

3 INPUT PROGRAM - GENERAL OVERVIEW

To carry out a finite element analysis using the PLAXIS 2D program, the user has to create a two-dimensional geometry model composed of points, lines and other components, in the xy-plane and specify the material properties and boundary conditions. This is done in the first two tabsheets (*Geometry modes*) of the Input program. The mesh generation and the definition of the calculation phases is done in the last three tabsheets (*Calculation modes*) of the Input program.

3.1 STARTING THE INPUT PROGRAM



This icon represents the Input program.

Start PLAXIS by double clicking the shortcut on your desktop or clicking the entry in the *Windows > Programs* menu.

Depending on your settings, you may see the PLAXIS News Feed window, see Figure 3.1. You can disable this by unchecking the *Show on startup* at the bottom of the window.

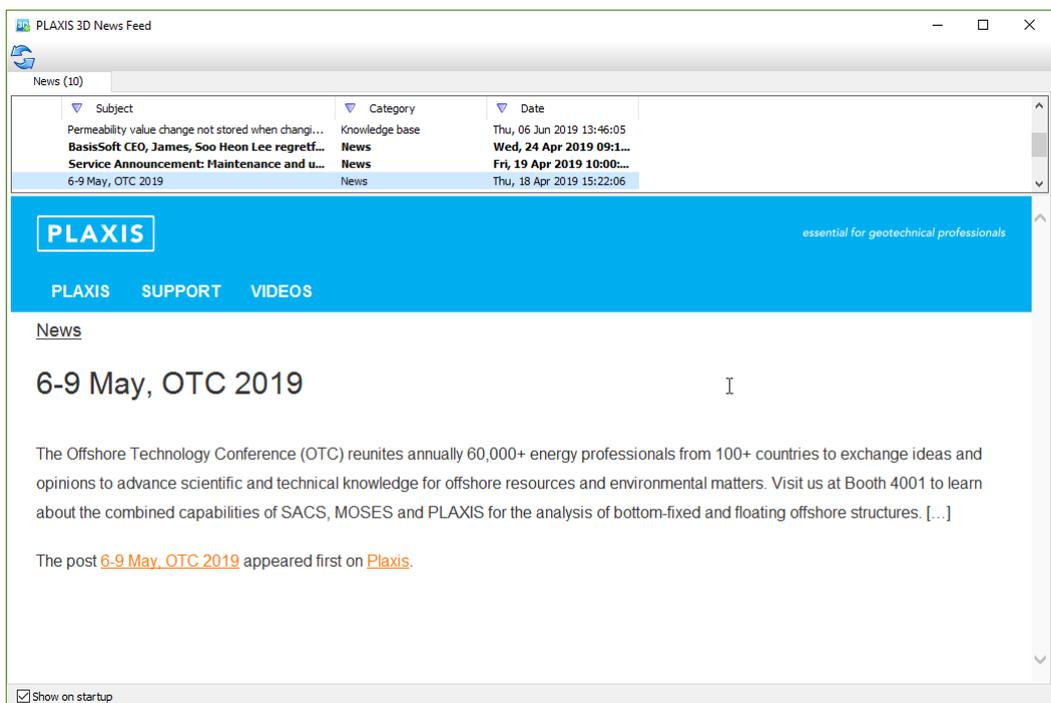


Figure 3.1 News feed window

PLAXIS will first check if you are entitled to use the software and present you with the available options and licences, see Figure 3.2.

If you need to change the licence configuration, e.g. if you have to add some modules, click *Configure licences*.

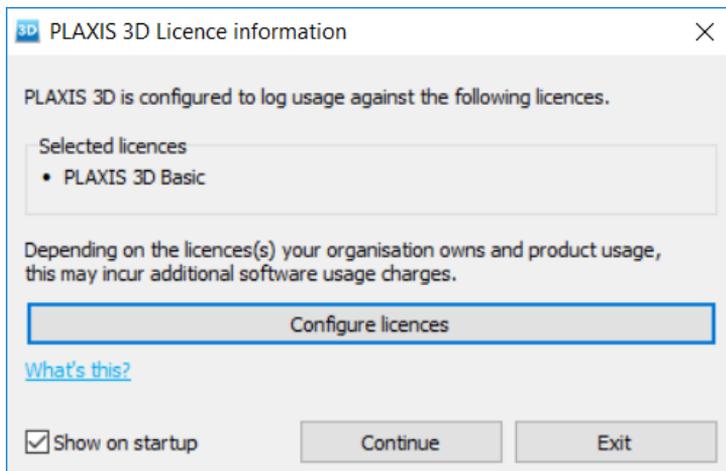


Figure 3.2 Licence check



NOTE: The licence configuration is also available from the menu *Windows > Programs* and from the menu *Help > License configuration*.

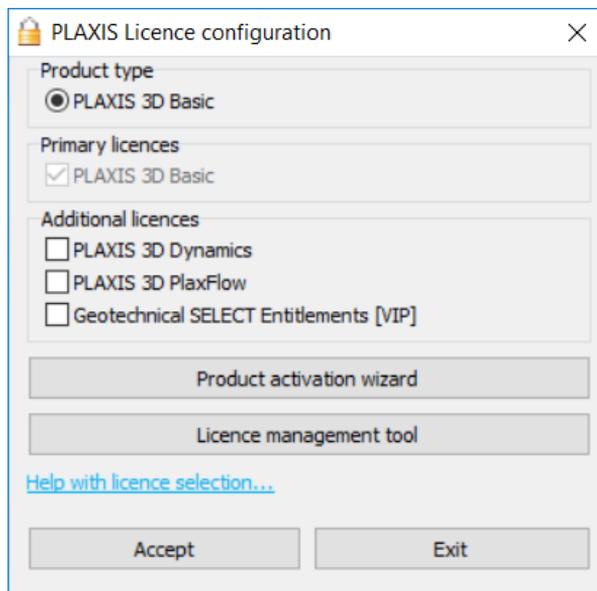


Figure 3.3 Licence configuration

In the configuration dialog, see Figure 3.3, you may add/remove modules that you want to use.



NOTE: If you change the license configuration while PLAXIS is open, you have to restart PLAXIS for the changes to take effect.

The licence administrator grants access to software licences. If you miss licences in this list, contact the licence administrator of your company.

For more information on the Product activation wizard and the Licence management tool, see this info on Bentley communities: [Info on Product Activation/License Management](#)

When you have chosen the licences to use, click *Continue* to start using PLAXIS.

At the start of the Input program the *Quick select* window appears in which a choice must be made between the selection of an existing project and the creation of a new project (Figure 3.4).

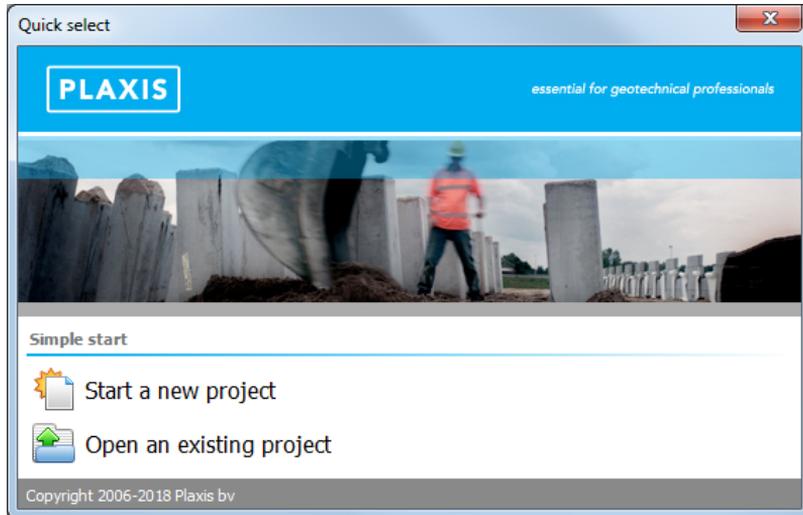


Figure 3.4 *Quick select* window

3.1.1 NEW PROJECT

When the *Start a new project* option is selected, the *Project properties* window appears in which the basic model parameters of the new project can be defined. The *Project properties* window contains the *Project* and the *Model* tabsheets. The *Project* tabsheet (Figure 3.5) contains the project name and description and offers the possibility to select a company logo. The *Model* tabsheet (Figure 3.6) contains the type of the model, the basic units for length, force and time (see Section 3.8), the unit weight of water and the initial dimensions of the model. The default values can be replaced by the current values when selecting *Set as default* and clicking the *OK* button. A more detailed description of all these options is given below.

Project

The title, folder and the file name of the project are available in the *Project* group box available in the *Project* tabsheet.

<i>Title</i>	The defined title appears as a default name for the file of the project when it is saved.
<i>Company</i>	The company that holds the license for this software.
<i>Directory</i>	The address to the folder where the project is saved is displayed. For a new project, there is no information shown.

File name The name of the project file is displayed. For a new project, no information is shown.

Comments

The *Comments* box in the *Project* tabsheet gives the possibility to add some extra comments about the project.

Company logo

Clicking the *Company logo* box in the *Project* tabsheet will open a file requester from which the desired file can be selected. This logo can be included in the output plots.

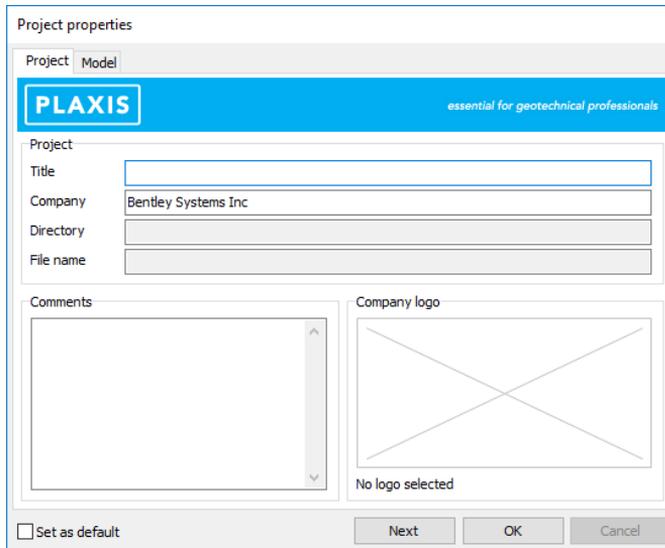


Figure 3.5 *Project properties* window (*Project* tabsheet)

General model properties

The general options of the project are available in the *Model* tabsheet of the *Project properties* window (Figure 3.6).

Model

PLAXIS 2D may be used to carry out two-dimensional finite element analysis. The finite element model is defined by selecting the corresponding option in the *Model* drop down-menu in the *Project* tabsheet.

Plane strain: A *Plane strain* model is used for geometries with a (more or less) uniform cross section and corresponding stress state and loading scheme over a certain length perpendicular to the cross section (*z*-direction). Displacements and strains in *z*-direction are assumed to be zero. However, normal stresses in *z*-direction are fully taken into account.

In earthquake problems the dynamic loading source is usually applied along the bottom of the model resulting in shear waves that propagate upwards. This type of problem is

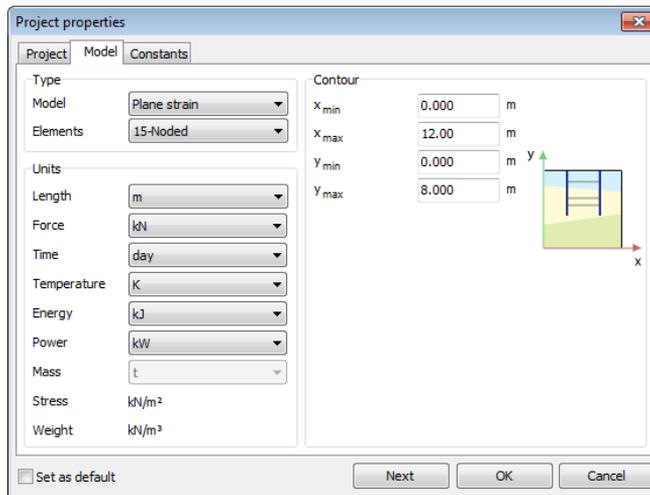


Figure 3.6 *Project properties* window (*Model* tabsheet)

generally simulated using a plane strain model.

Axisymmetric: An *Axisymmetric* model is used for circular structures with a (more or less) uniform radial cross section and loading scheme around the central axis, where the deformation and stress state are assumed to be identical in any radial direction. Note that for axisymmetric problems the x -coordinate represents the radius and the y -axis corresponds to the axial line of symmetry. Negative x -coordinates cannot be used.

Single-source vibration problems are often modelled with axisymmetric models. This is because waves in an axisymmetric system radiate in a manner similar to that in a three dimensional system. In this case, the energy disperses leading to wave attenuations with distance. Such effect can be attributed to the geometric damping (or radiation damping), which is by definition included in the axisymmetric model.

The selection of *Plane strain* or *Axisymmetric* results in a two dimensional finite element model with only two translational degrees of freedom per node (x - and y -direction).

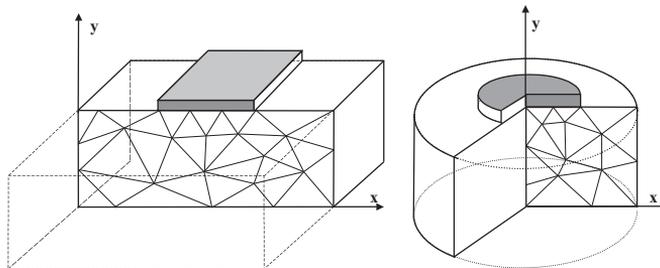


Figure 3.7 Example of a plane strain (left) and axisymmetric problem (right)

Elements

The user may select either 6-node or 15-node triangular elements (Figure 3.5) to model soil layers and other volume clusters.

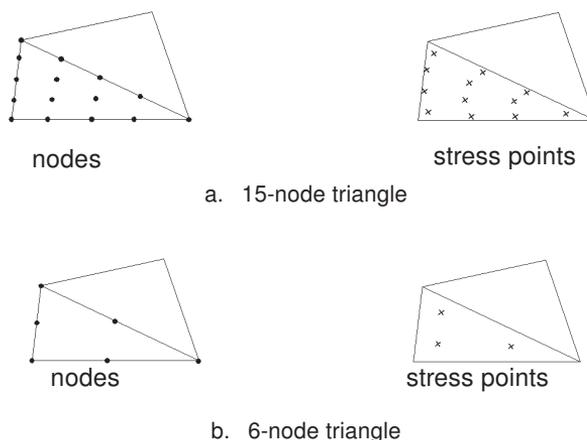


Figure 3.8 Position of nodes and stress points in soil elements

15-Node: The 15-node triangle is the default element. It provides a fourth order interpolation for displacements and the numerical integration involves twelve Gauss points (stress points). The type of element for structural elements and interfaces is automatically taken to be compatible with the soil element type as selected here.

The 15-node triangle is a very accurate element that has produced high quality stress results for difficult problems, as for example in collapse calculations for incompressible soils (Nagtegaal, Parks & Rice, 1974, Sloan, 1981 and Sloan & Randolph, 1982). The 15-node triangle is particularly recommended to be used in axi-symmetric analysis. The use of 15-node triangles leads to more memory consumption and slower calculation and operation performance. Therefore a more simple type of elements is also available.

6-Node: The 6-node triangle provides a second order interpolation for displacements and the numerical integration involves three Gauss points. The type of element for structural elements and interfaces is automatically taken to be compatible with the soil element type as selected here.

The 6-node triangle is a fairly accurate element that gives good results in standard deformation analyses, provided that a sufficient number of elements are used. However, care should be taken with axi-symmetric models or in situations where (possible) failure plays a role, such as a bearing capacity calculation or a safety analysis by means of *phi-c reduction*. Failure loads or safety factors are generally overpredicted using 6-noded elements. In those cases the use of 15-node elements is preferred.

One 15-node element can be thought of a composition of four 6-node elements, since the total number of nodes and stress points is equal. Nevertheless, one 15-node element is more powerful than four 6-node elements.

In addition to the soil elements, compatible plate elements are used to simulate the behaviour of walls, plates and shells (Section 5.7.3) and geogrid elements are used to simulate the behaviour of geogrids and wovens (Section 5.7.4). Moreover, compatible interface elements are used to simulate soil-structure interaction (Section 5.7.5). Finally, the geometry creation mode allows for the input of embedded beam rows, fixed-end anchors and node-to-node anchors (Section 5.7, Section 5.7.1).

Units

Units for length, force, time, temperature, energy, power and mass to be used in the analysis need to be specified. These basic units are entered in the *Model* tabsheet of the *Project properties* window (Figure 3.6).

The default units, as suggested by the program, are m (metre) for length, kN (kilonewton) for force, day for time, K (kelvin) for temperature, kJ (kilojoule) for energy, kW (kilowatt) for power and t (tonne) for mass. The corresponding units for stress and unit weights are listed in the box below the basic units.

In a dynamic analysis, the time is usually measured in [seconds] rather than the default unit [days]. 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*.

All input values should be given in a consistent set of units (Section 3.8). The appropriate unit of a certain input value is usually given directly behind the edit box, based on the basic set of units.

Hint: Note that for calculations involving thermal effects, it is strongly recommended to use the default units.

Contour

At the start of a new project the user needs to specify the contours of the geometry model. The initial setting of the x_{\min} , x_{\max} , y_{\min} and y_{\max} parameters set the outer boundaries of the geometry model. The initial view of the drawing area is such that the model dimensions are fully visible within the drawing area. The aforementioned parameters are entered in the *Model* tabsheet of the *Project properties* window.

Constants

In thermal calculations, pore water may appear in different states, i.e. in liquid state (water), solid state (ice) and gas state (vapour). Thermal properties of soils highly depend on the state of water in the pores. Rather than specifying these properties in material data sets, the thermal properties of pore water are specified for the project as a whole in the *Constants* tab sheet of the *Project properties* window (Figure 3.9).

Gravity: By default, the earth gravity acceleration, g , is set to 9.810 m/s^2 , assuming the default basic length unit is [m] irrespective of the selected unit of time. The direction of gravity coincides with the negative y -axis. Gravity is implicitly included in the unit weights given by the user (Section 6.1). In this way, the gravity is controlled by the total load multiplier for weights of materials, ΣM_{weight} (Section 7.8.1).

In dynamic calculations, the value of the gravity acceleration, g , is used to calculate the material density, ρ , from the unit of weight, γ ($\rho = \gamma/g$).

Reference temperature: In projects that involve temperature, the input of reference temperature is required. The reference temperature is the temperature at the ground surface in the case that no specific thermal conditions are given here. The reference temperature (T_{ref}) can be entered in the *Constants* tabsheet of the *Project properties*

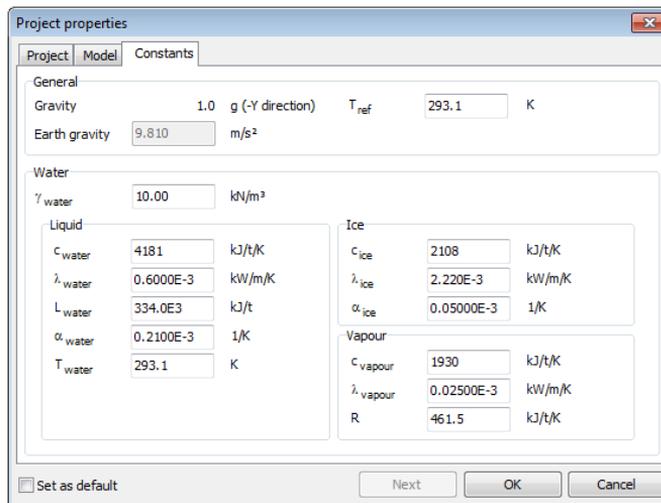


Figure 3.9 *Project properties* window (*Constants* tabsheet)

window. By default, the reference temperature is set to 293.1 K (= 20 °C).

Unit weight of water: In projects that involve pore pressures, the input of a unit weight of water is required to determine the effective stresses and pore pressures. The water weight (γ_{water}) can be entered in the *Constants* tabsheet of the *Project properties* window. By default, the unit weight of water is set to 10.00, assuming the default basic units of [kN] and [m] are used.

Specific heat capacity: The specific heat capacity of water, c_{water} , is a parameter that describes the amount of energy (heat) that can be stored in the pore water per unit of mass. It is specified in the unit of energy per unit of mass per unit of temperature (Section 6.1). Similarly, specific heat capacity can be given for given for ice and vapour.

Thermal conductivity: The thermal conductivity of water, λ_{water} , is a parameter that describes the rate of energy (heat) that can be transported in the pore water. It is specified in the unit of energy per unit length per unit of temperature. Similarly, thermal conductivity can be given for given for ice and vapour.

Latent heat of water: The latent heat of water is the amount of energy that is required to change the state of water from liquid state to solid state and vice versa. It is specified in the unit of energy per unit of mass.

Thermal expansion coefficient: The thermal expansion coefficient, α is the (change of) volumetric strain per unit of temperature. The thermal expansion coefficient can be given for water and ice. However, thermal expansion of vapour is generally suppressed and therefore ignored. Note that the thermal expansion for ice starts to *increase* from the freeze temperature downward.

Water temperature: In the case that water is entering into the soil (by means of inflow boundary conditions or infiltration wells), this water is assumed to have a temperature equal to the specified water temperature, T_{water} .

Specific gas constant for vapour: The specific gas constant for vapour, R , is a parameter that is used in the equation of relative humidity (see scientific documentation). It is recommended to use the default value.

Temperature-dependent pore fluid flow: To enable *Temperature-dependent* pore fluid properties, click on the *Use temperature-dependent table* checkbox. A table is displayed and the values are visualised in small graphs. The table contains five columns: *Temperature (T)*, *Density (ρ)*, *Thermal conductivity (λ)*, *Specific heat capacity (c)* and *Dynamic viscosity (μ)*. The table is pre-filled with default values for water and its phase transition to ice. If this option is selected, the constant parameters for liquid water and ice are not editable to emphasise that the *Temperature-dependent* water properties are taken into account. The table editor allows you to edit the table directly, or to import and export tables from and to a text file.

Hint: Note that the viscosity below 273.15 K is not relevant in the case of the default properties. It is set to a constant value to ease visualisation and allow for modifications if needed, in the case of brines for example, where the freeze point is low.

3.1.2 EXISTING PROJECT

When the Input program is started, a list of the recent projects appears in the *Quick select* window. In the case when a project other than the listed recent ones is required, the *Open an existing project* option should be selected. As this selection is made, the Windows® file requester (Figure 2.2) pops up. It enables the user to browse through all available folders and to select the desired PLAXIS project file (*.p2dx). After the selection of an existing project, the corresponding geometry is presented in the main window.

An existing PLAXIS 2D AE project can also be read by selecting the *Open project* option in the *File* menu. In the file requester, the type of the file is, by default, set to 'PLAXIS 2D AE files (*.p2dx)'.

3.1.3 PACKING A PROJECT



The created project can be compressed using the *Pack project* application (Figure 3.10), which is available in the *File* menu of the Input program. This application can be executed directly from the PLAXIS 2D AE installation folder by double clicking the corresponding file (PackProject.exe). A shortcut to the application can be created as well.

The project to be compressed and the archive can be located using the *Browse* button. The options available in the *Purpose* box are:

<i>Backup</i>	All the files in the project are included in the compressed project as well as the mesh information, phase specification and the results of all the saved calculation steps. The extension of the project file, indicating in which program it was created, and the archiving date are included in the archive name.
<i>Support</i>	Selecting this option enables including all the information required to give support for the project at hand. Note that support is only provided to VIP users.
<i>Custom</i>	The user can define the information to be included in the archive.