

Finite Element Solutions

Programs for the analysis of geotechnical problems by the finite element method (FEM) allow the user to closely model the behavior of real soils and structures. They also address problems which cannot be solved by standard analytical methods. FEM programs can model the behavior of any soil mass and its interaction with structures such as tunnels, collectors, deep foundations, sheeting etc. They are also used to calculate settlement and to solve various stability problems.

A key advantage of the GEO5 program suite is the ability to transfer data between programs. The structure can be designed or examined using traditional analytical or semi-analytical methods. Data can then be transferred into the FEM program for an independent assessment of its behavior. Such an approach ensures not only a more reliable design, it also saves time.

The basic program features include:

- User-friendly application environment
- Comprehensive graphical outputs
- Integrated contextual help
- Metric and imperial units
- Built-in database of soil characteristics



FEM

The program is applicable to the modeling of a wide range of geotechnical problems such as terrain settlement, sheeting and underpinning of structures, slope stability, beams on foundation, excavation, etc.

Material models

A reliable and accurate response of a given soil or rock mass can be obtained by using a suitable material model. The present release offers the following models:

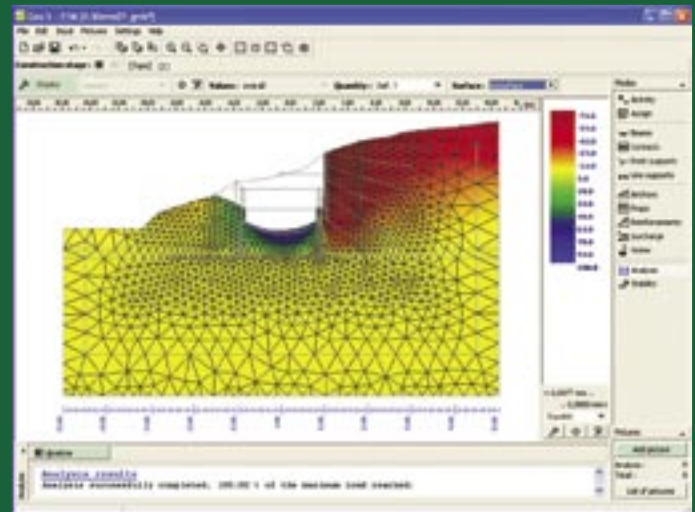
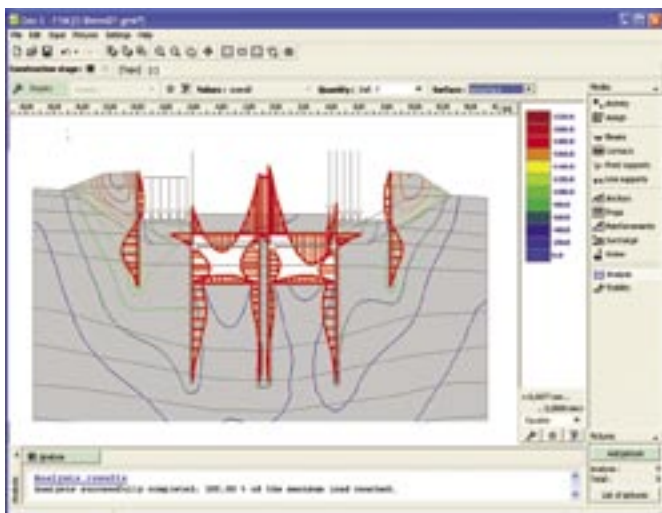
- Linear elastic model
- Modified linear elastic model
- Mohr-Coulomb model
- Drucker-Prager model
- Cam Clay model

Built-in geometrical model correction, automatic mesh generation

The program includes built-in correction of the input geometrical model. All material interfaces, new structures or calculation stages (tunnel excavation, excavation of a foundation ditch) can be specified independently from each other. Before running the mesh generator the program automatically locates the intersections of all specified lines, all closed regions and creates a corresponding geometrical model. Introducing even a fairly complicated structure into the program thus becomes a relatively simple task.

Mesh generator

The program contains a built-in automatic mesh generator that considerably simplifies construction of the finite element mesh. Both 3-node and 6-node triangular elements are available. The density of the finite elements mesh is assumed constant for the entire model but it can easily be refined in local areas as required. Preparing an effective mesh for a complex structure thus requires only a few minutes of the user's time.



Geotechnical Software

Boundary conditions

The program contains an automatic generator of boundary conditions, so that in most applications the user does not have to worry about this task. If needed, additional boundary conditions (fixed, hinge, spring, forced displacement) can be introduced anywhere in the soil body.

Beams

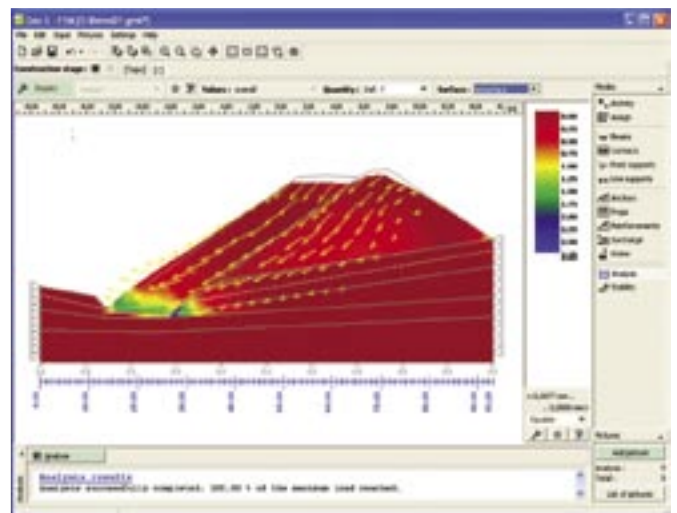
Beam elements serve to introduce beams, tunnel lining or sheeting structures into the analysis. As a result of the analysis the program gives the distribution of internal forces (bending moment, normal and shear forces) along the beam. Beam elements are specified along lines already existing in the topology. The beam parameters, however, can be adjusted in individual stages (e.g., gradually increasing beam thickness) or the beam can be completely removed in one step.

Anchors, geotextiles, geogrids

The program allows the introduction of any number of anchors into the analysis. Each anchor is fully determined by its starting point, end point and its stiffness. The anchor end points are automatically mapped onto the existing finite element mesh, so that an anchor can be placed anywhere within the soil body. To further simplify input, the anchor starting point can be placed on the terrain or on individual interfaces.

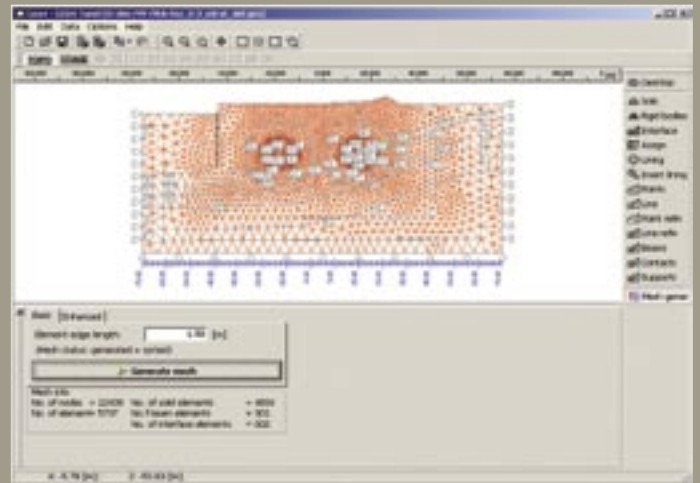
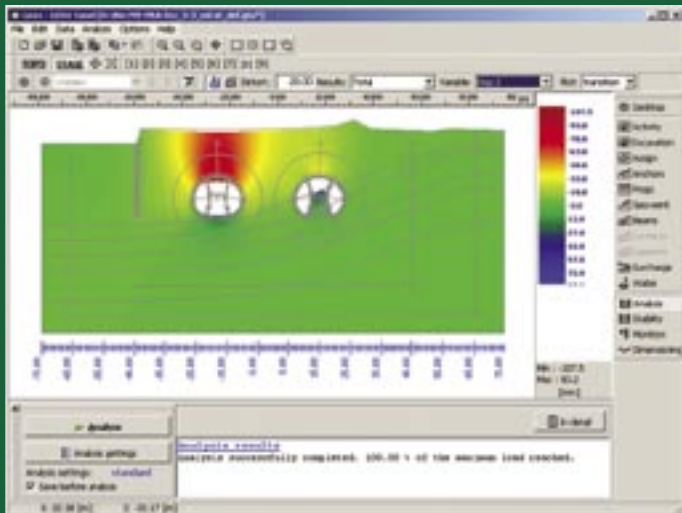
Surcharge

The program allows the specification of any number of surcharges (strip, trapezoidal, line loading). The surcharge can act either on a specified interface or anywhere in the soil/rock body. In a subsequent stage the existing surcharge can be either modified by changing its magnitude or entirely removed from the analysis.



geotechnical software suite

GEO5



www.finesoftware.eu

Geotechnical Software

Water

There are the following options to introduce water into the analysis:

- The ground water table is input as a continuous interface above or below the terrain.
- The pore pressure values (or values of coefficient R_u) are provided by pore pressure isolines. The first isoline always coincides with the terrain, the other ones can be placed anywhere in the body. The values between individual isolines are found through linear interpolation.

Presentation of results

The program allows visualization of the:

- scalar variables in the form of isosurfaces or isolines (e.g., components of stress and strain fields and their invariants, displacements)
- deformed or undeformed structure
- distribution of internal forces along the beams
- vectors of deformation
- forces in anchors

Either the total values of individual variables or their changes from stage to stage can be displayed.

Stability analysis

At each stage of construction the program makes it possible to carry out a slope stability analysis of a given structure. In the course of analysis the program gradually reduces the basic strength parameters of the soil until failure. As a result the program provides a factor of safety corresponding to that obtained by classical analyses.



Plate

The program performs the analysis of foundation and roof plates of arbitrary shape using the finite element method. It features:

- Automatic mesh generator for both triangular and quadrilateral elements with mesh refinement around points or lines
- Modeling of subsoil or piled foundation under the plate
- Concentrated and distributed supports (fixed, spring)
- Simple introduction of beams using the cross-sectional parameters
- Loading due to prescribed forces or displacements
- Any number of loadcases
- Determination of the required bending and shear reinforcement

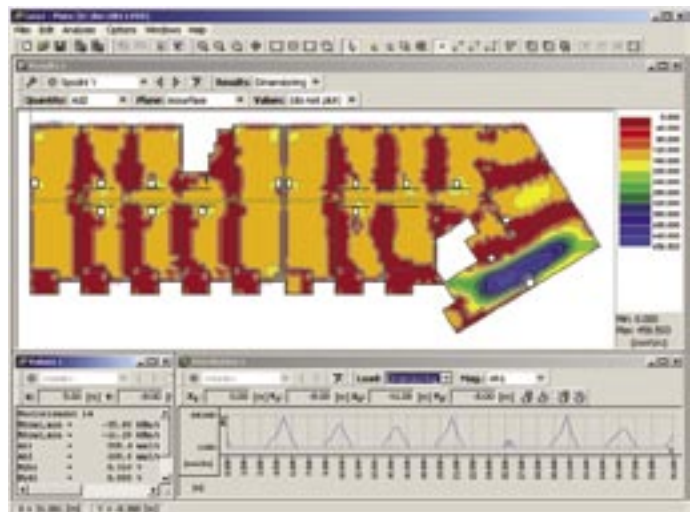


Tunnel

The Tunnel program is built upon the FEM program platform.

In addition it features:

- Two options for modeling lining (Beam elements - 1D or standard 2D elements)
- Option to create a geometrical model of the lining in a separate pre-processor using data in an external format (such as DXF).
- Simple introduction of the lining into the soil/rock body (topology correction and mesh generation are fully automatic)
- Any number of construction stages
- Loading of the lining geometry in DXF format
- Use of a relaxation factor to account for the 3D behavior of the body during excavation
- Introduction of hinges
- Loading due to temperature changes
- Consideration of soil swelling
- Consideration of lining shrinkage
- Analysis of a secondary lining either by changing the geometrical parameters (1D elements) or by changing the parameters of the original lining to allow for the transfer of stresses (2D elements)
- Introduction of monitoring points anywhere in the body
- Graphical representation of the distribution of internal forces on active parts of the lining
- Introduction of anchors fixed at their end points or along their length(s)
- Introduction of anchors as a region of improved soil mass (anchored region)



Your local dealer:



Fine Ltd.
Zaverka 12, 169 00 Prague 6
Czech Republic
hotline: +420 233 324 889
fax: +420 233 321 754
e-mail: hotline@finesoftware.eu
www.finesoftware.eu