

Driven tunnel – dynamic analysis of earthquake

Program: GEO5 FEM – Earthquake

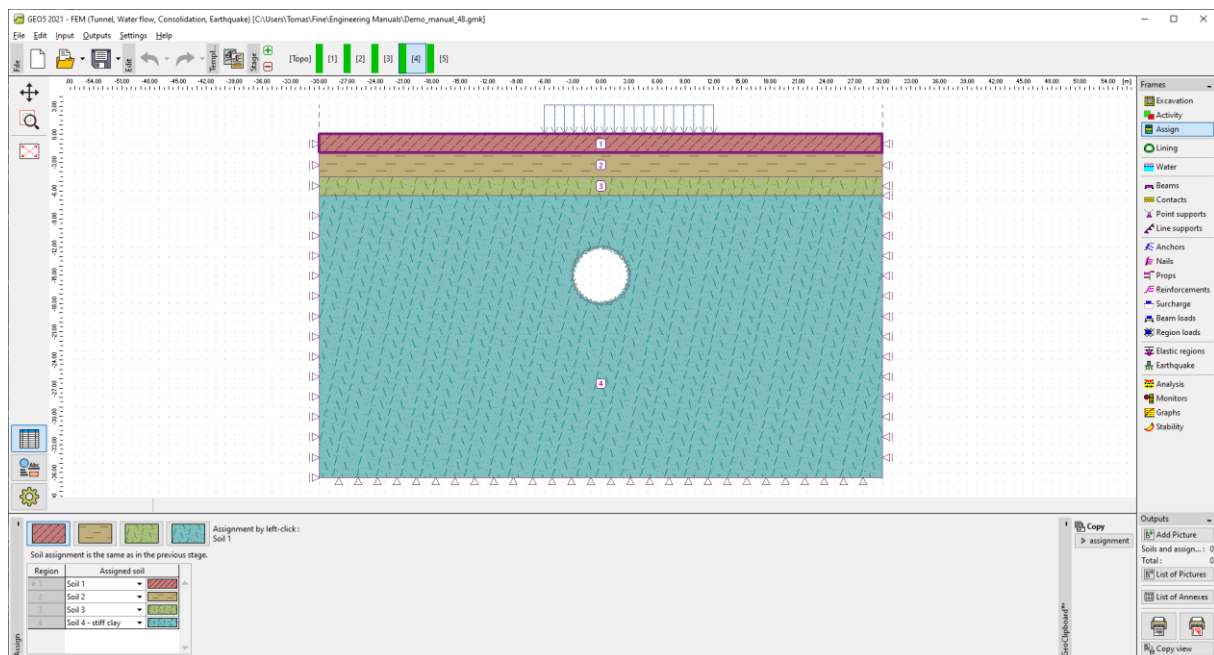
Files: Demo_manual_48_init.gmk, Demo_manual_48.gmk

The GEO5 FEM Earthquake module is used to perform a dynamic analysis of geotechnical structures loaded by the ground motion. The earthquake loading can be applied in an arbitrary construction stage of the existing model and its effect is analyzed as a dynamic mechanical problem. The result of the dynamic analysis is the time evolution of the displacement field, the total and the plastic strain field, and the stress field. Besides that, the program calculates the evolution of internal forces in the structural elements such as beams, anchors, etc. These quantities can be displayed at an arbitrary time step of the dynamic analysis.

In this manual, we provide an example of a dynamic analysis of a driven tunnel.

Problem description

We want to find the evolution of soil deformation and the evolution of internal forces in the lining of a driven tunnel during an earthquake. The tunnel is subjected to an earthquake in the last construction stage when all excavation steps are completed, and the lining is installed. The FE model adopted for this static analysis of all construction stages is stored in the “Demo_manual_48_init.gmk” file. The model contains the definition of the geological profile, material parameters, geometry, construction stages, and other inputs needed in the static analysis.



Geological profile and geometry of driven tunnel model

The assumed earthquake has the design peak ground acceleration $a_g = 2 \text{ m/s}^2$. For the individual soil layers, we assume the following values of dynamic elastic modulus and viscous damping ratio:

Soil	Dynamic modulus E_{dyn} [MPa]	Damping ratio ξ [-]
Layer 1	200	0.05
Layer 2	300	0.05
Layer 3	1200	0.05
Layer 4	1965	0.05

The main area of the model denoted as Layer 4 corresponds to clayey soil that extends dozens of meters below the bottom boundary of the existing FEM model.

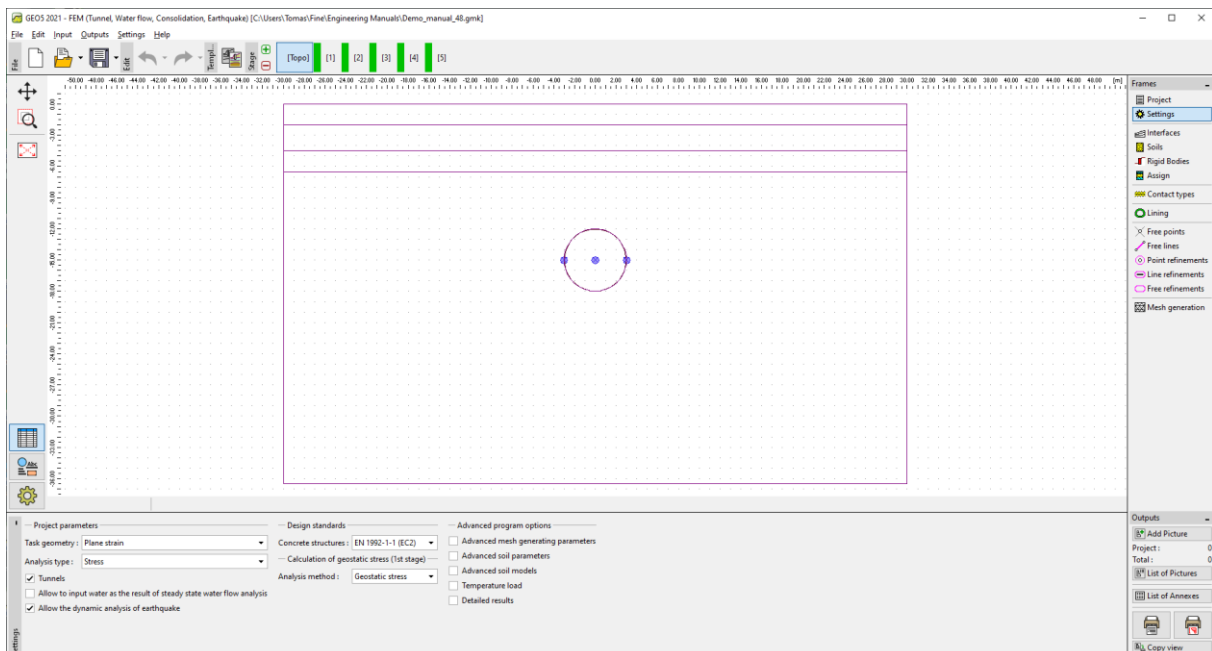
Besides the existing FE model, this manual builds on the “Eurocode 8: Design of structures for earthquake resistance” (Eurocode 8). This design code specifies the ground types A – E, S_1 and S_2 and corresponding elastic response spectra. GEO5 FEM Earthquake makes it possible to generate a response spectrum-compatible accelerogram which then enters the dynamic analysis.

Solution

We begin with the existing FE model which we enhanced by the dynamic analysis of earthquake. We start by opening the “Demo_manual_48_init.gmk” file and by saving it as a working copy “Demo_manual_48.gmk”.

Activation of dynamic earthquake analysis

The dynamic analysis of earthquake becomes available by checking the corresponding checkbox in the [Topo], Settings.



Frame Settings - here we activate the dynamic analysis of earthquake

With this option checked, the program requires inputting additional material parameters for the dynamic analysis. This allows us to run the dynamic analysis in each construction stage.

Material parameters for dynamic analysis

The next step in the model preparation is specifying the dynamic material parameters for each soil. These parameters are specified in the *[Topo]*, *Soils* frame. In the *Edit soil parameters* window in section *Earthquake*, we input:

- *Dynamic modulus of elasticity* – this is the Young modulus the soil exhibits in small deformations.
- *Type of damping* – we may choose between inputting the proportional damping ratio ξ or directly the Rayleigh damping parameters α and β . The meaning of these parameters is explained in “GEO5 - FEM - Earthquake - Theoretical Guide” available at <https://www.finesoftware.eu/user-guides/>. The most common value of damping ratio in geotechnical applications is $\xi = 5\%$.

The screenshot shows the 'Edit soil parameters' dialog box. The 'Earthquake' tab is selected, displaying the following parameters:

- Dynamic modulus of elasticity:** $E_{dyn} = 200.00$ [MPa]
- Type of damping:** proportional damping ratio
- Proportional damping ratio:** $\xi = 5.0$ [%]

Other tabs visible include:

- Identification:** Name: Soil
- Material model:** Mohr - Coulomb
- Basic data:** Unit weight: $\gamma = 20.00$ [kN/m³], Elastic modulus: $E = 15.00$ [MPa], Stiffness according to depth: constant, Poisson's ratio: $\nu = 0.35$ [-]
- Uplift pressure:** Calc. mode of uplift: standard, Saturated unit weight: $\gamma_{sat} = 20.00$ [kN/m³]
- Model Mohr - Coulomb:** Modulus unloading / reloading: $E_{ur} = 45.00$ [MPa], Angle of internal friction: $\varphi_{int} = 28.00$ [°], Cohesion of soil: $c_{int} = 8.00$ [kPa], Dilatation angle: $\psi = 0.00$ [°]
- Draw:** Pattern category: GEO, Search: , Subcategory: Soils (1 - 16), Pattern: 1 Silt, Color: , Background: automatic, Saturation <10 - 90>: 50 [%]

The dynamic modulus and damping are required to input in the Soil properties window

Next, we input the values of the dynamic modulus and the damping ratio listed in the table above. The materials are now defined, so we may continue with the settings of the earthquake parameters in the chosen construction stage.

Activating earthquake analysis in selected stage

We may run the dynamic analysis in the chosen construction stage by checking the *Analyze earthquake* in the *Earthquake* frame. With this option the program runs the following analyses in a sequel:

1. Static stress analysis
2. Modal analysis
3. Free-field analysis
4. Dynamic analysis of earthquake effect

We can specify the input for the static analysis (adding surcharge, activating or deactivating regions, installing structural elements, etc.) as usual. The dynamic analysis then starts from the equilibrated state at the end of the static analysis.

Our existing model with static analysis has four construction stages: 1. calculation of geostatic stress, 2. surcharge on terrain, 3. partial excavation of the tunnel cross-section, 4. installation of lining and completing the excavation. The fourth stage, therefore, corresponds to the final construction stage. It would be possible to choose *Analyze earthquake* in this stage. Nevertheless, since it is good practice to keep construction stages as simple as possible, we add yet another calculation stage (the fifth one), in which we define the earthquake. Since there is no additional loading acting in the static analysis in this stage, its results will be identical to the results of the previous stage.

Boundary conditions at the bottom of the model

At this fifth stage, we choose in the *Earthquake* frame the boundary conditions at the bottom of the model. Available options are:

- *Fixed* boundary conditions
- *Absorbing* boundary conditions

The *fixed boundary conditions* are used in cases when the bottom boundary of the model represents an interface between a relatively soft and very stiff material, e.g. overlying soil and bedrock. These boundary conditions reflect the downwards traveling waves back to the model.

The *absorbing boundary conditions* correspond to a situation where there is no material interface at the level of the bottom boundary and the material assigned to the lowest region extends below the model region. Assuming that the underlying material deforms only elastically, these boundary conditions completely damp the downwards traveling waves thus simulating that the departing waves freely pass through the bottom boundary and continue to travel downwards outside the model region.

Since the clayey soil in layer 4 in our model extends dozens of meters below the model and therefore there is no material interface at the bottom boundary, we choose the *absorbing* boundary conditions.

Direction of accelerogram

The course of the earthquake is defined in a form of a list of time instants and corresponding accelerations (an accelerogram) of the incoming wave at the bottom boundary. The program allows for inputting either horizontal or vertical components of the accelerogram, or a combination of both. A horizontal accelerogram prescribed on the bottom boundary creates an upwards traveling shear wave (S-wave). A vertical accelerogram prescribed at the bottom boundary creates an upwards traveling pressure wave (P-wave). For simplicity, we will use only the horizontal accelerogram in our model.

Artificial accelerogram

Eurocode 8 allows us to describe the seismic motion using artificially generated accelerograms. These accelerograms, however, must be generated to match the required elastic response spectrum. Eurocode 8 defines different elastic response spectra for different ground types denoted A-E. Therefore, we first need to determine the ground type and then find the corresponding response spectrum parameters. For these parameters, the program generates a spectrum-compatible accelerogram.

Ground types

The ground types are defined in Eurocode 8. The ground in our model meets the description of Type B: "Deposits of very dense sand, gravel, or a very stiff clay, at least several tens of meters in thickness, characterized by a gradual increase of mechanical properties with depth." For Type B the design code further defines the typical range of an average shear wave velocity in the upper 30 meters of the soil profile. The range is between 360 m/s and 800 m/s.

The average shear wave velocity in the upper 30 meters is according to the Eurocode 8 calculated as

$$v_{s,30} = \frac{30}{\sum_{i=1}^N \frac{h_i}{v_i}}$$

where h_i denotes the thickness of the i -th layer measured in meters.

The shear wave velocity of the i -th layer is calculated as

$$v_i = \sqrt{\frac{G_{dyn,i}}{\rho_i}}$$

where

$$G_{dyn,i} = \frac{E_{dyn,i}}{2(1 + \nu_i)}$$

denotes the dynamic shear modulus, $\rho_i = \gamma_i/g$ is the mass density, $E_{dyn,i}$ is the dynamic Young modulus, ν_i is the Poisson ratio, γ_i is specific weight, and g represents the gravitational acceleration.

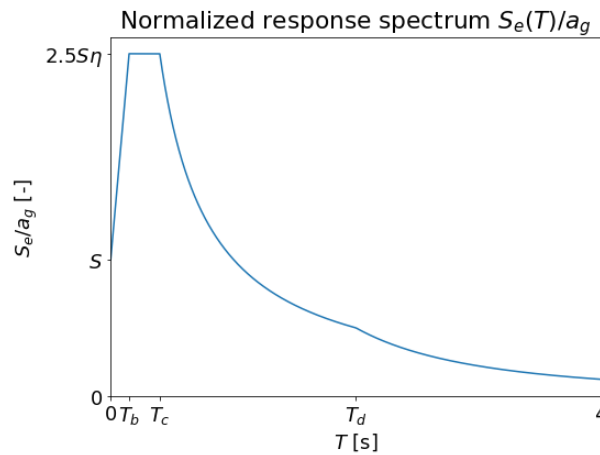
Calculation of average shear wave velocity

i	E [Mpa]	ν [-]	G [Mpa]	γ [kN/m3]	ρ [kg/m3]	v [m/s]	h [m]	h/v [s]
1	200	0.35	74.1	20	2000	192.5	2	0.0104
2	300	0.35	111.1	20	2000	235.7	2.5	0.0106
3	1200	0.3	461.5	21	2100	468.8	2	0.0043
4	1965	0.3	755.8	21	2100	599.9	23.5	0.0392
							sum(h_i/v_i) =	0.0644
							vs,30 =	465.6

The obtained shear wave velocity $v_{s,30} = 465.6$ m/s confirms that the ground is indeed of type B.

Elastic response spectrum

The Eurocode 8 introduces the elastic ground acceleration response spectrum defined by the parameters listed for each type of ground type. For ground type B and Type* 1 of the response spectrum, the document prescribes $S = 1.2$, $T_B = 0.15$ s, $T_C = 0.5$ s, $T_D = 2.0$ s. The meaning of these parameters shows the following figure.



Parametric elastic response spectrum defined in Eurocode 8

Generating accelerogram

If we want to generate a stationary accelerogram, i.e. an accelerogram without the characteristic rise, strong motion, and decay periods, we check the *Stationary* option. We assume a nonstationary accelerogram in our model, so we leave the option unchecked.

* Eurocode 8 also distinguishes between two types of spectra denoted as Type 1 and Type 2. For simplicity, we consider only Type 1 in this example.

In the next step, we enter the parameters common to the horizontal and the vertical accelerogram. These are:

Duration of earthquake t_s – takes input in seconds. It is the total duration of the generated accelerogram including the rise, the strong motion, and the decay periods.

Number of steps – is the number of time points for which the acceleration is generated. GEO5 FEM employs the Fast Fourier transform (FFT) in the accelerogram generating algorithm and therefore it is recommended to use 2^n time steps, e.g. 256, 512, 1024, etc. Reasonable time step size has the order of magnitude of hundredths of seconds.

Proportional damping ratio ξ – this damping ratio appears in the definition of required elastic response spectra. It is entered as a value between 0 and 1.

Damping correction factor η – for the most common value of damping ratio $\xi = 5\%$ the Eurocode 8 gives the value of the damping correction factor $\eta = 1$. For different values, the document prescribes the formula $\eta = \sqrt{10/(5 + \xi)}$. Damping ratio ξ is specified in units of percent.

Alternative – this is the seed of the pseudorandom number generator. When this value is changed, the program generates different accelerograms with analogous elastic response spectrum. This option is useful when exposing the model to several different artificial accelerograms as required for example by the Eurocode 8.

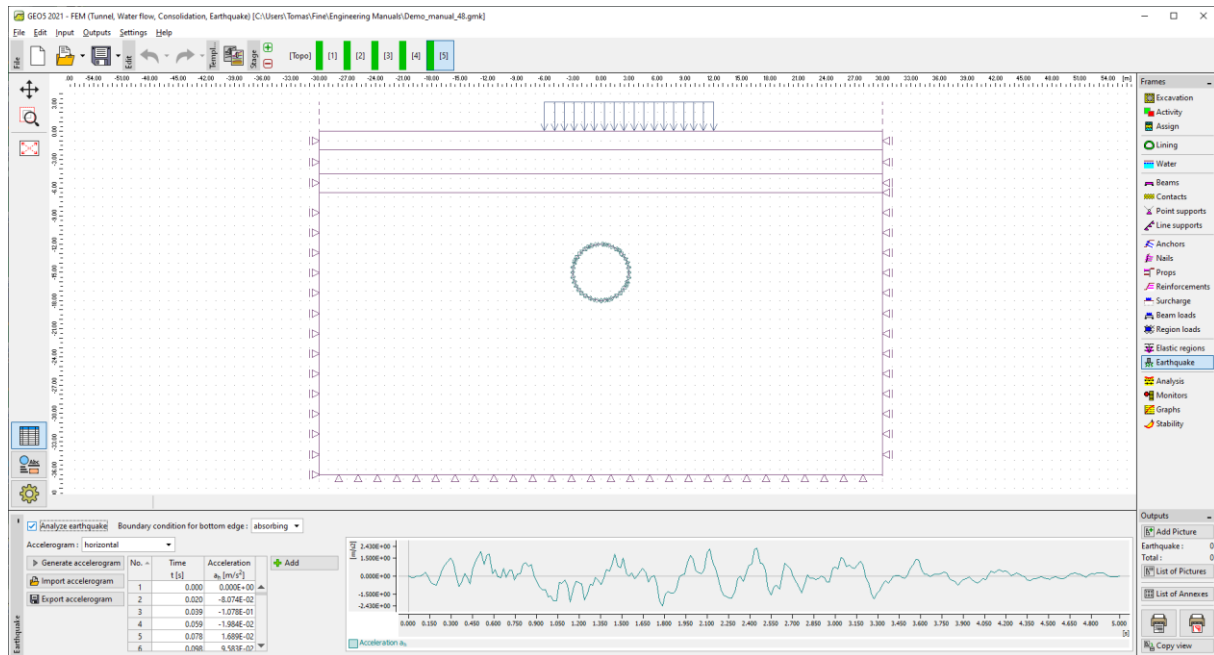
No.	Time t [s]	Acceleration a _h [m/s²]
1	0.000	0.000E+00
2	0.020	-8.074E-02
3	0.039	-1.078E-01
4	0.059	-1.984E-02
5	0.078	1.689E-02
6	0.098	9.583E-02
7	0.118	3.465E-01
8	0.137	3.360E-03
9	0.157	-5.320E-01
10	0.176	-7.026E-01
11	0.196	-7.188E-01
12	0.216	-1.070E-01
13	0.235	5.440E-01
14	0.255	9.114E-01
15	0.275	1.223E+00
16	0.294	1.494E+00
17	0.314	1.163E+00
18	0.333	-6.029E-02
19	0.353	-8.387E-01
20	0.373	-5.320E-01

The Generating accelerogram window with generated values of time steps and accelerations

Next, we need to enter the parameters of the required elastic response spectrum. We specify these parameters separately for the horizontal and vertical accelerogram. The values of these parameters for a particular ground type can be found in Eurocode 8.

The last part of the form serves to set the parameters defining the envelope function. Parameter ε_m determines time $t = \varepsilon_m t_s$, at which the envelope function attains its maximum. Parameter η_m controls the intensity of accelerations at the end of the accelerogram. It is a ratio between the value of the envelope function at the end of the accelerogram in time t_s and the maximum at $t = \varepsilon_m t_s$.

After defining all the above parameters, we generate the accelerogram by clicking *Generate*. Clicking *OK* closes the window and the generated record is shown in the *Earthquake* frame.



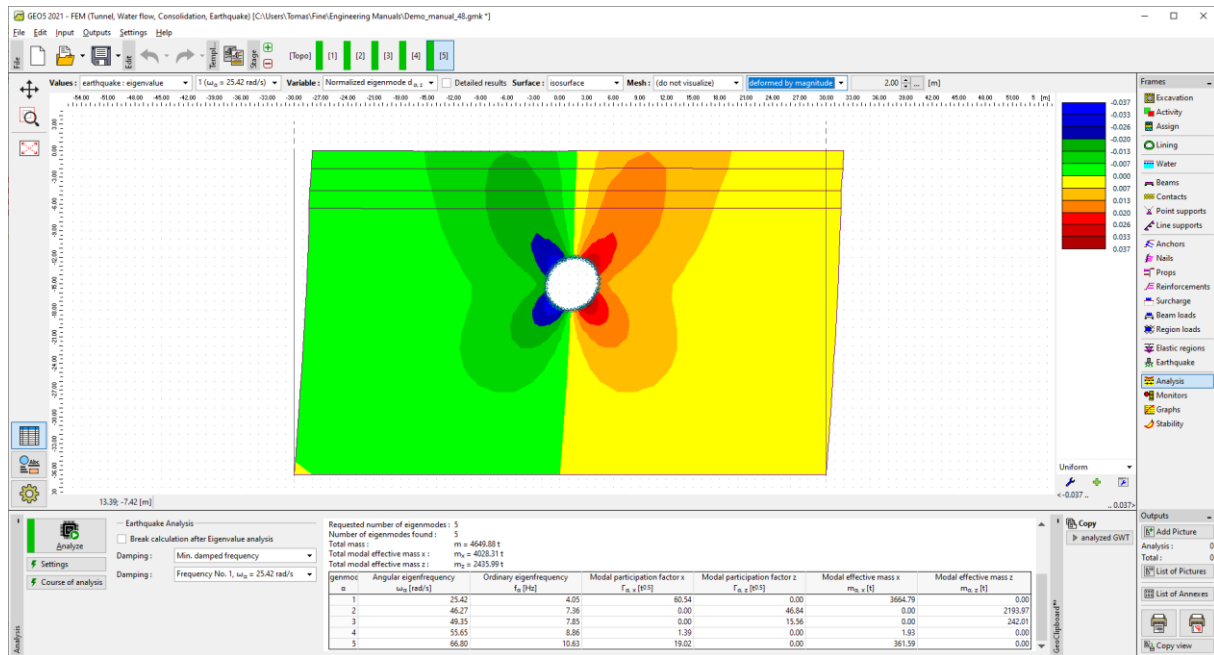
Earthquake frame with generated accelerogram

Analysis of eigenmodes and eigenfrequencies

We defined material damping via the damping ratio ξ and therefore it is necessary to specify which frequency (or pair of frequencies) will be dampened by this value. In GEO5 FEM Earthquake, these frequencies can be chosen from eigenfrequencies of the particular model. The eigenfrequencies are computed prior to running the dynamic analysis.

Let's assume that the least damped frequency in our model is the first eigenfrequency that corresponds to horizontal motion. Nevertheless, for better insight, we wish to calculate the five lowest eigenfrequencies and the corresponding eigenmodes. In the *Analysis settings* dialog, tab *Earthquake*, we set the *Requested number of eigenmodes* to five. This ensures that the algorithm will search at least the first five eigenmodes and eigenfrequencies. We also set the *Boundary support for eigenvalue analysis* to *vertical*. This choice enforces zero vertical displacements for the eigenmodes. Such a requirement is reasonable since we assume the horizontal accelerogram only.

We close the *Analysis settings* dialog and check *Break calculation after Eigenvalue analysis* in the *Analysis* frame. This tells the program to run the static and the eigenmode analyses. The analysis is executed by clicking the *Analyze* button.



Results of the modal analysis

The results of the modal analysis are available in the lower part of the *Analysis* frame. The program informs us that the algorithm found the first five eigenmodes as requested. The associated eigenfrequencies are listed in a table together with modal participation factors and modal effective masses. A modal effective mass tells us how much the associated eigenmode contributes to motion in a given direction (horizontal or vertical). In our model, the first modal effective mass in x -direction is equal to 3657.92 metric tons. The total mass of the model is equal to 4649.88 metric tons. This shows that the predominant frequency of horizontal motion is the first eigenfrequency $\omega_1 = 25.36$ rad/s and therefore we select it as the least damped frequency. The individual eigenmodes can be viewed if we choose *Values: earthquake: eigenvalue*. It is also possible to select *Mesh: deformed by magnitude* for more visualization.

Dynamic analysis

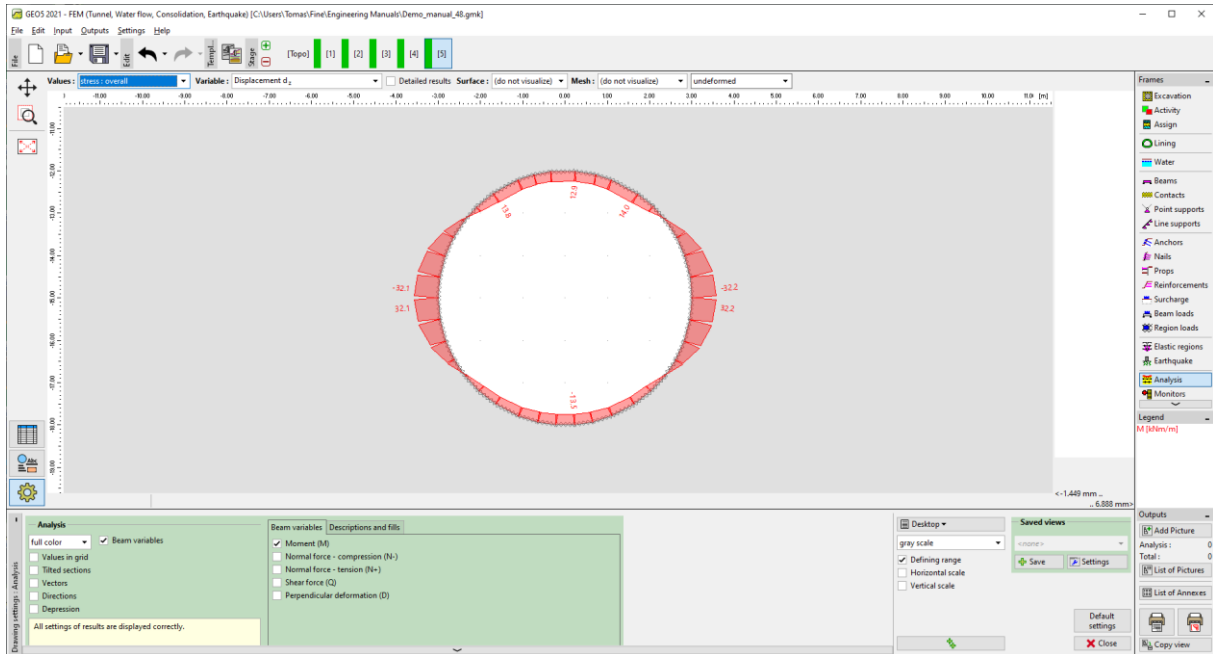
The damping details are set, and we can execute the dynamic analysis. We clear the option *Break calculation after Eigenvalue analysis* and click the *Analyze* button. As written earlier in this manual, the 2D dynamic analysis is preceded by the so-called free-field analysis. This is a 1D dynamic analysis of shear and pressure wave traveling through a layered subsoil that corresponds to the layers found along the left and right boundary of the model. The results of the free-field analysis then enter the special type of boundary conditions applied along the vertical boundaries in the 2D dynamic analysis. These *free-field boundary conditions* ensure that the departing mechanical waves are not reflected back to the model[†].

The results at the end of the earthquake are displayed by choosing *Values: earthquake: overall*.

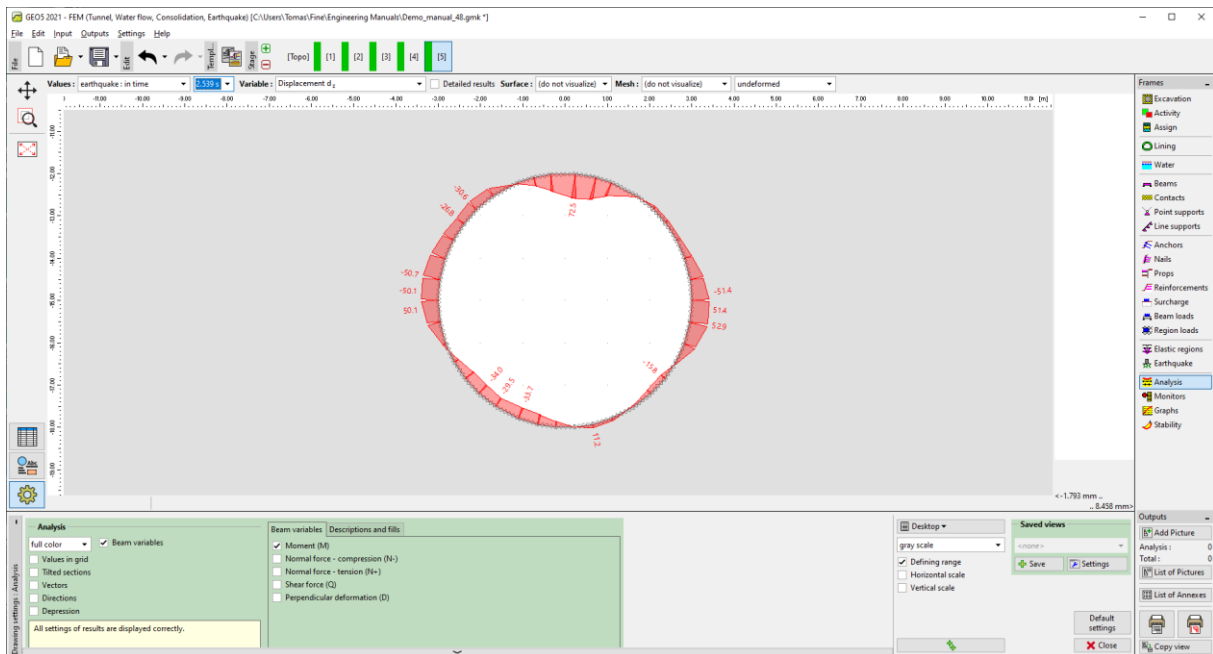
The results at individual time steps are displayed by choosing *Values: earthquake: in time*. If we go through individual time steps and want to display the deformed model, it is convenient to unify the scaling of the deformed mesh by choosing *Mesh: deformed by value*.

[†] The fixed as well as the free boundary conditions reflect the mechanical waves back to the model. Therefore, neither of these two types can be used for the vertical boundaries and the free-field boundary conditions are implemented in GEO5 FEM Earthquake.

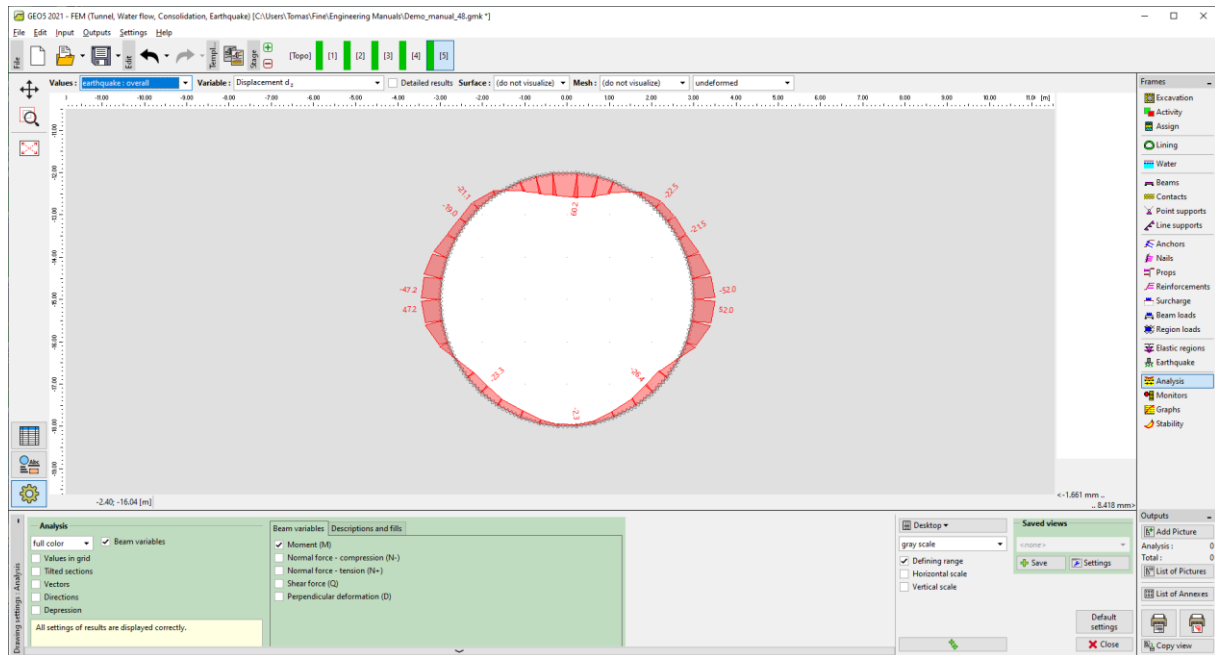
To visualize the bending moments, we need to set *Mesh: undeformed* and check *Beam variables* and *Moment* in the *Drawing settings: Analysis* panel which is opened by clicking the gear button in the bottom-left part of the program's window. The following figures compare the bending moments in the lining at the end of static analysis (before the earthquake), at time $t = 2.539$ s when the bending moment reaches its maximum, and at the end of the dynamic analysis.



Distribution of bending moments before the earthquake

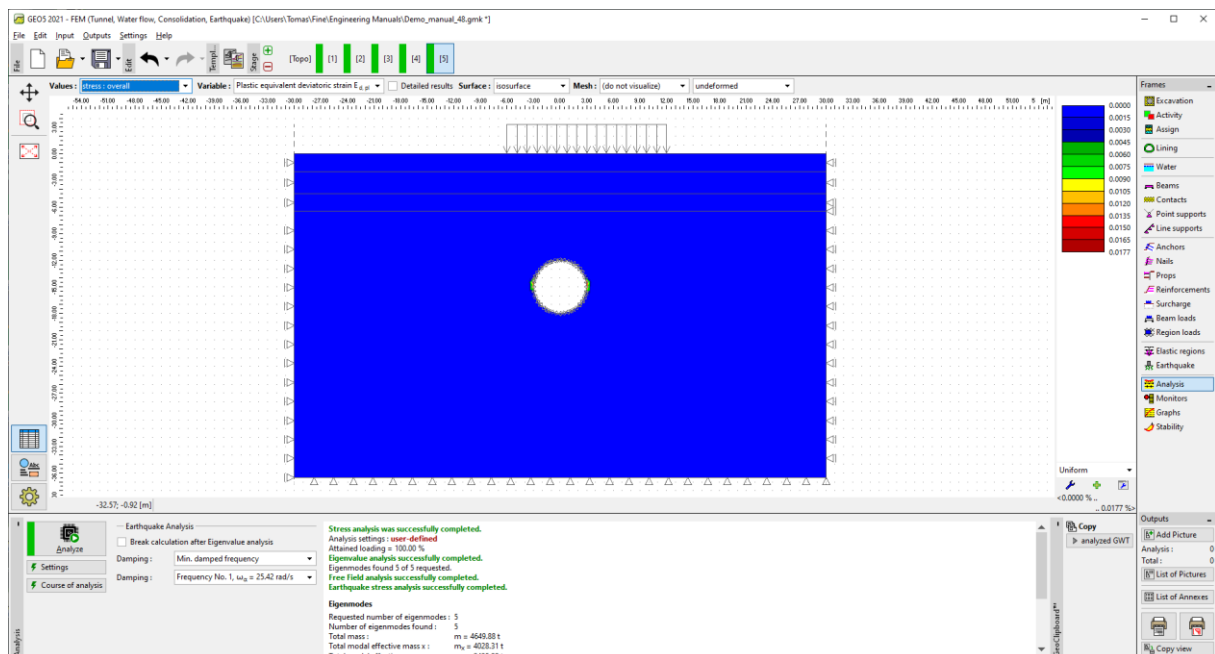


Distribution of bending moments at time $t = 2.539$ s

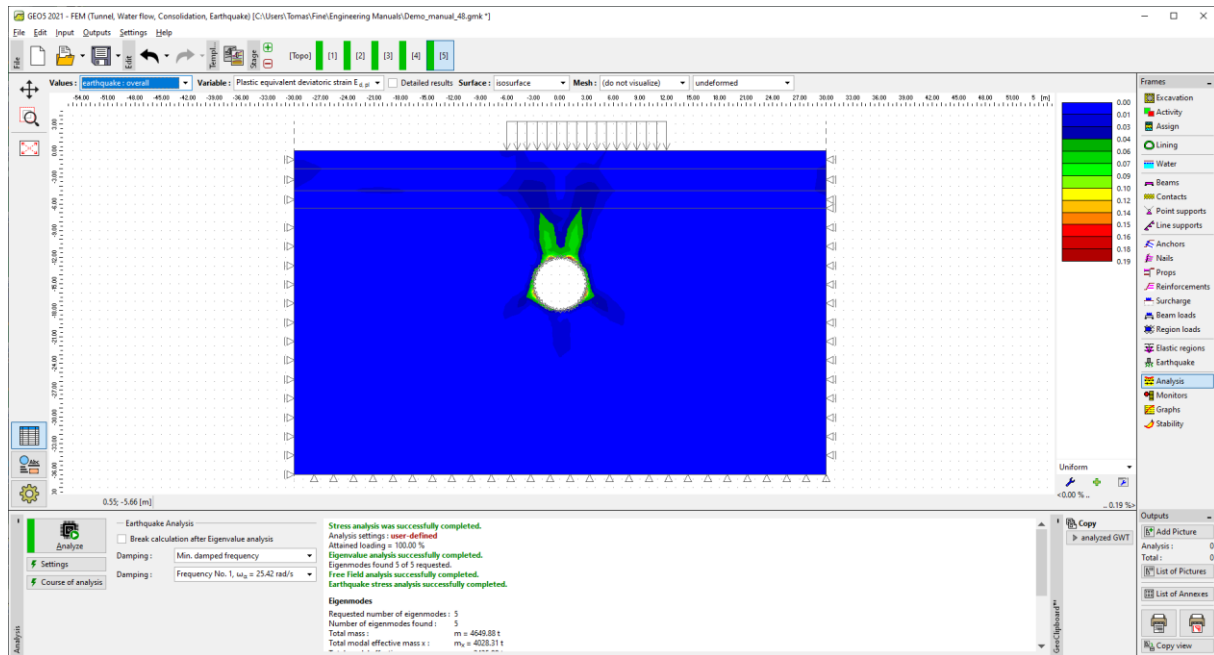


Distribution of bending moments at the end of dynamic analysis

The permanent change in the bending moment distribution is related to the redistribution of the stress due to the evolution of plastic zones and associated permanent deformations of the soil body. The plastic zones at the end of static and dynamic analyses are plotted in the following figures.



Areas of the equivalent deviatoric plastic strain before the earthquake



Areas of the equivalent deviatoric plastic strain at the end of dynamic analysis

Conclusions

In this manual, we extended the existing model in "Demo_manual_48_init.gmk" file with inputs that are necessary for a dynamic analysis of the earthquake. According to Eurocode 8 we determined the ground type and generated one response spectrum-compatible accelerogram. Based on the analysis of eigenmodes and eigenfrequencies we chose the first eigenfrequency as the least damped frequency.

The results of the dynamic analysis show a significant evolution of plastic regions around the tunnel and an increase of the maximum bending moment from 32.1 kNm (the maximum moment before earthquake located on the sides of the tunnel tube) to 60.2 kNm (the maximum moment at the end of the dynamic analysis located at the tunnel crown). The maximum bending moment during the earthquake was 72.5 kNm.