, such as for earthquake problems, Rayleigh damping may be required to obtain realistic results.
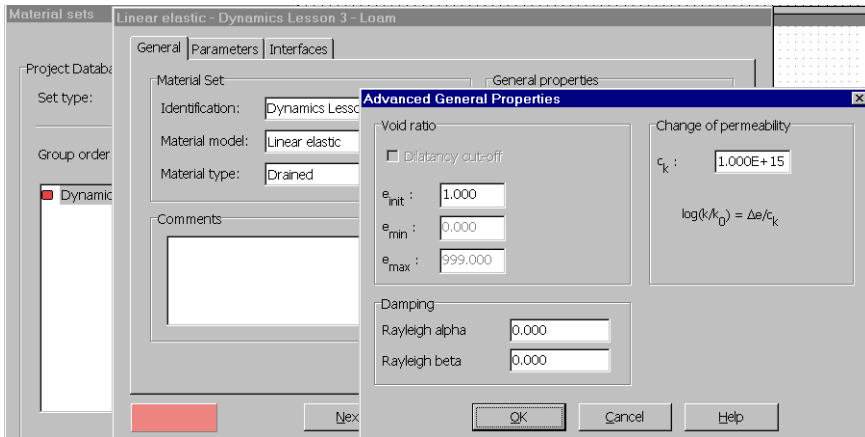
(a)
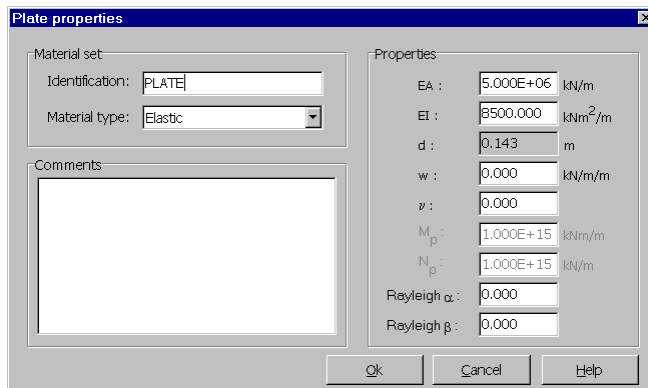
(b)

Figure 3.2 Setting values for the Rayleigh damping (a) for soil and (b) for plates

Determination of the Rayleigh damping coefficients

It is well known fact that damping in a soil structure influences significantly the magnitude and shape of its response. However and despite the considerable amount of research work in this field, little have yet been achieved for the development of a commonly accepted procedure for damping parameter identification. Instead, for engineering purposes, some measures are made to account for the material and geometrical damping. A commonly used engineering parameter is the damping ratio ξ .

In finite element method, the Rayleigh damping constitutes one of the convenient measures of damping that lumps the damping effects within the mass and stiffness matrices of the system. Rayleigh alpha is the parameter that determines the influence of the mass in the damping of the system. The higher the alpha value, the more the lower frequencies are damped. Rayleigh beta is the parameter that determines the influence of the stiffness in the damping of the system. The higher the beta value, the more the higher frequencies are damped.

The Rayleigh damping coefficients α and β can be determined from at least two given damping ratios ξ_i that correspond to two frequencies of vibration ω_i. The relationship between α, β, ξ_i and ω_i can be presented as

$$\alpha + \beta\,\omega_i^2 = 2\,\omega_i\,\xi_i$$

This relation entails that if two damping ratios at given frequencies are known, two simultaneous equations can be formed, from which α and β can be calculated. For example, assume that at $\omega_1 = 3$ rad/s and $\omega_2 = 5$ rad/s the damping ratios were $\xi_1 = 0.01$ and $\xi_2 = 0.05$ respectively. Substituting these parameters into the above equation results to

$$\alpha + 9\,\beta = 0.06$$

$$\alpha + 25\,\beta = 0.5$$

The solution of these two equations is $\alpha = -0.187$ and $\beta = 0.027$. If, however, more than two pairs of data are available, average quantities must be made to produce two equations (see Bathe, 1996, [9]).

The damping ratios can be obtained experimentally by means of the resonant column test. For this see Das, 1995, [6].

3.2   CALCULATIONS

In the Calculations program you can define the dynamic loads by activating displacements and loads as a function of time by their corresponding multipliers.

For general information on parameters and multipliers you are referred to the general Reference manual Section 4.8. In this part of the manual you can find information on the following:

- Selecting dynamic analysis

- Dynamic parameters:
  Time stepping
  Newmark parameters

- Dynamic loading:
  Harmonic loading
  Load multipliers from file
  Block loads

3.2.1 SELECTING DYNAMIC ANALYSIS

A dynamic calculation can be defined by selecting *Dynamic analysis* in the Calculation type box of the General tab sheet. With PLAXIS it is possible to perform a dynamic analysis after a series of plastic calculations. However, there are some restrictions:

- It is not possible to use updated mesh in a dynamic analysis.

- It is not possible to use staged construction type of loading for a dynamic calculation.

3.2.2 DYNAMIC ANALYSIS PARAMETERS

In the parameters tab of the calculations program you can define the control parameters of the dynamic calculation.

Dynamic time

A dynamic analysis uses a different time parameter than other types of calculations. The time parameter in a dynamic analysis is the *Dynamic time* parameter, which is always expressed in seconds [s], regardless of the unit of time as specified in the General Setting window. In a series of calculation phases in which some of them are dynamic, the *Dynamic time* is only increased in the dynamic phases (even non-successive), while the *Dynamic time* is kept constant in other types of calculations (whether before, in-between or after the dynamic phases).

The general *Time* parameter is, by default, expressed in days, which is generally more suitable for other types of time-dependent calculations like consolidation and creep. During dynamic calculations, the *Time* parameter is also increased, taking into account a proper conversion from the dynamic unit of time to the general unit of time.

Time stepping

The time step used in a dynamic calculation is constant and equal to $\delta t = \Delta t / (n \cdot m)$, where Δt is the duration of the dynamic loading (*Time interval*), n is the number of A*dditional steps* and m is the number of *Dynamic sub steps* (see Dynamic sub steps in Section 3.2.3 and the theory on the Critical time step in Section 5.2.2).

Time interval

For each phase in the calculation you have to specify the *Time interval* in the Parameter tab sheet. The estimated end time is calculated automatically by adding the time

intervals of all consecutive phases. When the calculation is finished, the real end time is given.

In a dynamic analysis the *Time interval* is always related to the *Dynamic time* parameter, expressed in seconds [s].

Estimated and realised end time

The estimated and realised end time are based on the general *Time* parameter expressed in the unit of time as specified in the General Setting window. PLAXIS will automatically convert the *Time interval* in a dynamic calculation into the general unit of time to calculate the *Estimated end time* or the *Realised end time*.

Additional steps

PLAXIS stores the calculation results in several steps. By default, the number of *Additional steps* is 250 but you can enter any value between 1 and 250 in the parameter tab sheet of the calculation program.

Delete intermediate steps

In PLAXIS it is possible to create animated output from dynamic analysis (Section 3.3). If you want the output to show more than only the start and end situation, you have to retain all the output steps. The option *Delete intermediate steps* should then be disabled.

3.2.3 ITERATIVE PROCEDURE MANUAL SETTING

Figure 3.3 Default parameters of iterative procedure for dynamic analysis

The iterative procedure for a dynamic analysis can be defined manually. The following parameters will be highlighted (see Figure 3.3):

- Dynamic sub steps
- Newmark alpha and beta
- Boundaries C_1 and C_2

Dynamic sub steps

For each additional time step, PLAXIS calculates the number of sub steps necessary to reach the estimated end time with a sufficient accuracy on the basis of the generated mesh and the calculated $\delta\,t_{critical}$ (see theory on critical time, Section 5.2.2). If the wave velocities (functions of material stiffness) in a model exhibit remarkable differences and/or the model contains very small elements, the standard number of sub steps can be very large. In such situations it may not always be vital to follow the automatic time stepping criterion.

You can change the calculated number of *Dynamic sub steps* in the manual settings of the iterative procedure in the parameter tab sheet. Be aware that in doing so the calculation results may be less accurate.

Newmark alpha and beta

The Newmark Alpha and Beta parameters in the manual settings for the iterative procedure determine the numeric time-integration according to the implicit Newmark scheme. In order to obtain an unconditionally stable solution, these parameters have to satisfy the following conditions: Newmark $\beta \geq 0.5$ and Newmark $\alpha \geq 0.25\,(0.5 + \beta)^2$

- For a damped Newmark scheme you can use the standard settings:
  $\alpha = 0.3025$ and $\beta = 0.6$. (Figure 3.3).
- For an average acceleration scheme you can use:
  $\alpha = 0.25$ and $\beta = 0.5$.

Boundary C_1 and C_2

C_1 and C_2 are relaxation coefficients used to improve the wave absorption on the absorbent boundaries. C_1 corrects the dissipation in the direction normal to the boundary and C_2 does this in the tangential direction. If the boundaries are subjected only to normal pressure waves, relaxation is not necessary ($C_1 = C_2 = 1$). When there are also shear waves (which is normally the case), C_2 has to be adjusted to improve the absorption. The standard values are $C_1 = 1$ and $C_2 = 0.25$ (Figure 3.3).

3.2.4   DYNAMIC LOADS

A dynamic load can consist of a harmonic load, a block load or a user defined load (ASCII file or SMC file for an earthquake model).

The input of dynamic loads is done in the Input program and is similar to that of the static load inputs (point loads and disptributed loads in system A and B or prescribed displacements). In addition, the user must set the dynamic loads system(s) in the Input program (Section 3.1.4). The dynamic load is then activated in the Calculation program by means of dynamic load multipliers.

The following paragraphs describe:

- How to activate a dynamic load

- Harmonic loads

- Load multiplier time series from ASCII file or SMC file

- Block loads

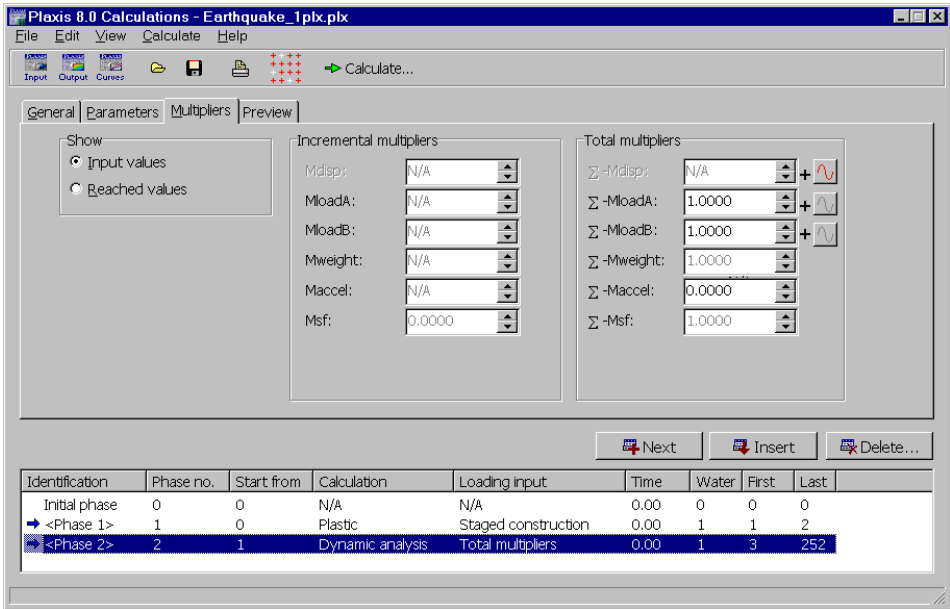### 3.2.5   ACTIVATING DYNAMIC LOADS

Figure 3.4 Multipliers in the Multiplier tab sheet

In the calculation program, multipliers are used to activate the dynamic loads. When the option *Dynamic analysis* is selected, you can click [icon] to the right of the multipliers ΣMdisp, ΣMloadA and ΣMloadB, Figure 3.4, to define the parameters for a harmonic

load or to read a dynamic load multiplier from a data file. This option is only available if the corresponding load is set as dynamic load in the *Loads* menu of the Input program.

The Active load that is used in a dynamic calculation is the product of the input value of the load, as specified in the Input program, and the corresponding dynamic load multiplier:

*Active load = Dynamic multiplier * Input value*

3.2.6   HARMONIC LOADS

In PLAXIS harmonic loads are defined as:

$$F = \hat{M}\,\hat{F}\,\sin\left(\omega\,t + \phi_0\right)$$

in which

$\hat{M}$ = Amplitude multiplier

$\hat{F}$ = Input value of the load

$\omega = 2\pi\,f$, with f = Frequency in cycles per unit of dynamic time (seconds)

ϕ_0 = Initial phase angle in degrees in the sine function.

Harmonic load can be applied by means of a harmonic load multiplier as shown in Figure 3.5. The load magnitude and frequency are specified by means of the *Amplitude multiplier* and *Frequency* respectively. The load phase angle can be defined, optionally, by the *Initial phase angle*.
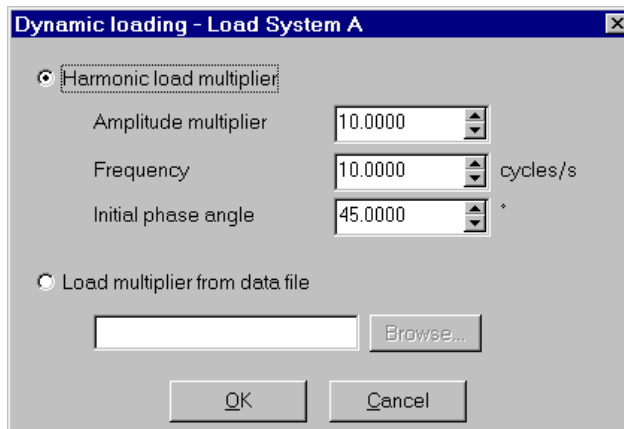

Figure 3.5 Harmonic load.

Example

Consider the load functions:

$F_1 = \sin (6 \pi t)$ $\qquad\qquad\qquad$ $F_2 = 2 \sin (2\pi t + \pi/2)$

Amplitude multiplier	:	1	Amplitude multiplier	:	2
Frequency	:	3 cycles/s	Frequency	:	1 cycles/s
Initial phase angle	:	0	Initial phase angle	:	90

In both functions, the input value of the load, $\hat{F}$, is set to 1.
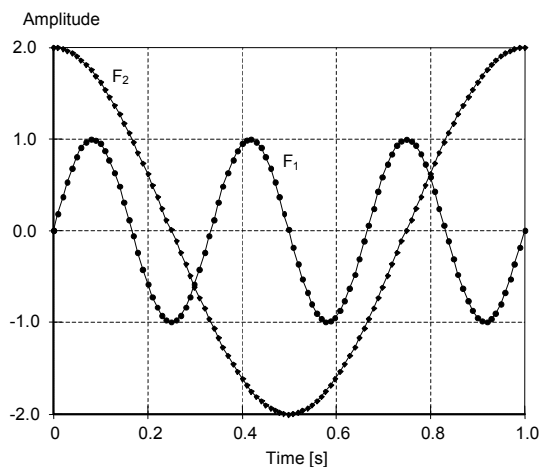The shape of these functions are shown in Figure 3.6.

Figure 3.6 Example $F_1 = \sin (6 \pi \text{ t})$, $F_2 = 2 \sin (2\pi \text{ t} + \pi/2)$.

3.2.7 LOAD MULTIPLIER TIME SERIES FROM DATA FILE.

Besides harmonic loading there is also the possibility to read data from a file with digitised load signal. This file may be in plain ASCII or in SMC format.

For prescribed displacements as entered in the Input program and activated in the Dynamic load multiplier input window of the Calculation program (the dynamic button besides the *ΣMdisp* multiplier) one can choose among displacements, velocities and accelerations. The velocities or accelerations are converted into displacements in the calculation program, taking into account the time step and integration method.

Alternatively, one can activate an external load in load system A or B by creating an ASCII file containing time and load multiplier in two couloms separated by a space. This file can be referred to in the *Load multipliers from data file* parameter (see Figure 3.7).

> **Hint:** PLAXIS assumes the data file is located in the current project directory when no directory is specified in the *Dynamic loading* window.

ASCII file

This file is an ASCII file that can be created by the user with any text editor. In every line a pair of values (actual time and corresponding multiplier) is defined, leaving at least one space between them. The time should increase in each new line. It is not necessary to use constant time intervals.

If the time steps in the dynamic analysis are such that they do not correspond with the time series given in the file, the multipliers at a given (Dynamic) time will be linearly interpolated from the data in the file. If the Dynamic time in the calculation is beyond the last time value in the file a constant value, equal to the last multiplier in the file will be used in the calculations.

SMC file

The SMC (Strong Motion CD-ROM) format is currently used by the U.S. Geological Survey National Strong-motion Program to record data of earthquakes and other strong vibrations. This format uses ASCII character codes and provides text headers, integer headers, real headers, and comments followed by either digitised time-series coordinates or response values. The header information is designed to provide the user with information about the earthquake and the recording instrument.

Most of the SMC files contain accelerations, but they may also contain velocity or displacement series and respones spectra (Figure 3.7). It is strongly recommended to use corrected earthquake data records, i.e. time series, that are corrected for final drift and non-zero final velocities.

The strong motion data are collected by the U.S. Geological Survey and are available from the National Geophysical Data Center (NGDC) of the National Oceanic and Atmospheric Administration.

Information on NGDC products is available on the World-wide Web at http://www.ngdc.noaa.gov/seg/fliers/ or by writing to:

> National Geophysical Data Center NOAA/EGC/1
> 325 Broadway
> Boulder, Colorado 80303
> USA

> **Hint:** The unit of length used in the SMC files is [cm], so accelerations are given in [cm/s2]. This has consequences for the input value of the prescribed displacements.

SMC files should be used in combination with prescribed boundary displacements at the bottom of a geometry model. When using the standard unit of length [m] it is necessary to use input values of 0.01 [m] for prescribed displacements. Alternatively, when using a unit of length of [ft] it is necessary to use input values of 0.0328 [ft] (1/[feet in cm]) for prescribed displacements. In this way the SMC file can directly be used for dynamic analysis of earthquakes.
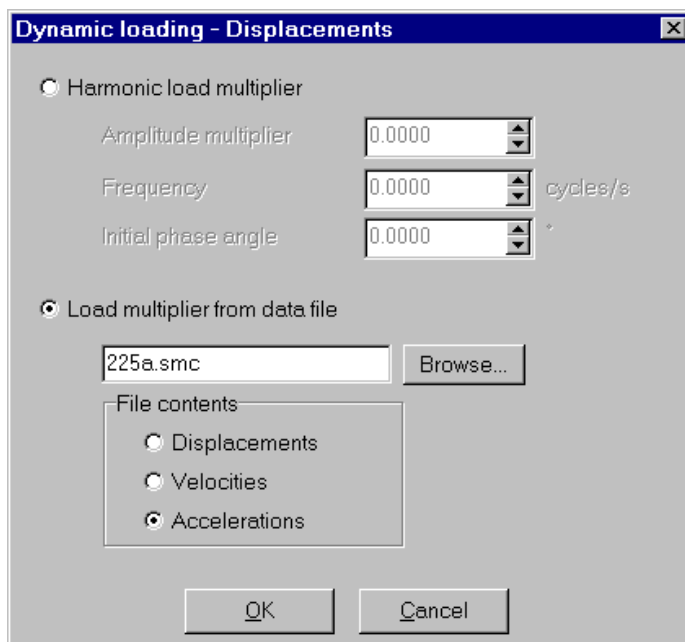

Figure 3.7 Selection of SMC file for earthquake loading

3.2.8   MODELLING BLOCK LOADS

A dynamic load can also suddenly be applied in a single time step or sub step (block load). You can model the block load using any of the following ways:

- Click on and define a harmonic load with an amplitude multiplier equal to the magnitude of the block load; $f = 0$ and $\phi 0 = 90°$. In this way, using the relation in Section 3.2.6, $F = \hat{M}\,\hat{F}$.

- Load a user defined ASCII file in which the block load is defined.

3.3   OUTPUT

The Dynamics module provides you with special options to display the results of a dynamic analysis.

- With the menu option *Create Animation* … in the *View* menu you can actually see the motion of the geometry in time. The number of steps in the animation can be influenced by the number of *Additional steps* you define in the Calculation phase. When you want to have all steps available for the output, the option *Delete intermediate steps* in the parameter tab sheet of the Calculations program should be disabled.

- For dynamic steps several velocity and acceleration options are available in the *Deformations* menu. You can select the total velocities, the total accelerations, the horizontal components and the vertical components.

Selection of phases from Create Animation window

By selecting *Create Animation* from the *View* menu, a window appears in which the phase(s) must be selected that are to be included in the animation (Figure 3.8). In order to select a range of phases, select the first phase of the range and then the last phase of the range while holding down the <Shift> key. After selecting the phase(s), click OK to start the process. The progress of this process is indicated in a separate window.

If a large number of steps is to be included in the animation, the process may take some minutes after which the animation is presented. The result is stored in an animation file (*.AVI) in the project. DTA directory and can be reused without regenerating using the window's media player.
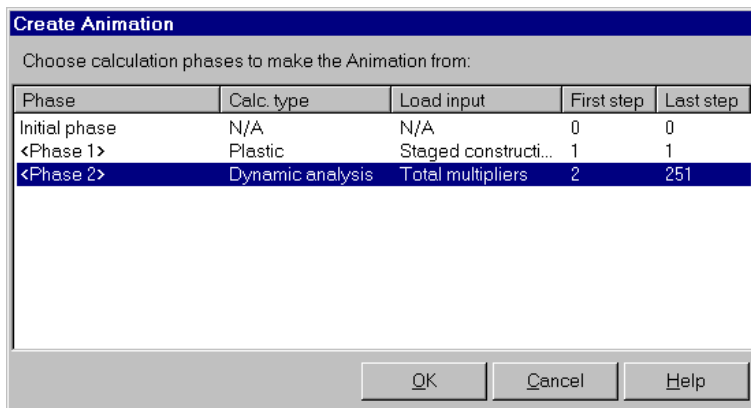

Figure 3.8 Selection of phases from *Create Animation* window

3.4   CURVES

For the general use of the Curves program you are referred to section 6 of the general PLAXIS Reference manual Version 8. In addition to the standard version of PLAXIS with the Dynamic analysis module you can display the velocity or acceleration as well as displacements as a function of time.

> **Hint:**   If you want to display motion for a particular point in the geometry you have to select this point with the option *Select points for curves* in the Calculations program.

Plots of time versus displacements, velocities, accelerations or multiplier or any other combination can be displayed. Select the corresponding radio buttons (Figure 3.9).
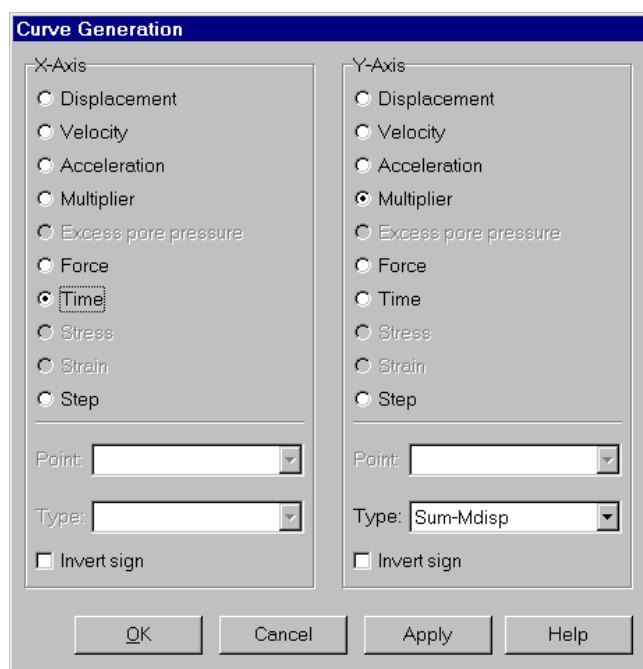

Figure 3.9 Dynamic fields in *Curve generation* window.

Transformation of curves from time to frequency domain

Once a time curve has been generated, it is possible to transform this curve into a frequency spectrum using the Fast Fourier Transform (FFT). This can be done by selecting the *Chart* option from the *Format* menu or clicking on the *Chart settings*

button in the toolbar. In the Chart setting window, you can select the option *Use frequency representation (spectrum)* and one of the three types of spectrum (Standard frequency (Hz), Angular frequency (rad/s) or Wave period (s)). Upon clicking on <OK> button the existing time curve will be transformed into a spectrum. The original curve can be reconstructed by selecting again the Chart settings and de-selecting the frequency representation.

4   VALIDATION AND VERIFICATION OF THE DYNAMIC MODULE

In addition to the Validation and Verification examples described in the last part of the general PLAXIS manual, this chapter contains some benchmark problems to demonstrate the correctness of the dynamic module. The examples involve comparisons with known analytical, semi-analytical or numerical solutions.

- One-dimensional wave propagation

- Simply supported beam

- Determination of velocity of Rayleigh wave

- Lamb's problem

- Surface waves on a half-space

- Pulse load on multi-layered media

4.1   ONE-DIMENSIONAL WAVE PROPAGATION

The first benchmark problem involves wave propagation in a one dimensional soil column. For this situation there is a simple analytical solution. The wave velocity V_p, in a confined one-dimensional body depends on the stiffness E_{oed}, and the mass ρ, of the medium:

$$V_p = \sqrt{\frac{E_{oed}}{\rho}} \qquad where \qquad E_{oed} = \frac{(1-v)\ E}{(1+v)(1-2v)} \qquad and \qquad \rho = \frac{\gamma}{g} \qquad (4.1)$$

in which E = Young's modulus, v = Poisson's ratio, γ = total unit weight and g is the gravity acceleration (9.8 m/s^2).

Input: Let's consider a soil column of 10 m high, with a radius of 0.2 m and standard fixities. The axisymmetric mesh with 15-noded elements is shown in Figure 4.1. The soil is considered to be linear elastic with the following homogeneous properties.

$\gamma = 20$ kN/m^3 $\qquad\qquad$ $E = 18000$ kN/m^2 $\qquad\qquad$ $v = 0.2$

For this material, and by means of equation (4.1), the wave velocity $V_p = 99$ m/s. Hence, it takes about 0.05 s for a wave generated at the top to reach the middle of the column and about 0.1 s to reach its bottom.

Output: After generation of initial stresses using $K_0 = 0.5$, an initial prescribed vertical displacement $u_y = 0.001$ m is given at the top. Two different conditions are considered at the bottom: (1) full fixity and (2) absorbent boundary. The calculation is performed with standard settings. Figure 4.2 and Figure 4.3 show the results for a point at the top (A), in the middle (B) and at the bottom (C) of the soil column for both models.
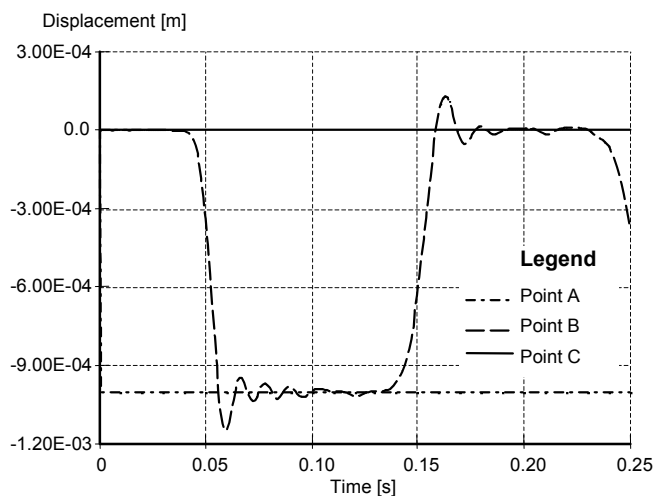
Figure 4.1 Finite element mesh

Figure 4.2 Time-displacement curve; full fixity at bottom of column (model 1)

Verification: Model 1: Figure 4.2 shows the response of at the top (prescribed displacements with u_y = 0.001 m, point A), at the middle, point B, and at the bottom (fixed displacement with u_y = 0.000 m, point C). It can be seen that the response at points A and C is as inputted. On the other hand, point B starts to move just before 0.05 s with displacement u_y = 0.001 m just after 0.05 s. Due to finite element discretisation and integration time step, the response at B is not as sharp and steady as at point A. However, as an average, the displacement in B agrees with the theory. After being reflected at the bottom, the wave reaches the middle point B again at about 0.15 s.

Then the displacement starts to diminish. After having reached the top again, the process continues in a similar way.
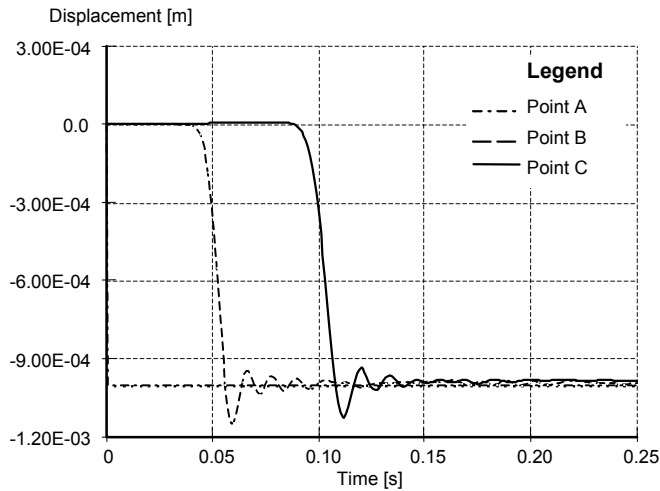

Figure 4.3   Time-displacement curve; absorbent boundary at bottom of column
(model 2)

Model 2: In contrast to the first model with the fixed bottom, Figure 4.3 shows that the bottom point also moves down 0.001 m around 0.1 s, after which it becomes steady. This is due to the absorbent boundary that prevents wave reflections. In fact, this agrees with the theoretical solution for an infinitely deep soil column.

In conclusion, the wave velocity is in agreement with the theoretical solution. However, some perturbations exist in the response that can be attributed to the numerical procedure of the finite element method.

4.2   SIMPLY SUPPORTED BEAM

The second benchmark problem is the dynamic behaviour of a simply supported beam. In this test the natural frequency of a simply supported beam will be determined and compared with the analytical solution.

The natural frequency of a system is the frequency that characterises the response of the system in a free vibration condition.

A simply supported beam has a number of fundamental modes of vibration. The natural frequency of the first mode of vibration excluding transversal shear and rotational effects is given by the following expression.

$$\omega_n = \pi^2 \sqrt{\frac{EI}{mL^4}} \qquad and \qquad m = \frac{w}{g} \qquad\qquad (4.2)$$

in which E = Young's modulus, I = Moment of inertia, m = Mass per unit of length, w = weight per unit of length, g is the gravity acceleration (9.8 m/s^2) and L = Length of the beam.

Input: In order to obtain the natural frequency with PLAXIS, a triangular pulse load of 100 kN/m is applied on the beam (plate element), Figure 4.4, for 0.01 s, Figure 4.5, after which the beam is allowed to vibrate freely. The period T of the beam can be determined by measuring the time between two successive oscillations. This measurement should be made after the transient vibrations have faded away.
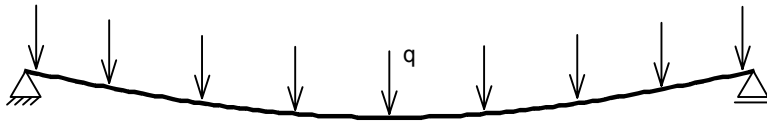

Figure 4.4 Simply supported beam.

A plate with the following characteristics is analysed:

$EA = 1.2 \cdot 10^6 \ kN$ $EI = 1000 \ kNm^2$ $v = 0.0$

$L = 10 \ m$ $q = pulse \ load$ $w = 0.98 \ kN/m/m$

According to equation (4.2), the natural frequency and period are respectively:

$$\omega_n = 9.87 \quad rad/s \qquad and \qquad T_n = \frac{2\pi}{\omega_n} = 0.637 \quad s$$

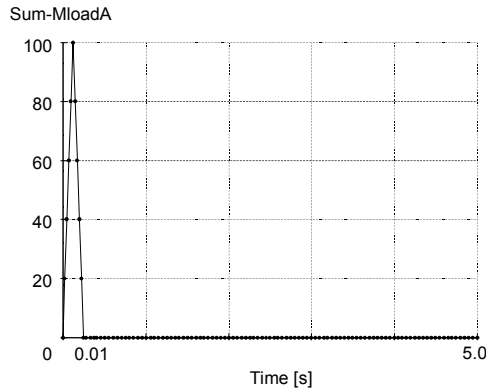The numerical integration is performed with the standard settings.

Figure 4.5 Pulse load.

Output: In Figure 4.6 the vertical displacements under the load are plotted. From this curve a period of 0.645 s is measured, resulting in a frequency ϖ_n = 9.74 rad/s.
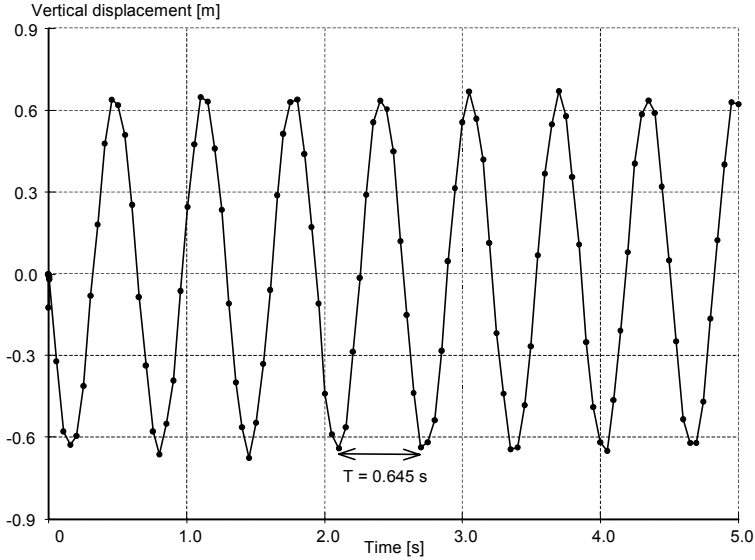

Figure 4.6 PLAXIS solution. Vertical displacements - time.

Verification: The natural frequency of the beam calculated in PLAXIS is ϖ_n = 9.74 rad/s and agrees well with the analytical solution obtained by equation (4.2): ϖ_n = 9.87 rad/s.

4.3   DETERMINATION OF THE VELOCITY OF THE RAYLEIGH WAVE

When body waves (compression and shear) encounter a free boundary of a solid body, a new type of wave generates. Such wave is known as the Rayleigh wave. The study of this type of waves is important because, in the event of earthquake, the largest disturbances, which are recorded on a distant seismogram, are usually caused by the Rayleigh waves.

Here the analytical velocity of the Rayleigh wave will be compared with the velocity calculated with the dynamic module.

The theoretical value of the velocity of the Rayleigh wave, V_R, has been computed by Knopoff (1952, 1954):

$$V_R = 0.54 V_p \qquad where \qquad V_p = \sqrt{\frac{(1-v)\,E}{(1+v)(1-2v)\rho}} \qquad and \qquad \rho = \frac{\gamma}{g} \qquad (4.3)$$

where V_p is the velocity of the compression (pressure) wave on an elastic medium, γ = total unit weight and g is the gravity acceleration (9.8 m/s^2).
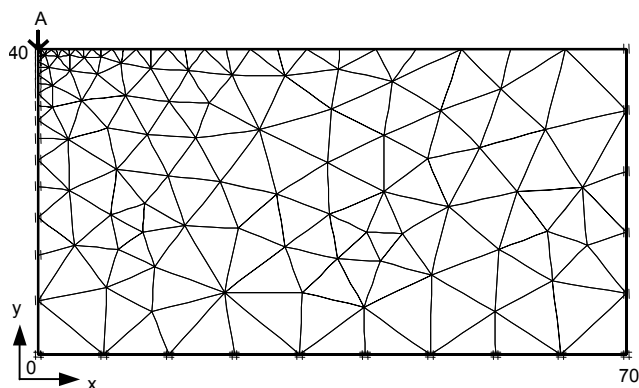
Figure 4.7 Geometry and finite element mesh

Input: The geometry is analysed with an axisymmetric finite element model, with the load in the centre of symmetry. The model is 70 m wide and 40 m high (See Figure 4.7). A local refinement in the mesh is made around the load. Absorbent boundaries are placed to avoid the reflection of outgoing waves in the soil body. The pulse load is approximated by a triangular load with a magnitude of 100 kN and a duration of 0.005 s. The load data is provided by an external text file, which reads:

Time [s]	Force [KN]
0.	0.
0.0025	100.
0.005	0.
1.	0.

Please note that in preparing the ASCI file, the time and force magnitudes are separated by a space and no titles are necessary.

The parameters of the soil are:

γ = 19.6 kN/m^3 E = 1000 kN/m^2 v = 0.25

According to equation (4.3) the velocity of the Rayleigh wave is V_R = 13.23 m/s

The Newmark parameters used in the numerical integration are α = 0.3025 and β = 0.6 (the standard setting in Version 8). No damping is included.

Output: Two output points are selected at the surface. They are located at A = 3.25 m and B = 5.38 m away from the load centre. In Figure 4.8 the vertical displacements at

these points are plotted versus time. Knowing the distance between these points, and measuring the time taken by the wave to travel from one point to the other, it is possible to calculate the velocity of the Rayleigh wave.
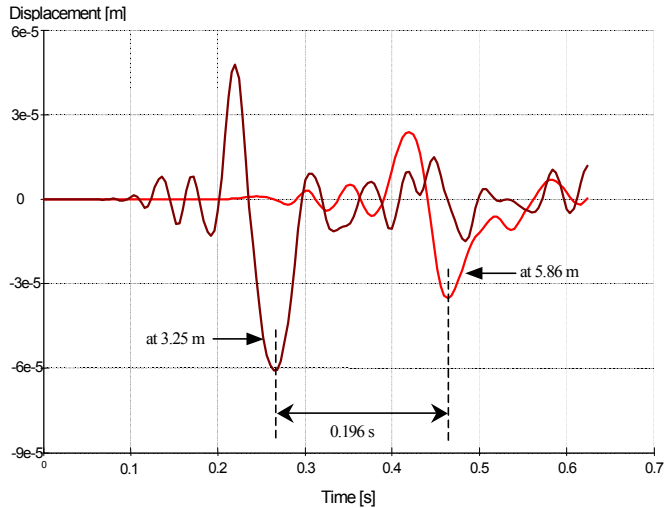
Figure 4.8 Rayleigh wave generated by a pulse load.

Verification: From Figure 4.8 it can readily be shown that the velocity of the Rayleigh wave calculated by PLAXIS is V_R = 13.3 m/s. Apparently this result is in good agreement with the analytical one, equation (4.3).

Reference:
Knopoff, L., On Rayleigh Wave Velocities, Bull. Seism. Soc. Amer. vol. 42, pp. 307-308, 1952.
Knopoff, L., Seismic Wave Velocities in Westerly Granite, Trans. Am. Geophys. Union, vol. 35, pp. 969-973, 1954.

4.4   LAMB'S PROBLEM

Lamb (1904) studied wave propagation in a semi-infinite elastic medium subjected to an impulsive force applied at the surface. Lamb's solution has been considered by, among others, Miklowitz (1978), Cagniard (1962) and Foinquinos & Roesset (2000).

In this section, comparison is made between PLAXIS results and results obtained from the closed form solution introduced by Foinquinos & Roesset (2000) for solving Lamb's far field problem.

Input: In PLAXIS, Lamb's far field problem is simulated by use of an axisymmetric geometry extending to 100 m in the radial direction and 30 m in the vertical direction (see Figure 4.9). The structure is assumed to consist of a linear elastic soil with Young's modulus of 50000 kPa, Poisson's ratio of 0.25 and unit weight of 20 kN/m^3. The structure is subjected to a point load at the centre of symmetry. The time distribution of the load pulse is approximated by a triangle with 0.025 s duration and amplitude of 50 kN. The load starts after 0.05 s. Absorbing boundaries are introduced at the bottom and right hand side.

In solving the closed form formulation for the vertical displacements, some damping was introduced to avoid numerical waviness that may occur due to the presence of singularities. Because of this, in PLAXIS, some small damping is necessary. As such, Rayleigh damping with $\alpha_R = 0.001$ and $\beta_R = 0.002$ is included. Also the default damped Newmark scheme ($\alpha = 0.3025$ and $\beta = 0.6$), is used.
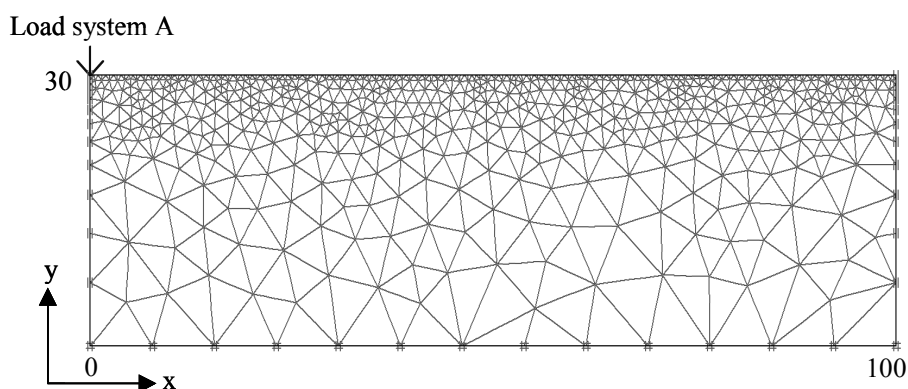

Figure 4.9 Pulse load on a half space (plain strain).

Output: Figure 4.10 shows vertical displacements at a point on the surface, 50 m away from the source, as computed by PLAXIS and Foinquinos et al formulation. It can be seen that despite the difference in the response pulse period, the displacement amplitude and shape are in good agreement. This difference can be attributed to the finite element discretisation and integration time step. However, knowing the difficulties involved in simulating far fields problems by finite element method, the computational result, with relatively small number of elements (1246 element), encourages the utilization of PLAXIS for simulating wide range of dynamic problems. Also, it can be noticed in the figure that PLAXIS is capable to capture the arrival time of the P wave (compression) and the S wave (shear). This is not possible in the Lamb's formulation because it takes only the Rayleigh wave into consideration.
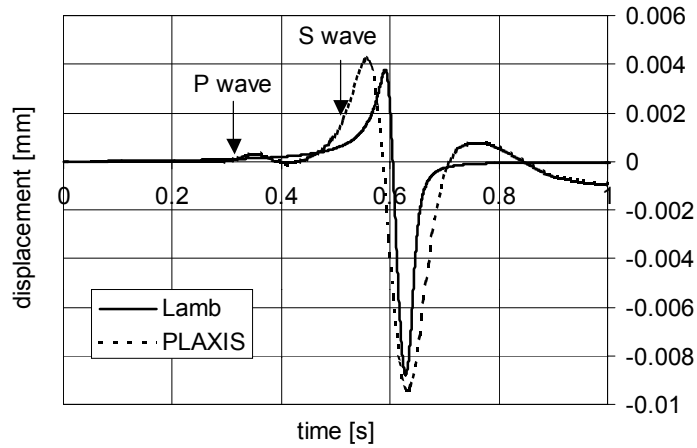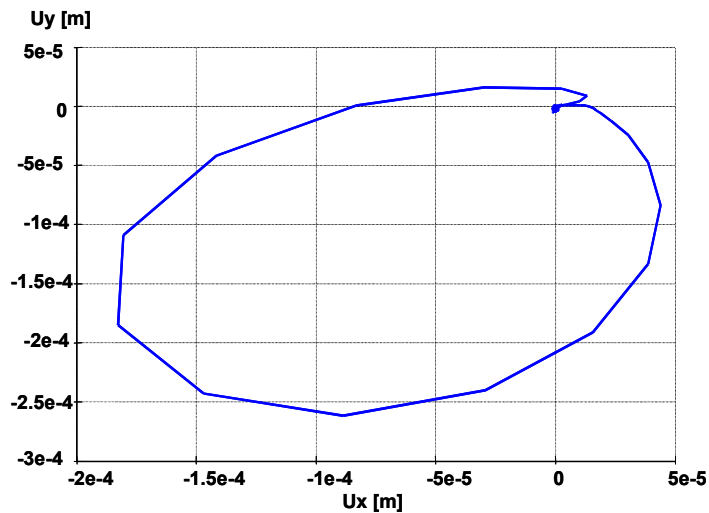
Figure 4.10 Vertical displacements.

The path followed by the particles on the surface as a result of the Rayleigh wave is shown in Figure 4.11. The trajectory describes an ellipse, which is in agreement with the theory.

Figure 4.11 Particle path (x=50 m).

The deformed meshes at different times provide insight in the behaviour of the Rayleigh wave, as displayed in Figure 4.12 to Figure 4.15. (Remember that the load pulse starts at 0.05 s.) Even better insight can be obtained by using the PLAXIS feature to create animations.
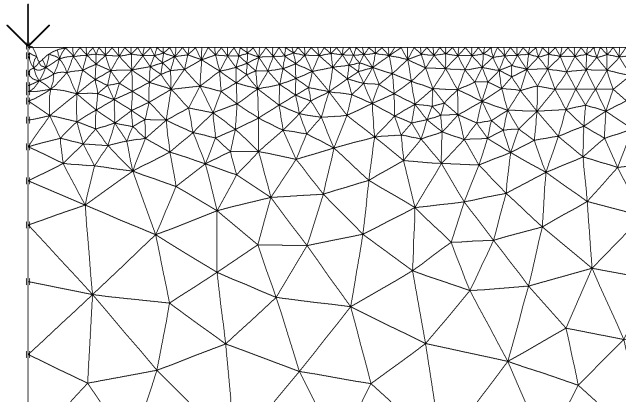
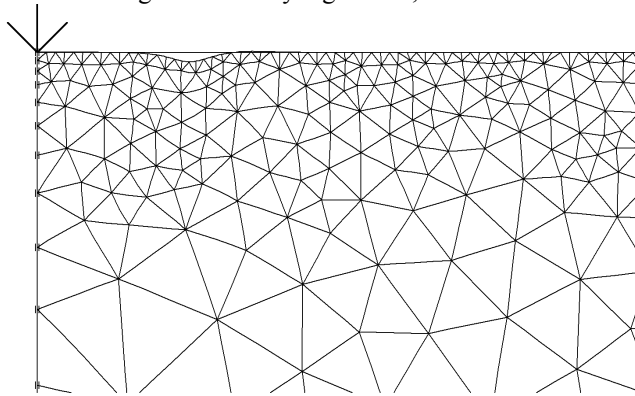Figure 4.12 Rayleigh wave, $t = 0.09$ s.

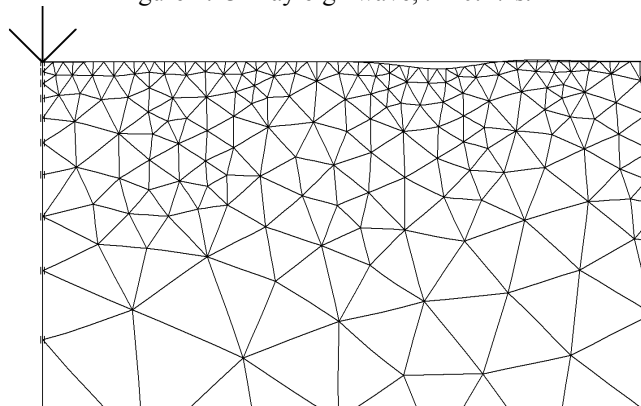Figure 4.13 Rayleigh wave, $t = 0.17$ s.

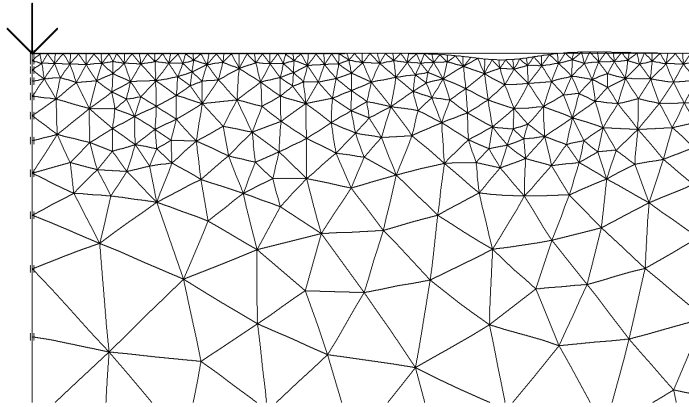Figure 4.14 Rayleigh wave, $t = 0.29$ s.

Figure 4.15 Rayleigh wave, $t = 0.41$ s.

Reference:
Lamb, H., On the Propagation of Tremors Over the Surface of an Elastic Solid, Phil. Trans. Roy. Soc. (London) A, vol. 203, pp. 1-42, 1904.
Cagniard, L., E.A.Flinn and C.H.Dix, 1962, Reflection and Refraction of Progressive Seismic Waves. New York, McGraw-Hill, 282pp.
Miklowitz, J., 1978, The Theory of Elastic Waves and Wave guides, Amsterdam: North–Holland.
Foinquinos, R. and Roesset, J.M., 2000. Elastic Layered Half Space Subjected to Dynamic Surface Loads. In: Wave Motion in Earthquake Engineering, E. Kausel and G. Manolis, Editors.

4.5   SURFACE WAVES
COMPARISON WITH BOUNDARY ELEMENTS

In this section, again, surface waves on an elastic half space are considered. In this case the surface waves are generated by a prescribed displacement that is activated harmonically. A comparison is made between the results from the PLAXIS dynamic module and results obtained by Chouw et al. (1991) using boundary elements.

Input: Because of symmetry, only the right half is considered as presented in Figure 4.16. The plane strain mesh is composed of 15-noded elements and is 200 m wide and 30 m deep. The soil is assumed linear elastic with the following properties:

$\gamma = 17.64$ kN/m^3 $\nu = 0.33$ $G = 53280$ kN/m^2

The prescribed displacement acts in the centre over a width of 1.5 m. Absorbent boundaries are applied at the bottom and right-hand boundary. The displacement is activated harmonically with an amplitude of 0.48 m and a frequency of 5 Hz. There is no damping.
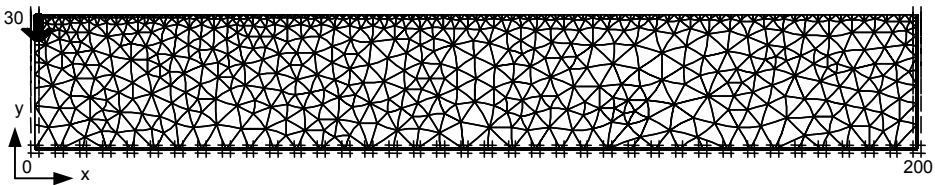
Figure 4.16 Finite Element mesh.

Output: Figure 4.17 shows the amplitudes of the surface waves at different distances from the source. Figure 4.18 shows similar results obtained by Chouw et al., 1991 using boundary elements. The results are quite similar.
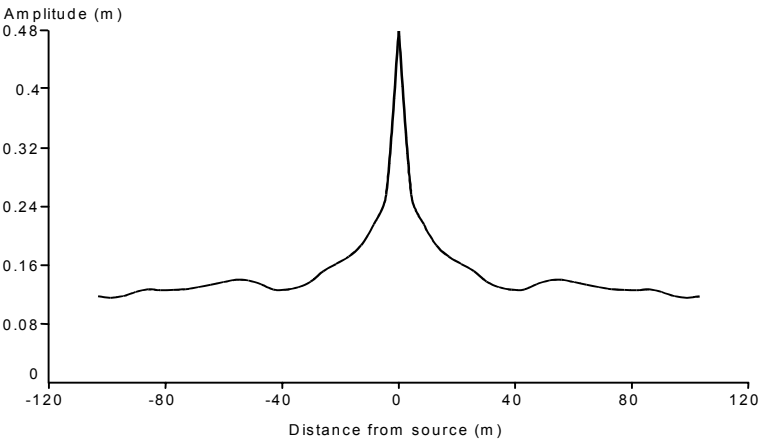
Figure 4.17 Amplitudes using the PLAXIS dynamic module at different distances from the source.

Figure 4.18 Amplitudes using boundary elements at different distances from the source (after Chouw et al., 1991).

Verification: From the results it can be concluded that surface waves due to prescribed displacement can be correctly simulated using the dynamic module.

Reference:
Chouw, N., R. Le, G. Schmid, 1991, Impediment of Surface Waves in Soil, Mathematical and Numerical aspects of wave propagation phenomena.

4.6 PULSE LOAD ON A MULTI LAYER SYSTEM COMPARISON WITH SPECTRAL ELEMENTS

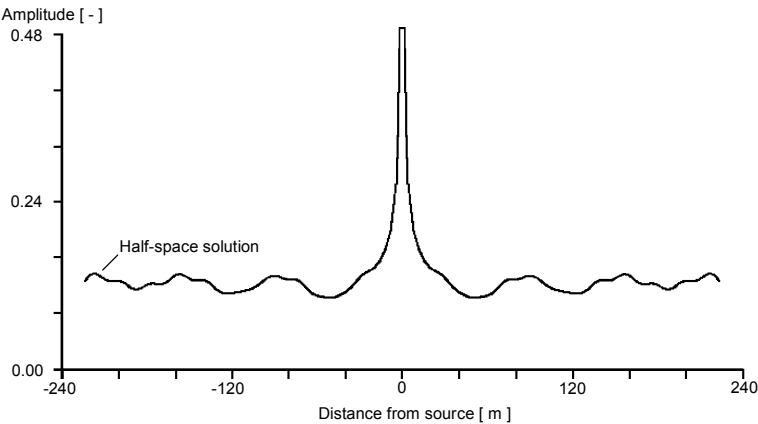In this section the dynamic response of a multi layer system due to a transient load pulse is considered. For such a system, no analytical solution exists. For verification, the computational results of PLAXIS are compared with those obtained from a semi-analytical technique using the spectral element method (Al-Khoury et al, 2001).

Input: In PLAXIS, the geometry is modelled with an axisymmetric model using 15-noded elements. The geometry is 16 m wide and 15.4 m deep, as presented in Figure 4.19. A distributed load with 0.15 m radius is used to simulate a typical Falling Weight Deflectometer (FWD) load pulse. FWD is a non-destructive dynamic test used commonly for the evaluation of pavement structures. Figure 4.20 shows the load pulse in time. The load starts at $t = 0.052$ s and ends at $t = 0.076$ s.

A three-layered system representing pavement structure and consisting of asphalt, subbase and subgrade layers is considered. Two cases are evaluated: First, the pavement structure is based on a stiff subgrade. Second, the pavement structure is based on a soft subgrade. The properties of the layers are listed in Table 4.1. Absorbent boundaries are applied at the bottom and right-hand boundary. There is no damping.

Table 4.1 Soil properties

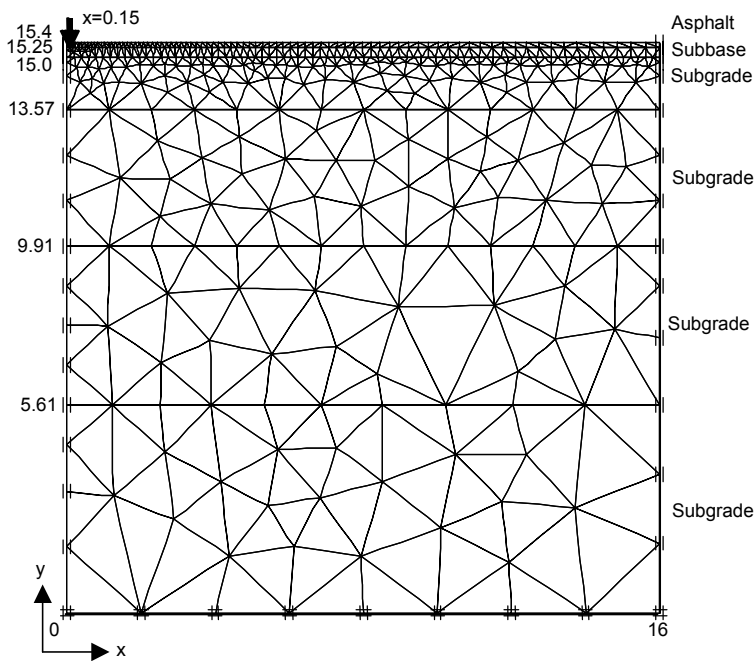| Parameter	Name	Asphalt	Subbase	Stiff subgrade	Soft subgrade	Unit
Material model	Model	Elastic	Elastic	Elastic	Elastic	-
Soil weight	γ	23.0	19.0	15.0	15.0	kN/m^3
Young's modulus (constant)	E	$1 \cdot 10^6$	$2 \cdot 10^5$	$1 \cdot 10^5$	$2.5 \cdot 10^4$	kN/m^2
Poisson's ratio	ν	0.35	0.35	0.35	0.35	-

Figure 4.19 Finite element mesh.

Figure 4.20 Pulse load.

Displacement (mm)
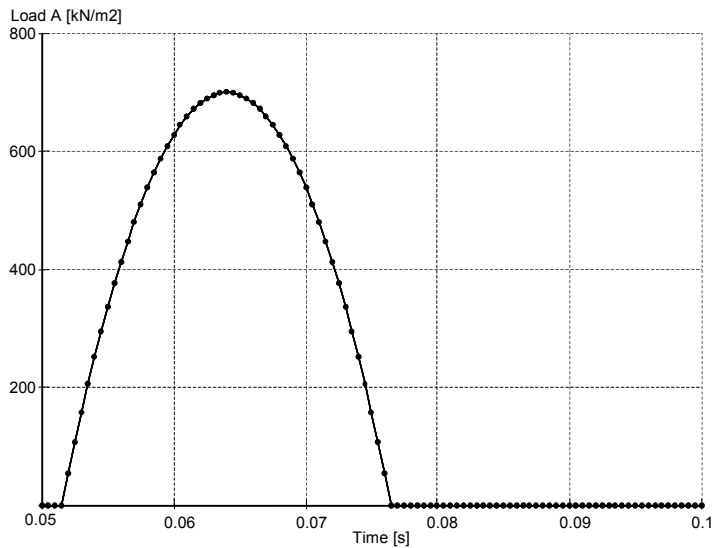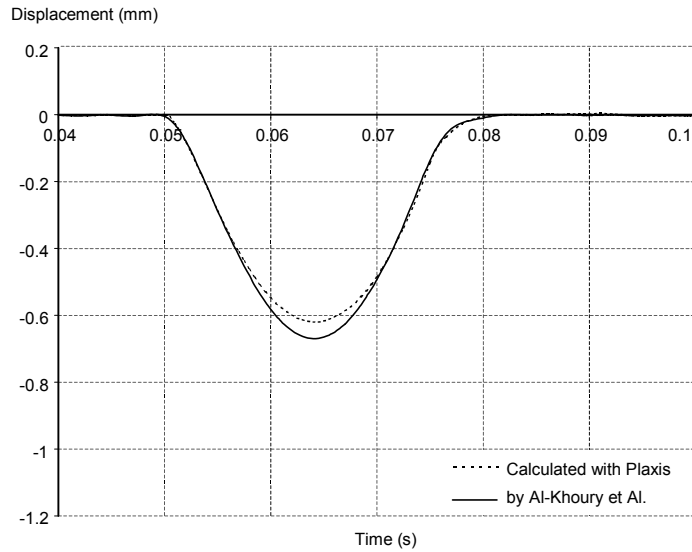
Figure 4.19 Vertical displacements at the centre using a stiff subgrade, calculated with PLAXIS and by using spectral elements.
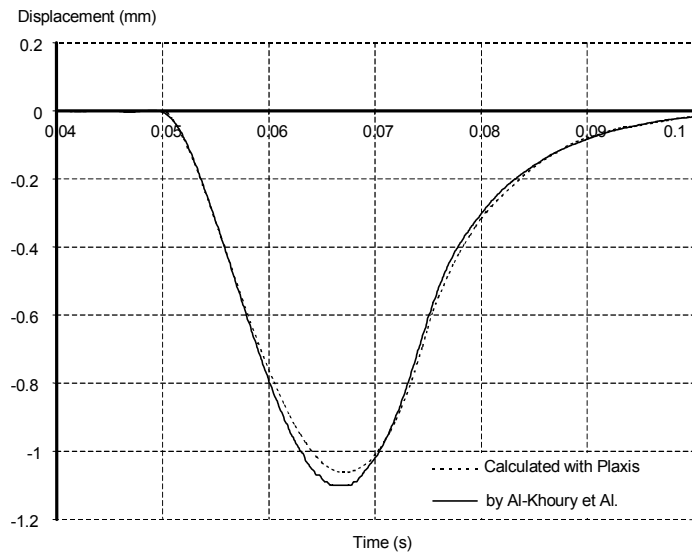
Displacement (mm)

Figure 4.20 Vertical displacements at the centre using a soft subgrade, calculated with PLAXIS and by using spectral elements.

Output: Figure 4.19 and Figure 4.20 show the vertical displacements at the centre directly under the load for both stiff and soft subgrade structures. For the stiff subgrade, the maximum displacements calculated by PLAXIS is about -0.62 mm and for the soft subgrade is about –1.06 mm. Even though the pulse load period is 0.025 s, the response takes about 0.03 s for the stiff subgrade and about 0.05 s for the soft subgrade.

Verification: Figure 4.19 and Figure 4.20 also show the results obtained from the spectral element method. For the stiff subgrade, the maximum displacement is about –0.67 mm and for the soft subgrade is about –1.10 mm. The deflection for the stiff subgrade last about 0.03 s and about 0.05 s for the soft subgrade. The results show that calculations by both PLAXIS and R. Al-Khoury et al. are in good agreement.

Reference:
R. Al-Khoury, A. Scarpas, C. Kasbergen and J. Blaauwendraad, Spectral Element Technique for Efficient Parameter Identification of Layered Media, Part I: Forward Calculation. International Journal of Solids and Structures, Vol. 38, 1605-1623, 2001.

5   THEORY

In addition to the theoretical formulations and aspects presented in the scientific section of the general PLAXIS manual, this chapter highlights some of the theoretical backgrounds of the dynamic module. The chapter does not give a full theoretical description of the dynamic modelling. For a more detailed description you are referred to the literature [1, 2, 6].

The following topics will be discussed:

- Basic equations for dynamic behaviour

- Time integration

- Wave velocities

- Critical time step

- Model boundaries

- Absorbent boundaries

- Initial stresses and stress increments at the model boundaries

5.1   BASIC EQUATION DYNAMIC BEHAVIOUR

The basic equation for the time-dependent movement of a volume under the influence of a (dynamic) load is:

$$\underline{\underline{M}}\,\ddot{\underline{u}} + \underline{\underline{C}}\,\dot{\underline{u}} + \underline{\underline{K}}\,\underline{u} = \underline{F} \tag{5.1}$$

Here, M is the mass matrix, u is the displacement vector, C is the damping matrix, K is the stiffness matrix and F is the load vector. The displacement, u, the velocity, $\dot{u}$, and the acceleration, $\ddot{u}$, can vary with time. The last two terms in the equation (5.1) ($K\,u = F$) correspond to the static deformation.

Here the theory is described on the bases of linear elasticity. However, in principle, all models in PLAXIS can be used for dynamic analysis. The soil behaviour can be both drained and undrained. In the latter case, the bulk stiffness of the groundwater is added to the stiffness matrix K, as is the case for the static calculation.

In the matrix M, the mass of the materials (soil + water + any constructions) is taken into account. In PLAXIS the mass matrix is implemented as a lumped matrix.

The matrix C represents the material damping of the materials. In reality, material damping is caused by friction or by irreversible deformations (plasticity or viscosity). With more viscosity or more plasticity, more vibration energy can be dissipated. If elasticity is assumed, damping can still be taken into account using the matrix C. To determine the damping matrix, extra parameters are required, which are difficult to

determine from tests. In finite element formulations, C is often formulated as a function of the mass and stiffness matrices (Rayleigh damping) [1, 2] as:

$$\underline{\underline{C}} = \alpha_R \, \underline{\underline{M}} + \beta_R \, \underline{\underline{K}} \tag{5.2}$$

This limits the determination of the damping matrix to the Rayleigh coefficients α_R and β_R. Here, when the contribution of M is dominant (for example, $\alpha_R = 10^{-2}$ and $\beta_R = 10^{-3}$) more of the low frequency vibrations are damped, and when the contribution of K is dominant (for example, $\alpha_R = 10^{-3}$ and $\beta_R = 10^{-2}$) more of the high-frequency vibrations are damped. In the standard setting of PLAXIS, $\alpha_R = \beta_R = 0$.

5.2   TIME INTEGRATION

In the numerical implementation of dynamics, the formulation of the time integration constitutes an important factor for the stability and accuracy of the calculation process. Explicit and implicit integration are the two commonly used time integration schemes. The advantage of explicit integration is that it is relatively simple to formulate. However, the disadvantage is that the calculation process is not as robust and it imposes serious limitations on the time step. The implicit method is more complicated, but it produces a more reliable (more stable) calculation process and usually a more accurate solution [3].

The implicit time integration scheme of Newmark is a frequently used method. With this method, the displacement and the velocity at the point in time $t+\Delta t$ are expressed respectively as:

$$u^{\,t+\Delta t} = u^{\,t} + \dot{u}^{\,t} \, \Delta t + \left(\left(\tfrac{1}{2} - \alpha \right) \ddot{u}^{\,t} + \alpha \, \ddot{u}^{\,t+\Delta t} \right) \Delta t^{2} \tag{5.3a}$$

$$\dot{u}^{\,t+\Delta t} = \dot{u}^{\,t} + \left(\left(1 - \beta \right) \ddot{u}^{\,t} + \beta \, \ddot{u}^{\,t+\Delta t} \right) \Delta t \tag{5.3b}$$

In the above equations, Δt is the time step. The coefficients α and β determine the accuracy of the numerical time integration. They are not equal to the α and β for the Rayleigh damping. In order to obtain a stable solution, the following condition must apply:

$$\beta \geq 0.5 \qquad \alpha \geq \tfrac{1}{4} \left(\tfrac{1}{2} + \beta \right)^{2} \tag{5.4}$$

The user is advised to use the standard setting of PLAXIS, in which the damped Newmark scheme with $\alpha = 0.3025$ and $\beta = 0.60$ is utilised. As an alternative, the commonly used average acceleration method with $\alpha = 0.25$ and $\beta = 0.50$ can be used. Other combinations are also possible, however.

Implementation of the integration scheme in PLAXIS

Equations (5.3) can also be written as:

$$\ddot{u}^{\,t+\Delta t} = c_0\,\Delta u - c_2\,\dot{u}^{\,t} - c_3\,\ddot{u}^{\,t}$$

$$\dot{u}^{\,t+\Delta t} = \dot{u}^{\,t} + c_6\,\ddot{u}^{\,t} + c_7\,\ddot{u}^{\,t+\Delta t}$$

$$u^{\,t+\Delta t} = \qquad\qquad u^{\,t} + \Delta u \qquad\qquad (5.5a)$$

or as:

$$\ddot{u}^{\,t+\Delta t} = c_0\,\Delta u - c_2\,\dot{u}^{\,t} - c_3\,\ddot{u}^{\,t}$$

$$\dot{u}^{\,t+\Delta t} = c_1\,\Delta u - c_4\,\dot{u}^{\,t} - c_5\,\ddot{u}^{\,t}$$

$$u^{\,t+\Delta t} = \qquad\qquad u^{\,t} + \Delta u \qquad\qquad (5.5b)$$

where the coefficients c_0 .. c_7 can be expressed in the time step and in the integration parameters α and β. In this way, the displacement, the velocity and the acceleration at the end of the time step are expressed by those at the start of the time step and the displacement increment. With implicit time integration, equation (5.1) must be obtained at the end of a time step ($t+\Delta t$):

$$\underline{\underline{M}}\,\underline{\ddot{u}}^{\,t+\Delta t} + \underline{\underline{C}}\,\underline{\dot{u}}^{\,t+\Delta t} + \underline{\underline{K}}\,\underline{u}^{\,t+\Delta t} = \underline{F}^{\,t+\Delta t} \qquad\qquad (5.6)$$

This equation, combined with the expressions (5.5) for the displacements, velocities and accelerations at the end of the time step, produce:

$$\left(c_0\,\underline{\underline{M}} + c_1\,\underline{\underline{C}} + \underline{\underline{K}}\,\right)\Delta\underline{u} = \underline{F}_{ext}^{\,t+\Delta t} + \underline{\underline{M}}\left(c_2\underline{\dot{u}}^{\,t} + c_3\underline{\ddot{u}}^{\,t}\right) + \underline{\underline{C}}\left(c_4\underline{\dot{u}}^{\,t} + c_5\underline{\ddot{u}}^{\,t}\right) - \underline{F}_{int}^{\,t} \qquad (5.7)$$

In this form, the system of equations for a dynamic analysis reasonably matches that of a static analysis. The difference is that the 'stiffness matrix' contains extra terms for mass and damping and that the right-hand term contains extra terms specifying the velocity and acceleration at the start of the time step (time Δt).

5.2.1   WAVE VELOCITIES

The compression wave velocity, V_p, in a confined one-dimensional soil is a function of stiffness, E_{oed}, and the mass, ρ, as:

$$V_p = \sqrt{\frac{E_{oed}}{\rho}} \qquad where \qquad E_{oed} = \frac{(1-v)\ E}{(1+v)(1-2v)} \qquad and \qquad \rho = \frac{\gamma}{g} \quad (5.8)$$

in which E = Young's modulus, v = Poisson's ratio, γ = total unit weight and g is the gravity acceleration (9.8 m/s^2).

A similar expression can be found for the shear wave velocity, V_s:

$$V_s = \sqrt{\frac{G}{\rho}} \qquad where \qquad G = \frac{E}{2(1+v)} \qquad\qquad (5.9)$$

5.2.2   CRITICAL TIME STEP

Despite the advantages of the implicit integration, the time step used in the calculation is subject to some limitations. If the time step is too large, the solution will display substantial deviations and the calculated response will be unreliable. The critical time step depends on the maximum frequency and the coarseness (fineness) of the finite element mesh. In general, the following expression can be used for a single element [7]:

$$\Delta t_{critical} = \frac{B}{\alpha\ \sqrt{\dfrac{E(1-v)}{\rho(1+v)(1-2v)}}\sqrt{1+\dfrac{B^4}{4\,S^2}-\dfrac{B^2}{2S}\left[1+\dfrac{1-2v}{4}\dfrac{2S}{B^2}\right]}} \qquad (5.10)$$

The first root term represents the velocity of a (compression) wave, equation 5.8. The factor α depends on the element type. For a 6-node element $\alpha = 1/(6\sqrt{c_6})$, with $c_6 \approx 5.1282$, and for a 15-node element $\alpha = 1/(19\sqrt{c_{15}})$, with $c_{15} \approx 4.9479$ [1]. The other determining factors are the Poisson's ratio, v, the average length of an element B (see Section 3.6.1 of the general PLAXIS Reference Manual, Version 8, for a description of the average element length) and the surface of the element S. In a finite element model, the critical time step is equal to the minimum value of Δt according to (5.10) over all elements. This time step is chosen to ensure that a wave during a single step does not move a distance larger than the minimum dimension of an element.

5.3   MODEL BOUNDARIES

In the case of a static deformation analysis, prescribed boundary displacements are introduced at the boundaries of a finite element model. The boundaries can be completely free or fixities can be applied in one or two directions. Particularly the vertical boundaries of a mesh are often non-physical (synthetic) boundaries that have been chosen so that they do not actually influence the deformation behaviour of the construction to be modelled. In other words: the boundaries are 'far away'. For dynamic calculations, the boundaries should in principle be much further away than those for

static calculations, because, otherwise, stress waves will be reflected leading to distortions in the computed results. However, locating the boundaries far away requires many extra elements and therefore a lot of extra memory and calculating time.

To counteract reflections, special measures are needed at the boundaries – in this context, we speak of 'silent or absorbent boundaries'. Various methods are used to create these boundaries, which include:

- Use of half-infinite elements (boundary elements).

- Adaptation of the material properties of elements at the boundary (low stiffness, high viscosity).

- Use of viscous boundaries (dampers).

All of these methods have their advantages and disadvantages and are problem dependent. For the implementation of dynamic effects in PLAXIS (Version 8) the absorbent boundaries are created with the last method (use of viscous boundaries). The way this method works is described below.

5.3.1 ABSORBENT BOUNDARIES

In opting for absorbent boundaries, a damper is used instead of applying fixities in a certain direction. The damper ensures that an increase in stress on the boundary is absorbed without rebounding. The boundary then starts to move.

The use of absorbent boundaries in PLAXIS is based on the method described by Lysmer and Kuhlmeyer [4].

The normal and shear stress components absorbed by a damper in x-direction are:

$$\sigma_n = - c_1 \, \rho \, V_p \, \dot{u}_x \tag{5.11a}$$

$$\tau = - c_2 \, \rho \, V_s \, \dot{u}_y \tag{5.11b}$$

Here, ρ is the density of the materials. V_p and V_s are the pressure wave velocity and the shear wave velocity, respectively. V_p and V_s are determined using equations 5.8 and 5.9. C_1 and C_2 are relaxation coefficients that have been introduced in order to improve the effect of the absorption. When pressure waves only strike the boundary perpendicular, relaxation is redundant ($C_1 = C_2 = 1$).

In the presence of shear waves, the damping effect of the absorbent boundaries is not sufficient without relaxation. The effect can be improved by adapting the second coefficient in particular. The experience gained until now shows that the use of $C_1 = 1$ and $C_2 = 0.25$ results in a reasonable absorption of waves at the boundary. However, it is not possible to state that shear waves are fully absorbed so that in the presence of shear waves a (limited) boundary effect is noticeable. Additional research is therefore

necessary on this point, but the method described will be sufficient for practical applications.

For an inclined boundary, an adjusted formulation based on equation (5.11) is used that takes the angle of the boundary into account.

5.4 INITIAL STRESSES AND STRESS INCREMENTS

By removing the boundary fixities during the transition from a static analysis to a dynamic analysis, the boundary stresses also cease. This means that the boundary will start to move as a result of initial stresses. To prevent this, the original boundary stress will be converted to an initial (virtual) boundary velocity. When calculating the stress, the initial boundary velocity must be subtracted from the real velocity:

$$\sigma_n = -c_1 \, \rho \, V_p \, \dot{u}_n + \sigma_n^0 = -c_1 \, \rho \, V_p \left(\dot{u}_n - \dot{u}_n^0 \right) \tag{5.12}$$

This initial velocity is calculated at the start of the dynamic analysis and is therefore based purely on the original boundary stress (preceding calculation or initial stress state).

At present, situations can arise where a new load is applied at a certain location on the model and is continuously present from that moment onward. Such a load should result in an increase in the average boundary stress. If it involves an absorbent boundary, the average incremental stress cannot be absorbed. Instead, the boundary will start to move. In most situations, however, there are also fixed (non-absorbent) boundaries elsewhere in the mesh – for example, on the bottom. The bottom of the mesh, at the location of the transition from a non-rigid to a hard (stiff) soil layer, is often chosen for this. Here, reflections also occur in reality, so that such a bottom boundary in a dynamic analysis can simply be provided with standard (fixed) peripheral conditions. In the above-mentioned case of an increased load on the model, that increase will eventually have to be absorbed by the (fixed) bottom boundary — if necessary, after redistributing the stresses.

6   REFERENCES

[1]   Zienkiewicz O.C., Taylor R.L. (1991), The finite element method (4th edition), volume 2, Solid and Fluid mechanics, Dynamics and Non-Linearity. Mc Graw-Hill, U.K.

[2]   Hughes, TH.J.R. (1987), The finite element method, linear static and dynamic analysis, Prentice Hall Int.

[3]   Sluys L.J. (1992), Wave propagation, Localisation and Dispersion in softening solids, dissertation, Delft University of Technology.

[4]   Lysmer J., Kuhlmeyer R.L. (1969) Finite Dynamic Model for Infinite Media. ASCE J. of the ENg. Mech. Div., p.859-877

[5]   Kramer S.L. (1996), Geotechnical earthquake engineering. Prentice Hall, New Jersey.

[6]   Das B.M. (1995), Fundamentals of soil dynamics. Elsevier.

[7]   Pal, O. (1998) Modélisation du comportement dynamique des ouvrages grâce à des éléments finis de haute précision, thesis, L'université Joseph Fourier - Grenoble I

[8]   Chouw, Le, Schmid (1991) "Impediment of Surface Waves in Soil". In: Mathematical and Numerical aspects of wave propagation phenomena.

[9]   Bathe, Klaus-Jurgen (1996) "Finite Element Procedures". Prentice Hall.