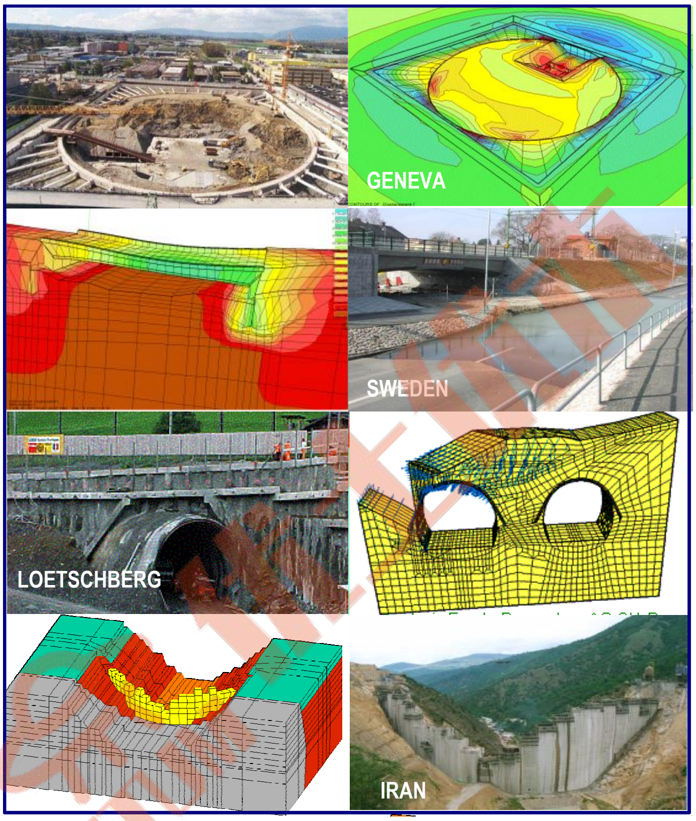
ZSOIL.PC 2020

USER MANUAL

TUTORIALS



Soil, Rock and Structural Mechanics
in dry or partially saturated media

Zace Services Ltd, Software engineering
P.O.Box 2, 1015 Lausanne, Switzerland
Tel.+41 21 802 46 05, Fax 802 46 06
https://www.zsoil.com, Hotline hotline@zsoil.com

TUTORIALS

ZSoil®.PC 2020 manual

A. Truty Th. Zimmermann K. Podleś R. Obrzud
with contribution by A. Urbański and S. Commend

+STRUCTURES
since 1985

Zace Services Ltd, Software engineering
P.O.Box 2, CH-1015 Lausanne
Switzerland
(T) +41 21 802 46 05
(F) +41 21 802 46 06
http://www.zsoil.com,
hotline: hotline@zsoil.com

WARNING

ZSoil.PC is regularly updated for minor changes. We recommend that you send us your e-mail, as ZSoil owner, so that we can inform you of latest changes. Otherwise, consult our site regularly and download free upgrades to your version.

Latest updates to the manual are always included in the online help, so that slight differences with your printed manual will appear with time; always refer to the online manual for latest version, in case of doubt.

ZSoil.PC 2020 manual:
1. Data preparation
2. Tutorials and benchmarks
3. Theory

ISBN 2-940009-08-2

Published by Elmepress International, Lausanne, Switzerland

END-USER LICENSE AGREEMENT FOR ZACE's ZSoil® SOFTWARE

Applicable to all V2020 versions: professional & academic, single user & networks, under Windows 7, 8, 10

Read carefully this document, it is a binding agreement between you and Zace Services SA (Zace) for the software product identified above. By installing, copying, or otherwise using the software product identified above, you agree to be bound by the terms of this agreement. If you do not agree to the terms of this agreement, promptly return the unused software product to the place from which you obtained it for full refund of price paid. ZACE SERVICES SA OFFERS A 60 DAYS MONEY-BACK GUARANTEE ON ZSOIL.

ZSOIL (the Software & associated hotline services when applicable) SOFTWARE PRODUCT LICENSE:

ZSOIL Software is protected by copyright laws and international copyright treaties, as well as other intellectual property laws and treaties. The **ZSOIL** software product is licensed, not sold.

1. GRANT OF LICENSE

   A: **Zace Services SA (Zace)** grants you, the customer, a non-exclusive license to use $\mathbf{N}_{\text{bought}}$ (= the number of licenses bought) copies of **ZSOIL**. You may install copies of **ZSOIL** on an unlimited number of computers, provided that you use only $\mathbf{N}_{\text{bought}}$ copies at the time.

   B: You may make an unlimited number of copies of documents accompanying **ZSOIL**, provided that such copies shall be used only for internal purposes and are not republished or distributed to any third party.

   C: Duration of the agreement may be limited or unlimited, depending on license purchased. Time unlimited licenses of ZSOIL V2020 will be supported for a period of 4 years starting from date of purchase. This support is limited to ZSOIL V2020 upgrades, under Windows 7,8,10.

2. COPYRIGHT
   All title and copyrights in and to **the Software** product (including but not limited to images, photographs, text, applets, etc), the accompanying materials, and any copies of **ZSOIL** are owned by **Zace Services SA**. ZSOIL is protected by copyright laws and international treaties provisions. Therefore, you must treat **ZSOIL** like any other copyrighted material except that you may make copies of the software for backup or archival purposes or install the software as stipulated under section 1 above.

3. OTHER RIGHTS AND LIMITATIONS

   A: Limitations on Reverse Engineering, Decompilation, Disassembly. You may not reverse engineer, decompile, or disassemble **the Software**.

   B: No separation of components. **ZSOIL** is licensed as a single product and neither **the Software's** components, nor any upgrade may be separated for use by more than $\mathbf{N}_{\text{bought}}$ user(s) at the time.

   C: Rental. You may not lend, rent or lease the software product.

   D: Software transfer. You may permanently transfer all of your rights under this agreement and within the territory (country of purchase and delivery), provided you do not retain any copies, and the recipient agrees to all the terms of this agreement.

   E: Termination. Without prejudice to any other rights, **Zace Services SA** may terminate this agreement if you fail to comply with the conditions of this agreement. In such event, you must destroy all copies of **the Software**.

WARRANTIES & LIMITATIONS TO WARRANTIES

1. DISCLAIMER
   ZSOIL, developed by **Zace Services SA** is a finite element program for the analysis of above- and underground structures in which soil/rock & structural models are used to simulate the soil, rock and/or structural behaviour. The ZSOIL code and its soil/rock & structural models have been developed with great care. Although systematic testing and validation have been performed, it cannot be guaranteed that the **ZSOIL** code is free of errors. Moreover, the simulation of geotechnical and/or structural problems by means of the finite element method implicitly involves some inevitable numerical and modelling errors. **ZSOIL** is a tool intended to be used by trained professionals only and is not a substitute for the user's professional judgment or independent testing. The accuracy at which reality is approximated depends highly on the expertise of the user regarding the modelling of the problem, the understanding of the soil and structural models and their limitations, the selection of model parameters, and the ability to judge the reliability of the computational results. Hence, **ZSOIL** may only be used by professionals that possess the aforementioned expertise. The user must be aware of his/her responsibility when he/she uses the computational results for geotechnical design purposes. **Zace Services SA** cannot be held responsible or liable for design errors that are based on the output of **ZSOIL** calculations. The user is solely responsible for establishing the adequacy of independent procedures for testing the reliability, accuracy and completeness of any output of **ZSOIL** calculations.

2. LIMITED WARRANTY
   Zace Services SA warrants that **ZSOIL** will a) perform substantially in accordance with the accompanying written material for a period of 90 days from the date of receipt, and b) any hardware accompanying the product will be free from defects in materials and workmanship under normal use and service for a period of one year, from the date of receipt.

3. CUSTOMER REMEDIES
   Zace Services SA entire liability and your exclusive remedy shall be at **Zace's** option, either a) return of the price paid, or b) repair or replacement of the software or hardware component which does not meet Zace's limited warranty, and which is returned to **Zace Services SA**, with a copy of proof of payment. This limited warranty is void if failure of **the Software** or hardware component has resulted from accident, abuse, or misapplication. Any replacement of software or hardware will be warranted for the remainder of the original warranty period or 30 days, whichever is longer.

NO OTHER WARRANTIES.

YOU ACKNOWLEDGE AND AGREE THAT ZSOIL IS PROVIDED ON AN “AS IS” AND “AS AVAILABLE” BASIS AND THAT YOUR USE OF OR RELIANCE UPON ZSOIL AND ANY THIRD PARTY CONTENT AND SERVICES ACCESSED THEREBY IS AT YOUR SOLE RISK AND DISCRETION. ZACE SERVICES SA AND ITS AFFILIATES, PARTNERS, SUPPLIERS AND LICENSORS HEREBY DISCLAIM ANY AND ALL REPRESENTATIONS, WARRANTIES AND GUARANTIES REGARDING ZSOIL AND THIRD PARTY CONTENT AND SERVICES, WHETHER EXPRESS,IMPLIED OR STATUTORY. TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, ZACE SERVICES SA DISCLAIMS ALL OTHER WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE, WITH REGARD TO THE SOFTWARE PRODUCT, AND ANY ACCOMPANYING HARDWARE. NO LIABILITY FOR CONSEQUENTIAL DAMAGES. TO THE MAXIMUM EXTENT PERMITTED BY LAW, IN NO EVENT SHALL ZACE SERVICES SA HAVE ANY LIABILITY (DIRECTLY OU INDIRECTLY) FOR ANY SPECIAL INCIDENTAL, INDIRECT, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING, WITHOUT LIMITATION, DAMAGES FOR LOSS OF BUSINESS, PROFITS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION, OR ANY OTHER PECUNIARY LOSS) ARISING OUT OF THE USE OF OR INABILITY TO USE ZSOIL, EVEN IF ZACE SERVICES SA HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

OTHER PROVISIONS.

HOTLINE: If included, during the first year following purchase, hotline assistance will be provided by **Zace Services SA**, by e-mail exclusively. This service excludes all forms of consulting on actual projects. This hotline assistance can be renewed, for following years (max.3), at an average cost of 15% of current full package price.
PROFESSIONAL VERSIONS of **ZSOIL** are meant to be used in practice & in research centers.
ACADEMIC VERSIONS of **ZSOIL** are meant to be used exclusively for teaching and research in academic institutions.
ACADEMIC WITH CONSULTING VERSIONS of **ZSOIL** are meant to be used exclusively for teaching, research and consulting in academic institutions.

APPLICABLE LAW AND JURISDICTION THIS AGREEMENT IS GOVERNED BY THE (SUBSTANTIVE) LAWS OF SWITZERLAND, ALL DISPUTES ARISING OUT OF OR IN CONNECTION WITH THIS AGREEMENT OR THE USE OF ZSOIL SHALL EXCLUSIVELY BE SETTLED BY THE ORDINARY COURTS OF CANTON DE VAUD (ARRONDISSEMENT DE LAUSANNE)

LAUSANNE 18.02.2020

Contents of Tutorial

PREFACE

Examples contained in this part are designed:

- to illustrate some aspects of data preparation technique,
- to demonstrate program capabilities to deal with different real-live cases,
- to show most important aspects of post-processor usage.

Actions described during model generation are optimal ones, in the sense of minimizing user effort, and to automate data generation process as much as possible.

All examples included in this chapter will be shortly described first, then all major steps to create the model, compute and visualize results will be presented in form of video films activated from the online manual.

For the theoretical background, see *THEORETICAL MANUAL*. For the systematic description od data preparation technique, see *DATA PREPARATION*.

2D PROBLEMS

3D PROBLEMS

Chapter 1

2D PROBLEMS

1.1 CUT STABILITY ANALYSIS

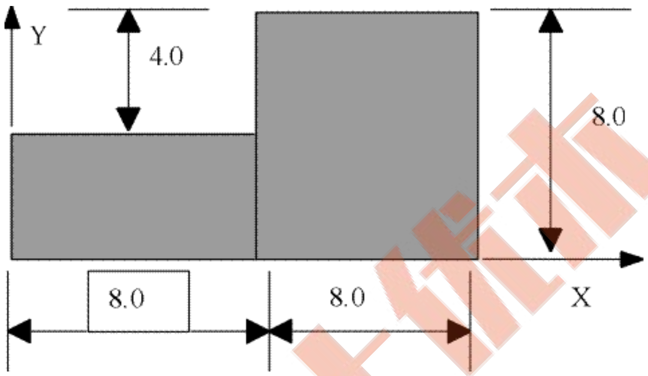
- **Data file: tutorials/cut.inp**

- **Description of the problem**

  The objective of this lesson is to get familiar with the Z_SOIL.PC environment, in particular with the graphic object-oriented preprocessor and post-processor. In this tutorial a macro-modeling approach, with semi-automatic meshing, will be used.

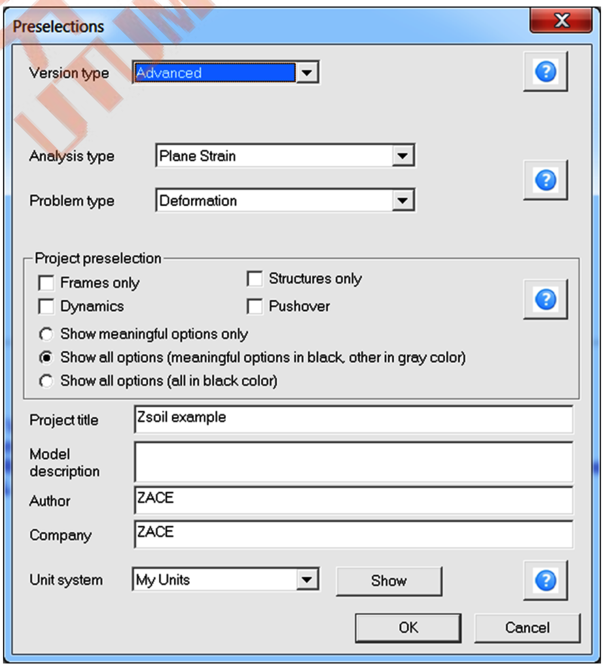
- **Engineering draft**

  The geometry of a 4 meter-high cut is shown in the figure below.



- **Project preselection**

  In the dialog box for project preselection (it appears automatically for option `File/New` in main ZSoil® menu) set ◎ **Plane strain** to ON and select `Deformation` item from the `Problem type` list. This way a single-phase, plane strain analysis is to be carried out.



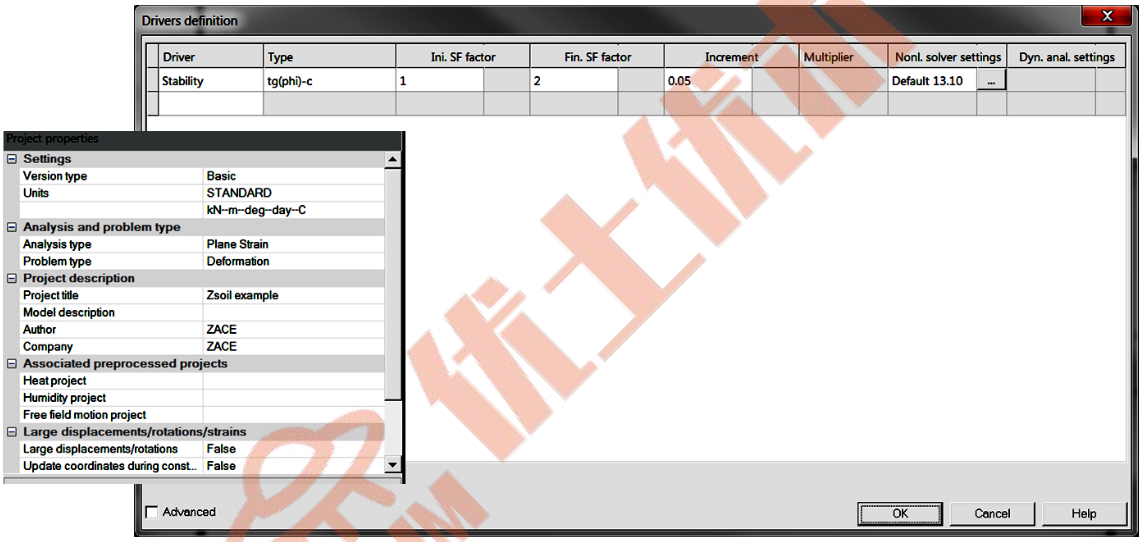
- **Drivers**

In ZSoil® program drivers list contains an information what is to be computed by the calculation module and in which order. In the geotechnical engineering to obtain correct results all major steps like excavation, stage construction, modification of the boundary conditions for solid and fluid must be taken into account and must reproduce all events as in the reality.

To set up the list of drivers use menu `Control/Analysis & drivers`. From list of drivers select `Stability`, from list Type select $tan(\phi) - c$ reduction algorithm, specify the initial value of the safety factor, its final value and increment and accept the setting with button **Modify** (note that the default driver is always set to the `Initial state` and for that reason it must be modified in our case).

In the considered case computation will start with the SF factor equal to 1.0, then 1.05, 1.10 etc... until divergence occurs (loss of stability is detected) or final value of the safety factor is achieved.

To learn on how to set up drivers list watch the video Set drivers



- **Materials**

In this example a single material, described by the Mohr-Coulomb elastic-perfect plastic model, is used. The material properties are given in the following table.

Material	Model	Data group	Properties	Unit	Value
1 soil	Mohr-Coulomb	Elastic	E	[kN/m^2]	40000
			ν	–	0.3
		Density	γ	[kN/m^3]	19
		Non-linear	ϕ	[°]	30
			ψ	[°]	0
			C	[kN/m^2]	16
			Rankine cut-off	–	OFF
			Dilatancy cut-off	–	OFF

To edit material properties use menu `Assembly/Materials`.

To learn on how to enter material data watch the video Edit materials .

- **Summary of all data preparation steps (video films)**

1 Create a new project
2 Create macro-model
3 Create meshes in subdomains
4 Create solid boundary conditions
5 Edit materials
6 Set drivers
7 Run computation
8 Visualize results

1.2 HOW TO RUN AN AXISYMMETRIC PROBLEM WITH A DRIVEN EXTERNAL FORCE

- **Data file: tutorials/foota.INP**
- **Description** The typical case of a circular footing is considered in this example. Figure 1.1 illustrates the general configuration of the footing.

  The conventional approach to the bearing capacity problem is to combine Rankine fields with Prandtl fields to satisfy statically admissible failure zones. Different combinations of these failure zones are possible leading to a plethora of possible solutions. To verify all these solutions there is a need to show that a unique displacement field corresponds to the stress field adopted in these solutions. This is almost impossible to achieve. ZSoil®.PC on the other hand uses an elegant scheme to find a solution that satisfies the requirements of the plasticity theory with compatible displacements and stresses.
- **Engineering draft**

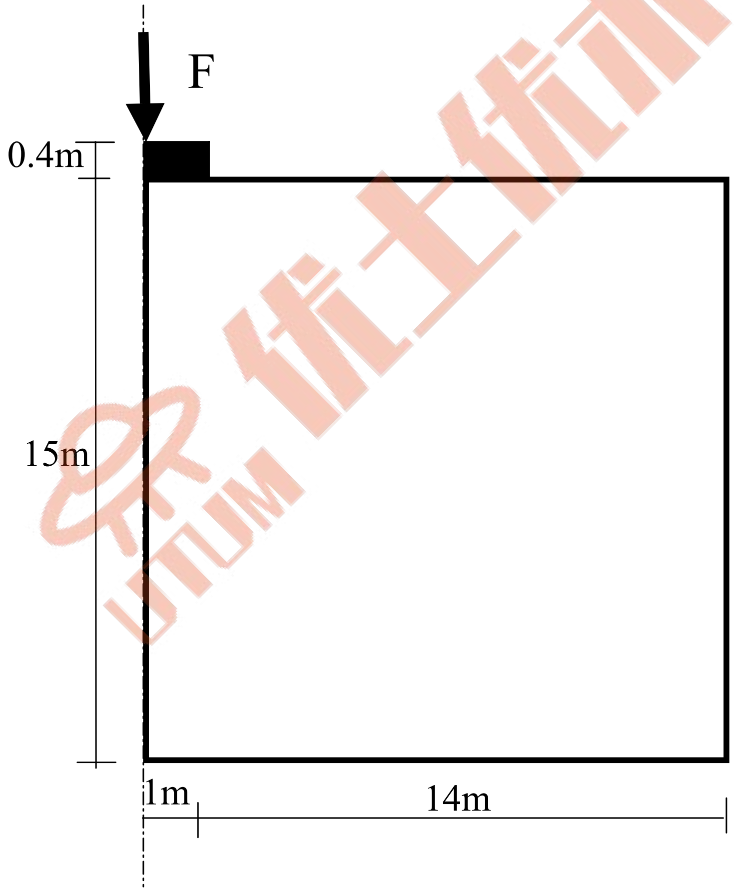
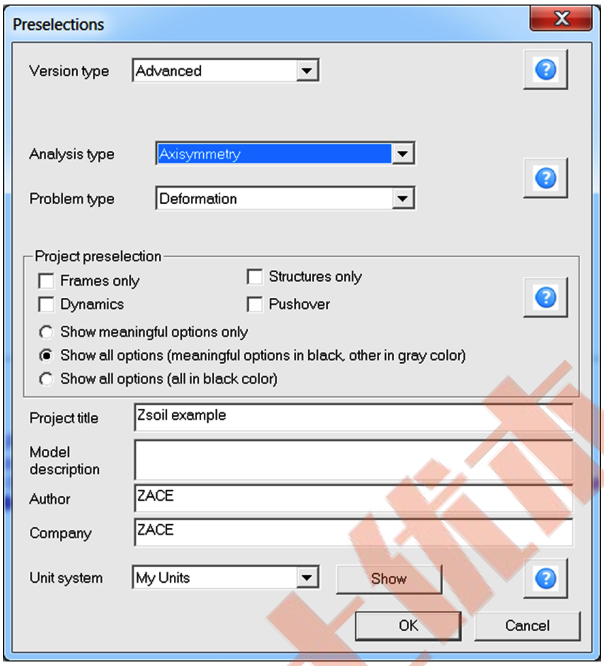


Figure 1.1: Axisymmetric footing problem

- **Project preselection**

In the dialog box for project preselection (it appears automatically for option `File/New` in main ZSoil® menu) set ⊙ **Axisymmetry** to ON and select `Deformation` item from the `Problem type` list. This way a single-phase, axisymmetric bearing capacity problem will be carried out.

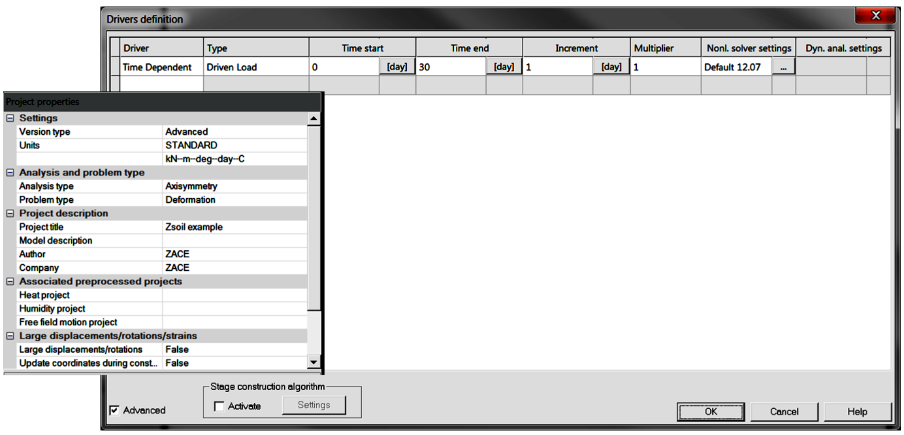


- **Drivers**

To set up the list of drivers use menu `Control/Analysis & drivers`. From list of drivers select `Time dependent`, from list `Type` select `Driven load`, specify the initial time value, final time value, initial time step, time step multiplier and accept the setting with button **Modify** (note that the default driver is always set to the `Initial state` and for that reason it must be modified in our case).

In the considered case computation will start at time $t = 1$ [day] (here time is a fictitious parameter), then the next time step will be $t = 2$ [days], $t = 3$ [days] etc. until divergence occurs (limit loads is detected) or time $t = 30$ [days] is achieved. As the `Multiplier` is equal to 1.0 the time step is kept constant and in the considered case is equal to $\Delta t = 1.0$ [day].

To learn on how to set up drivers list watch the video Set drivers



- **Materials**

The two materials are used in this example, first for the subsoil and the second for the concrete foundation. All the parameters are given in table below:

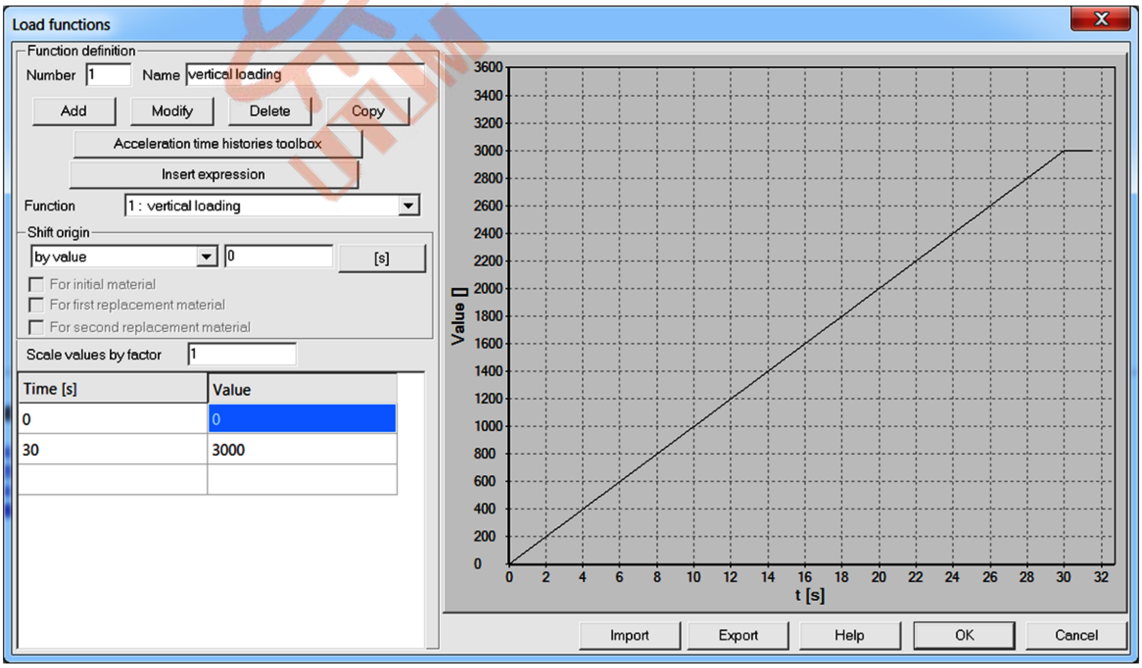
Material	Model	Data group	Properties	Unit	Value
1 soil	Drucker-Prager	Elastic	E	[kN/m^2]	30000
			ν	–	0.3
		Density	γ	[kN/m^3]	0
		Non-linear	ϕ	[°]	25
			ψ	[°]	0
			C	[kN/m^2]	15
			Adjustment	–	Intermediate
			cut off	–	OFF
2 concrete	Elastic	Elastic	E	[kN/m^2]	30000000
			ν	–	0.2
		Density	γ	[kN/m^3]	0

To edit material properties use menu `Assembly/Materials`.

To learn on how to enter material data watch the video Edit materials .

- **Load function**

Load functions are used in ZSoil® program to describe variation (in time, real or fictitious) of applied loads, imposed displacements, pressures, temperatures, humidities and also for description of progressive unloading after excavation. In the considered case the evolution of the vertical force is driven by the load time function number 1, labeled as *vertical load* and specified in detailed way in the following figure.



To edit the load time function use menu `Assembly/Load function`.

To learn on how to enter load time function watch the video Edit load time function .

- **Summary of all data preparation steps (video films)**

1	Create a new project
2	Create macro-model/mesh/loads/BC
3	Edit materials
4	Set drivers
5	Edit load time function
6	Run computation
7	Visualize results

1.3 HOW TO RUN A CONSOLIDATION PROBLEM

- **File: CONSP2D.INP**
- **Description of the problem**

The process of ground compression due to extrusion of water from the voids in a fine-grained soil as a result of increased loading is known as "Primary Consolidation". Associated settlements are referred to as "Primary Consolidation Settlements". After primary consolidation has ended, soil compression and additional associated settlements continue at a very slow rate, the result of plastic readjustment of soil grains due to progressive breaking of clayed particles and their interparticle bonds. This phenomenon is known as "Secondary Compression or Creep", and their associated settlement are called "Secondary settlements".

ZSoil®.PC simulates both of these physical processes using a multiphase algorithm described in the "Theoretical Section" of the user manual. This algorithm follows the traditional approach in soil mechanics with, however, some new numerical tools developed to accurately simulate these natural processes. In the following example, the case of a symmetric foundation is used to highlight the basic features of the consolidation problem, as shown in the figure.

Consolidation analysis can be carried out exclusively in the mode `Deformation+Flow` to be set in the initial preselection dialog box or later on through menu `Control/Analysis & drivers`.

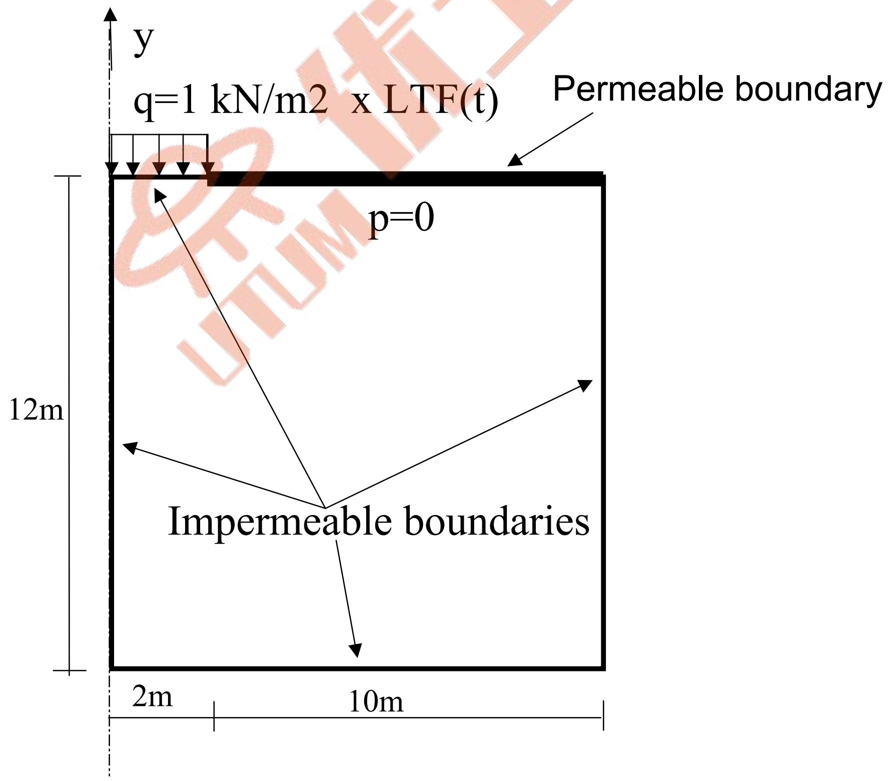
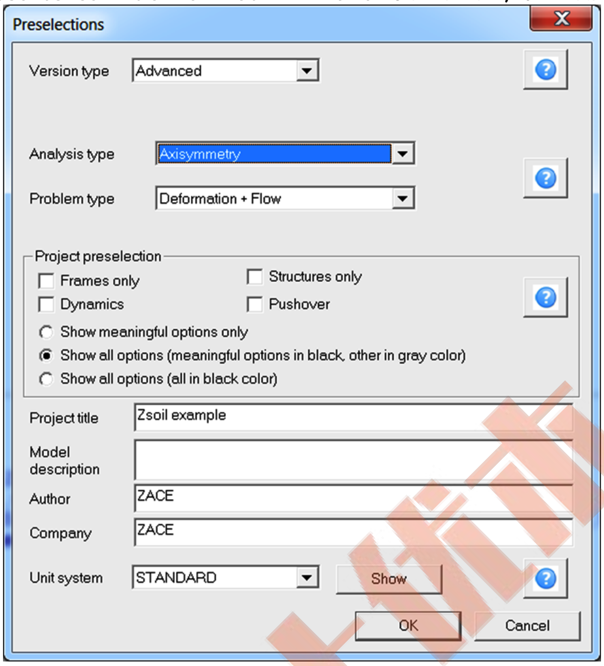


Figure 1.2: Axisymmetric consolidation problem

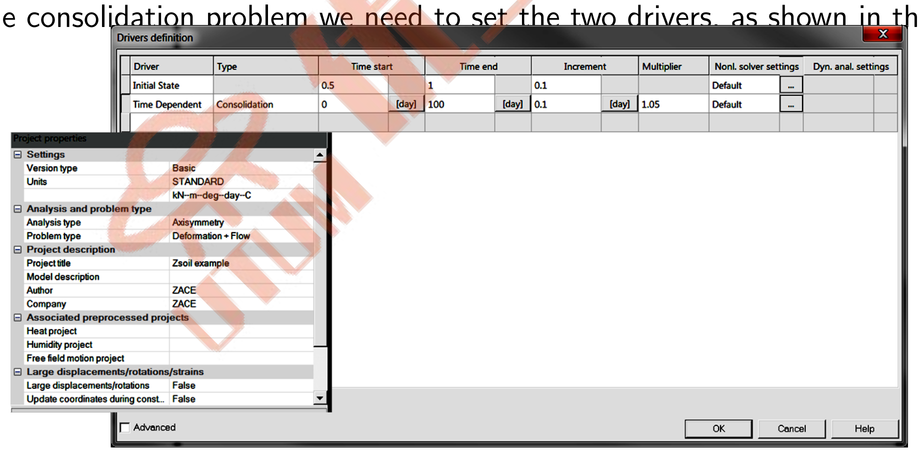
- **Project preselection**

In the dialog box for project preselection (it appears automatically for option `File/New` in main ZSoil® menu) set ⊙ **Axisymmetry** to ON and select `Deformation+Flow` item from the `Problem type` list. The predefined system of units for both data preparation and visualization of results can be verified in menu `Control/Units`.



- **Drivers**

To run the consolidation problem we need to set the two drivers, as shown in the figure.



The role of the `Initial state driver` is to set the initial condition for the fluid pressure and for the *in situ* effective stresses at time $t = 0$. This driver in the mode `Deformation+Flow` will enforce the two computational steps:

- ⋆ Steady state fluid flow analysis which yields pressure distribution in the domain at time $t = 0$ (for the assumed pore water pressure boundary condition ($p = 0$) at the permeable boundary)
- ⋆ Standard initial state computation with a pressure field obtained from the steady state

This setting is equivalent to the uncoupled total stress analysis.

The second driver activates fully coupled consolidation analysis, driven by a vertical load density q evolving in time according to the load time function defined later on.

To learn on how to set up the drivers list watch the video Set drivers

- **Materials**

Material properties of the consolidating soil are given in the following table:

Material	Model	Data group	Properties	Unit	Value
1 soil	Mohr-Coulomb	Elastic	E	[kN/m^2]	30000
			ν	–	0.3
		Density	γ_D	[kN/m^3]	16.52
			γ^F	[kN/m^3]	10
			e_o	–	0.60
		Non-linear	ϕ	[°]	25
			ψ	[°]	0
			C	[kN/m^2]	5
		Flow	β^F	[kN/m^2]	10^{38}
			k'_x	[m/d]	10^{-4}
			k'_y	[m/d]	10^{-4}
			α	[1/m]	1
			S_r	–	0.0
		$K_o^{in\ situ}$	$K_o^{x'}$	–	0.6
			$K_o^{z'}$	–	0.6

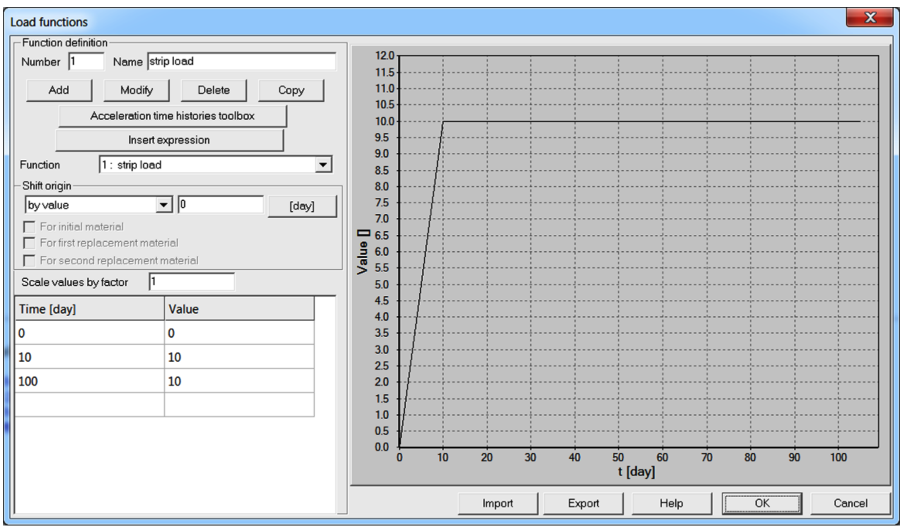
In the considered case the assumed fluid boundary condition yields the full saturation state of the medium. Therefore, parameters α and S_r, required for partially saturated medium, are not meaningful here. By assuming $k'_x = k'_y$ an isotropic Darcy flow conditions are fulfilled and $\beta^F = 10^{38}$ kPa is equivalent to the condition of quasi-incompressible fluid.

To edit material properties use menu `Assembly/Materials`.

To learn on how to enter material data watch the video Edit materials .

- **Load function**

The load time function associated with the unit distributed load $q = 1kN/m^2$ describes its evolution in time. At the initial state ($t = 0$) it must take value zero while later on it grows until final value. In the considered case the evolution of the load is driven by the load time function number 1, labeled as *vertical load* and specified in detailed way in the following figure.



To edit the load time function use menu `Assembly/Load function`.

To learn on how to enter load time function watch the video Edit load time function .

- **Generation of the model**

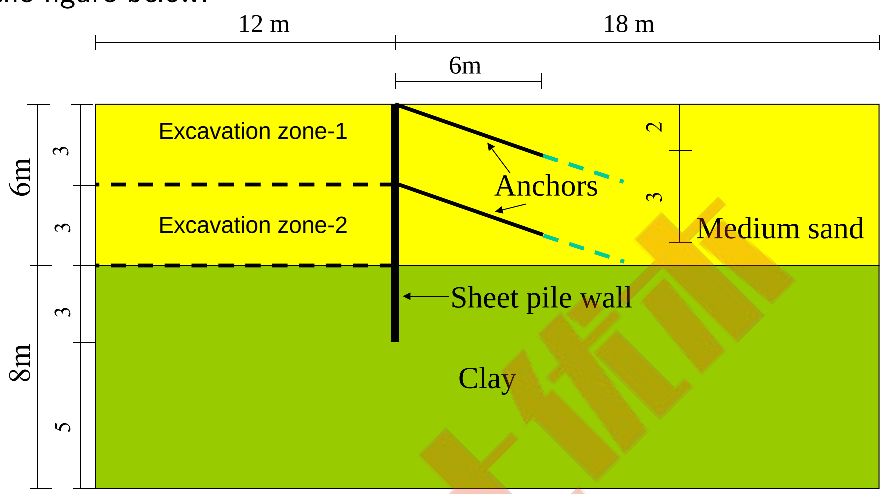
The computational model is built in the following steps documented in form of video films.

1	Create a new project
2	Edit load time function
3	Create macro-model/mesh/loads and boundary conditions
4	Edit materials
5	Set drivers
6	Run computation
7	Visualize results

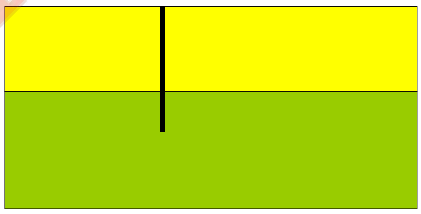
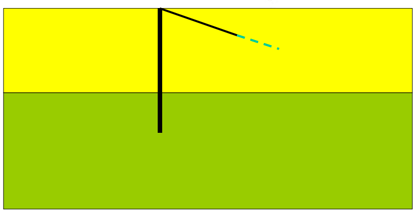
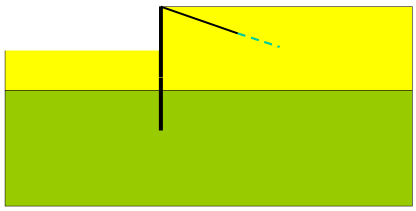
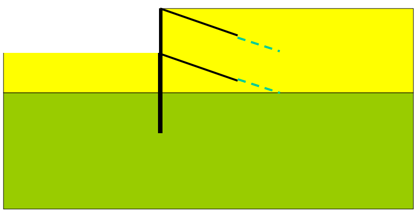
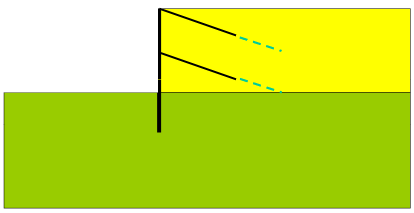
1.4 HOW TO RUN SHEET-PILE WALL PROBLEM

- **Data file: tutorials/sheet-pile-wall.INP**
- **Description**

Generation of a complex geotechnical model of installation of an anchored sheet pile wall, followed then by an excavation is the goal of this tutorial. The geometry of the model will evolve in time and some model components like wall, anchors or excavated soil layers will appear or disappear according to the assumed scenario. The geometry of the model is shown in the figure below.

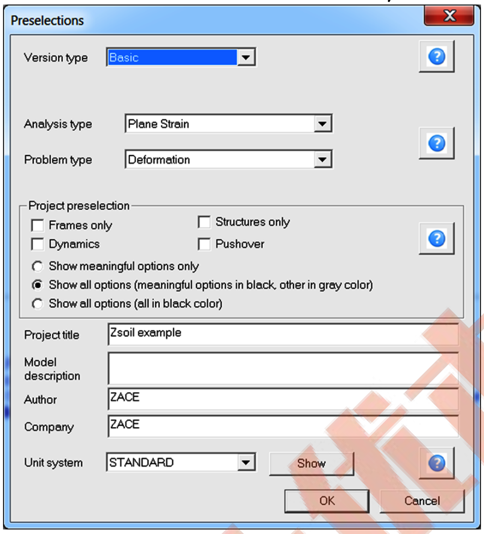


Sequence of all steps is shown in the following table.

 Initial state ($t = 0$)	 Installation of sheet pile wall ($t = 1$)
 Installation of first anchor ($t = 2$)	 Excavation of 1 sand layer ($t = 3$)
 Installation of second anchor ($t = 4$)	 Excavation of 2 sand layer ($t = 5$)

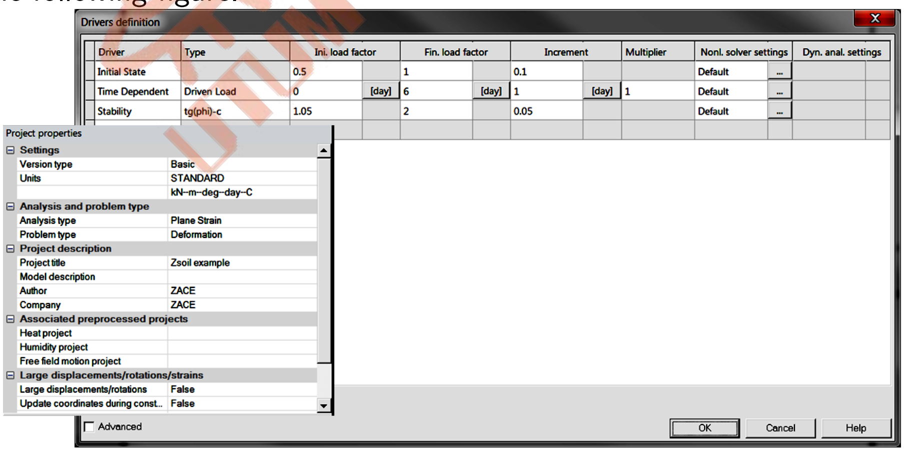
- **Project preselection**

  In the dialog box for project preselection (it appears automatically for option `File/New` in main Z_Soil menu) set ◎ **Plane strain** to ON and select `Deformation` item from the `Problem type` list. The predefined system of units for both data preparation and visualization of results can be verified in menu `Control/Units`.



- **Drivers**

  The whole computational process will consist of three drivers i.e. the `Initial state` which will yield the initial stress distribution (including user defined coefficient of *in situ* lateral pressure $K_o = 0.8$ in clay layer), `Time dependent/Driven load` to analyze all construction and excavation steps and at the end `Stability` (using $c - tan(\phi)$ reduction algorithm) will be carried to assess the global safety factor. The complete set of drivers is given in the following figure.



  To learn on how to set up the drivers list watch the video Set drivers

- **Materials**

Material properties for medium sand and clay layers are given in the following table:

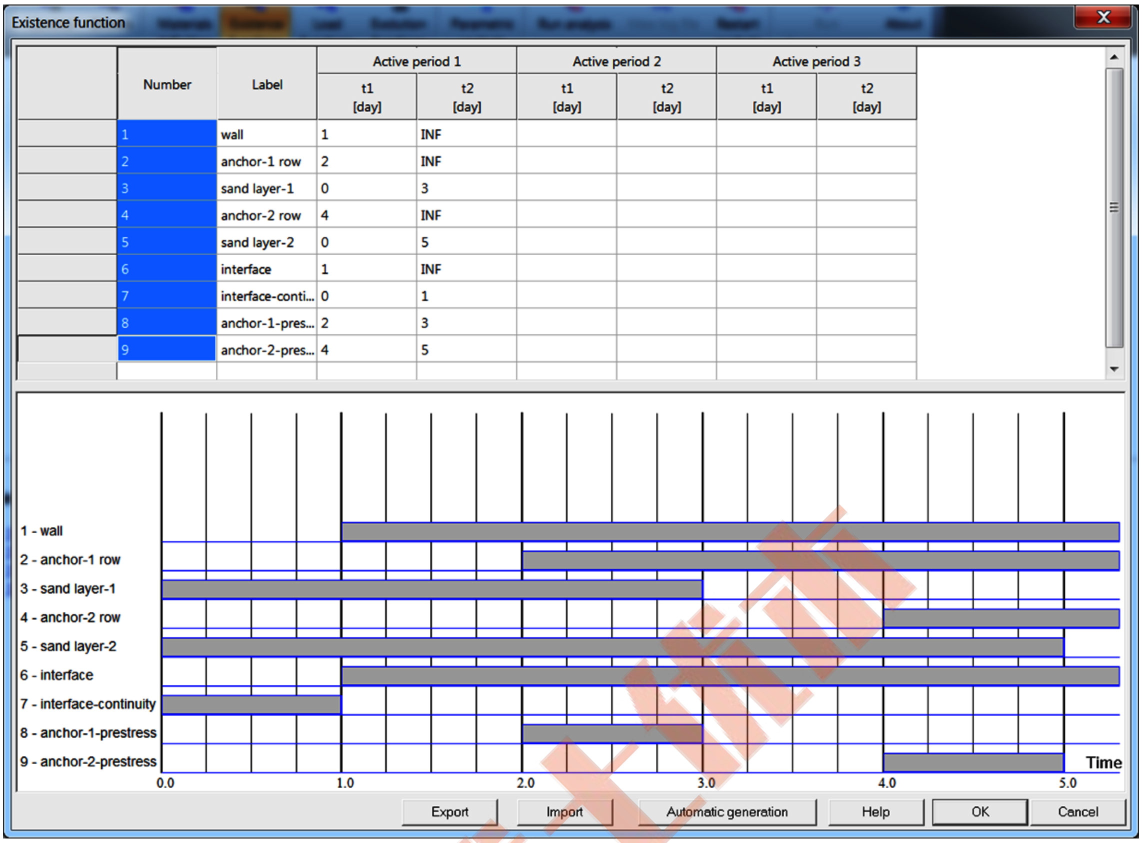
Material	Model	Data group	Properties	Unit	Value
1 Clay	Mohr-Coulomb	Elastic	E	[kN/m^2]	30000
			ν	–	0.32
		Density	γ_D	[kN/m^3]	18
			γ^F	[kN/m^3]	10
			e_o	–	0.0
		Non-linear	ϕ	[°]	20
			ψ	[°]	0
			C	[kN/m^2]	15
		Initial K_0 state	K'_{ox}	–	0.8
			K'_{oz}	–	0.8
			Inclination angle	–	0.0
2 Sand	Mohr-Coulomb	Elastic	E	[kN/m^2]	60000
			ν	–	0.25
		Density	γ_D	[kN/m^3]	17.5
			γ^F	[kN/m^3]	10
			e_o	–	0.0
		Non-linear	ϕ	[°]	30
			ψ	[°]	0
			C	[kN/m^2]	0.0
		Initial K_0 state	K'_{ox}	–	0.5
			K'_{oz}	–	0.5
			Inclination angle	–	0.0
3 Wall	Elastic beam	Elastic	E	[kN/m^2]	210000000
			ν	–	0.2
		Density	Unit weight	[kN/m^3]	0.0
		Geometry	Type	–	Values
			Interval	[m]	1.0
			I_z	[m^4]	$11352 \cdot 10^{-8}$
			A_x	[m^2]	$2462 \cdot 10^{-4}$
			A_y	[m^2]	$2462 \cdot 10^{-4}$
4 Anchors	Truss	Elastic	E	[kN/m^2]	300000000
		Density	Unit weight	[kN/m^3]	0.0
		Geometry	Interval	[m]	4.0
			Area	[m^2]	0.000829
5 Interface	Contact	Non-linear	ϕ	[°]	inherit*
			ψ	[°]	0
			C	[kN/m^2]	0.0

Friction angle in the interface satisfies the condition: $tan(\phi^{interface}) = 0.5\ tan(\phi^{soil})$. In the considered case initial void ratio and fluid specific weight do not play any role.

To edit material properties use menu `Assembly/Materials`.

To learn on how to enter material data watch the video Edit materials .

- **Existence function**



Introduction of contact elements leads to discontinues mesh connectivity along the interface. For that reason before the sheet-pile wall is installed, full compatibility of the displacement field must be preserved in the interface. This effect can easily be achieved during generation of interface elements, where contact is defined in dual mode (full continuity first and then real interface behavior). Both modes are controlled by the two existence functions (in our case continuity is controlled by the existence function number 7 while real contact behavior by function number 6). It is strongly recommended to apply a distinct label to each existence function.

To edit existence functions use menu `Assembly/Existence function`.

To learn on how to enter existence functions watch the video Edit existence functions .

- **Generation of the model**

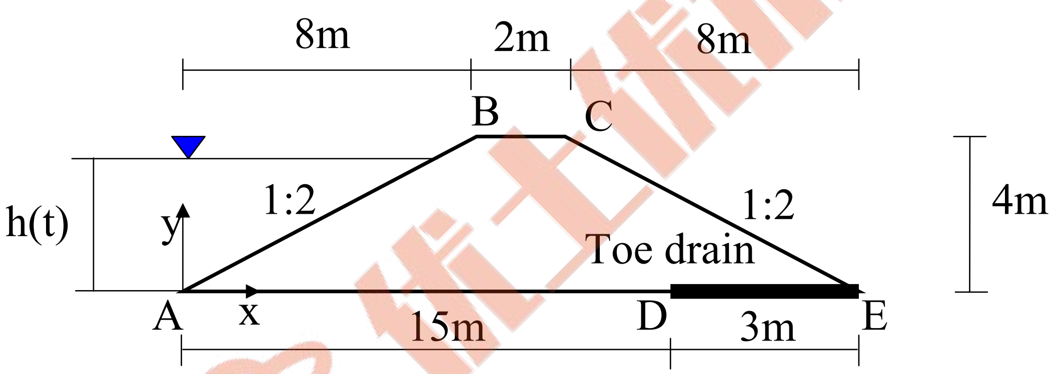
The computational model is built in the following steps and some of them are documented in form of video films.

1. Create a new project
2. Edit materials
3. Edit existence functions
4. Edit construction lines
5. Generate macro-model
6. Set drivers
7. Run computation
8. Visualize results

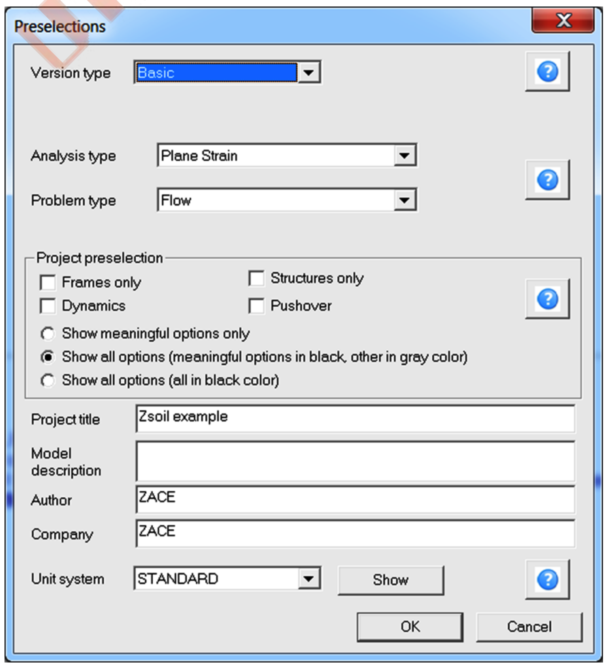
1.5 HOW TO RUN A STEADY-STATE FLOW PROBLEM

- **Data file: tutorials/ssf.INP**
- **Description**

  Consider the embankment illustrated in figure below which is submitted to the following flow conditions: a hydrostatic pressure distribution is applied along boundary AB and defined through the pressure head value $h(t)$, while a toe-drain is present at part DE. We will analyze the two steady state solutions for $h(t = 0d) = 3$m and then $h(t = 1d) = 2$m within a single run. Application of a varying in time pressure boundary condition, defined through the pressure head $h(t)$, and toe drain as well, require a special treatment realized through the so-called seepage elements. These elements allow for an automatic switch from pressure to the flux boundary condition (at the boundary AB, below the water table, we need to set the pressure boundary condition assuming the hydrostatic pressure profile while above a zero flux; in the toe drain in part where water flows to the drain we need to enforce zero pressure while in the remaining part zero flux).

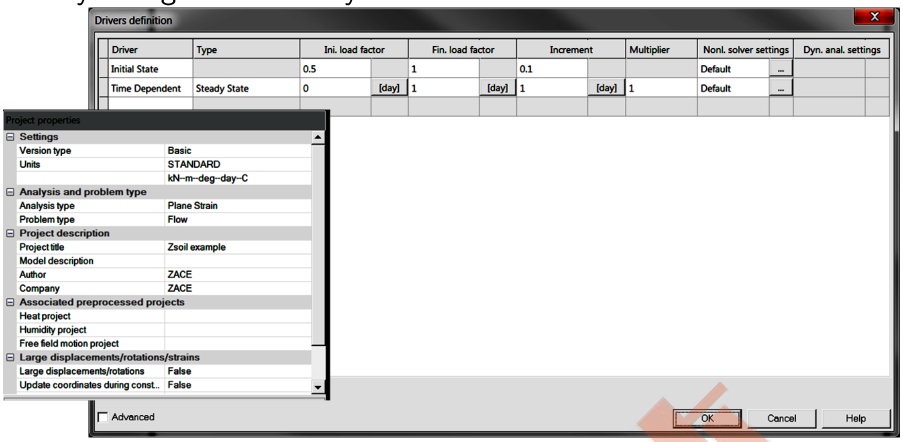


- **Project preselection**



- ## Drivers

The whole computational process will consist of the two drivers i.e. the `Initial state` and then `Time dependent/Steady state`. The fist driver will yield the steady state solution at time $t = 0$ for the pressure boundary condition assumed at that time while the second one will yield again the steady state solution but at time $t = 1d$.



To learn on how to set up the drivers list watch the video Set drivers

- ## Materials

Material properties for soil are given in the following table:

Material	Model	Data group	Properties	Unit	Value
1 Soil	Elastic	Density	γ_D	[kN/m^3]	0
			γ^F	[kN/m^3]	10
			e_o	–	0.0
		Flow	β^F	[kN/m^3]	10^{38}
			$k_x^{'}$	[m/d]	10^{-4}
			$k_y^{'}$	[m/d]	10^{-4}
			β	[°]	0.0
			α	[m^{-1}]	5.0
			S_r	–	0.0

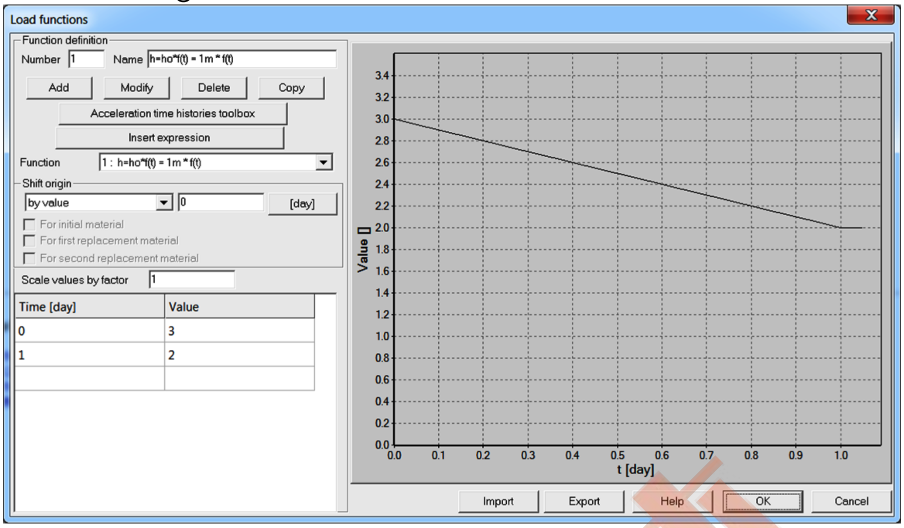
In the considered case any solid material model can be used and the only meaningful parameters to be set up are placed in groups `Density` and `Flow`. Initial void ratio e_o, fluid bulk modulus β^F are not meaningful for steady state analysis. As $k_x^{'} = k_y^{'}$ the flow orthotropy angle β is not meaningful either. The fictitious material (2) applied to seepage surface elements is not specified here (the multiplier for, automatically estimated, penalty parameter is equal to the default value 1.0).

To edit material properties use menu `Assembly/Materials`.

To learn on how to enter material data watch the video Edit materials .

- **Load time function**

  The evolution of the load time function associated with the pressure head boundary condition is shown in the figure.



  To learn on how to enter material data watch the video Edit load time function .

- **Generation of the model**

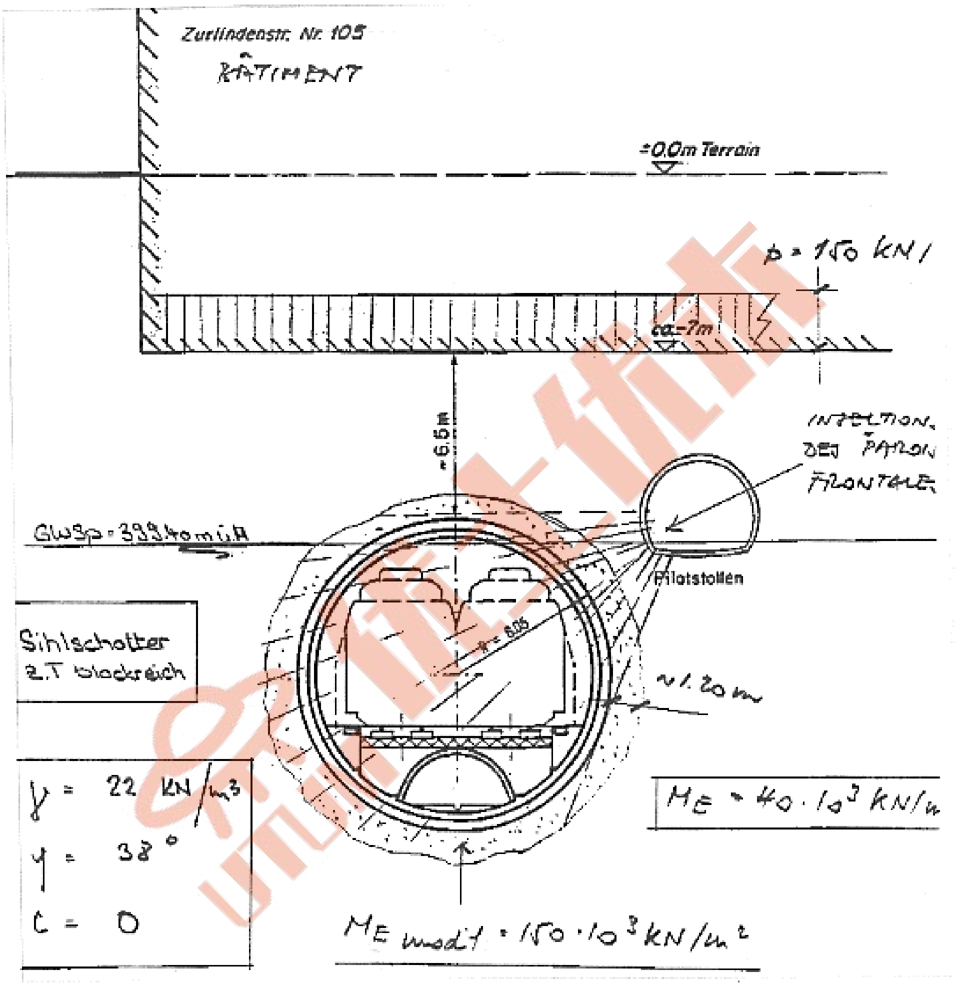
  The computational model is built in the following steps and some of them are documented in form of video films.

1	Create a new project under name : ssf.inp
2	Set drivers
3	Edit load time function
4	Edit materials
5	Create macro-model and mesh
6	Run computation
7	Visualize results

1.6 HOW TO SIMULATE A TUNNEL IN AN URBAN ENVIRONMENT

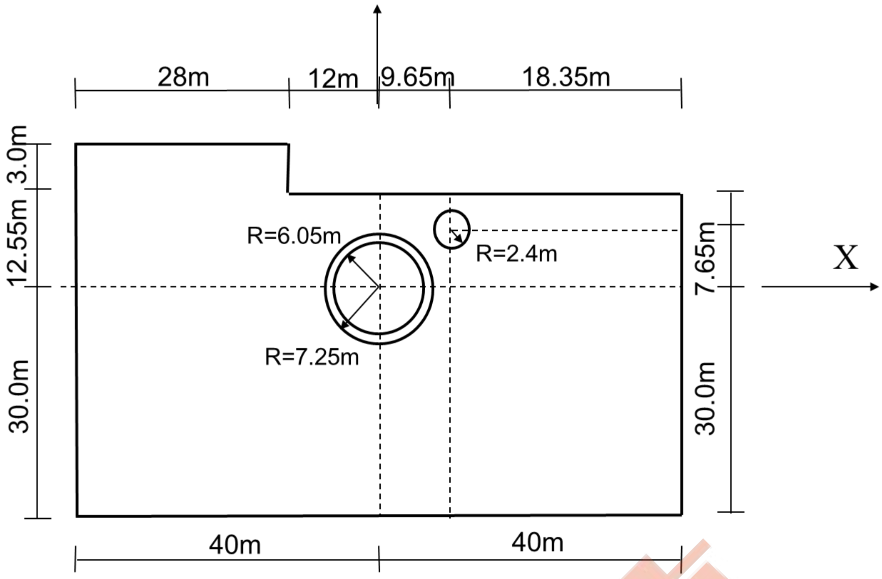
- **Data file: tutorials/tunnelzh.INP**
- **Description**

  This tutorial is devoted to the problem of tunneling in an urban environment. The engineering draft is shown in the figure below.

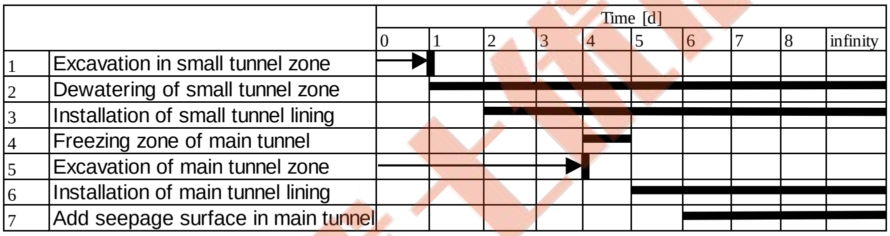


To build the tunnel with the external radius 6.05m a small technological tunnel (with the external radius 2.4m) is excavated first and concrete lining is constructed (thickness is 0.2m). Then the whole zone (with radius of 7.25m) around the main tunnel is frozen, main tunnel is excavated and lining is constructed (with thickness of 0.7m). Once the lining is installed freezing is stopped.

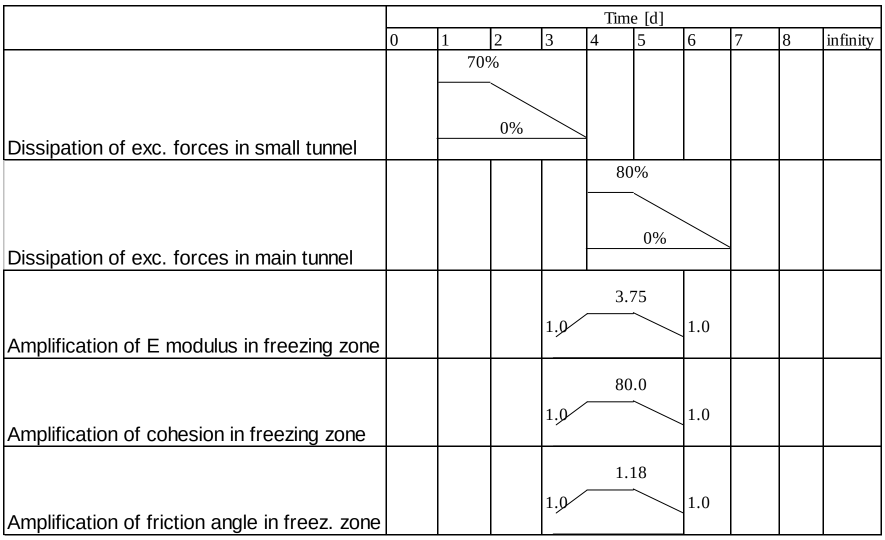
The geometrical model with all dimensions is shown in the following figure.



Sequence of events that will be considered in this example is shown in the following table.

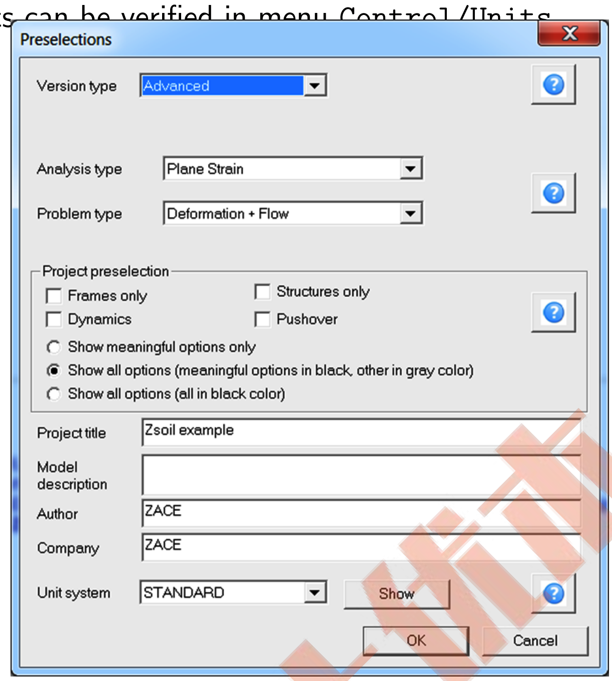


It has to be emphasized that the problem is three-dimensional due to effect of excavation front. In the example we take this into account assuming that just after excavation only some part of the excavation forces is dissipated (30 % in the zone of small tunnel and 20 % in the zone of main tunnel) while the remaining part is progressively applied to the installed linings. As far as freezing effect is considered we assume that the stiffness modulus E varies from E = 40000 kPa (in situ value) up to E =150000 kPa (this yields a factor of $\frac{150000}{40000} = 3.75$), cohesion varies from $c = 1$ kPa to c=80 kPa and friction angle from $\phi = 38^o$ up to $\phi = 45^o$. We assume the residual value for cohesion (c=1 kPa) to simplify the data setting for evolution of this parameter in time. Progressive unloading and variation of material properties in time are shown in the figure below.



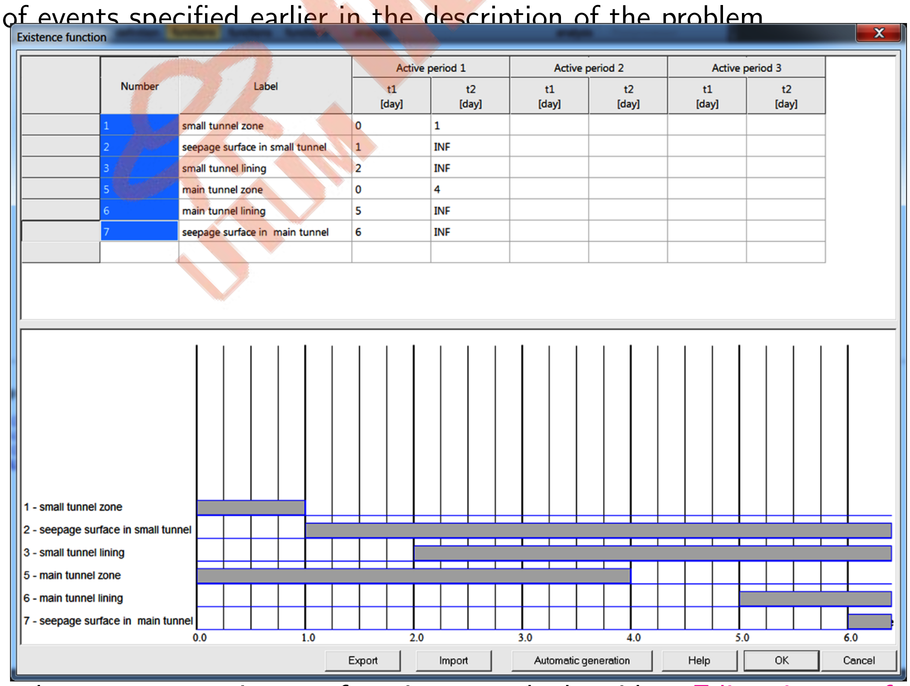
- **Project preselection**

  In the dialog box for project preselection (it appears automatically for option `File/New` in main Z_Soil menu) set ⊙ **Plane strain** to ON and select `Deformation+Flow` item from the `Problem type` list. The predefined system of units for both data preparation and visualization of results can be verified in menu `Control/Units`.



- **Existence functions**

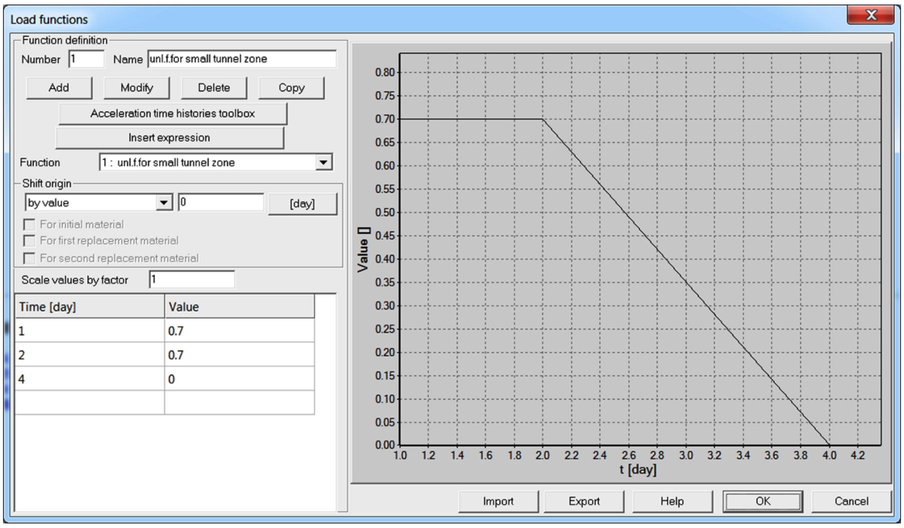
  The sequence of excavation/construction events is controlled by the existence functions shown in the figure below. All these existence functions are defined according to the sequence of events specified earlier in the description of the problem.



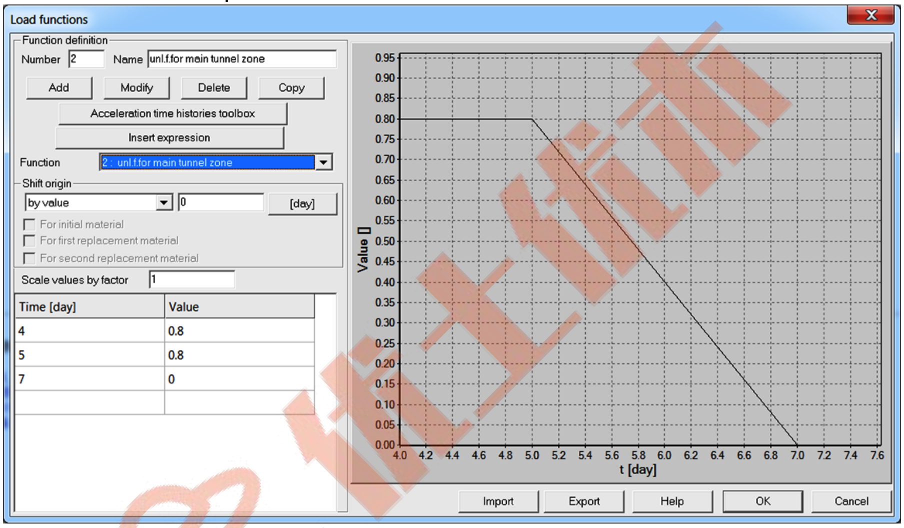
  To learn on how to enter existence functions watch the video Edit existence functions .

- **Load time function**

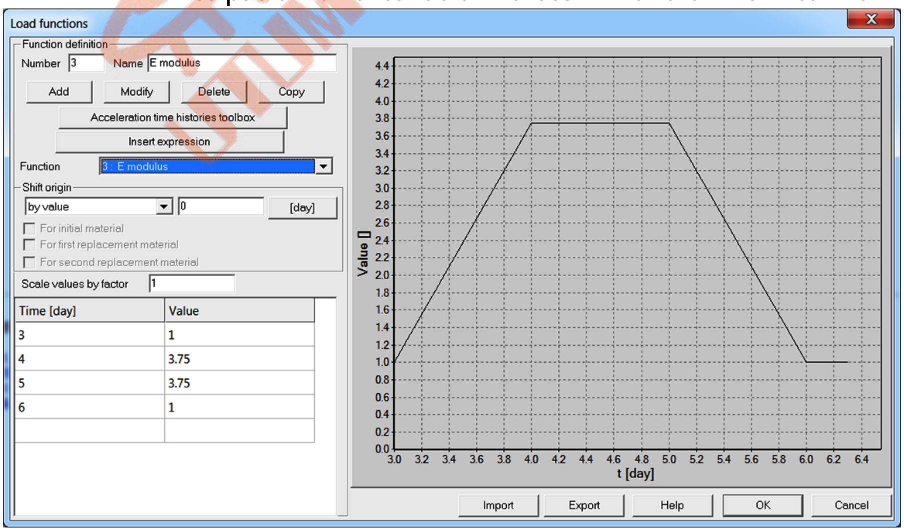
  The evolution of the load time functions related to the dissipation of excavation forces, but also to describe the evolution of stiffness and strength parameters in time due to freezing are shown in the following figures. Each load time function has its own distinct label given



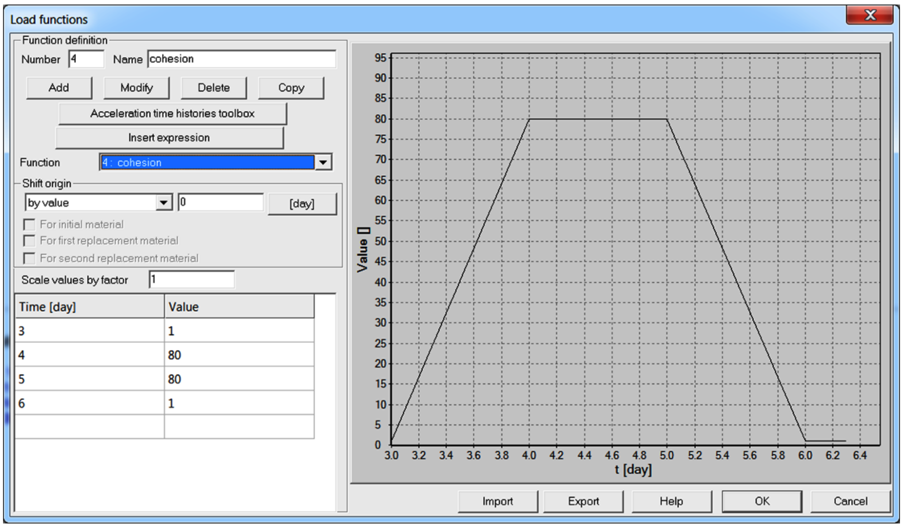
$LTF = 1$ Dissipation of excavation forces in zone of small tunnel



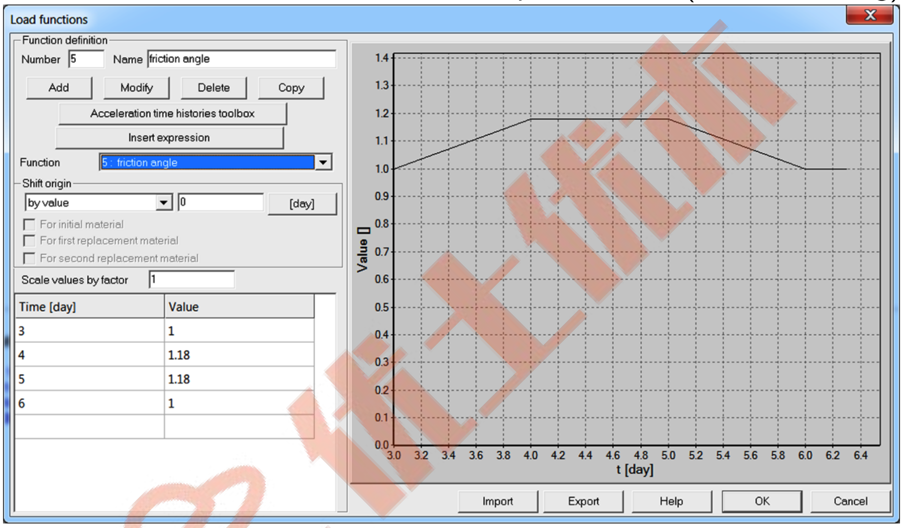
$LTF = 2$ Dissipation of excavation forces in zone of main tunnel



$LTF = 3$ Evolution of E modulus multiplier in time (due to freezing)



$LTF = 4$ Evolution of cohesion c multiplier in time (due to freezing)



$LTF = 5$ Evolution of friction angle ϕ multiplier in time (due to freezing)

To learn on how to edit load time functions watch the video Edit load time function .

- **Materials**

Material properties for subsoil and tunnel linings are given in the following table:

Material	Model	Data group	Properties	Unit	Value
1 Subsoil	Mohr-Coulomb	Elastic	E	[kN/m^2]	40000
			ν	–	0.25
		Density	γ_D	[kN/m^3]	18
			γ^F	[kN/m^3]	10
			e_o	–	0.32
		Flow	$k_x = k_y$	[m/day]	0.0864
			S_r	–	0
			α	[1/m]	5
		Non-linear	ϕ	[°]	38
			ψ	[°]	10
			C	[kN/m^2]	1
		Initial K_0 state	K'_{ox}	–	0.45
			K'_{oz}	–	0.45
			Inclination angle	–	0.0

2 Subsoil	Mohr-Coulomb	Elastic	E	[kN/m^2]	40000
			ν	–	0.25
		Density	γ_D	[kN/m^3]	18
			γ^F	[kN/m^3]	10
			e_o	–	0.32
		Non-linear	ϕ	[°]	38
			ψ	[°]	10
			C	[kN/m^2]	1
		Initial K_0 state	$K^{'}_{ox}$	–	0.45
			$K^{'}_{oz}$	–	0.45
			Inclination angle	–	0.0
3 Small tunnel	Elastic beam	Elastic	E	[kN/m^2]	25000000
			ν	–	0.2
		Density	Unit weight	[kN/m^3]	24.0
		Geometry	Type	–	User defined (rectangular)
			Interval	[m]	1.0
			b	[m]	1.0
			h	[m]	0.2
4 Main tunnel	Elastic beam	Elastic	E	[kN/m^2]	30000000
			ν	–	0.2
		Density	Unit weight	[kN/m^3]	24.0
		Geometry	Type	–	User defined (rectangular)
			Interval	[m]	1.0
			b	[m]	1.0
			h	[m]	0.7
5 Foundation	Elastic beam	Elastic	E	[kN/m^2]	30000000
			ν	–	0.2
		Density	Unit weight	[kN/m^3]	0.0
		Geometry	Type	–	User defined (rectangular)
			Interval	[m]	1.0
			b	[m]	1.0
			h	[m]	1.0

It can be noticed that material number 2 is equivalent to material number 1, however stiffness properties and strength parameters of material 2 will vary in time to simulate freezing process.

The fictitious material with number 6 applied to seepage surface elements is not specified here (the multiplier for, automatically estimated, penalty parameter is equal to the default value 1.0).

To edit material properties use menu `Assembly/Materials`.

To learn on how to enter material data watch the video Edit materials .

- **Material properties varying in time**

  Setting time dependent material properties can be made during editing of the material data. Any load time function (but with meaningful values) can be associated with any (if allowed) material parameter. To apply the load time function to the selected material parameter select an option `Load time function` from combo-box `Data mode` (placed always at the bottom of the material data dialog box). In the `Load time function` mode one may press the `Set` button and to select the load time function from the list of already defined load time functions. It is also possible to enter directly the load time function ID (when it does not exist it is automatically created and added as a zero valued function to the list of load time functions).

  To learn on how to define material properties varying in time watch the video Define time dependent material properties .

- **Generation of the model**

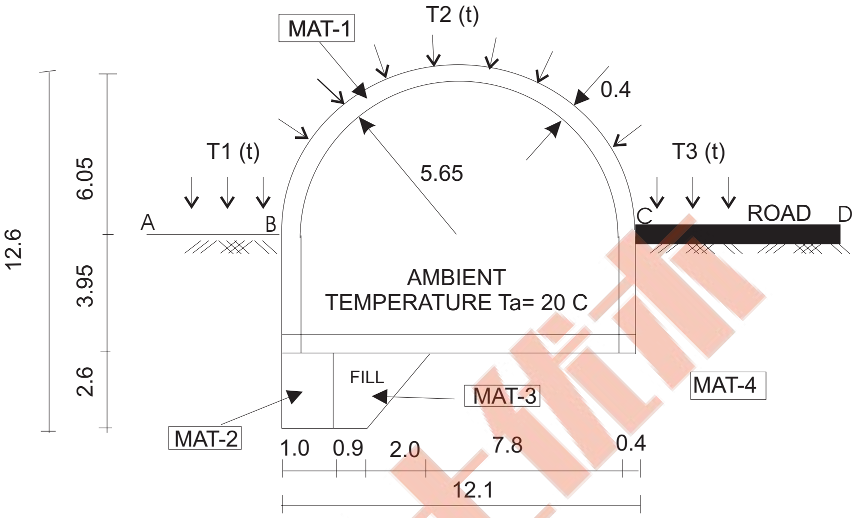
  The computational model is built in the following steps and some of them are documented in form of video films.

1	Create a new project under name : tunnelzh.inp
2	Edit load time function
3	Edit existence functions
4	Edit materials
5	Create construction lines
6	Draw model skeleton
7	Create macro-model and mesh
8	Set drivers
9	Run computation
10	Visualize results

1.7 HEAT TRANSFER FOLLOWED BY MECHANICAL ANALYSIS

- **Data file: tutorials/tunnelheat.INP, tutorials/tunnelmech.INP**
- **Description**

A problem of thermal stresses induced by imposed temperatures on the external surface of a tunnel lining is considered here. An engineering draft of the problem is shown in the figure below.



The problem will be solved in two steps. In the first one we will perform thermal analysis which will consist of the initial state computation (in the context of thermal analysis it is equivalent to the steady state at time $t = 0$) followed then by a transient analysis driven by varying in time temperatures T1 (t), T2 (t) and T3 (t). On the internal surface of the tunnel we assume convective type of the boundary condition driven by a constant ambient temperature equal to 20^o.

Once the thermal project is completed we will run mechanical analysis, driven by imposed strains evaluated from temperature fields resulting from thermal analysis. Solution of this complex problem will consist of the two separate projects i.e. thermal one run first and then a mechanical one run as second.

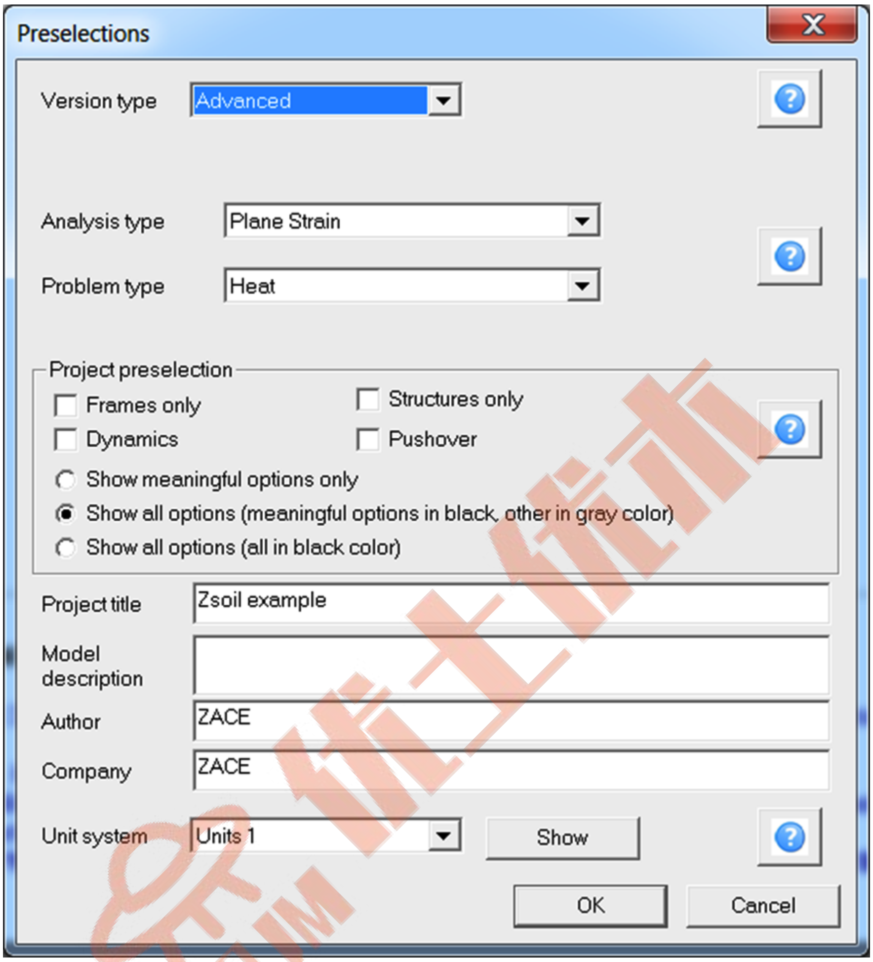
Both projects will be analyzed using the same mesh and all model components will be generated in a single preprocessing session.

Remarks:

1. Meshes used in thermal analysis and mechanical one can be completely different and in geometrical sense may even cover different domains. A projection of thermal solution on the mesh used in the mechanical analysis will be carried out by the code automatically
2. Beam elements cannot be used in thermal analysis
3. Beam elements can be used in the mechanical analysis (replacing equivalent continuum elements) but exclusively in the layered version (☒ **Nonlinear** is set to ON).
4. In the considered case the finite element model for both thermal and mechanical analyzes will be unique; features typical for heat analysis like convective BC, imposed temperatures will automatically be ignored in the mechanical project and features typical for mechanical analysis like displacement BC, etc.. will be ignored in the thermal one.

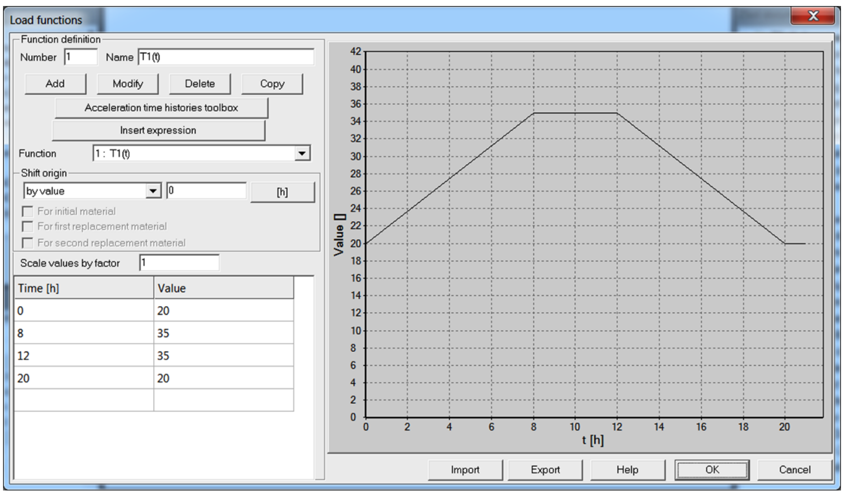
- **Project preselection**

  In the dialog box for project preselection (it appears automatically for option `File/New` in main Z_Soil menu) set ⊚ **Plane strain** to ON and select `Heat` item from the `Problem type` list. The predefined system of units for both data preparation and visualization of results can be verified in menu `Control/Units`.

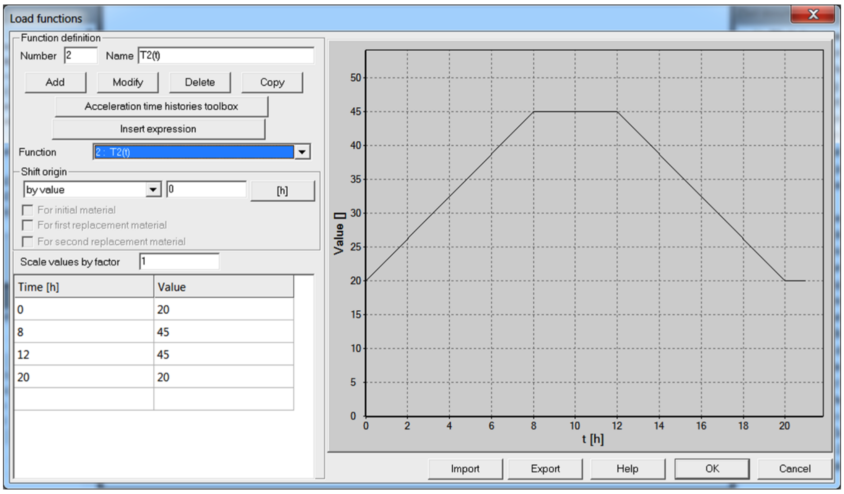


- **Load time function**

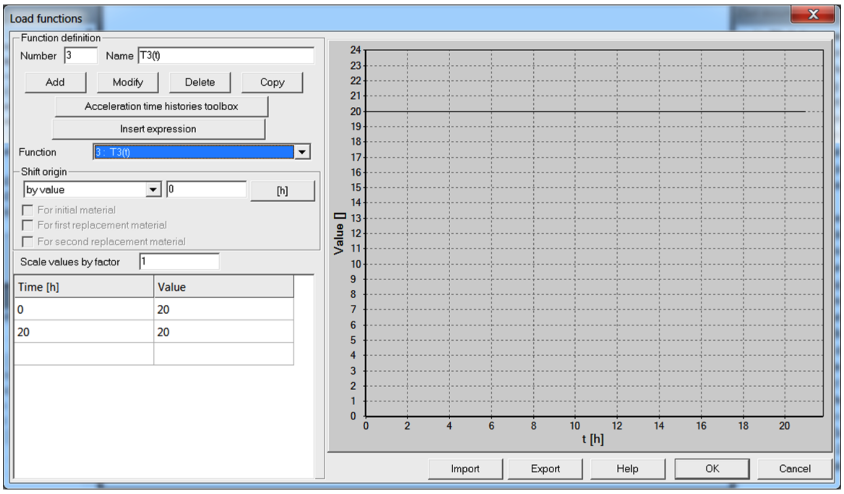
  The load time functions associated with the imposed temperature BC's along edges A-B, B-C, C-D are shown in the following figures.



$LTF_1(t)$ Evolution of $T_1(t) = 1°\text{C}\ LTF_1(t)$



$LTF_2(t)$ Evolution of $T_2(t) = 1°$C $LTF_2(t)$



$LTF_3(t)$ Evolution of $T_3(t) = 1°\text{C}\ LTF_3(t)$

Note that these load time function are meaningful only for thermal analysis.

To learn on how to edit load time functions watch the video Edit load time function .

- **Materials**

Material properties for subsoil, tunnel lining, fill, contact interface and convection surface are summarized in the following table:

Material	Model	Data group	Properties	Unit	Value
1 Tunnel lining	Elastic	Elastic	E	[kN/m^2]	30000000
			ν	–	0.2
		Density	γ_D	[kN/m^3]	24
			γ^F	[kN/m^3]	10
			e_o	–	0.0
		Heat	$c^* = c\,\rho$	[kJ/(m^3 K)]	3000
			λ	[kJ/(m K h)]	8.64
			α	[1/°C]	10^{-5}
			Source	–	OFF
2 Tunnel lining	Elastic	Elastic	E	[kN/m^2]	30000000
			ν	–	0.2
		Density	γ_D	[kN/m^3]	24
			γ^F	[kN/m^3]	10
			e_o	–	0.0
		Heat	$c^* = c\,\rho$	[kJ/(m^3 K)]	3000
			λ	[kJ/(m K h)]	8.64
			α	[1/°C]	10^{-5}
			Source	–	OFF
3 Fill	Mohr-Coulomb	Elastic	E	[kN/m^2]	100000
			ν	–	0.3
		Density	γ_D	[kN/m^3]	20
			γ^F	[kN/m^3]	10
			e_o	–	0.0
		Non-linear	ϕ	[°]	30
			ψ	[°]	10
			C	[kN/m^2]	0
		Initial K_o state	K_{ox}	–	1.0

			K_{oz}	–	1.0
		Heat	$c^* = c\ \rho$	[kJ/(m^3 K)]	3000
			λ	[kJ/(m K h)]	8.64
			α	[1/°C]	10^{-5}
			Source	–	OFF
4 Subsoil	Mohr-Coulomb	Elastic	E	[kN/m^2]	50000
			ν	–	0.3
		Density	γ_D	[kN/m^3]	20
			γ^F	[kN/m^3]	10
			e_o	–	0.0
		Non-linear	ϕ	[°]	30
			ψ	[°]	10
			C	[kN/m^2]	0
		Initial K_o state	K_{ox}	–	0.5
			K_{oz}	–	0.5
		Heat	$c^* = c\ \rho$	[kJ/(m^3 K)]	3000
			λ	[kJ/(m K h)]	8.64
			α	[1/°C]	10^{-5}
			Source	–	OFF
5 Interface soil-lining	Contact	Non-linear	ϕ	[°]	5
			ψ	[°]	0
			C	[kN/m^2]	0.0
6 Convection	Convection	Heat	h	[kJ/(m^2 K h)]	106

- **Generation of the model for both thermal and mechanical analysis**

  The computational model is built in the following steps and some of them are documented in form of video films.

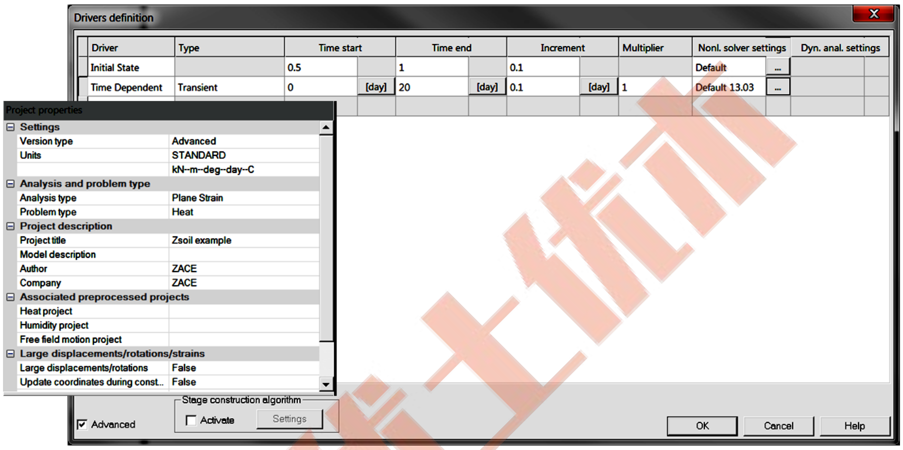
1	Create a new project under name : tunnelheat.inp
2	Edit load time function
3	Edit materials
4	Create skeleton model
5	Create macro-model
6	Create mesh

1.7.1 THERMAL ANALYSIS

The computational model to carry out both thermal and mechanical analysis was fully described in section 1.7. In this section we will customize the model to run thermal analysis exclusively. The only modification concerns setting of the list of drivers.

- **Data file: tutorials/tunnelheat.INP**
- **Drivers**

  As in the preselection dialog box we have already set up the `Problem Type` to `Heat` the only modification which is needed is the proper setting of the drivers set (shown in figure below).



  In this example we want to trace the evolution of temperatures in space and time due to transient BC, hence the initial condition must be set up. Here before running transient analysis we have declared an *Initial state* driver, equivalent to the steady state heat transfer case solved at time $t = 0$. All nodal temperature BC are such that nodal temperatures at time t=0 are equal to T=20°C and ambient temperature is also equal to T=20°C. Thus it is obvious that at t=0 we have to obtain a uniform temperature T=20°C in whole domain as the result of steady state analysis. NB. To assume another initial condition one may use `Initial temperature` super-element.

- **Customization of the model to carry out thermal analysis**

  The computational model is built in the following steps and some of them are documented in form of video films.

1	Open project: tunnelheat.inp
2	Set drivers
3	Run computation
4	Visualize results

1.7.2 MECHANICAL ANALYSIS (TUNNEL LINING MODELED WITH CONTINUUM ELEMENTS)

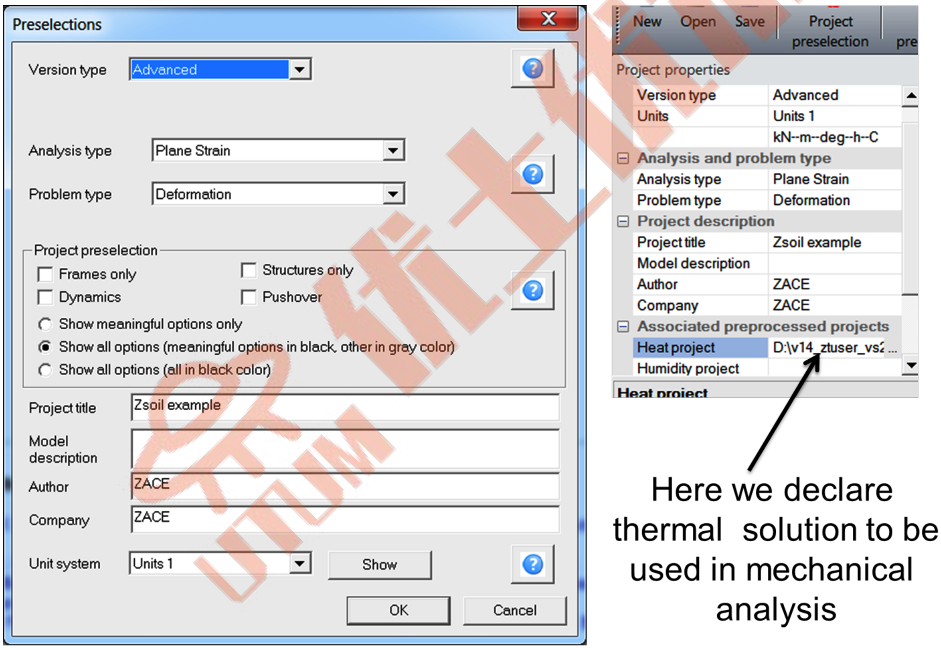
In this case the tunnel lining will be modeled with aid of special type continuum elements (EAS elements with enhanced shear and bending representation) which belong to the group of `Continuum for structures`. The only limitation when using these type of elements is such that `Elastic` model can only be applied. Bending moment, shear and axial forces can be recovered in the postprocessor.

The data set prepared under name **tunnelheat.inp** has to be saved under name **tunnelmech.inp** (without results saving).

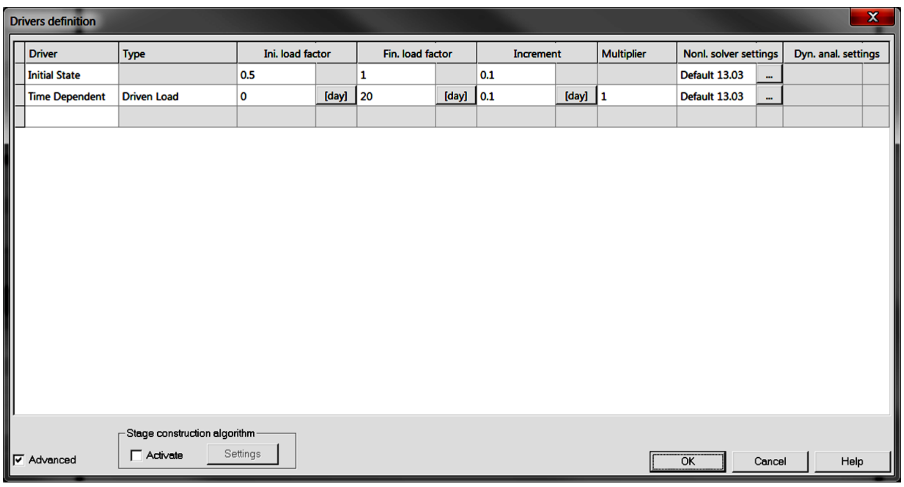
- ## Data file: tutorials/tunnelmech.INP

- **Project preselection**

  In the dialog box for project preselection (`Control/Project preselection`) switch from `Heat` to `Deformation Problem type`. NB. Time unit cannot be changed (same unit must be used as in the thermal analysis).



- **Drivers**



To apply the thermal solution to the mechanical model we have to browse for **tunnelheat.inp** project in the **Associated preprocessed projects** [ ] in dialog `Analysis and Drivers`.

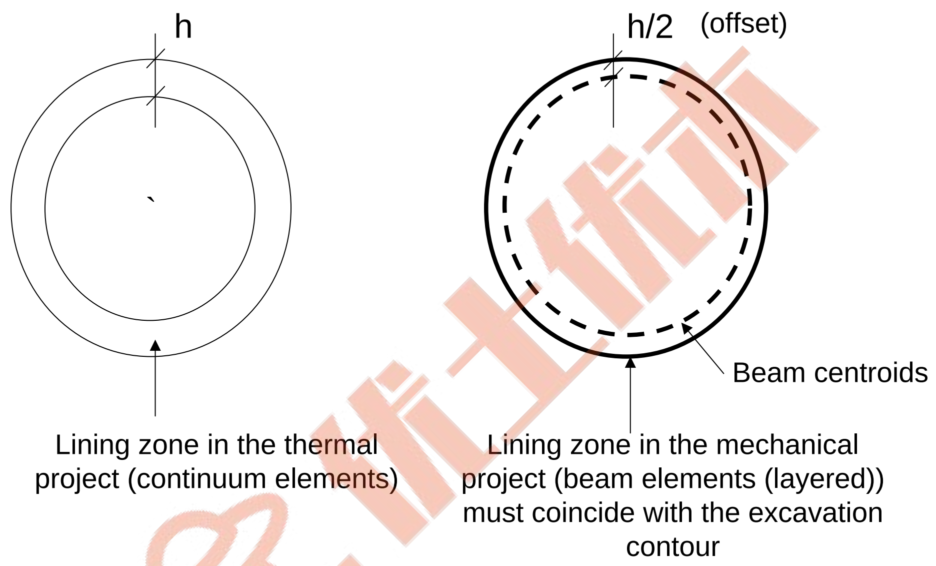
- **Customization steps to carry out mechanical analysis**

The computational model is built in the following steps and some of them are documented in form of video films. To see on how to recover M, N, T diagrams integrated from the stress fields see the video on visualization of results.

1	Open project: tunnelheat.inp
2	Save as: tunnelmech.inp
3	Set drivers and attach heat project
4	Run computation
5	Visualize results

1.7.3 MECHANICAL ANALYSIS (TUNNEL LINING MODELED WITH BEAMS)

In this case the tunnel lining will be modeled with aid of beam elements. The first requirement is such that to handle spatial distribution of the temperature field the layered beam model must be used (button ☒ **Nonlinear** must be switched ON in material definition for the beam). The next requirement is such that the beam offset option must be used to shift the centroid of the beam with respect to the master nodes (offset option). This setting can be made during creation of the beam both at the macro-modeling or FE-model levels. The aim of setting the offset is to position properly the cross section of the beam with respect to the cross section designed by the continuum elements used in thermal project (see explanation in the figure below).



- **Data file: tutorials/tunnelmech-b.INP**
- **Materials**

  Replacing continuum lining by a lining defined as a layered beam requires an additional modification of the material data. We have to add a new material which will have number 7.

Material	Model	Data group	Properties	Unit	Value
7 Lining	Elastic beam	Elastic	E	[kN/m^2]	210000000
			ν	–	0.2
		Density	Unit weight	[kN/m^3]	0.0
		Geometry	Type	–	User defined
			Interval	[m]	1.0
			Shape		rectangular
			b	[m]	1.0
			h	[m]	0.4
			ν	–	0.2

- **Customization steps to carry out mechanical analysis**

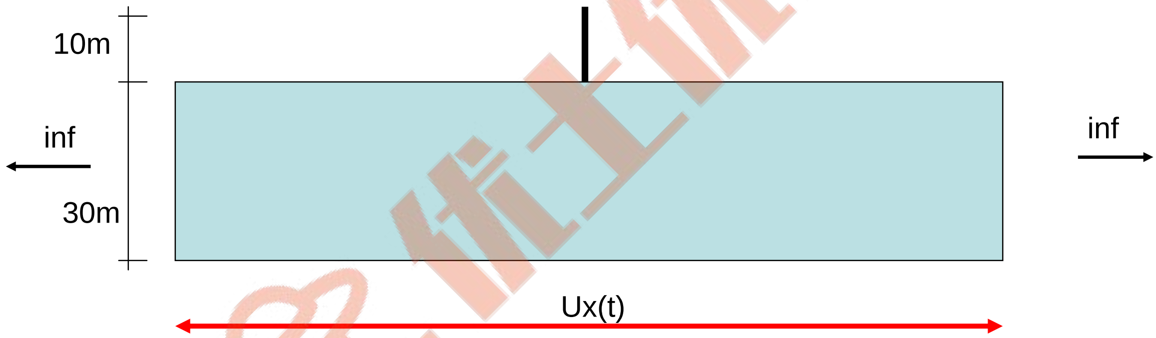
1 Open project: tunnelmech.inp
2 Save as: tunnelmech-b.inp
3 Add new material for lining (beam)
4 Replace continuum by beams
5 Run computation
6 Visualize results

1.8 DYNAMIC SOIL-STRUCTURE INTERACTION USING DOMAIN REDUCTION METHOD (DRM)

- **Data files: tutorials/dynamics/SOIL-COLUMN.inp, tutorials/dynamics/DRM-S-FF-SHL.inp, tutorials/dynamics/FF-SHL.inp**

- **Description**

A problem of dynamic concrete column-subsoil interaction is considered in this example. To reduce the computational effort DRM method will be used and hence the computational model will consist of the two models, the background one and the reduced model. The background model will be set up as a shear layer model (it is basically 1D model) discretized using quadrilateral elements and periodic solid boundary conditions to tie degrees of freedom on the opposite vertical walls. The background model will be excited by harmonic imposed displacement, with period of $T = 0.5$ s, applied to the base. The draft of the problem is shown in the figure below.



The reduced (DRM) problem will be solved in two steps. In the first one we will perform transient dynamic analysis of a shear layer that will consist of the initial state driver followed then by a transient dynamic one, driven by harmonic excitation $u_x(t)$ applied to the base (at depth of 30m) (FF-SHL.inp). Once the shear layer project is completed we will run transient dynamic analysis of a reduced model (DRM-S-FF-SHL.inp) including the column, driven by displacements, velocities and accelerations computed from the background model. To compare results of the reduced and full size models an additional full size model will be generated too (SOIL-COLUMN.inp).

- **Materials**

Material properties, common for all examples, defined for subsoil, concrete column and viscous dampers are summarized in the following table:

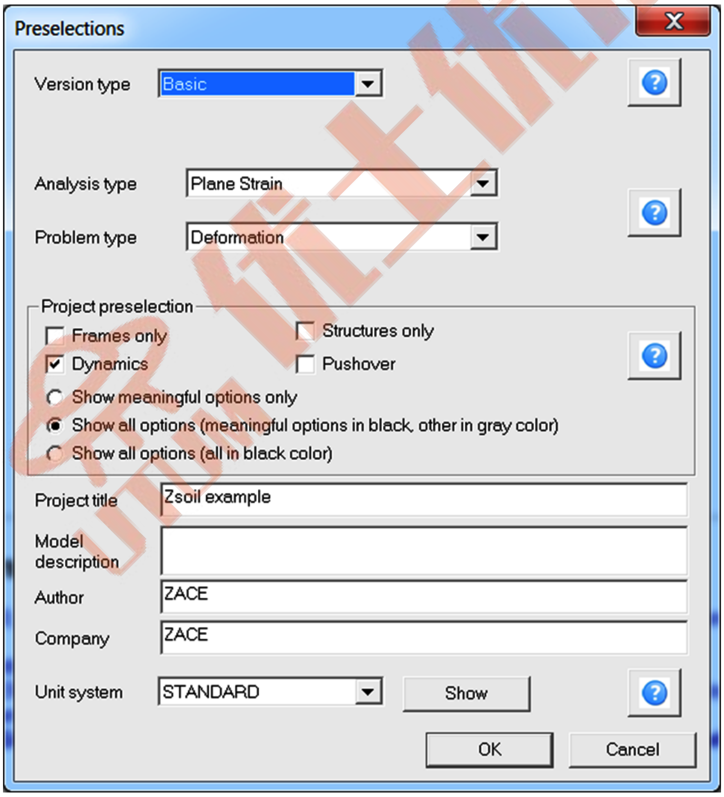
Material	Model	Data group	Properties	Unit	Value
1 Subsoil	Elastic	Elastic	E	[kN/m^2]	192000
			ν	–	0.2
		Density	γ_D	[kN/m^3]	19.61
			γ^F	[kN/m^3]	10
			e_o	–	0.0

2	Column	Elastic	Elastic	E	[kN/m^2]	20000000
				ν	–	0.2
			Density	γ	[kN/m^3]	24.52
			Geometry	Rectangular cross section	–	
				b	[m]	1.0
				h	[m]	1.0
3	Viscous damper	—	Elastic	E	[kN/m^2]	192000
				ν	–	0.2
			Density	γ_D	[kN/m^3]	19.61
				γ^F	[kN/m^3]	10
				e_o	–	0.0

1.8.1 FULL SIZE MODEL

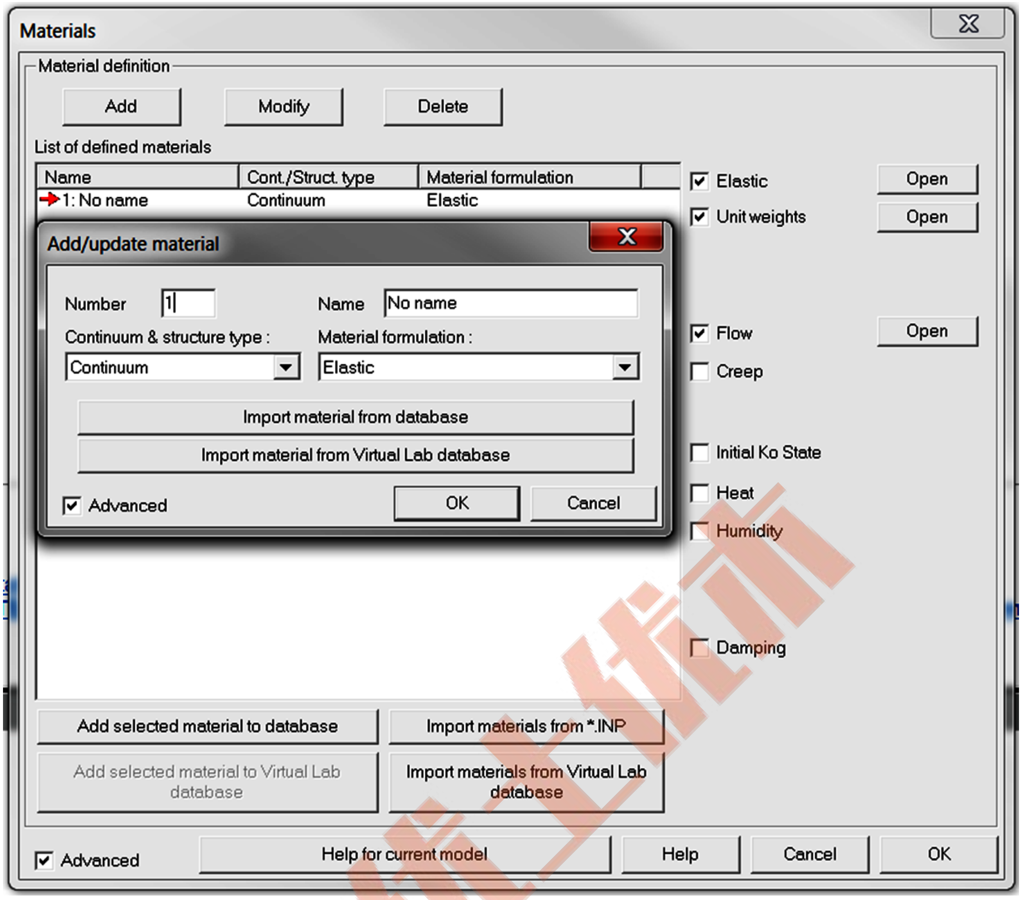
To carry out the time history analysis of column-subsoil interaction, without additional elements cancelling waves reflections from rigid boundaries, a large computational model must be generated. In the considered case 3600m long model was generated. It enables such an analysis during first 5s of the excitation, performed with 2Hz frequency. In the following table all major steps leading to the final model will be summarized.

Step 1: Open new project



1. Select `Plane Strain`
2. Set ☒ **Dynamics** ON
3. Accept these settings pressing button **OK**
4. `File/Save As` **SOIL-COLUMN.inp**

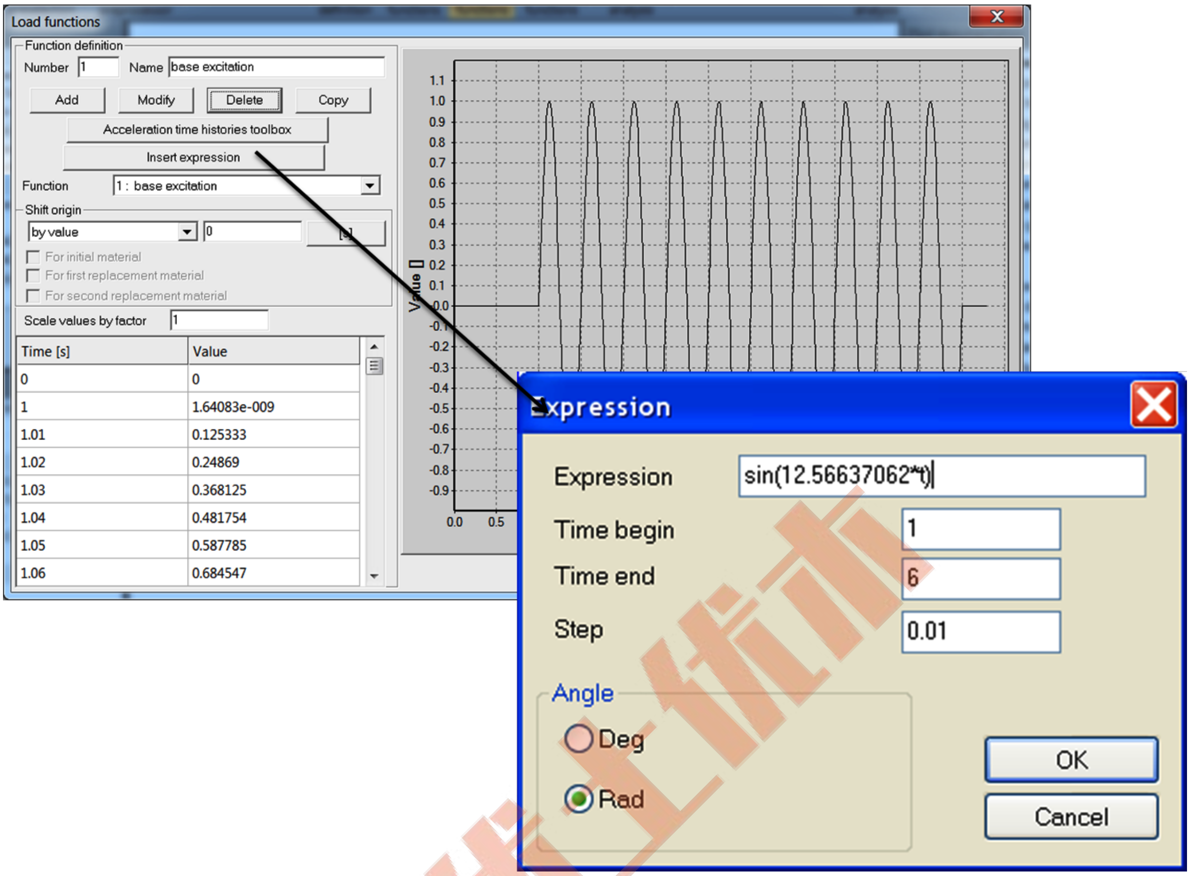
Step 2: Create list of materials



1. Add new material in the main menu through `Assembly/Materials` for `Continuum` with number 1 and use `Elastic` constitutive model
2. Add new material in the main menu through `Assembly/Materials` for `Beams` with number 2
3. Set properties for both materials according to the data specified in the main section for this tutorial

NB. If you want to learn more on how to edit material properties watch video
Edit material properties

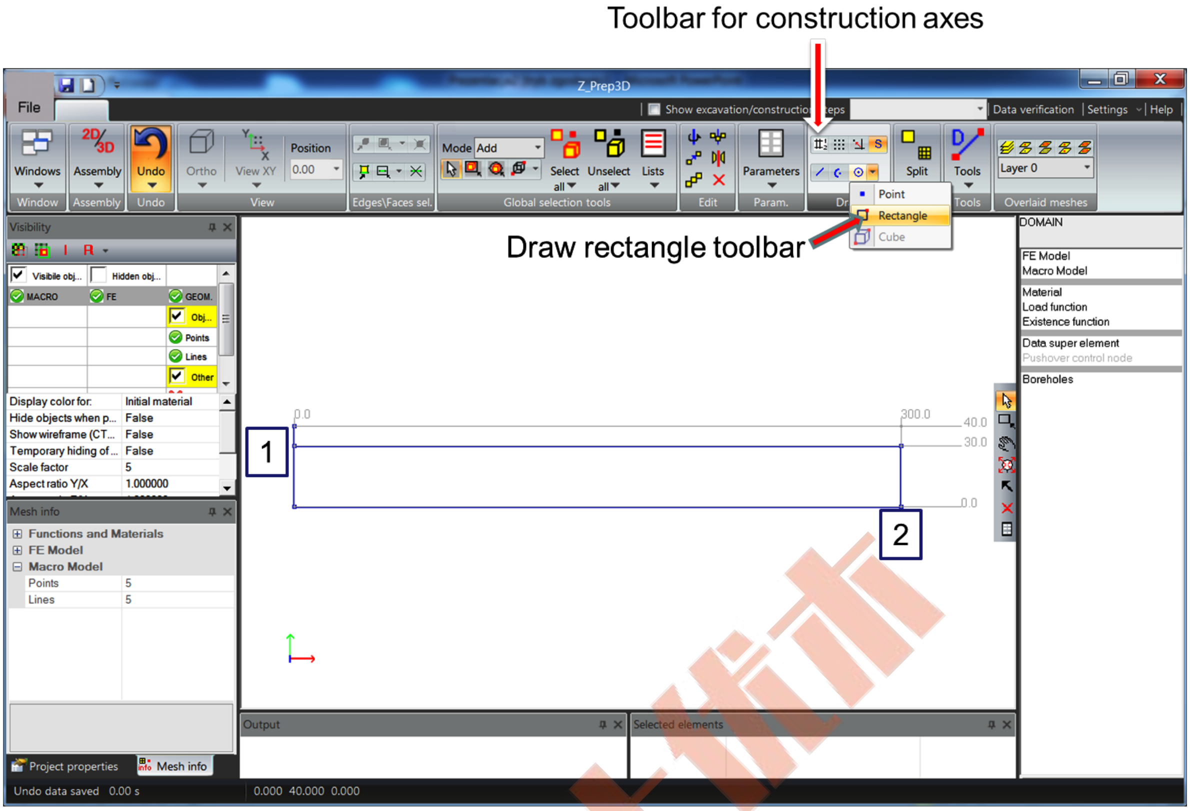
Step 3: Create list of load time functions



1. Open dialog box for editing load time functions through `Assembly/Load functions`
2. Add new function with index 1
3. Define this function through expression (press button **Expression**
4. Set the angle unit to radians (set ◎ **Rad** ON), set time range from $t = 1..6$ s, and in the edit **Expression** [ ] type $sin(12.56637062 * t)$ (period is equal to $T = 0.5$ s, hence $\omega = 4\,\pi$)

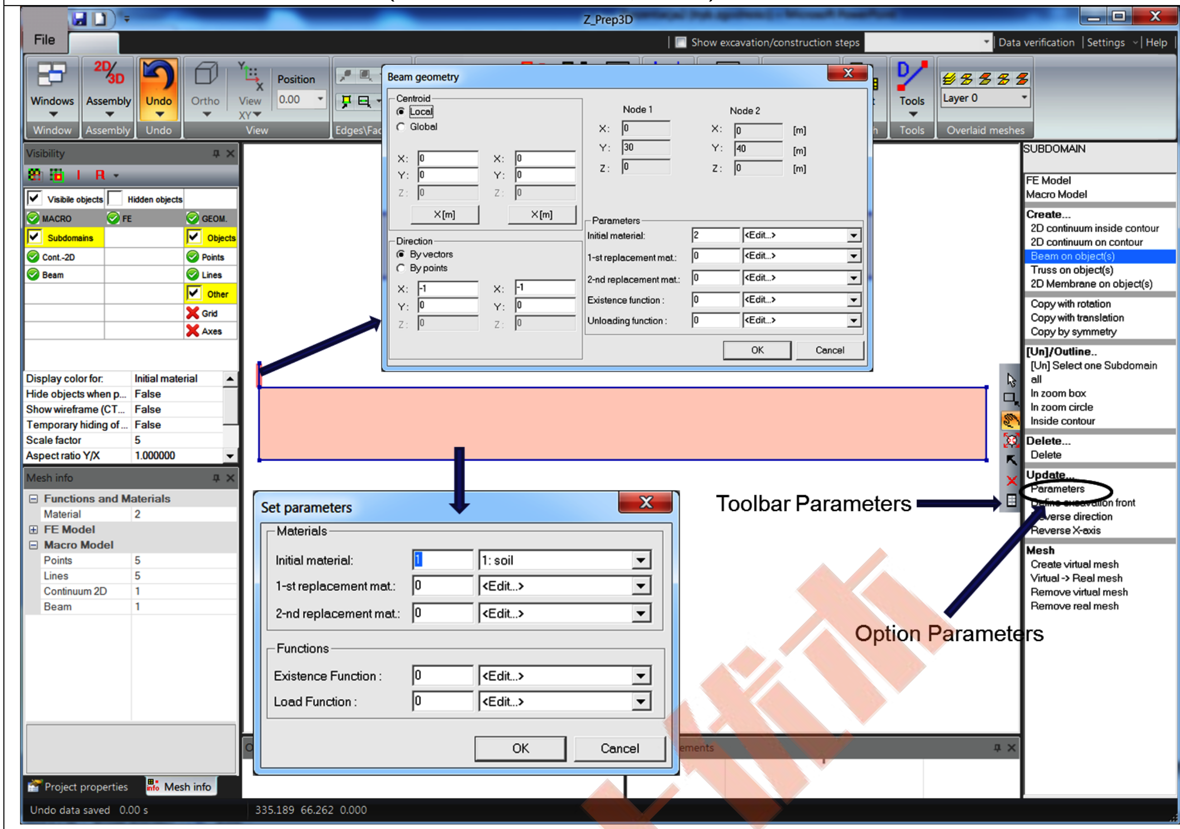
NB. If you want to learn more on how to edit load time functions watch video
Edit load time functions

Step 4: CAD drawing



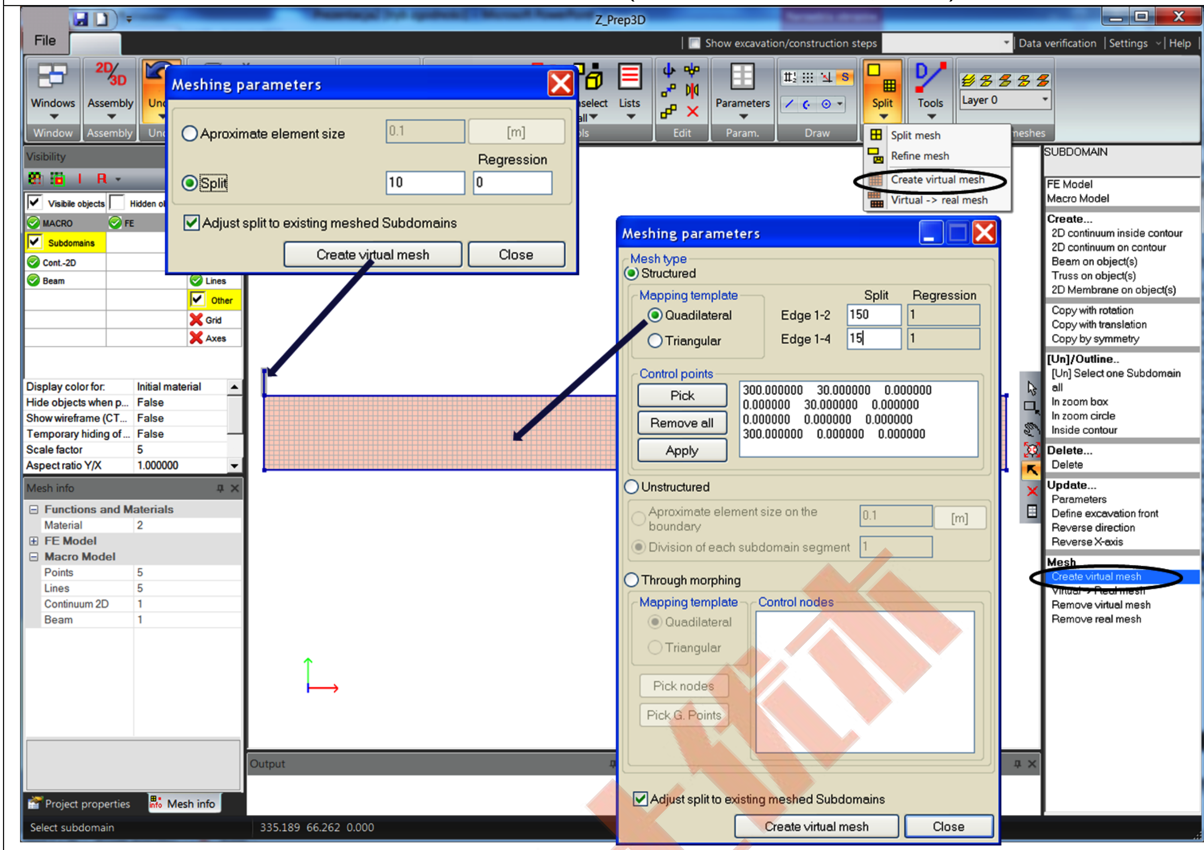
1. To speedup drawing of the model create construction axes using the toolbar in ribbon group `Draw`; add characteristic X coordinates 0 and 300m, and Y coordinates 0, 30 and 40m
2. Draw rectangle 300m $\times$ 30m (between points 1, 2) using toolbar for rectangle drawing (in ribbon group `Draw`); here we will plot only part of the model as aspect ratio is equal to $\dfrac{3600}{30} = 120$)
3. Draw line segment between 2 points (0,30)-(0,40) to be used later on for generation of concrete column

Step 5: Create subdomains (at macro-modeling level)



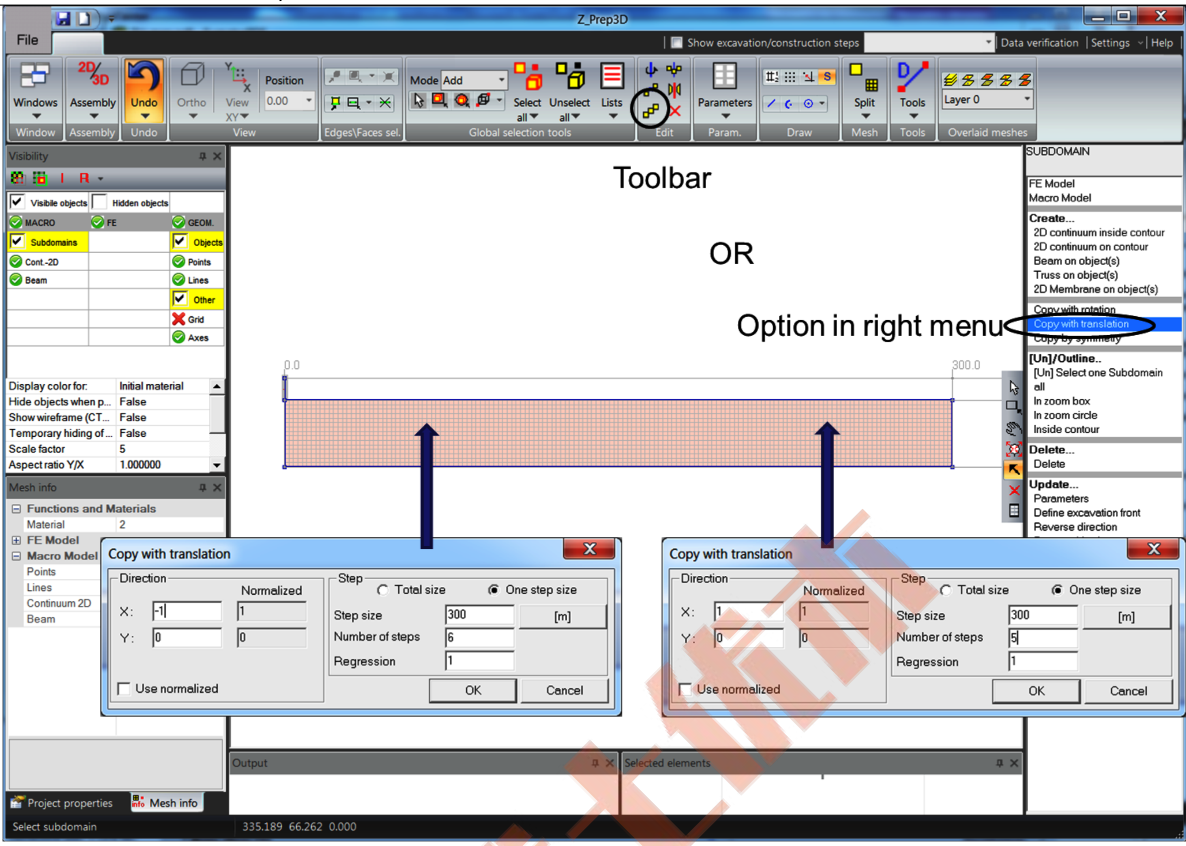
1. Create 2D continuum subdomain through method `Macro model/Subdomain/2D continuum in contour` by picking the point within the closed contour
2. Apply proper material number (1) to the continuum subdomain using option `Macro model/Subdomain/Update/Parameters`
3. Create beam subdomain through method `Macro model/Subdomain/Beam on object(s)` by picking the vertical line segment
4. Apply proper material number (2) to the beam subdomain using option `Macro model/Subdomain/Update/Parameters`

Step 6: Create virtual meshes for subdomains (at macro-modeling level)



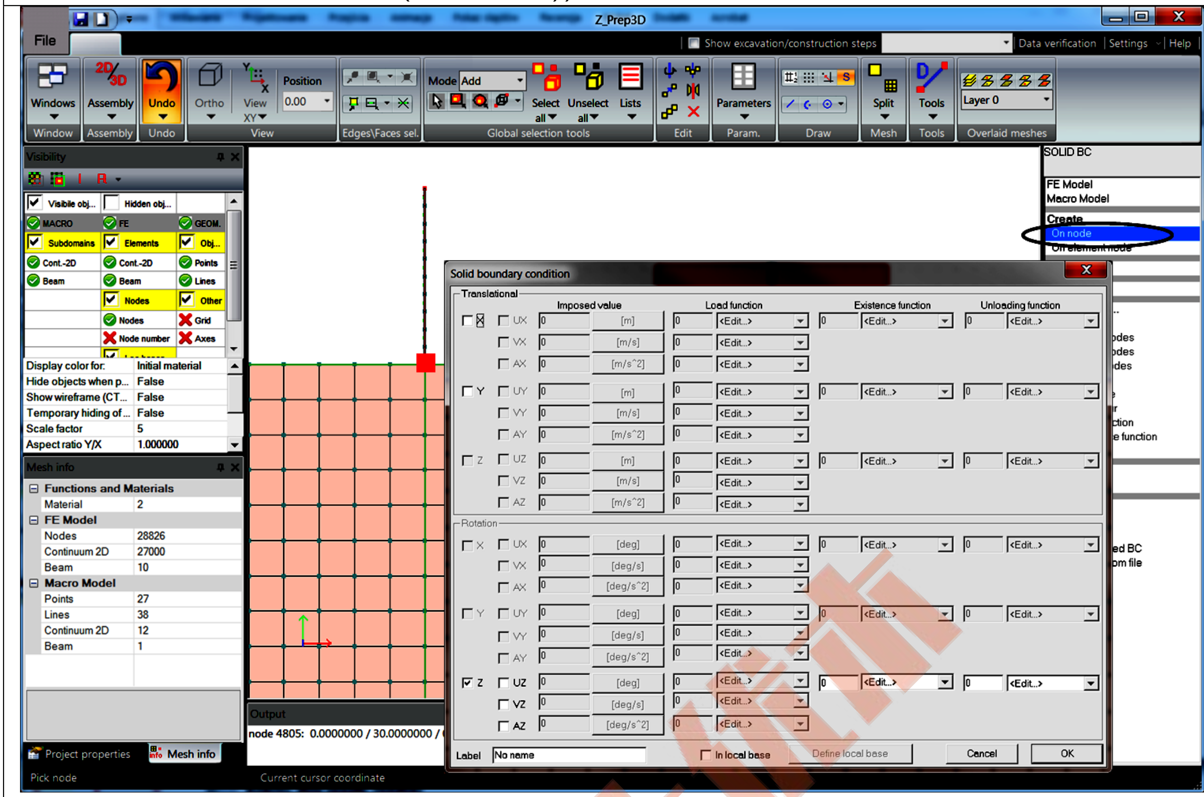
1. Create virtual structured mesh in continuum subdomain; use option `Macro model/Subdomain/Mesh/Create virtual` mesh and click on the subdomain to activate the dialog box (or use toolbar in ribbon menu `Mesh`)
2. Set number of elements along 1-2 and 1-4 edges
3. Create virtual structured mesh in beam subdomain; use option `Macro model/Subdomain/Mesh/Create virtual mesh` and click on the subdomain to activate the dialog box (toolbar in ribbon menu `Mesh` can be used as well)
4. Set number of elements for beam subdomain

Step 7: Create whole continuum domain by copying 2D subdomain and create real mesh(at macro-modeling level)



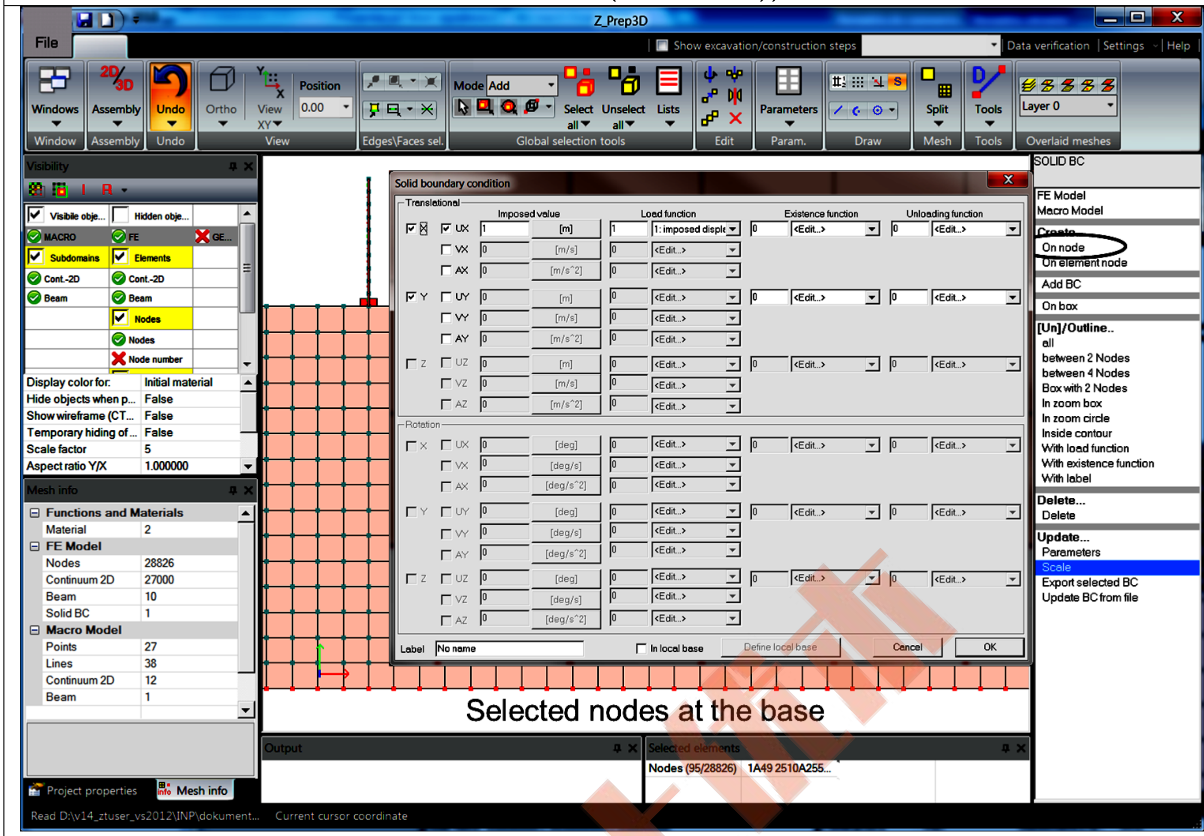
1. Copy selected 2D subdomain in right direction 5 times using option `Macro model/Subdomain/Copy with translation` or toolbar in ribbon menu `Edit`
2. Copy selected 2D subdomain in left direction 6 times using option `Macro model/Subdomain/Copy with translation` or toolbar in ribbon menu `Edit`
3. Select all subdomains (`Macro model/Subdomain/Outline/all`) and create real meshes using option (`Macro model/Subdomain/Mesh/Virtual`→`Real`)

Step 8: Create BC for beam (at FE level))



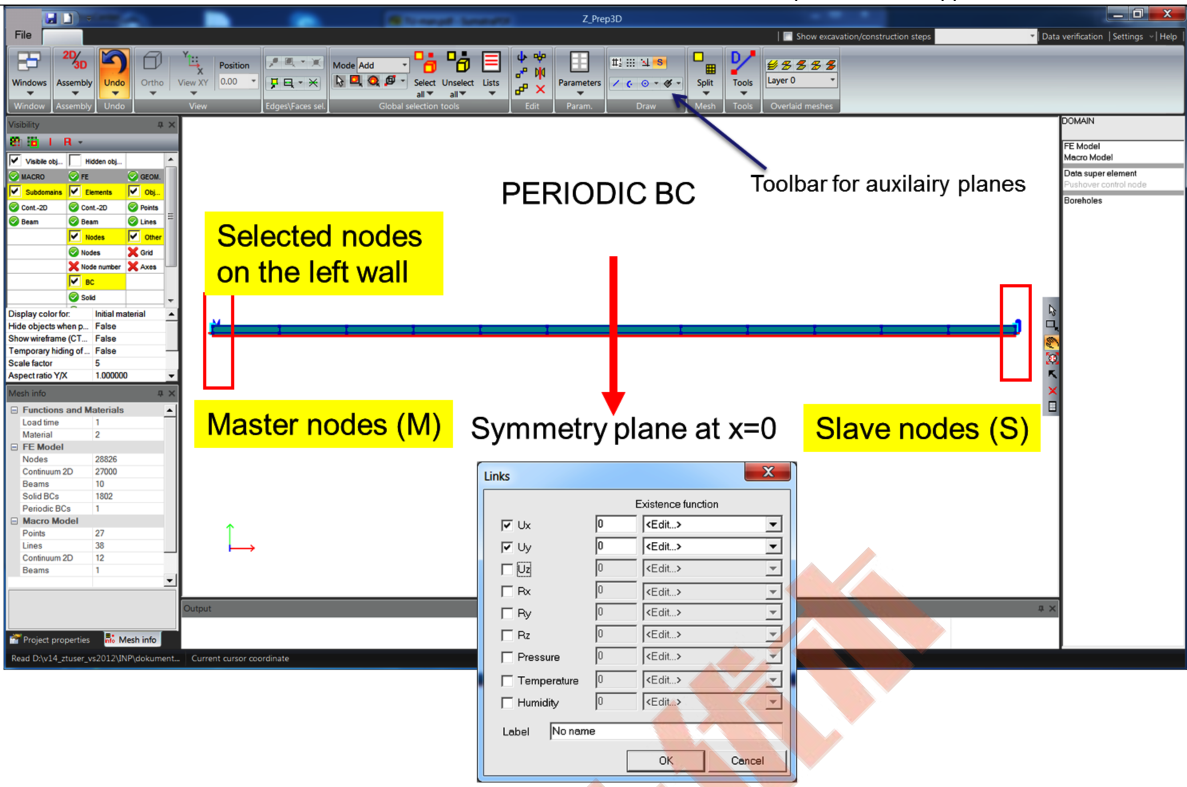
1. Fix rotation at base of the beam using option `FE model/Boundary conditions/Solid BC/On node` and picking the nodal point

Step 9: Create BC for the bottom boundary (at FE level))



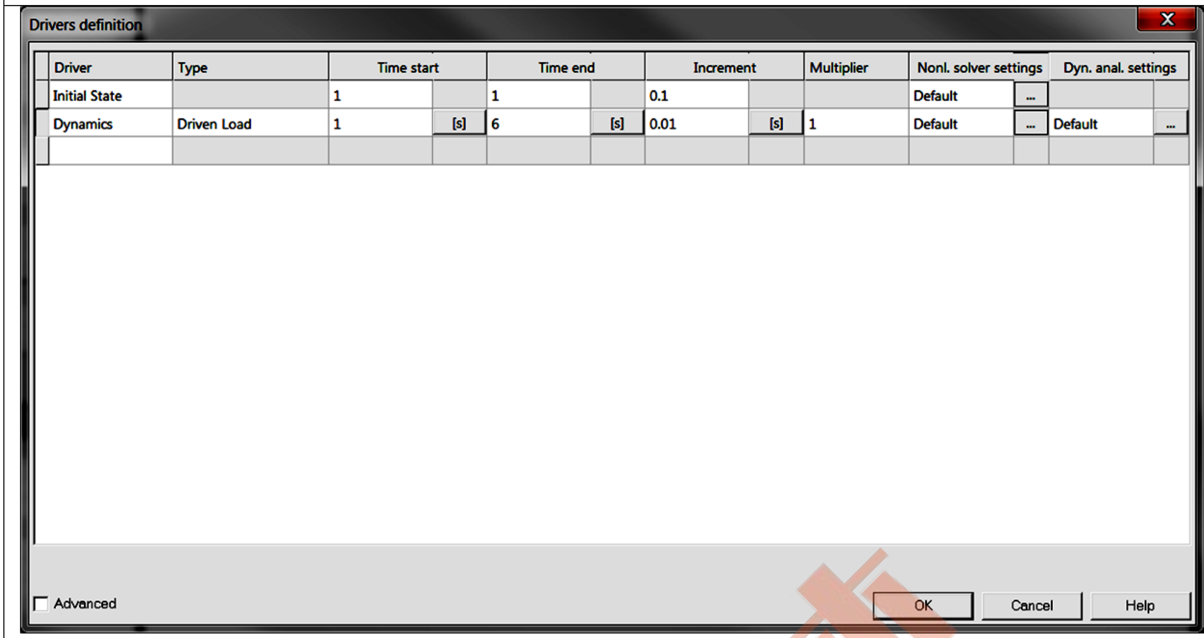
1. Select all nodes at the bottom boundary of the 2D domain
2. Fix UY and UX dofs; in addition set value of the imposed displacement amplitude along x direction to 1.0 and associate with it the load time function number 1; use FE `model/Boundary conditions/Solid BC/On node` option to make this setup

Step 10: Create periodic BC for left and right vertical walls (at FE level))



1. Create an auxiliary plane at x =0 with normal oriented towards x axis (option in ribbon menu `Draw`)
2. Select all nodes, except those at the base, at the left wall
3. Generate periodic boundary condition $\mathbf{u}_M = \mathbf{u}_S$ (option `FE model/Boundary Conditions/Periodic BC/Nodes` and `Plane`) to tie all degrees of freedom on both vertical boundaries (note that in this specific case there is no need to define standard displacement BC on vertical mesh walls and then to excavate them); to define these BC pick plane;

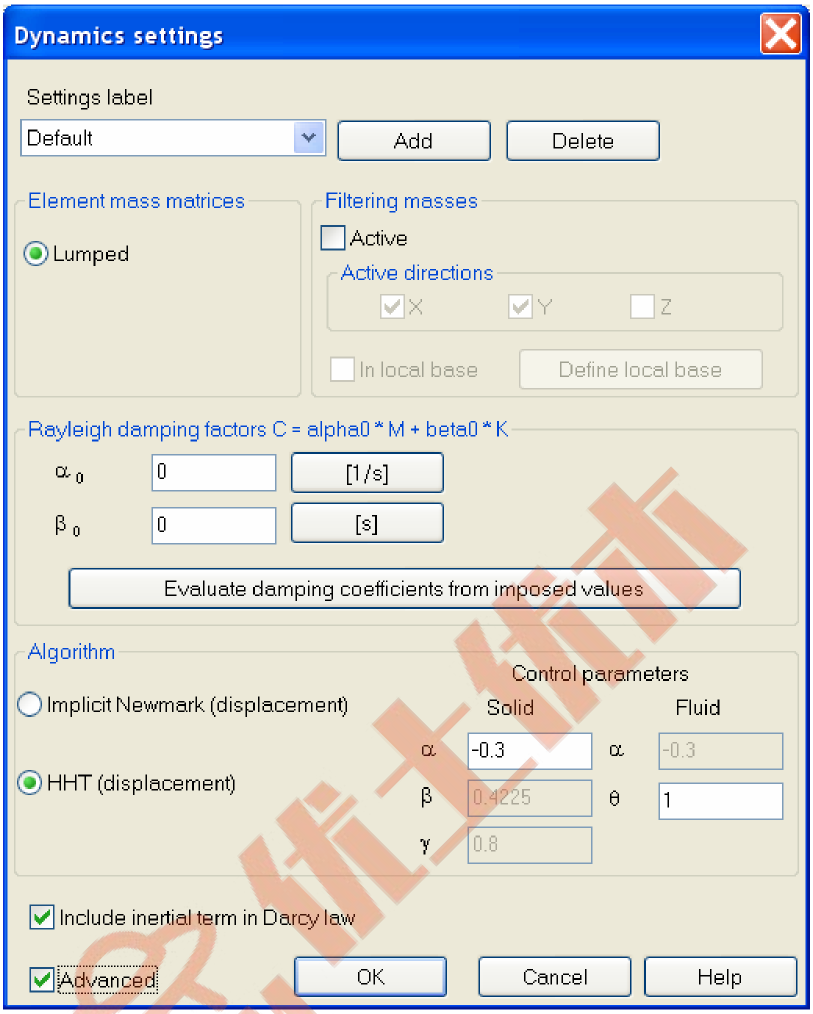
Step 11: Setup drivers



1. Add `Initial state` driver to generate the in situ stress state
2. Add `Transient dynamics` driver from time 1..6 s to perform dynamic time history analysis

NB. Drivers can be set both in the main ZSoil®menu and in the ribbon menu `Assembly/Drivers` in the preprocessing stage.

Step 12: Setup transient dynamics driver

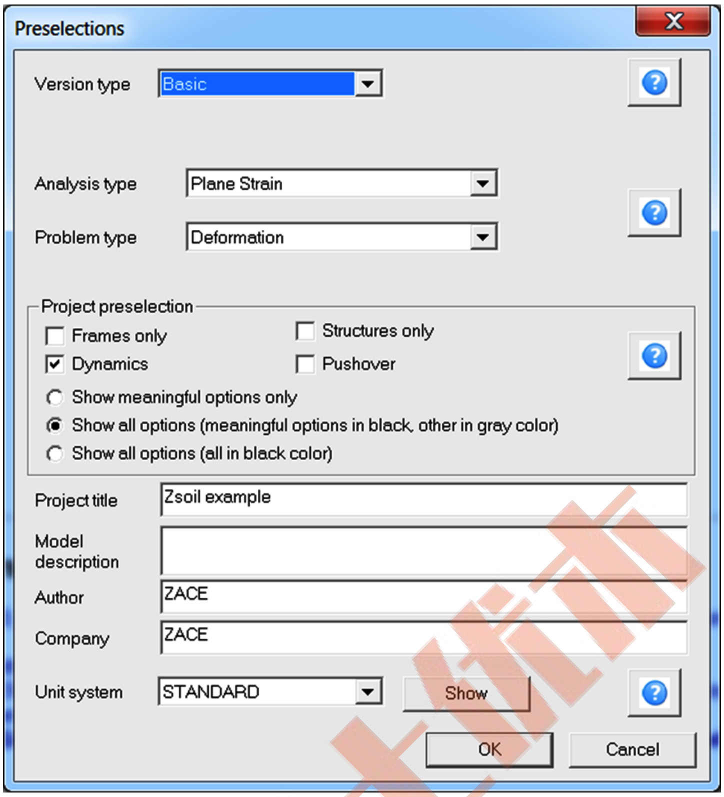


1. Use HHT integration scheme with default integration coefficients (main ZSoil®menu `Control/Dynamics`)

1.8.2 BACKGROUND MODEL

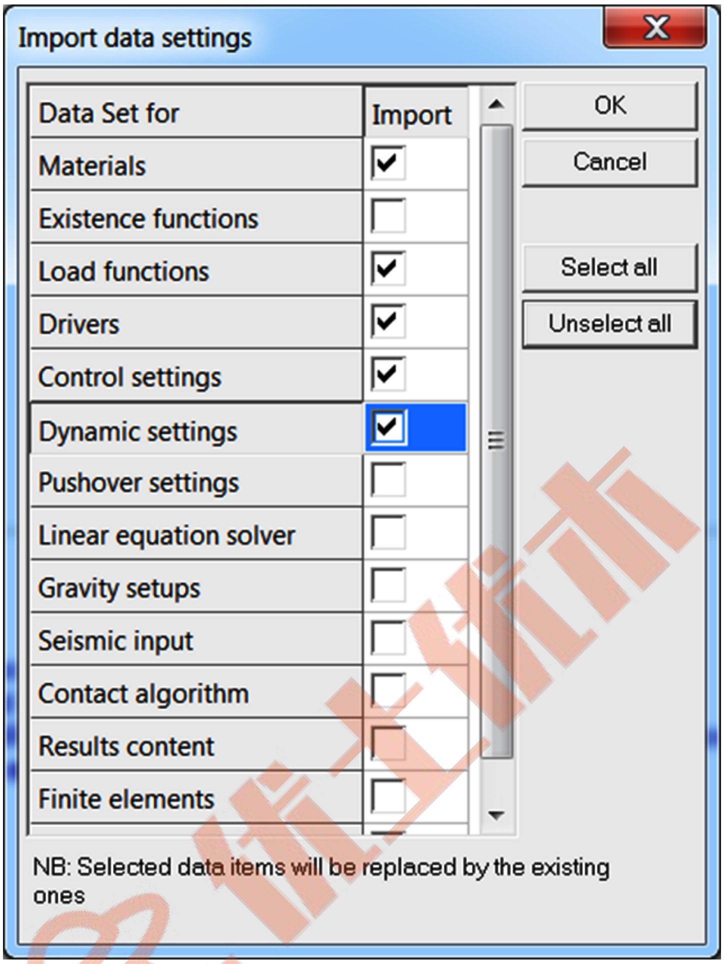
Background model is needed to carry out the time history analysis with aid of Domain Reduction Method. In the considered case we will make a simple model of shear layer with one column of Q4 elements and periodic BC on both vertical walls. Size of the model along x axis will be equal to 3600m although it would be enough to make it as long as the length of the DRM model described in the next subsection. This simplified model will be subject to harmonic excitation with frequency 2Hz in the same way as the full-size model.To carry out the time history analysis of column-subsoil interaction, without additional elements cancelling waves reflections from rigid boundaries, a large computational model must be generated. In the considered case 3600m long model was generated. It enables such an analysis during first 5s of the excitation, performed with 2Hz frequency. In the following table all major steps leading to the final model will be summarized.

Step 1: Open new project



1. Select `Plane Strain`
2. Set ☒ **Dynamics** ON
3. Accept these settings pressing button **OK**
4. `File/Save As` **FF-SHL.inp**

Step 2: Import list of materials, load time functions, drivers, control settings from SOIL-COLUMN.inp file



1. To reuse the material data, load time functions, drivers defined in the full size model (SOIL-COLUMN.inp) we can use option `File/Import data from another *.inp file`
2. The second material, for the concrete column, is not needed, but its presence in the data is not meaningful

Step 3: CAD drawing

1. To speedup drawing of the model create construction lines using the toolbar, or top menu `Settings/Construction lines`; add characteristic X coordinates -1800, 0 and 1800, and Y coordinates 0 and 30
2. Draw a rectangle (3600m $\times$ 30m) from $x = -1800$m to $x = 1800$m and $y = 0$ to $y = 30$m; in this case, due to huge aspect ratio of the model, it is recommended to enter coordinates of the vertices directly through the dialog box rather then to draw it; note that the horizontal size of this model can be reduced up to dimension of the reduced model (100m), presented in the next subsection

NB. Drawing of the domain contours can be made directly by mouse but also by setting coordinates of the endpoint in the dialog box; coordinates of the endpoint can be defined as absolute coordinates or as relative ones (in this case put @ character before X, Y, Z coordinates)

Step 4: Create 2D continuum subdomain

1. In the same way as for the full size project create one 2D subdomain, make virtual split 1 $\times$ 15 (1 element along the length) and then convert it to the real mesh (see this operation for full size project generation)

NB. Length of this model has to be longer or equal to the length of the DRM model

Step 5: Create boundary conditions at the bottom boundary

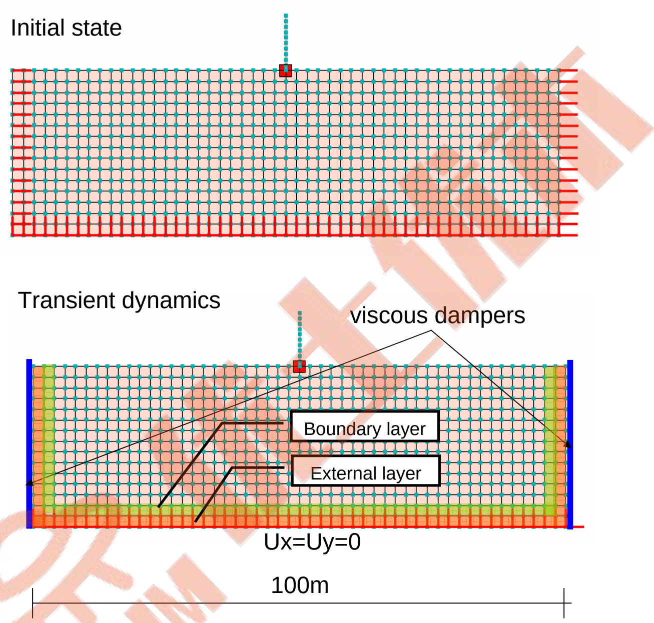
1. Fix UY and UX dofs; in addition set value of the imposed displacement amplitude along x direction to 1.0 and associate with it the load time function number 1; use `FE model/Boundary conditions/Solid BC/On node` option to make this setup; this operation is exactly the same as for full size model (see this operation for full size project generation)

Step 6: Create periodic BC for left and right vertical walls (at FE level))

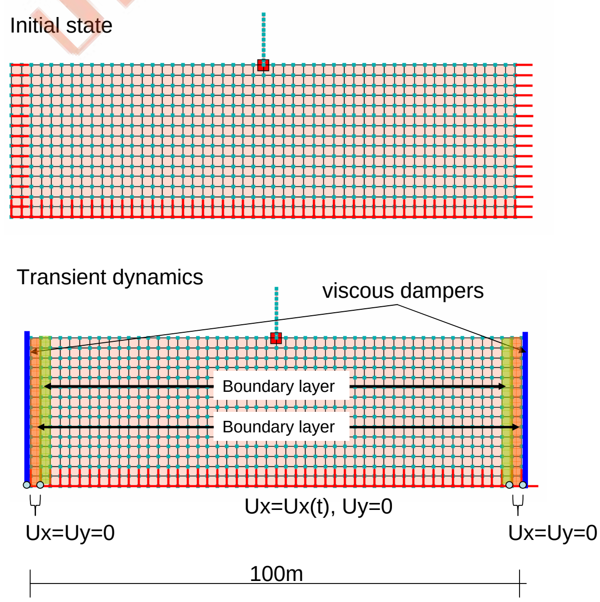
1. Generate periodic boundary condition $\mathbf{u}_M = \mathbf{u}_S$ (option `FE model/Boundary Conditions/Periodic BC`) to tie all degrees of freedom on both vertical boundaries; this operation is exactly the same as the one described for the full size model (see this operation for full size project generation)

1.8.3 DRM REDUCED MODEL

The reduced model will consist of a small part of subsoil adjacent to the concrete column plus the associated background model that was already solved. The DRM mdels can be constructed in several ways. The two possible cases are shown in figures below. In this tutorial we will use the first scheme as it is much simpler (both are practically equivalent). As the model is symmetric we can replace standard horizontal fixities on the two vertical mesh walls by periodic BC. However, in more complicated cases we may need to assume the standard box-shape type BC to solve the initial state and later on, when dynamic driver is activated, will have to remove them. Hence in the considered case the initial state will be solved first with box type BC, then horizontal fixities will be released (but reactions will be kept) and reduced model, with free field motion imported from the simplified background model, will be carried out.

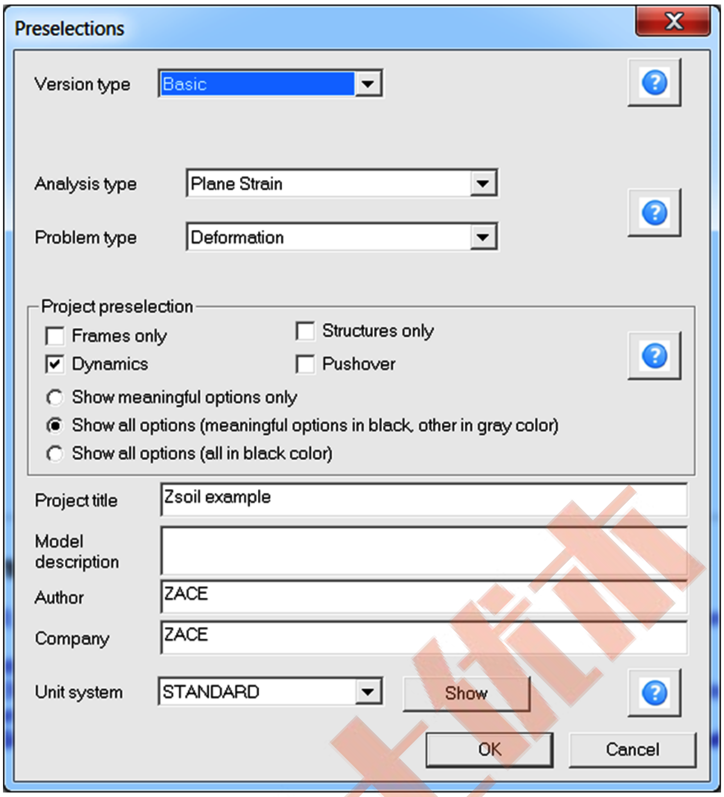


Case 1: exterior/boundary layers placed at the bottom and along vertical walls



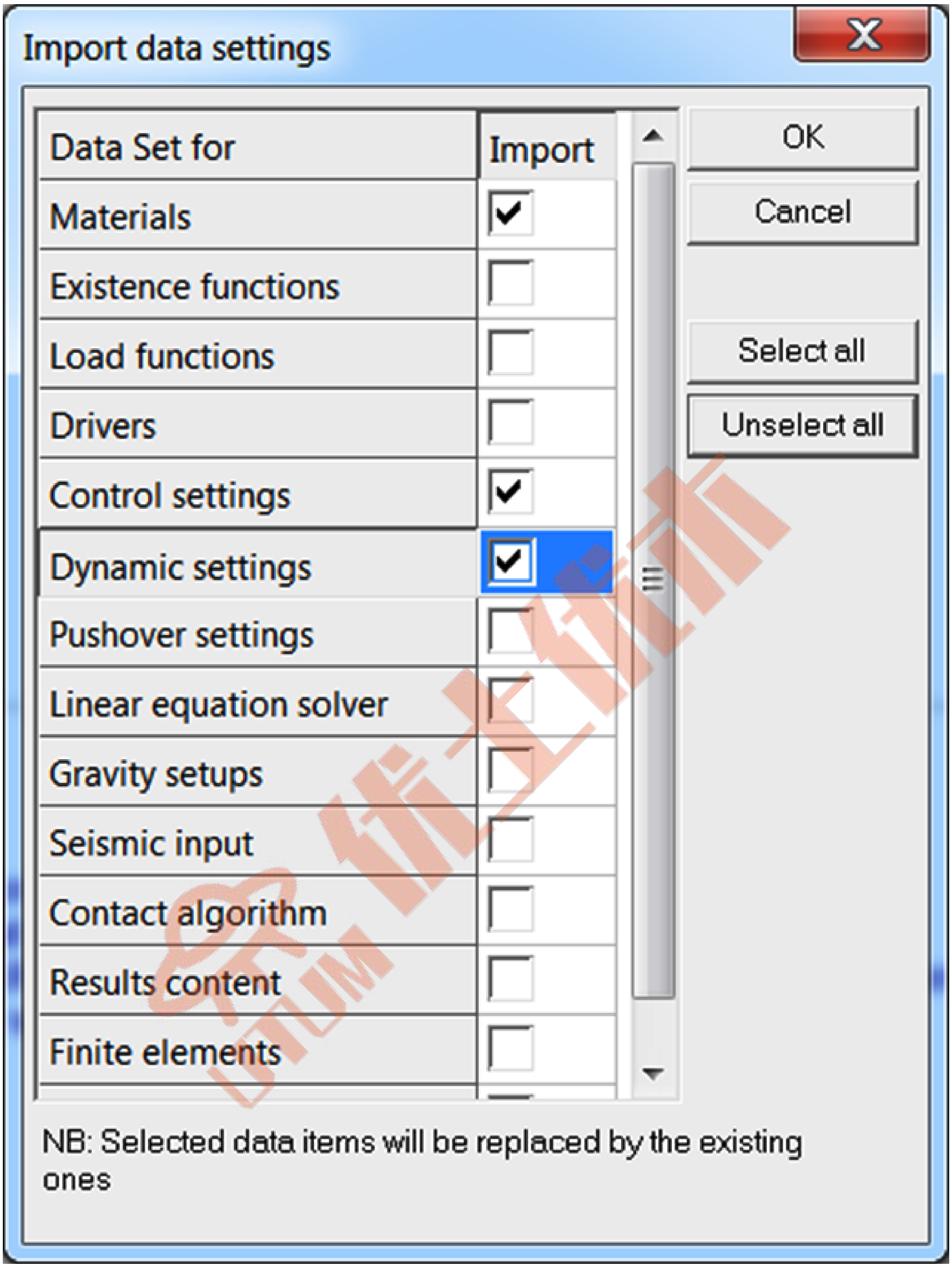
Case 2: exterior/boundary layers placed only along vertical walls

Step 1: Open new project



1. Select `Plane Strain`
2. Set ☒ **Dynamics** ON
3. Accept these settings pressing button **OK**
4. `File/Save As` **DRM-S-FF-SHL.inp**

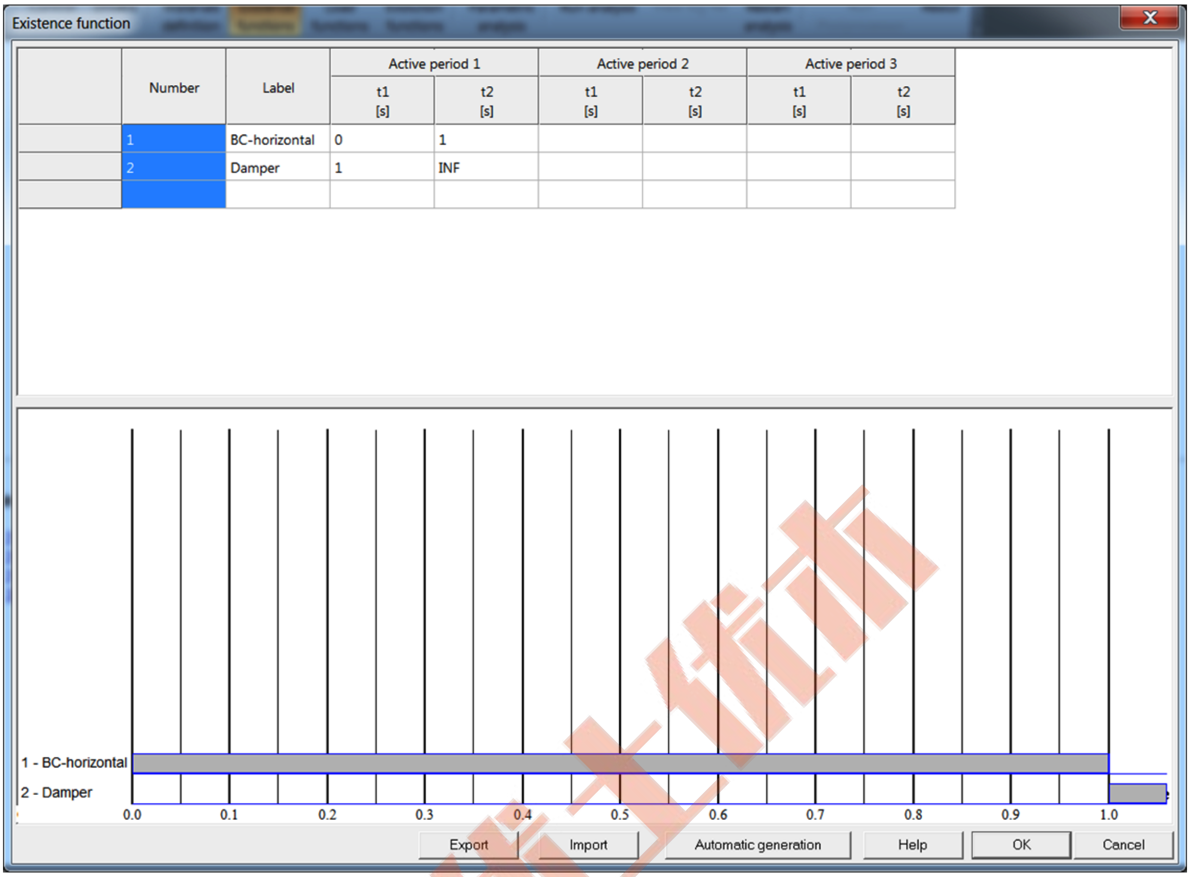
Step 2: Import materials, dynamic control from SOIL-COLUMN.inp file



1. Import materials and dynamic settings from the full size model (SOIL-COLUMN.inp) through main ZSoil®menu option `File/Import data from another *.inp file`

2. Add new material for viscous dampers `Assembly/Materials/Add material` and set the flag ⊠ **Inherit properties from adjacent continuum** to ON; this way each damper will inherit its mechanical and flow properties from the adjacent continuum element

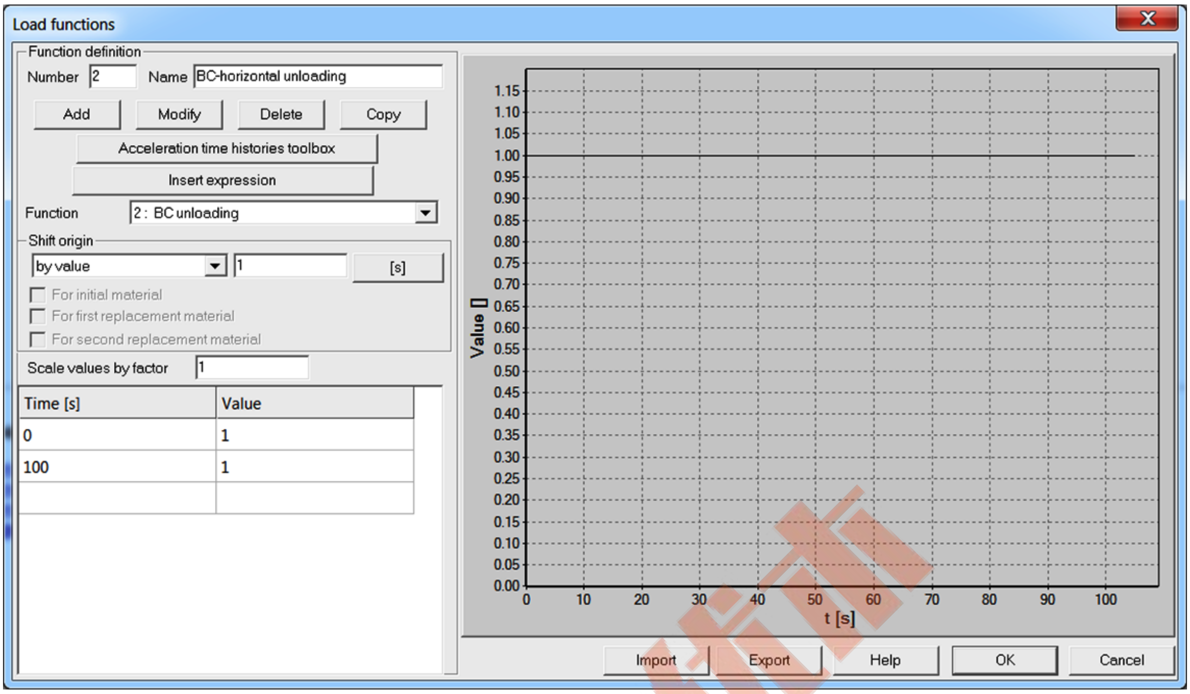
Step 3: Create list of existence functions



1. Open dialog box for editing existence functions through `Assembly/Existence functions` in main ZSoil®window or in the preprocessor
2. Add new function with index 1 and label *BC Horizontal*
3. Define this function as active in the period $< 0..1 >$; this function will be associated, later on, with the horizontal BC at vertical walls
4. Add new function with index 2 and label *Damper*
5. Define this function as active in the period $(1..\infty)$; this function will be associated, later on, with the viscous dampers

NB. If you want to learn more on how to edit existence functions watch video
Edit existence functions

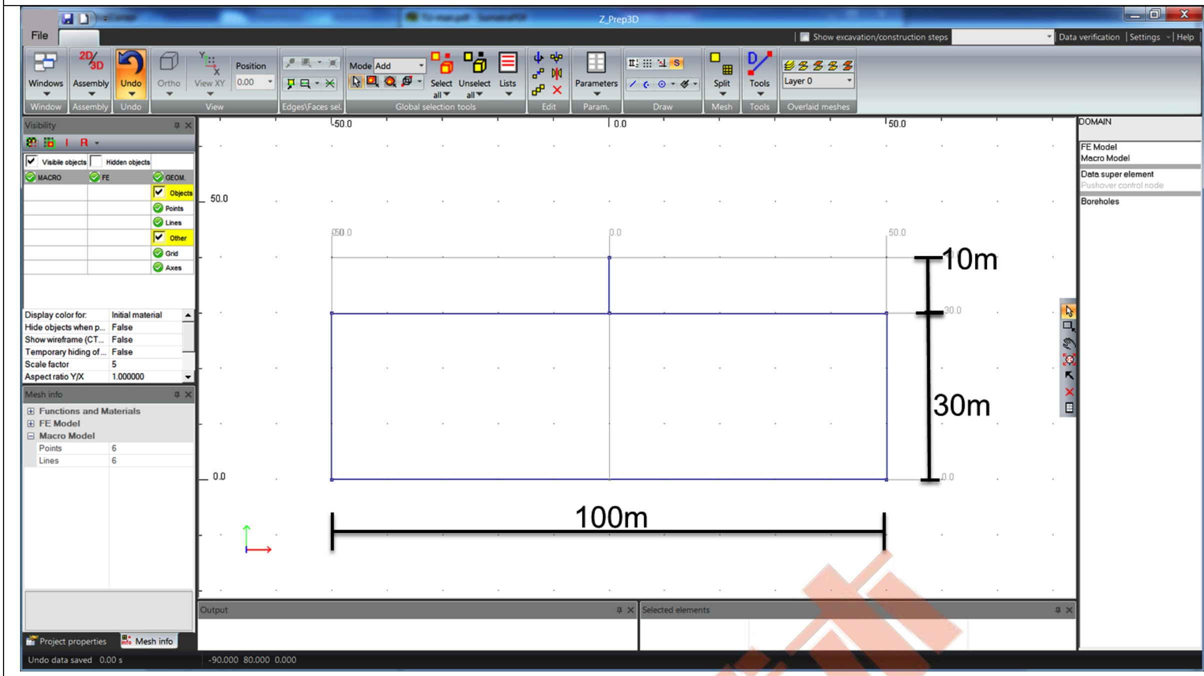
Step 4: Create list of load time functions



1. Open dialog box for editing load time functions through `Assembly/Load functions` in main ZSoil®window or in the preprocessor
2. Add new function with index 2
3. Assume that the time origin of this function is shifted by value 1.0
4. Edit this function and set three points $(LTF_2(t = 0) = 1,\ LTF_2(t = 100) = 1)$; this function will be used as an unloading function for horizontal reactions at vertical walls when running dynamic driver (reaction forces will not be dissipated)

NB. If you want to learn more on how to edit load time functions watch video
Edit load time functions

Step 5: CAD drawing

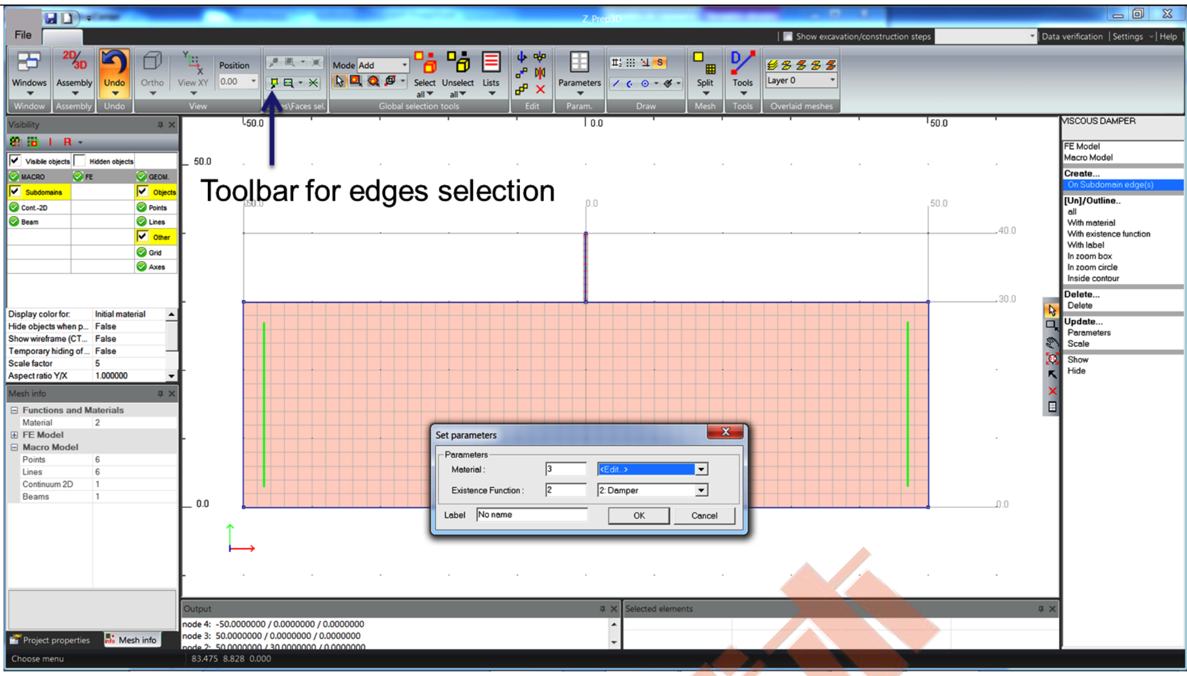


1. To speedup drawing of the model create construction lines using the toolbar, or top menu `Settings/Construction lines`; add characteristic X coordinates -50, 0 and 50, and Y coordinates 0, 30 and 40m
2. Draw a rectangle (100m × 30m) from $x = -50$m to $x = 50$m and $y = 0$ to $y = 30$m
3. Draw vertical line at $x = 0$ from $y = 30$ to $y = 40$ m

Step 6: Create 2D continuum subdomain

1. In the same way as for the full size project create 2D subdomain, make virtual split 50 × 15 (50 elements along the length)
2. Create beam subdomain with virtual split 10

Step 7: Create viscous dampers

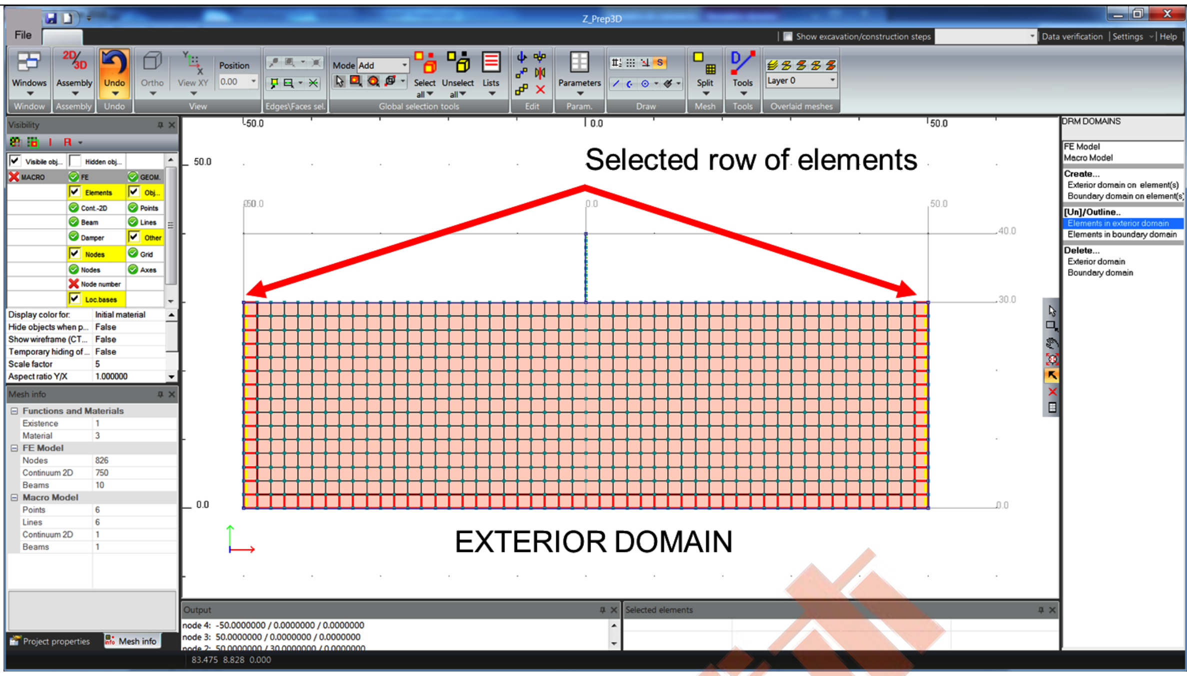


1. Select vertical external edges of 2D subdomain and create viscous dampers using option `Macro model/Viscous dampers/Create/On subdomain edge(s)`
2. Set material No 3 and existence function No 2 for dampers

Step 8: Convert virtual to real mesh

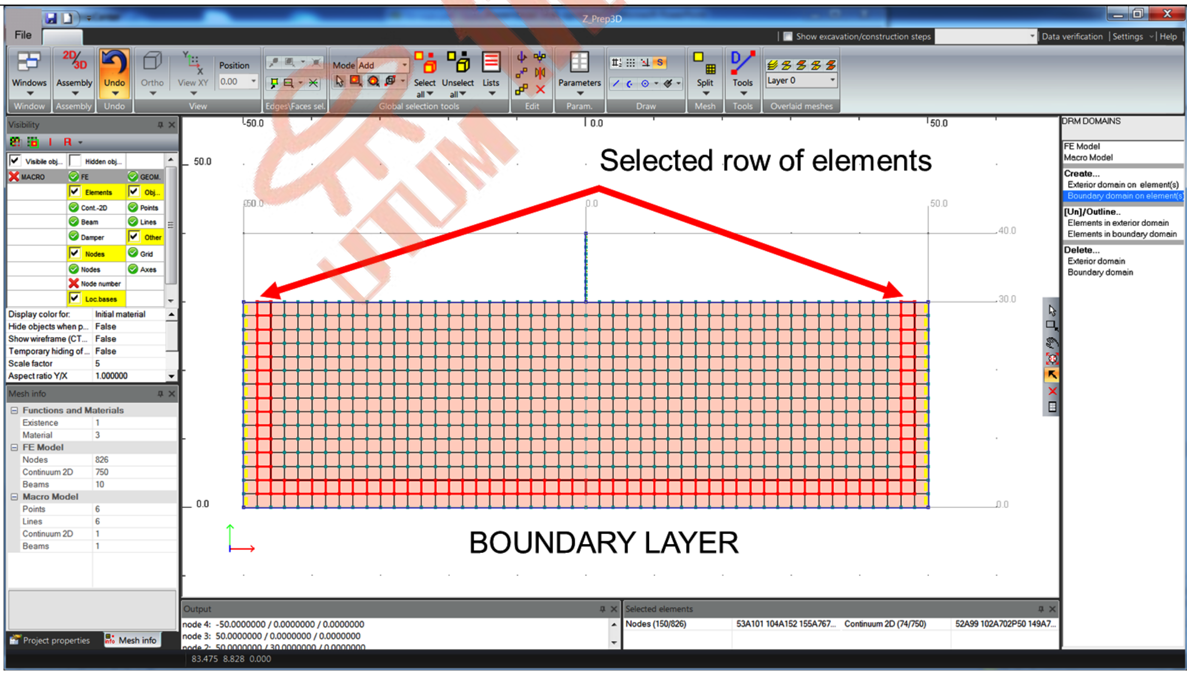
1. Select all subdomains (`Macro model/Subdomain/Outline/all` or toolbar from ribbon menu `Global selection tools/Select all`)
2. Convert virtual meshes to real ones (`Macro model/Subdomain/Mesh/Virtual to real` or ribbon menu toolbar `Mesh/Virtual to real`)

Step 9: Create DRM exterior domain



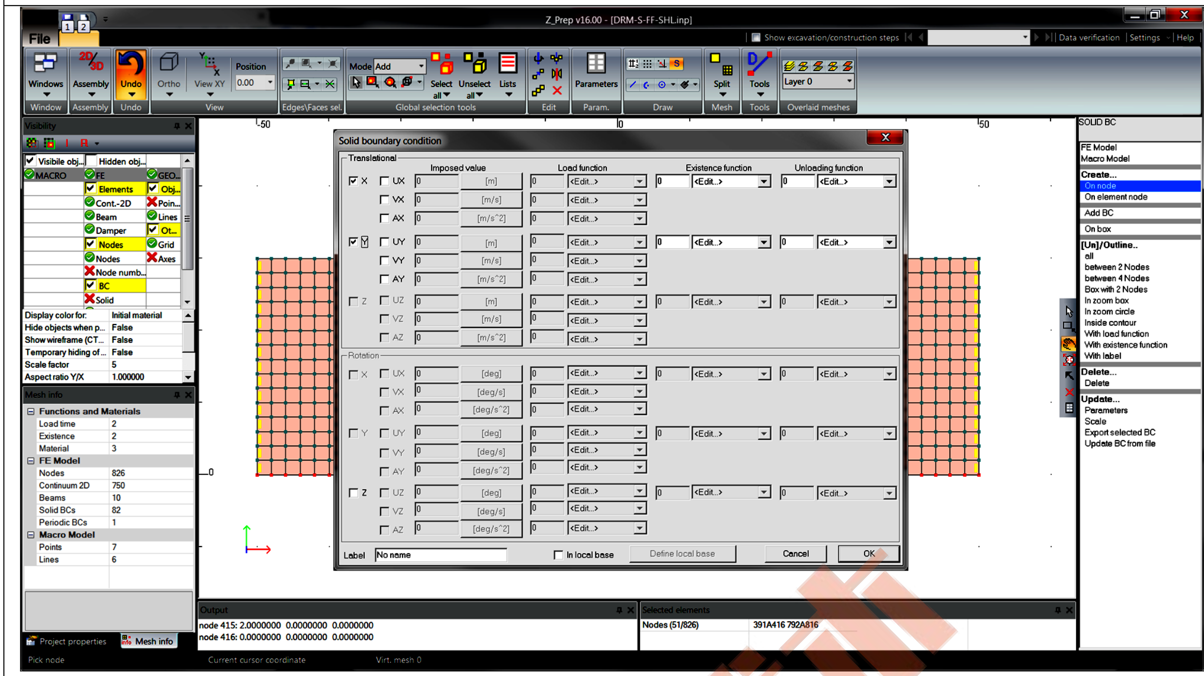
1. Select single row of elements that form the external layer and set the exterior DRM domain there; use `FE model/DRM domains/Create/Exterior domain on element(s)` option to make this setup

Step 10: Create DRM boundary layer



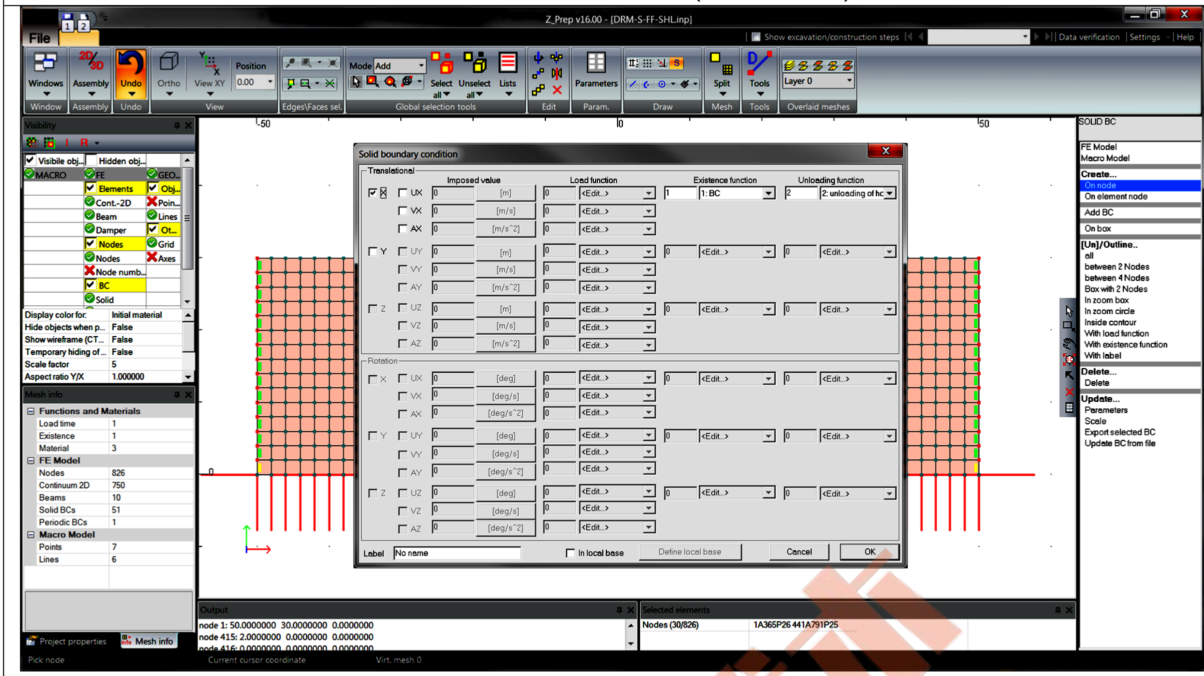
1. Select single row of elements that form the boundary layer and set the boundary DRM domain there; use `FE model/DRM domains/Create/Boundary domain on element(s)` option to make this setup

Step 11: Create boundary conditions at the bottom boundary



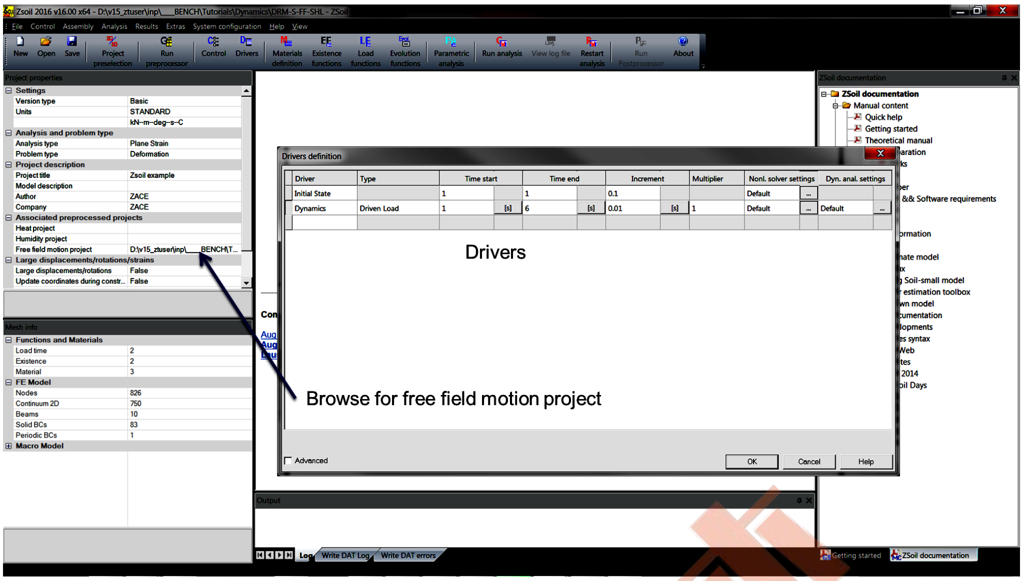
1. Fix UY and UX dofs at bottom nodes; note that in this specific case, nodes at the base belong to the exterior domain, therefore for the rigid base model the relative motion is exactly zero in all these points (in exterior domain we seek for the relative/residual motion during calculations);

Step 12: Create horizontal fixities for vertical walls (at FE level)



1. Select all nodes, except those at the base, at the left and right mesh wall
2. Generate boundary condition (option `FE model/Boundary Conditions/Solid BC`) $u_x = 0$ with the existence function No 1 and unloading function No 2; this way the horizontal fixities will be active during initial state computation and nonactive during dynamic analysis; the unloading function helps to replace fixities by reactions that preserve static equilibrium prior running dynamic driver

Step 13: Setup drivers and plug the free field motion project

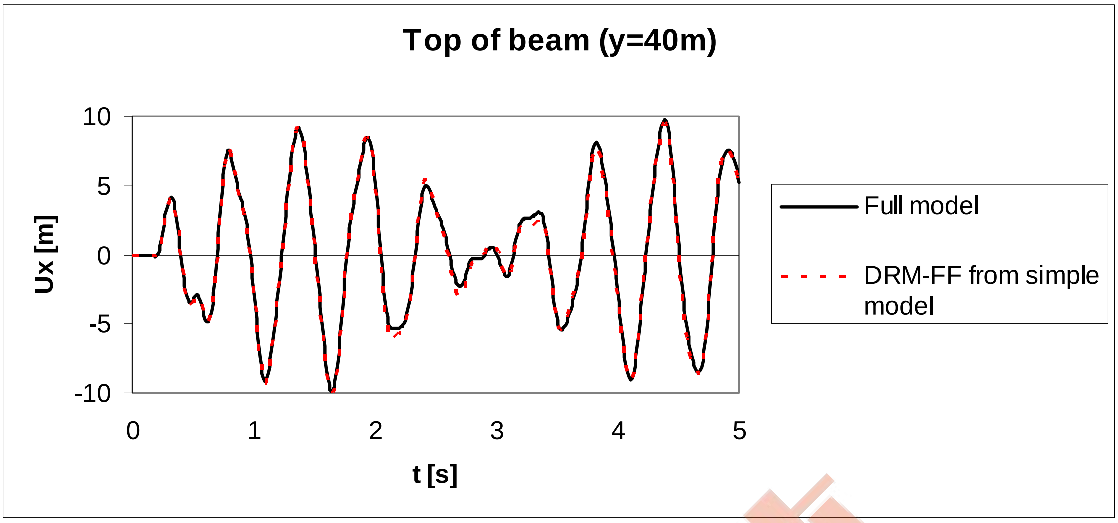


1. Create same list of drivers as for the full size model (see definition for the full size model)
2. Browse for the input file of the background model

Step 14: Setup transient dynamics driver

1. Use same settings for the dynamic driver as in the full size model (see definition for the full size model)

Step 15: Results

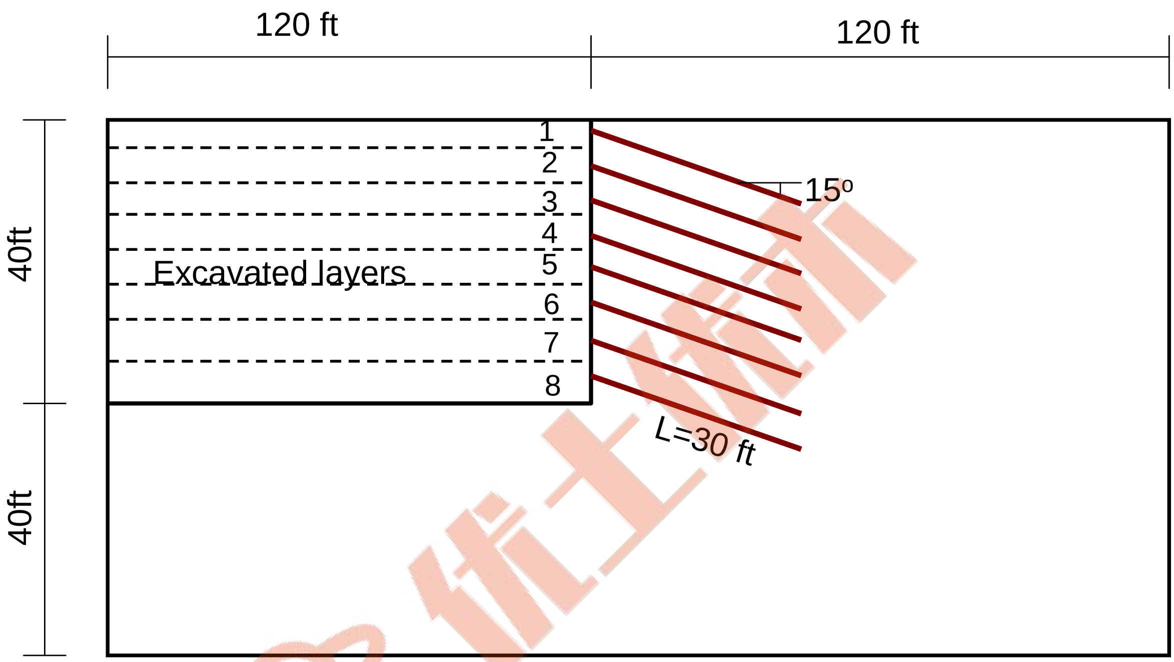


1. Open project for full model
2. Select top node of the concrete column
3. Open postprocessor and use option `Graph. option/Nodal time history` to plot evolution of $u_x(t)$ of the selected node
4. Save the time history in the *.csv format
5. Open project with reduced model
6. Select top node of the concrete column
7. Open postprocessor and use option `Graph. option/Nodal time history` to plot evolution of $u_x(t)$ of the selected node
8. Save the time history in the *.csv format
9. Use Excel application (or similar) to plot both results

1.9 NAILING OF VERTICAL CUT

- **Data file: tutorials/NAIL2Da.inp**
- **Description**

A problem of nailing of a vertical cut is considered in this example. Contrary to some simple limit equilibrium methods finite element model requires a multi-step excavation and nail installation procedure to eliminate spurious forces in nails and potential numerical divergence problems. The draft of the problem is shown in the figure below.



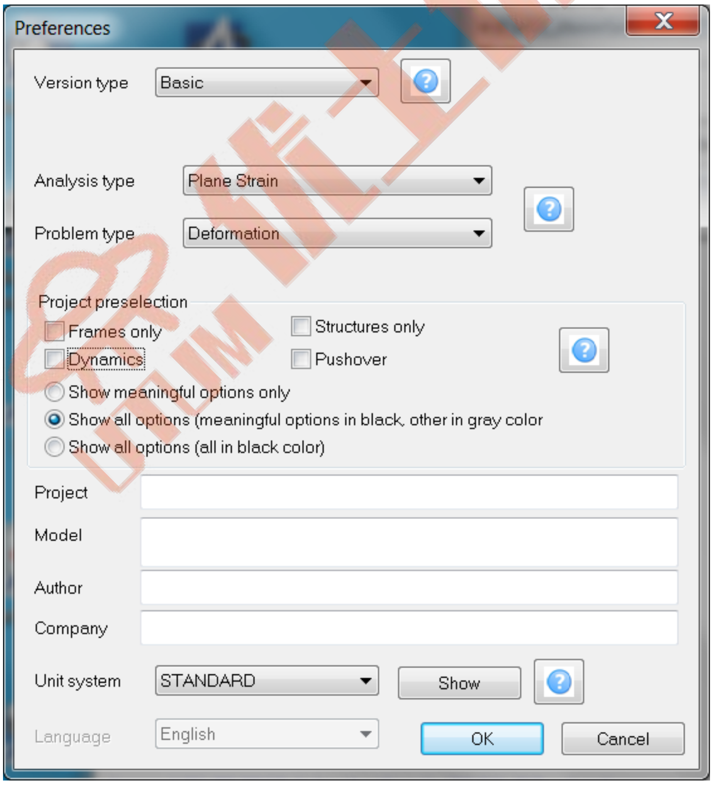
- **Materials**

Material properties defined for subsoil, concrete facing, nails and nail interface are summarized in the following table:

Material	Model	Data group	Properties	Unit	Value
1 Subsoil	Mohr-Coulomb	Elastic	E	[lbf/ft^2]	2088650
			ν	–	0.3
		Unit weights	γ_D	[lbf/ft^3]	120
			γ^F	[lbf/ft^3]	63.7
			e_o	–	0.0
		Nonlinear	c	[lbf/ft^2]	100
			ϕ	[^{o}]	32
			ψ	[^{o}]	0
		Initial State K_o	K_{ox}	–	0.5
			K_{oz}	–	0.5
2 Facing	Beams	Elastic	E	[lbf/ft^2]	417729000
			ν	–	0.2
		Unit weights	γ	[lbf/ft^3]	0
		Geometry	Rectangular	–	
			b	[ft]	1.0
			h	[ft]	0.5

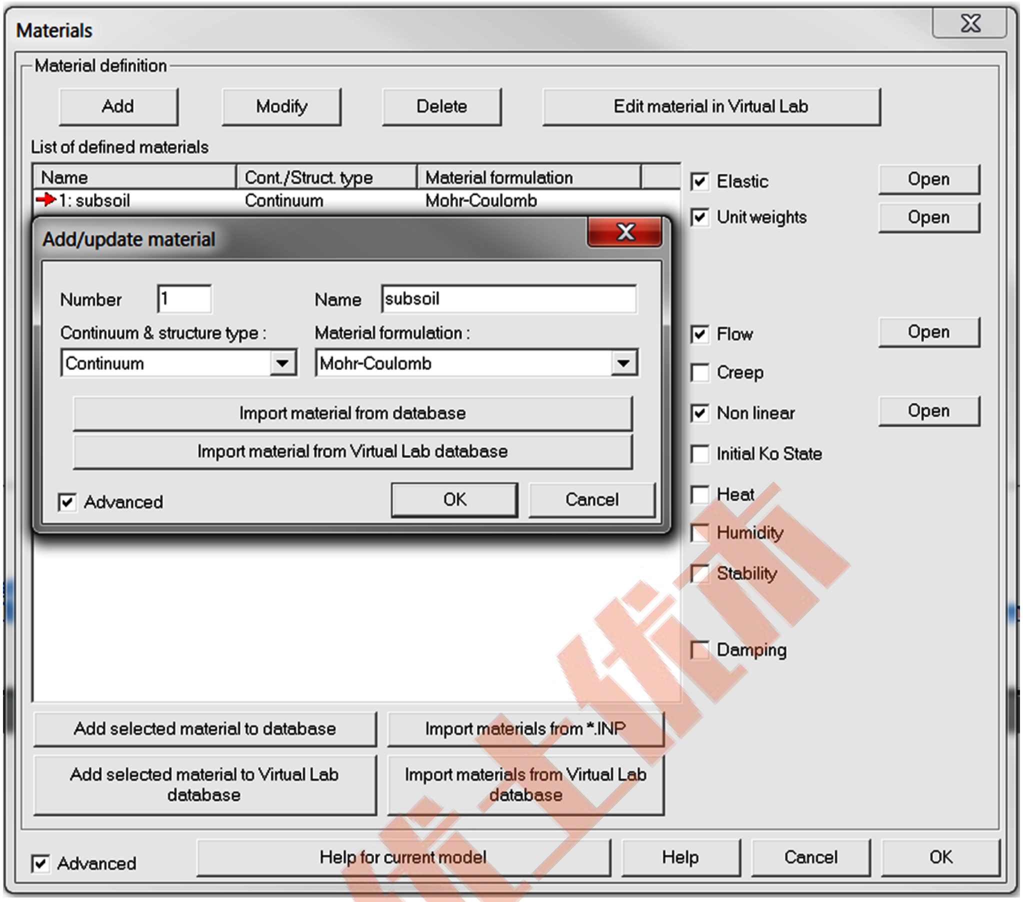
3	Interface	Nail interface	Elastic	k_n mult.	–	0.1
				$\frac{k_t}{k_n}$ mult.	–	1.0
			Nonlinear	Diameter D	[ft]	0.333
				τ_{ult}	[lbf/ft^2]	7200
4	Nail	Beams	Elastic	E	[lbf/ft^2]	4177290000
				ν	–	0.2
			Unit weights	γ	[lbf/ft^3]	0
			Geometry	Circular	—	
				Diameter	[ft]	0.083
				Interval..	[ft]	5

Step 1: Open new project



1. Select `Plane Strain`
2. Accept these settings pressing button **OK**
3. `File/Save As` **NAIL2Da.inp**

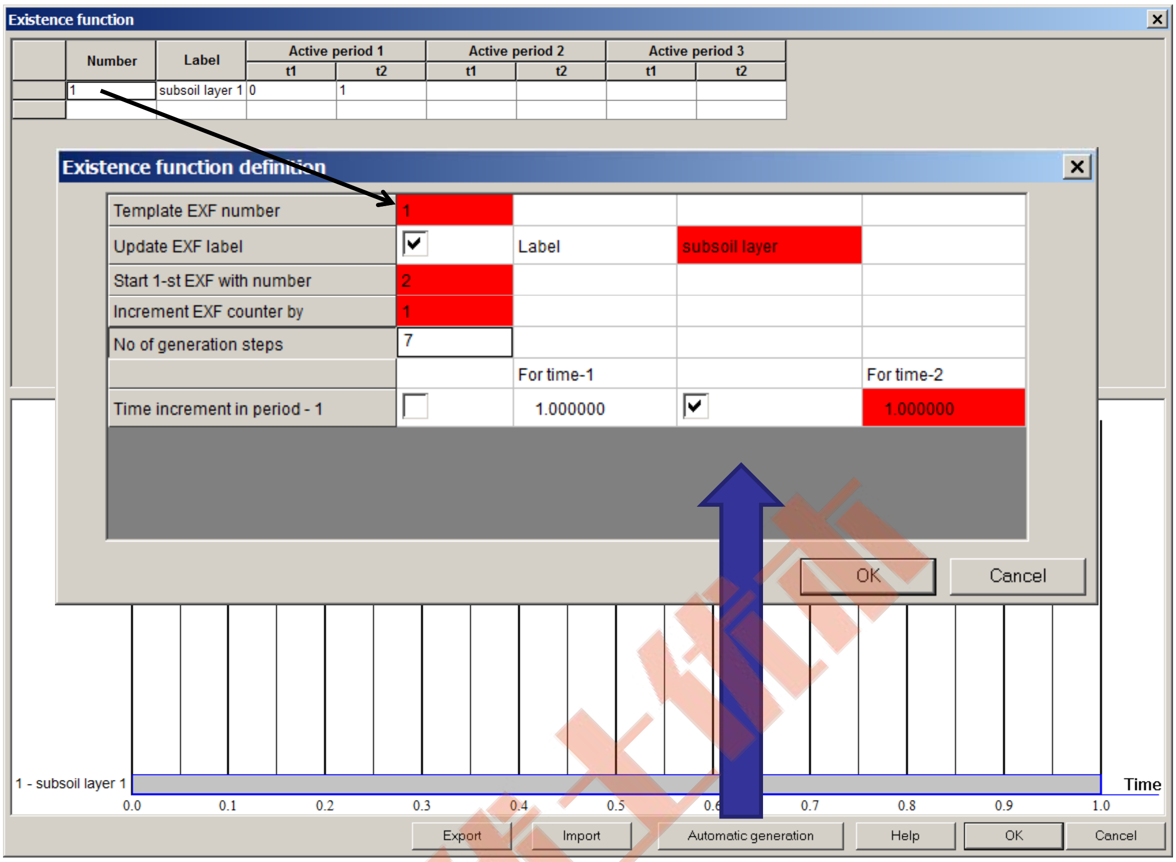
Step 2: Create list of materials



1. Add new material (subsoil) No 1 in the main menu through `Assembly/Materials` for `Continuum` and use `Mohr-Coulomb` constitutive model
2. Add new material (concrete facing) No 2 in the main menu through `Assembly/Materials` for `Beams`
3. Add new material (nail interface) No 3 in the main menu through `Assembly/Materials` for `Nail interface`
4. Add new material (steel nails) No 4 in the main menu through `Assembly/Materials` for `Beams`
5. Set properties for all materials according to the data specified in the main section for this tutorial

NB. If you want to learn more on how to edit material properties watch video
Edit material properties

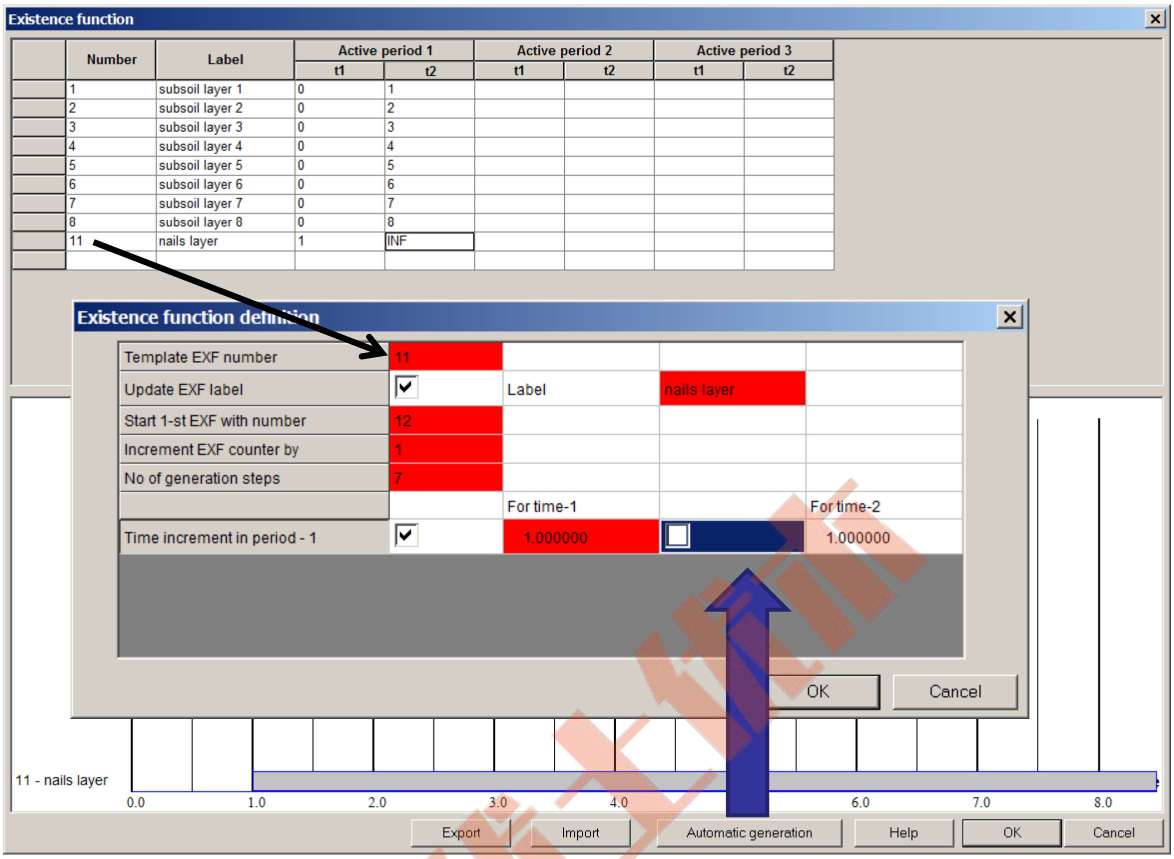
Step 3: Create list of existence functions for excavation of subsoil layers



1. Open dialog box for editing existence functions through `Assembly/Existence functions`
2. Add new function with index 1 and label it as *subsoil layer*
3. Define this function as active in the period $< 0..1 >$; this function will be associated, later on, with the first excavated subsoil layer
4. Press button **Automatic generation**; set ON the flag ☒ **Update existence function...**, set **Start existence f.. [ ]** to 1, set **Increment of existence f.. [ ]** to 1, set **Time increment [ ]** to 1 and **Number of steps [ ]** to 7; this way additional 7 existence functions will be generated

NB. If you want to learn more on how to edit existence functions watch video
Edit existence functions

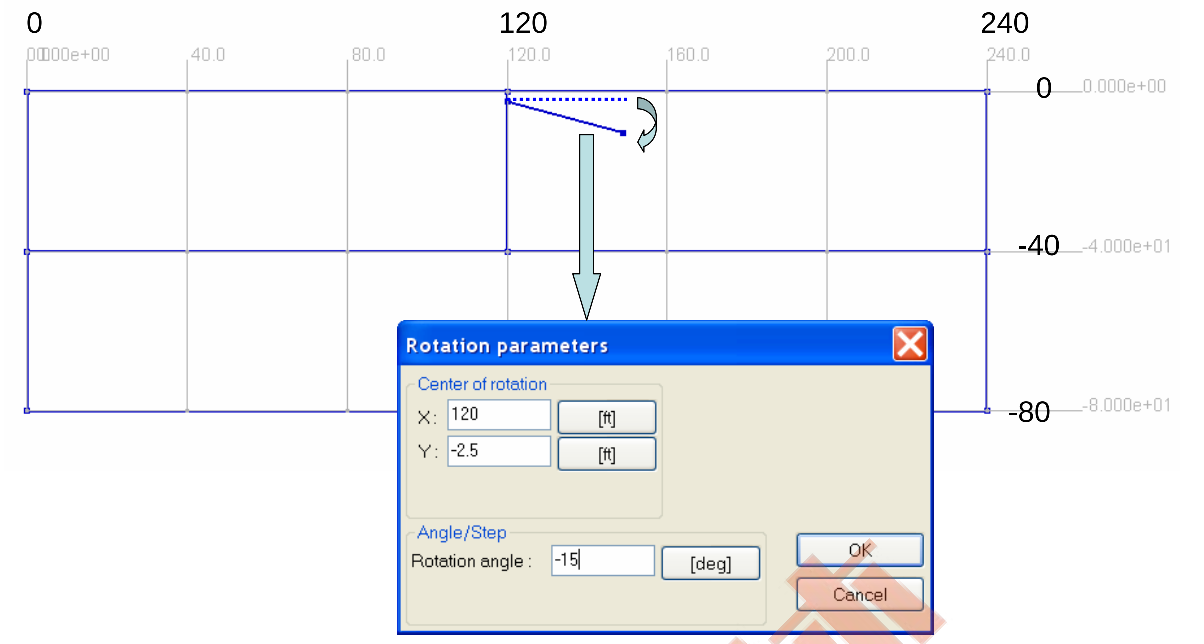
Step 4: Create list of existence functions for nails and facing segments



1. Open dialog box for editing existence functions through `Assembly/Existence functions`
2. Add new function with index 11 and label it as *nails layer*
3. Define this function as active in the period $< 1..\infty >$; this function will be associated, later on, with the first layer of nails
4. Press button **Automatic generation**; set ON the flag ☒ **Update existence function...**, set **Start existence f..** [ ] to 11, set **Increment of existence f..** [ ] to 1, set **Time increment** [ ] to 1 and **Number of steps** [ ] to 7; this way additional 7 existence functions will be generated

NB. If you want to learn more on how to edit existence functions watch video
Edit existence functions

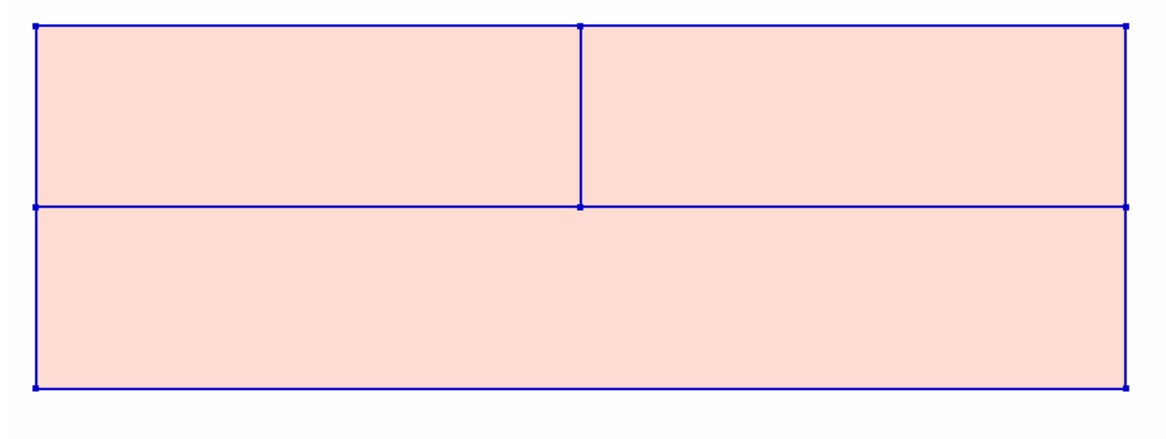
Step 5: CAD drawing



1. To speedup drawing of the model create construction lines using the toolbar, or top menu `Settings/Construction lines`; add characteristic X coordinates 0,120,240, and Y coordinates -80, -40 and 0
2. Draw a rectangular part (240 ft × 80 ft) of the domain using toolbar for line drawing
3. Draw line segment between 2 points (0,-40)-(240,-40) (accept intersection)
4. Draw line segment between 2 points (120,-40)-(120,0) (accept intersection)
5. Draw line segment between 2 points (120,-2.5)-(150,-2.5) (do not accept intersection)
6. Rotate last drawn line segment (`Macro model/Objects/Update/Rotate`) around the point (120,-2.5) by angle -15^o; this object will be used later on for generation of the first layer of nails

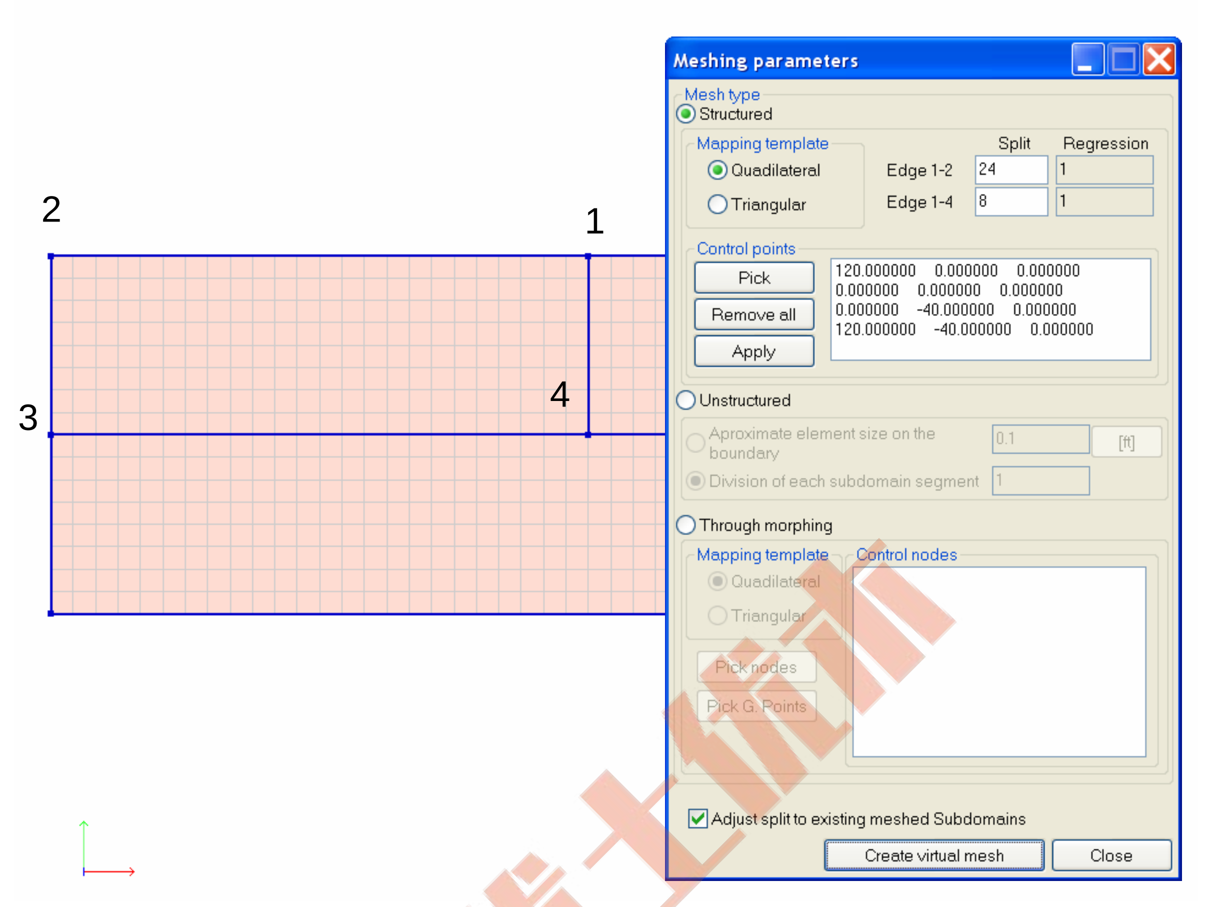
NB. Drawing of the domain contours can be made directly by mouse but also by setting coordinates of the endpoint in the dialog box; coordinates of the endpoint can be defined as absolute coordinates or as relative ones (in this case put @ character before X, Y, Z coordinates)

Step 6: Create subdomains (at macro-modeling level)



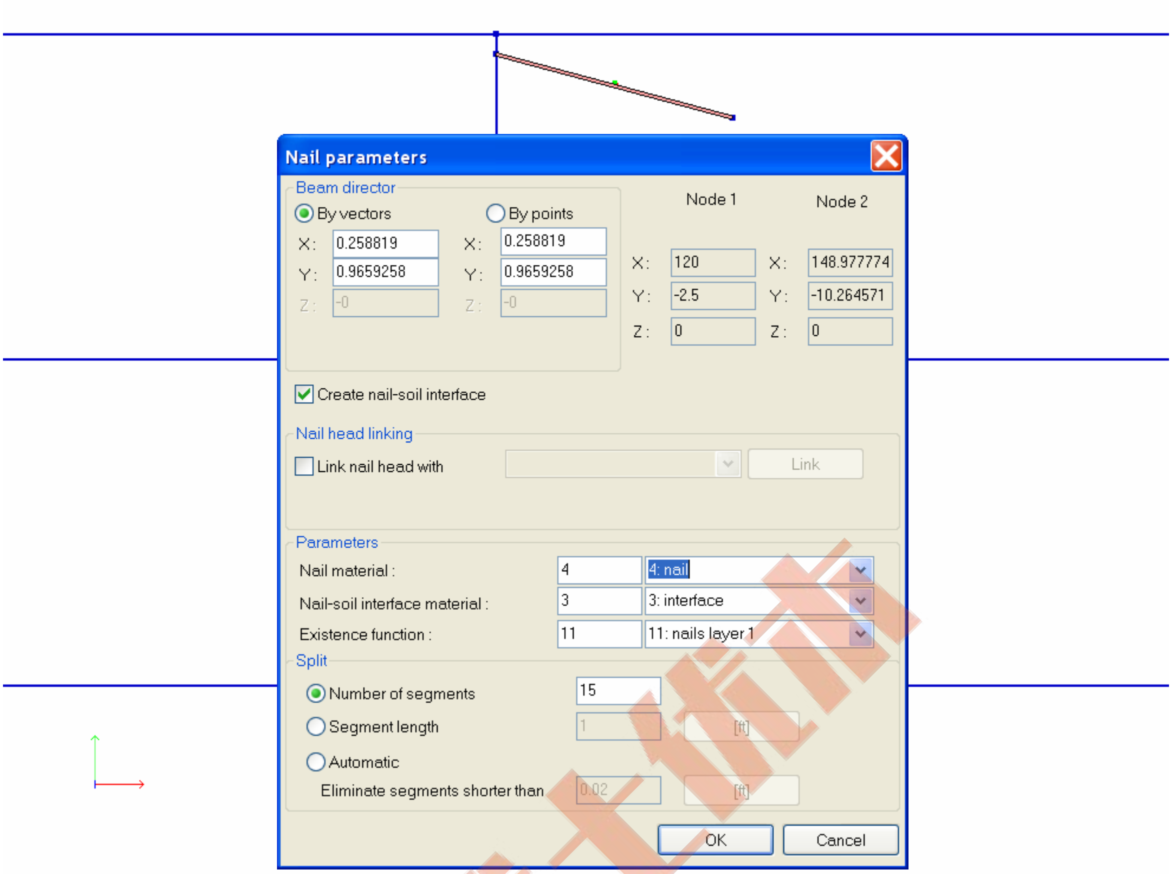
1. Create three 2D continuum subdomains through method `Macro model/Subdomain/2D continuum in contour` by picking the point within the closed contour
2. Apply proper material number (1) to the continuum subdomain using option `Macro model/Subdomain/Update/Parameters`
3. Create beam subdomain through method `Macro model/Subdomain/Beam on object(s)` by picking the vertical line segment
4. Apply proper material number (2) to the beam subdomain using option `Macro model/Subdomain/Update/Parameters`

Step 7: Create virtual meshes for subsoil and facing subdomains (at macro-modeling level)



1. Create virtual structured mesh in continuum subdomains; use option `Macro model/Subdomain/Mesh/Create virtual mesh` and click on the subdomain to activate the dialog box
2. Set number of elements along 1-2 and 1-4 edges; for top continuum subdomains use split 24×8 and for the bottom one 48×8
3. Create virtual structured mesh in beam subdomain; use option `Macro model/Subdomain/Mesh/Create virtual mesh` and click on the subdomain to activate the dialog box
4. Set number of elements for beam subdomain (if option ☒ **Adjust ..** is ON then setting split parameter is not needed as it will be inherited from the adjacent continuum subdomain)

Step 8: Create nail layers (at macro-modeling level)

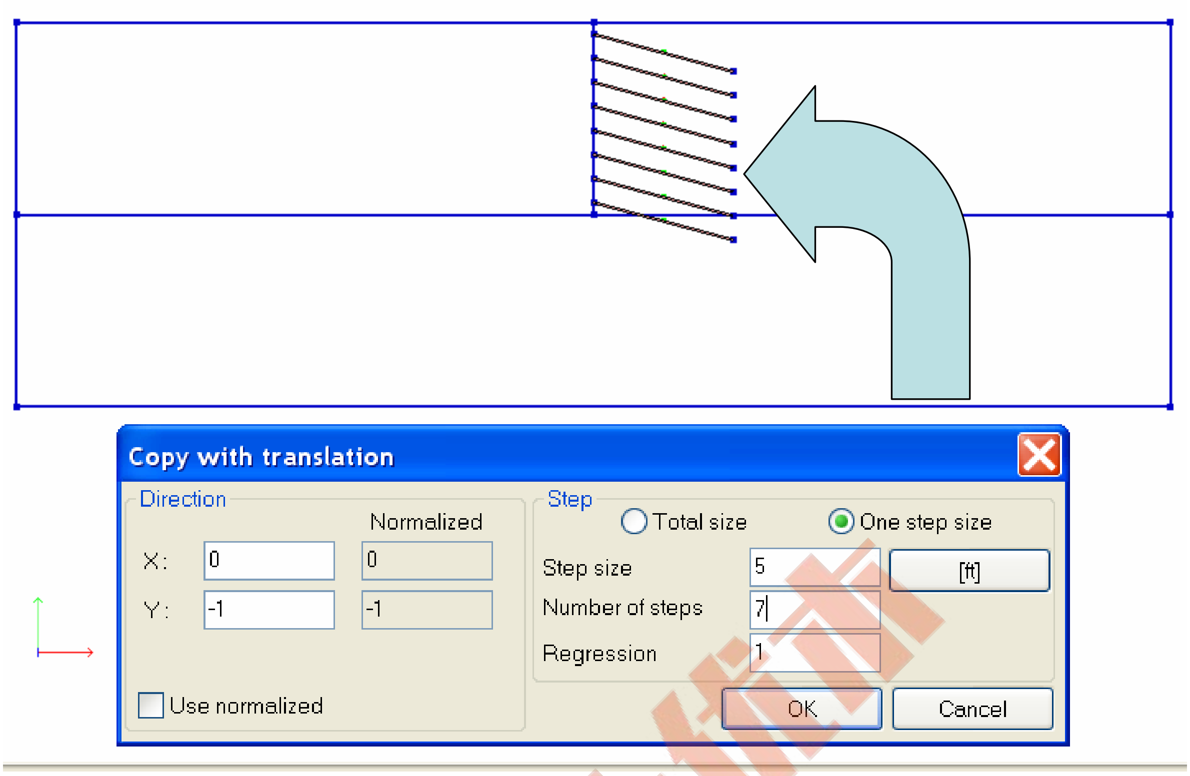


1. Create Nail on the picked graphical object using method `Macro model/Nail/Create/On object(s)` (one may select it first)
2. Edit properties of the nail by using method `Macro model/Nail/Update/Parameters`; in the dialog box set flag ☒ **Create nail-soil interface** to ON; apply material No 4 to the nail core and material No 3 to the nail interface; assume split value to 6 (to avoid force oscillations, mesh for the nail should be similar to the one used for subsoil)

Remarks

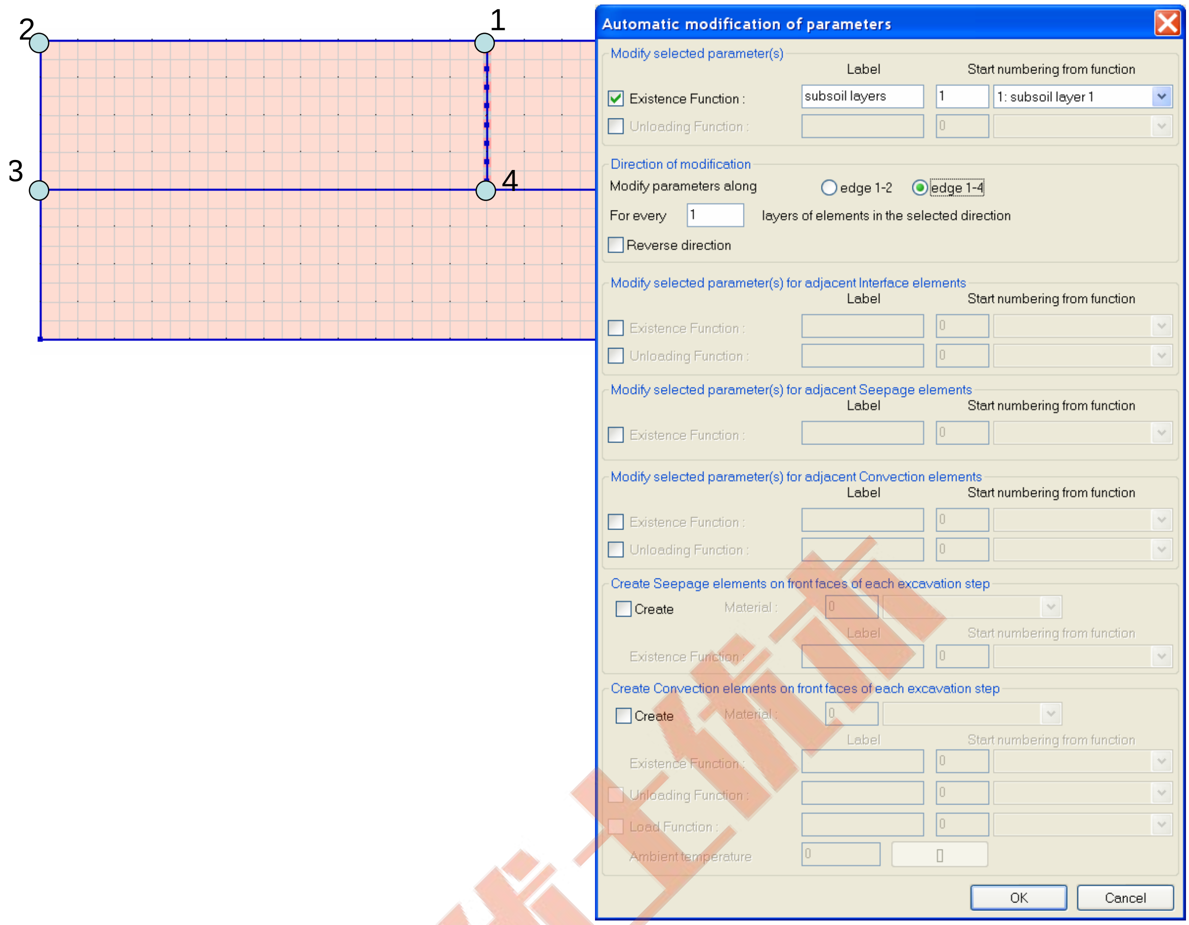
- **Linking of nail heads with the facing has to be made at the FE model level, once the real mesh is generated**
- Distance between nails in z direction is setup at the material data (material No 4) for nails (beams) in the dit field **Interval between beams** [ ]

Step 9: Copy nail layers (at macro-modeling level)



1. Select created nail and copy it downwards via `Macro model/Nail/Create/Copy with translation` seven times with step 5 ft (do not accept the intersection with the existing graphical objects)
2. Apply proper existence function to each nail layer one by one (use option `Macro model/Nail/Update/Parameters`)

Step 10: Set existence functions for excavated subsoil layers (at macro level))



1. Run method `Macro model/Subdomain/Update/Define excavation front` to set up existence functions for subsequent layers to be excavated

2. In the dialog box for the excavation front activate flag **Existence function [ ]**, set the label *subsoil layers*, select first defined existence function (No 1) that will be applied to first row of excavated elements from the top, activate option ◎ **Edge 1-4** that indicates the direction of excavation front propagation and set value 1 to the edit field **For every .... layers of elements... [ ]**

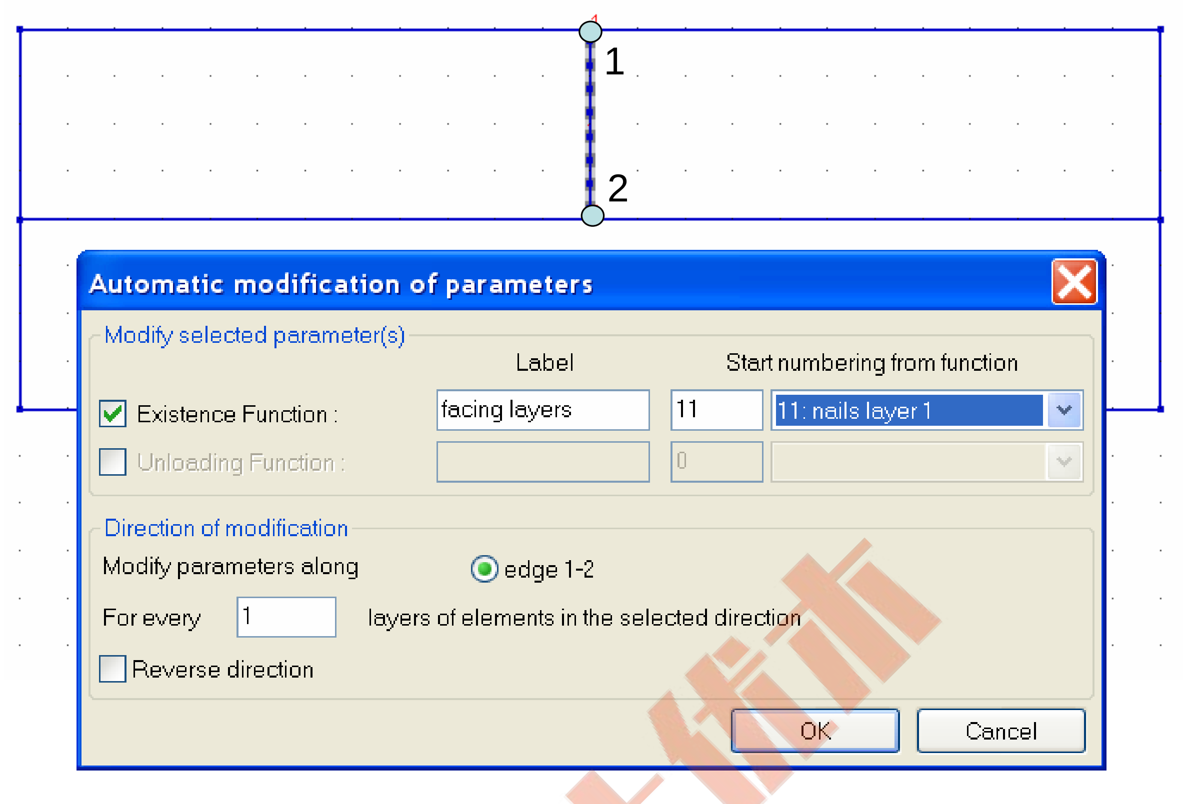
3. The above setting will enforce application of existence function No 1 to the first top row of elements in the real mesh, No 2 to the second one etc...

Remarks

- Existence functions for each excavated subsoil layer can be set up at the `FE model` level once the real mesh is generated

- Existence functions for each excavated subsoil layer could be also set up at the `Macro model` level but in this case we would need to create 8 subdomains in the considered zone instead of 1

Step 11: Set existence functions for constructed facing layers (at macro level))

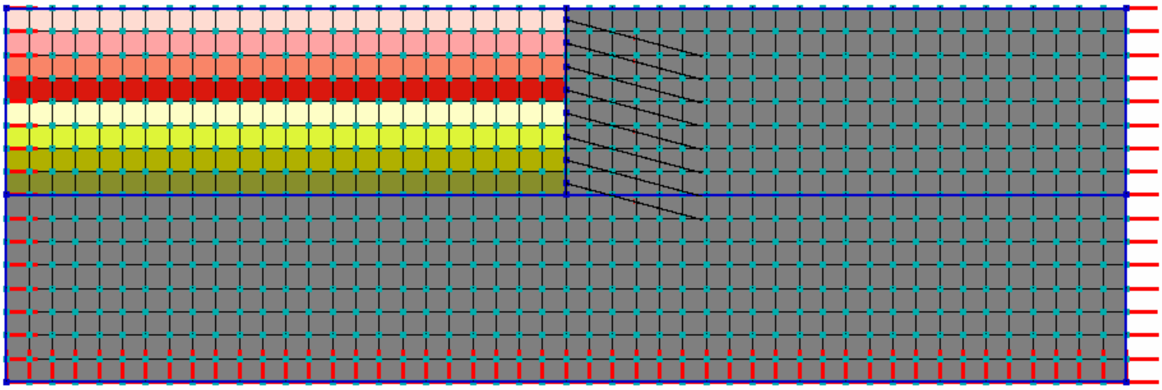


1. Run method `Macro model/Subdomain/Update/Define excavation front` to set up existence functions for subsequent facing layers that are to be constructed
2. In the dialog box for the excavation front activate flag **Existence function** [ ], set the label *nails layers*, select first defined existence function for nails (No 11) that will be applied to first row of facing elements from the top, activate option and set value 1 to the edit field **For every .... layers of elements...** [ ]
3. The above setting will enforce application of existence function No 11 to the first top row of facing elements (beams) in the real mesh, No 12 to the second one etc...

Remarks

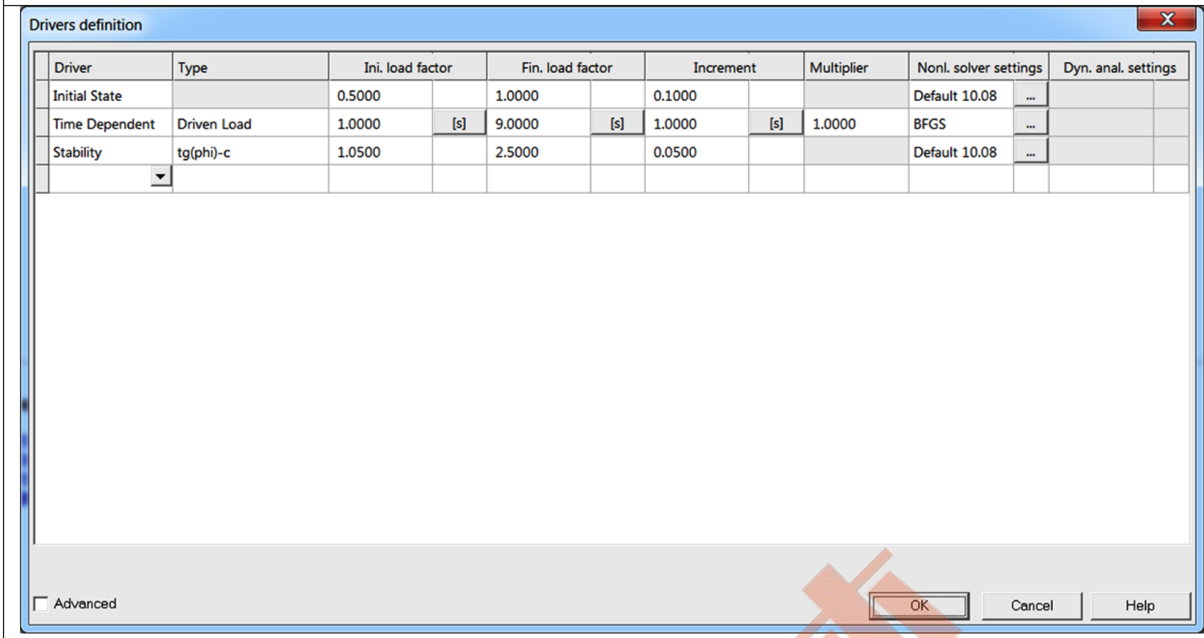
- Existence functions for each constructed facing layer can also be set up at the `FE model` level once the real mesh is generated
- Existence functions for each constructed facing layer could also be set up at the `Macro model` level but in this case we would need to create 8 subdomains in the considered zone instead of 1

Step 16: Create real mesh and set solid BC



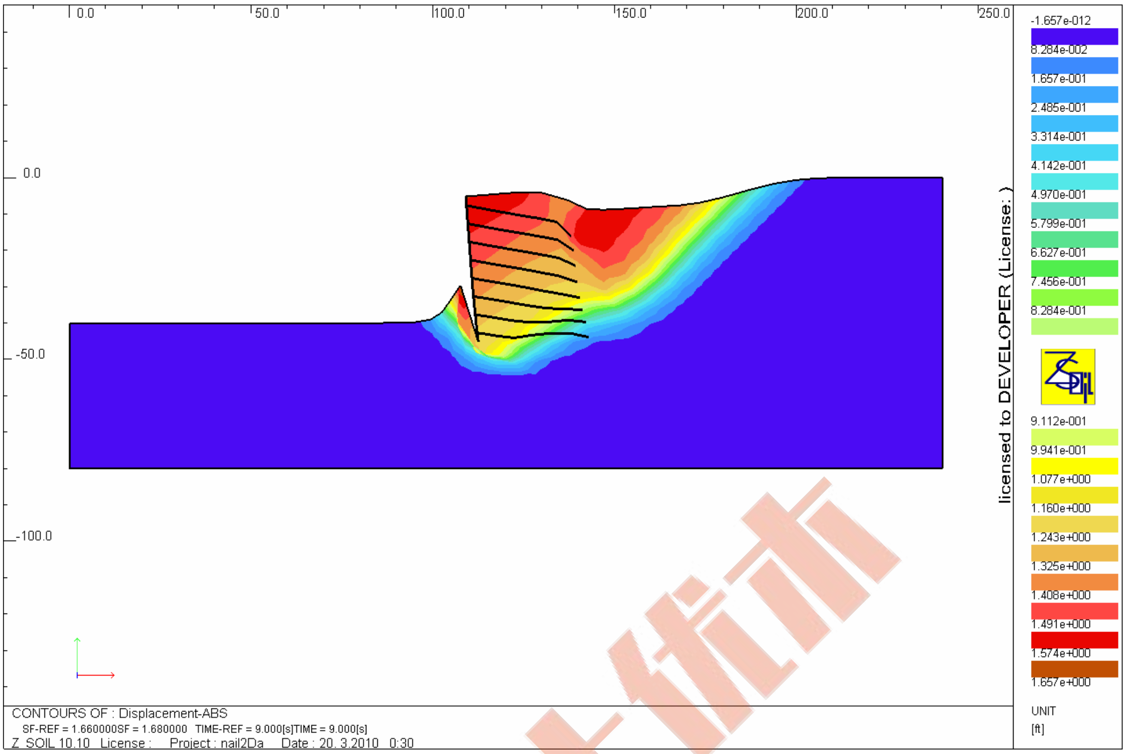
1. Select all subdomains (Ctrl-A from keyboard or through menu `Macro model/Subdomain/Outline/all` and convert virtual meshes to the real ones through `Macro model/Subdomain/Mesh/Virtual→Real mesh`
2. Switch to `FE model` menu
3. Generate standard box-type solid BC through `FE model/Boundary conditions/Solid BC/Create/On box`

Step 17: Setup drivers



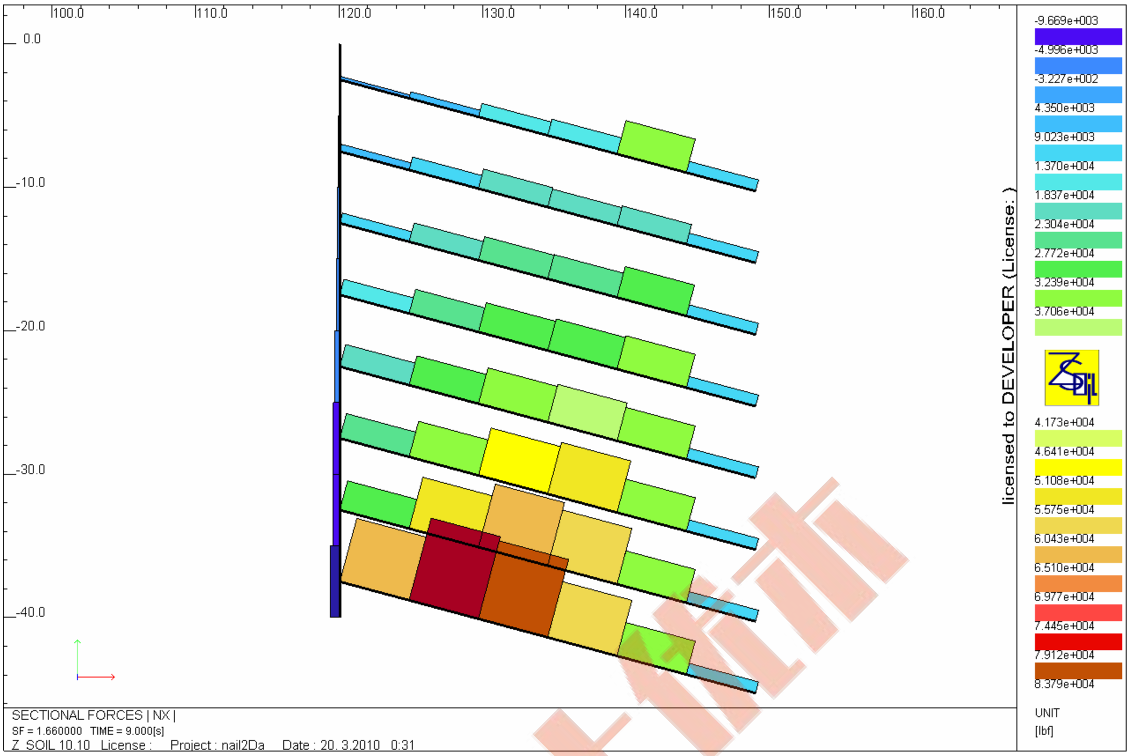
1. Add `Initial state` driver to generate the in situ stress state
2. Add `Time dependent/Driven load` driver from time 1..9 s to perform stage construction algorithm
3. Add `Stability`/$\tan(\phi) - c$ driver with SF_o=1.05...2.50 with step 0.05

Step 18: Results (failure mode)



1. Open postprocessor and set the reference time step (t =9, SF =1.66) (use option `Graph. option/Time/Select reference time step` to perform this operation) to visualize incemental deformation that helps to identify the failure mode
2. Use option `Maps` to plot color contours of incremental displacement amplitudes
3. Use keys $PgUp$ or $PgDn$ to increase or decrease the scale for deformed configuration

Step 19: Results (forces in nails)



1. Open postprocessor
2. Use option `MNT for beams/....` to plot force diagrams in nails
3. Use option `Settings` to switch to ploting of axial forces instead of bending moment
4. Use keys $PgUp$ or $PgDn$ to increase or decrease the scale for diagrams

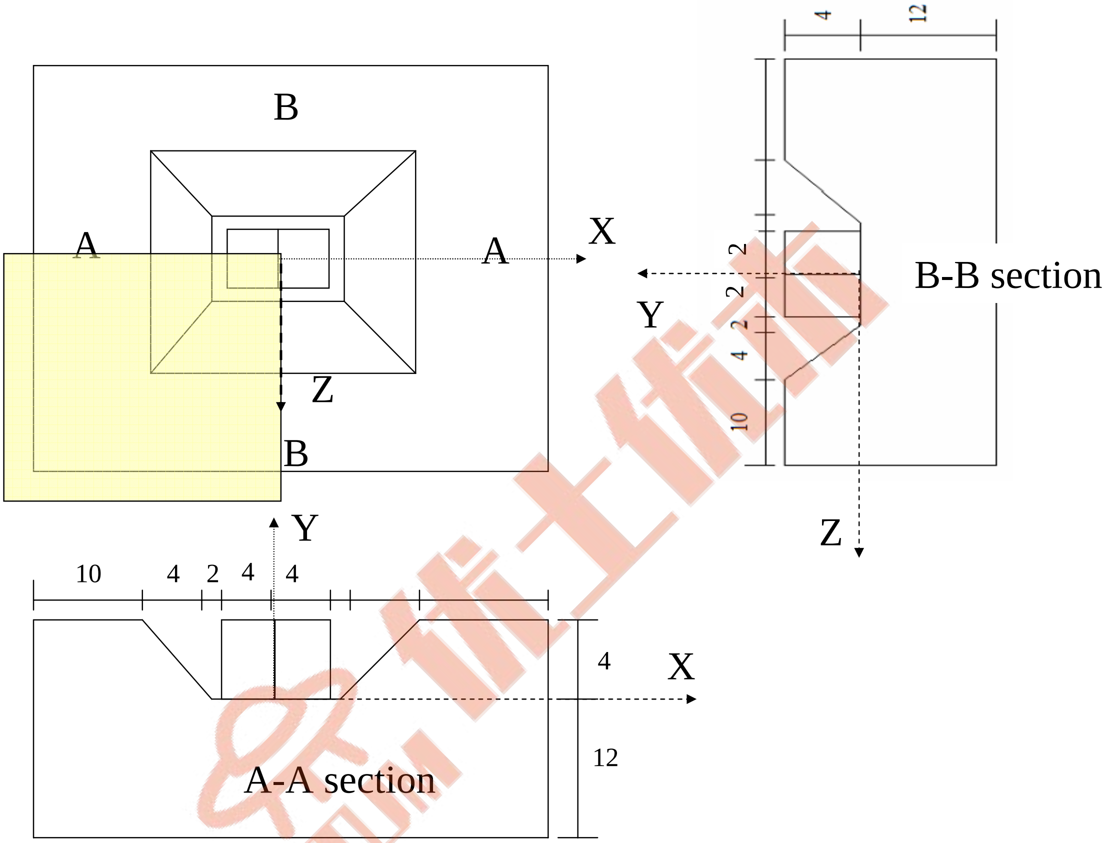
Chapter 2

3D PROBLEMS

2.1 CONCRETE BOX CONTAINER

- **Data file: tutorials/box-container.INP**
- **Description**

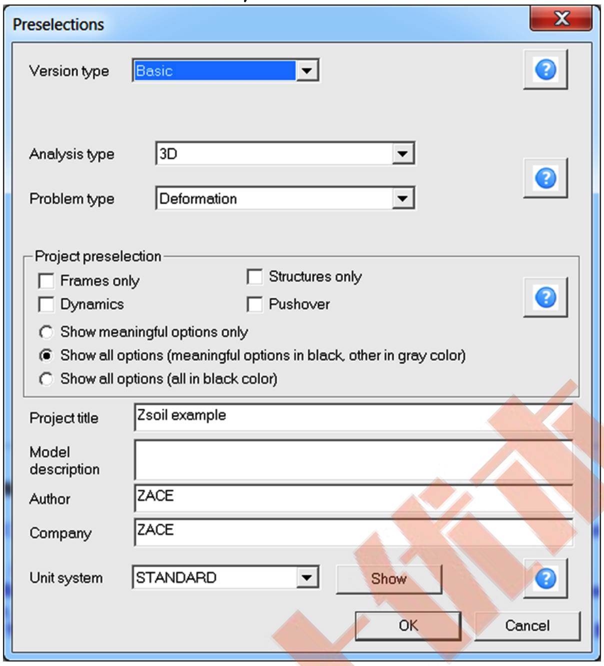
  This example concerns modeling of an excavation followed then by a construction of a reinforced concrete shell container. Geometry of the model is shown in the following figure. Due to the symmetry with respect to the X-Y plane only half of the model will be analyzed.



After the excavation a two chamber container is built, trial water pressure loading, induced first in the left and then in the right chamber, is applied, and finally space between the external container walls and excavation is filled by the soil.

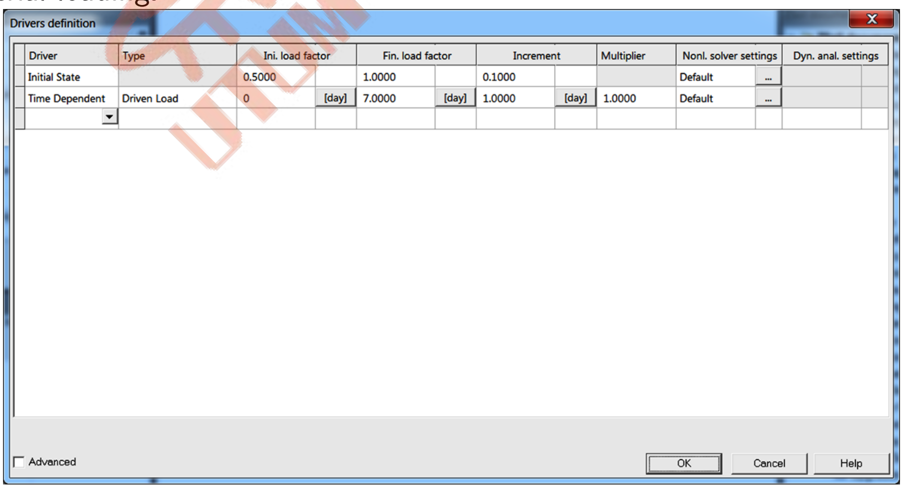
- **Project preselection**

  In the dialog box for project preselection (it appears automatically for option `File/New` in main Z_Soil menu) set ◎ **3D** to ON and select `Deformation` item from the `Problem type` list. The predefined system of units for both data preparation and visualization of results can be verified in menu `Control/Units`.



- **Drivers**

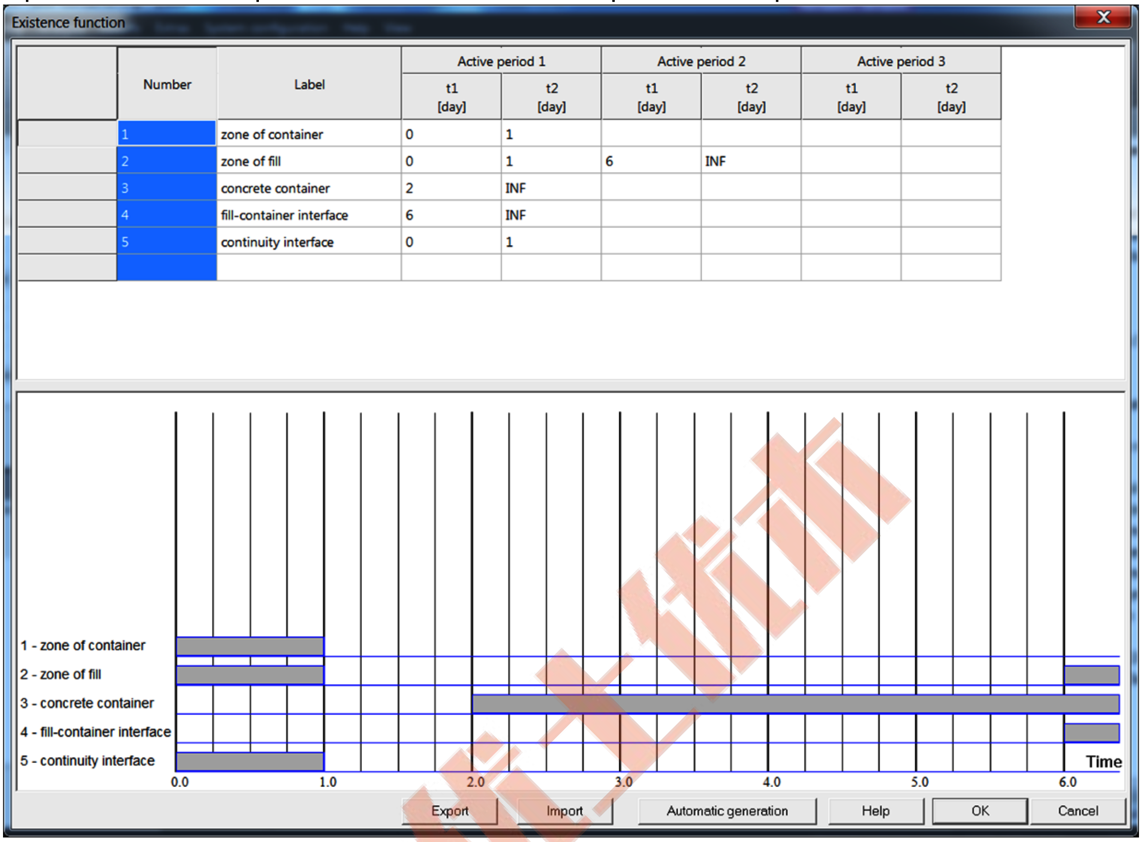
  The whole computational process will consist of the two drivers i.e. the `Initial state` and then `Time dependent/Driven load`. The fist driver will yield the in situ stress state at time $t = 0$ while the second one allows to trace all excavation/construction steps including trial loading.



To learn on how to set up the drivers list watch the video Set drivers

- **Existence functions**

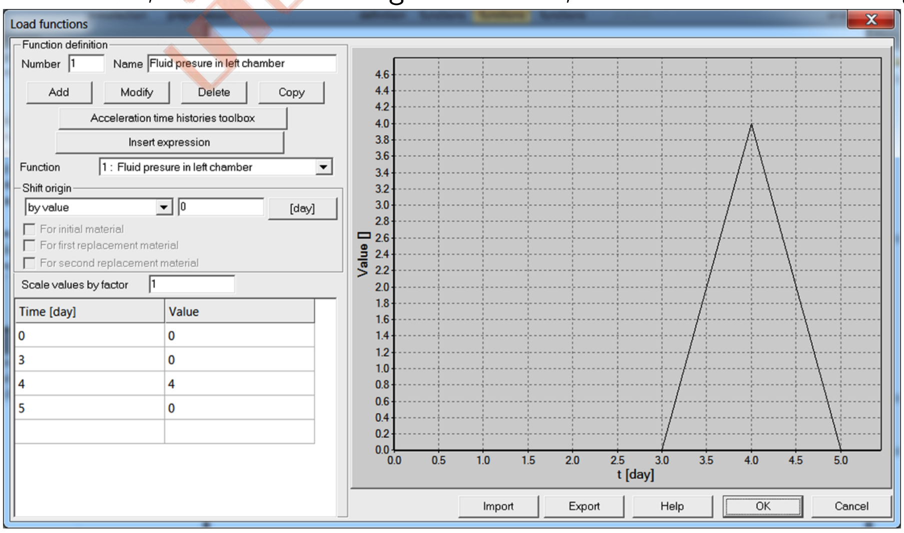
The sequence of excavation/construction events is controlled by the existence functions shown in the figure below. All these existence functions are defined according to the sequence of events specified earlier in the description of the problem.



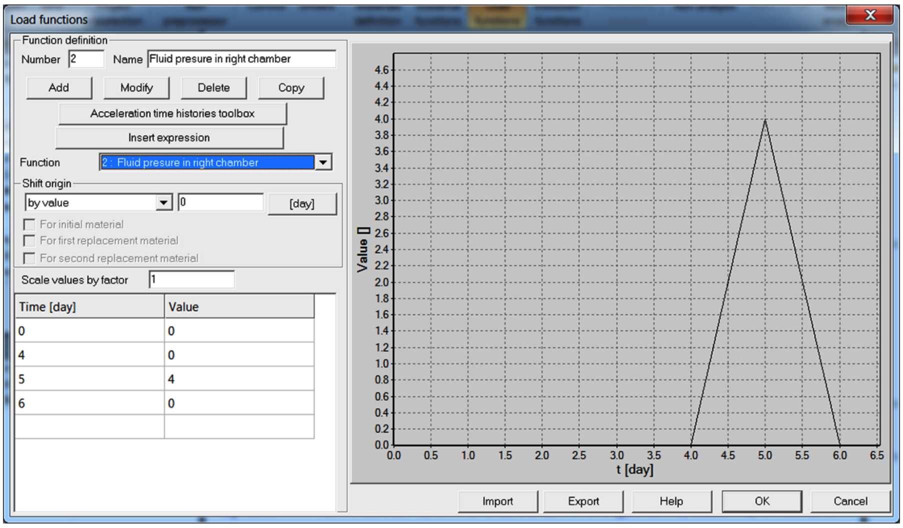
To learn on how to enter existence functions watch the video Edit existence functions .

- **Load time function**

The load time functions associated with the trial loading (defined as a pressure via fluid head), first in the left, and then in the right chambers, are shown in the following figures.



$LTF_1(t)$ Evolution of the pressure head in the left chamber



$LTF_2(t)$ Evolution of the pressure head in the right chamber

To learn on how to edit load time functions watch the video Edit load time function .

- **Materials**

Material properties for subsoil, concrete container, fill and contact interface are summarized in the following table:

Material	Model	Data group	Properties	Unit	Value
1 Concrete	Elastic shell	Elastic	E	[kN/m^2]	30000000
			ν	–	0.2
		Density	γ	[kN/m^3]	24
2 Subsoil	Mohr-Coulomb	Elastic	E	[kN/m^2]	60000
			ν	–	0.32
		Density	γ_D	[kN/m^3]	20
			γ^F	[kN/m^3]	10
			e_o	–	0.0
		Non-linear	ϕ	[°]	20
			ψ	[°]	0
			C	[kN/m^2]	15
		Initial state K_o	K_{ox}	–	0.7
			K_{oz}	–	0.7
3 Fill	Mohr-Coulomb	Elastic	E	[kN/m^2]	60000
			ν	–	0.32
		Density	γ_D	[kN/m^3]	20
			γ^F	[kN/m^3]	10
			e_o	–	0.0
		Non-linear	ϕ	[°]	20
			ψ	[°]	0
			C	[kN/m^2]	15
4 Interface	Contact	Non-linear	ϕ	[°]	5
			ψ	[°]	0
			C	[kN/m^2]	0.0

NB. Thickness of the concrete container is assumed to be 0.2m and is defined in the preprocessor.

- **Modelling process of filling**

  In this example a simplest model of filling is applied. This process is modeled here in single time step. In the real applications it is recommended to do it in few steps and to apply simultaneously initial stresses in newly built layers. This way we can model any technology of filling. To avoid generation of several fill layers in macro-model one can make it at the FE-model level.

- **Generation of the model**

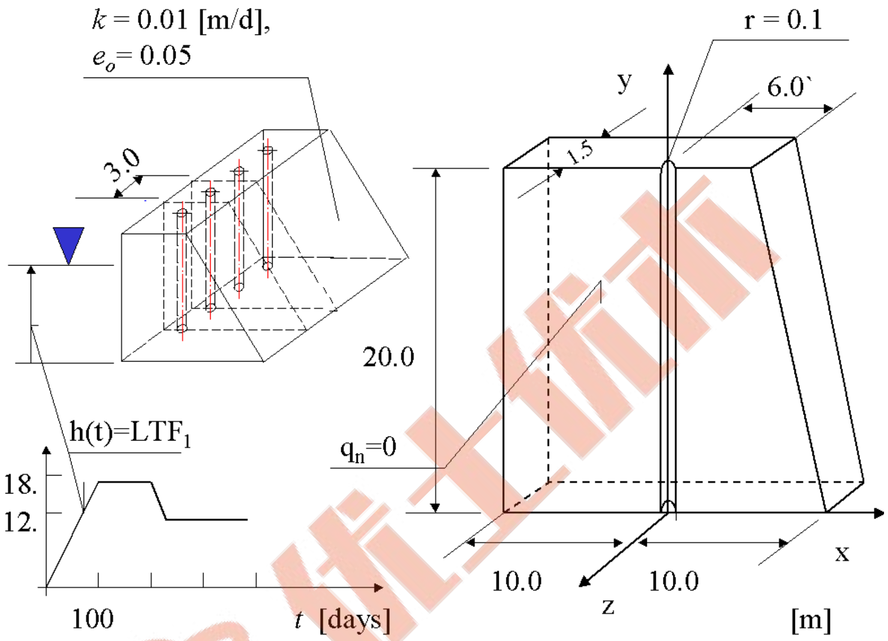
  The computational model is built in the following steps and some of them are documented in form of video films.

1	Create a new project under name : box-container.inp
2	Edit existence functions
3	Edit load time function
4	Edit materials
5	Edit drivers
6	Create macro-model
7	Create virtual meshes
8	Create loads
9	Create real mesh and boundary conditions
10	Run computation
11	Visualize results

2.2 DRAINING CONCRETE DAM

- **Data file: tutorials/drain.INP**
- **Description**

A 3D transient flow analysis in a concrete dam is considered in this tutorial. The dam is drained by equally spaced vertical holes ($r = 0.1m$). Due to the periodicity of the system the computational model is built for a single representative segment with thickness equal to one half of the distance between the drains. The upstream water elevation varies according to the assumed load time function. The engineering draft is given in the following figure.

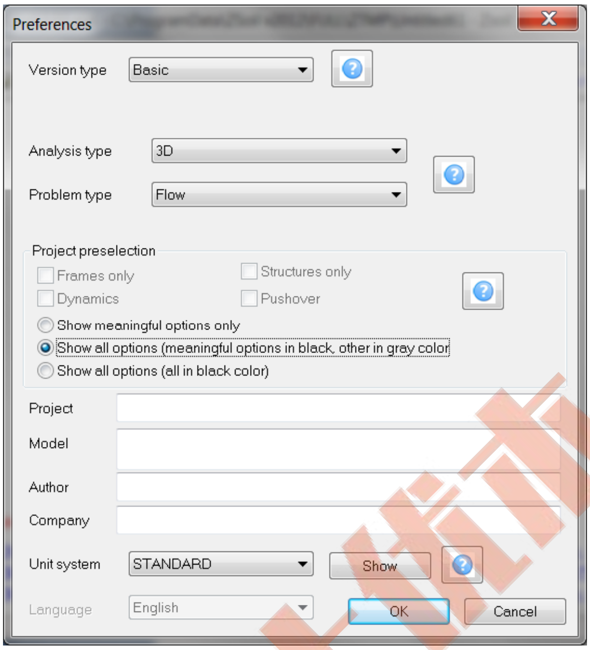


The following results are sought:

- ⋆ pressure field (free surface evolution, in particular)
- ⋆ velocity field
- ⋆ time histories of the outflow through the drainage and downstream face to assess the efficiency of the system

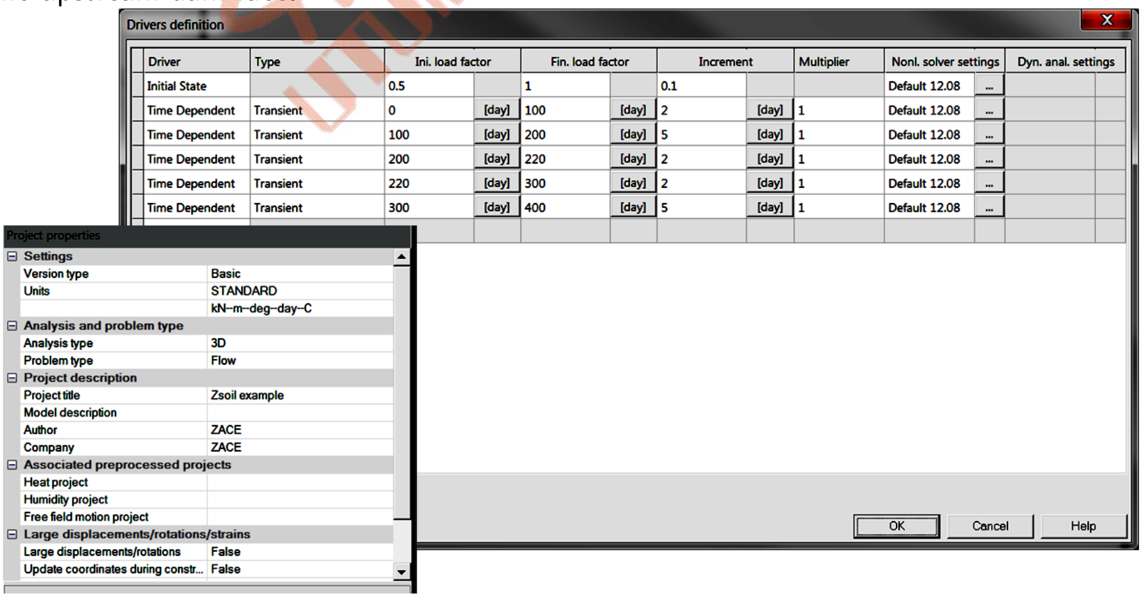
• Project preselection

In the dialog box for project preselection (it appears automatically for option `File/New` in main Z_Soil menu) set ⊙ **3D** to ON and select `Flow` item from the `Problem type` list. The predefined system of units for both data preparation and visualization of results can be verified in menu `Control/Units`.



• Drivers

The whole computational process will consist of the two drivers i.e. the `Initial state` and then `Time dependent/Transient`. The fist driver will yield the initial condition for the pore pressure at time $t = 0$ (the `Initial state` for `Flow` problem is equivalent to the steady state solution at time $t = 0$). The second driver is needed to trace the evolution of the pressure and velocity fields due to variable pressure boundary conditions applied to the upstream dam face.



To learn on how to set up the drivers list watch the video Set drivers

- ## Materials

Material properties for concrete are given in the following table:

Material	Model	Data group	Properties	Unit	Value
1 Concrete	Elastic	Density	γ_D	[kN/m^3]	0
			γ^F	[kN/m^3]	10
			e_o	–	0.05
		Flow	β^F	[kN/m^3]	10^{38}
			$k_x^{'}$	[m/d]	10^{-2}
			$k_y^{'}$	[m/d]	10^{-2}
			$k_z^{'}$	[m/d]	10^{-2}
			α	[m^{-1}]	0.5
			S_r	–	0.2

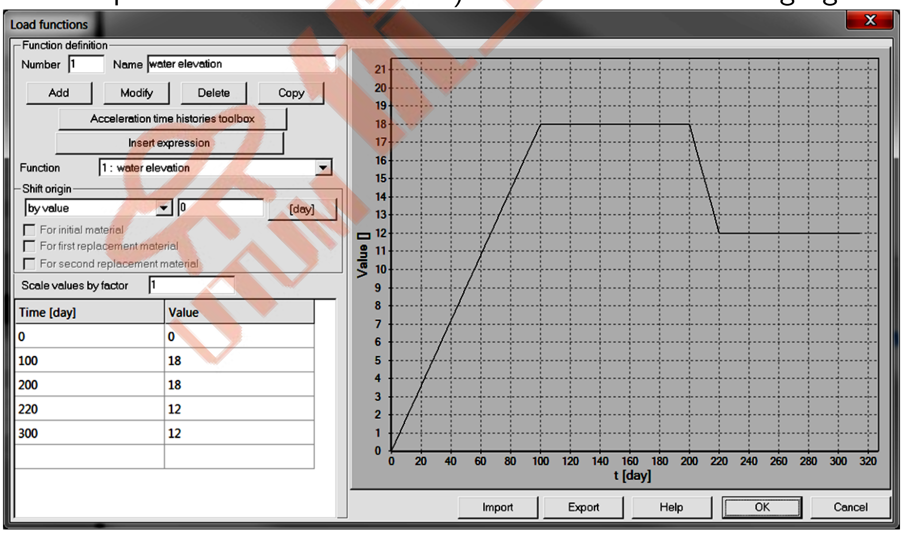
In the considered case any solid material model can be used and the only meaningful parameters to be set up are placed in groups `Density` and `Flow`. Initial void ratio e_o and fluid bulk modulus β^F are meaningful parameters for transient analysis. The fictitious material (2) applied to seepage surface elements is not specified here (the multiplier for, automatically estimated, penalty parameter is equal to the default value 1.0).

To edit material properties use menu `Assembly/Materials`.

To learn on how to enter material data watch the video Edit materials .

- ## Load time functions

The load time function associated with the water elevation on the upstream face of the dam (defined as a pressure BC via fluid head) is shown in the following figure.



To learn on how to edit load time functions watch the video Edit load time function .

- **Generation of the model**

  The computational model is built in the following steps and some of them are documented in form of video films.

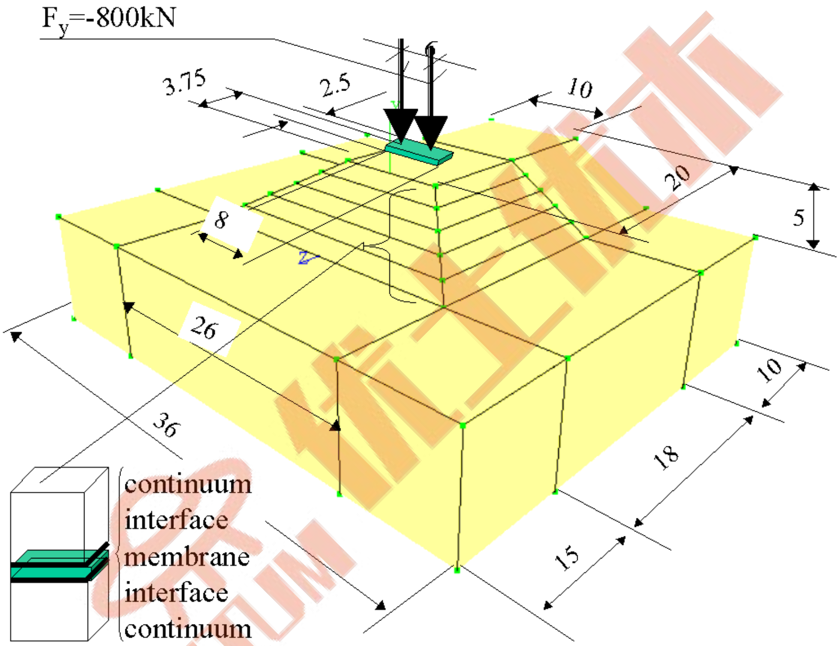
  1 Create a new project under name : box-container.inp
  2 Edit load time function
  3 Edit materials
  4 Create construction axes
  5 Create macro-model
  6 Create boundary conditions
  7 Create virtual/real meshes
  8 Run computation
  9 Visualize results

2.3 REINFORCED SOIL ABUTMENT

- **Data file: tutorials/rfslope.INP**
- **Description**

Safety analysis of a geotextile reinforced abutment is the aim of this tutorial. Five geotextile layers are modeled with aid of membrane elements using an anisotropic membrane model. The interface between geotextile and soil is modeled with aid of interface contact elements governed by the Colulomb's friction law. The abutment is loaded by the body loads and two concentrated forces applied to the concrete footing located at the crest. In this tutorial we will show on how to generate model evolving in time to take the effect of construction stage into account. The abutment will be constructed in six steps. Five steps are needed to build the abutment and in the sixth one the concrete slab will be added.

The engineering draft of the problem is given in the figure below.



The following aspects are of the interest in this example:

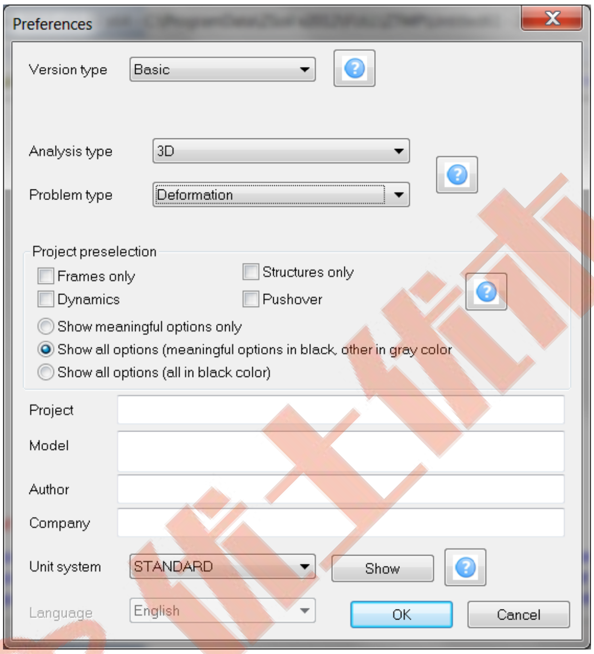
- ⋆ Deformation of the whole structure under strip load
- ⋆ Stress state in geo-grid membranes and interfaces
- ⋆ Global safety factor of the structure

Remark:

Due to the symmetry of the problem only half of the model will be generated.

- **Project preselection**

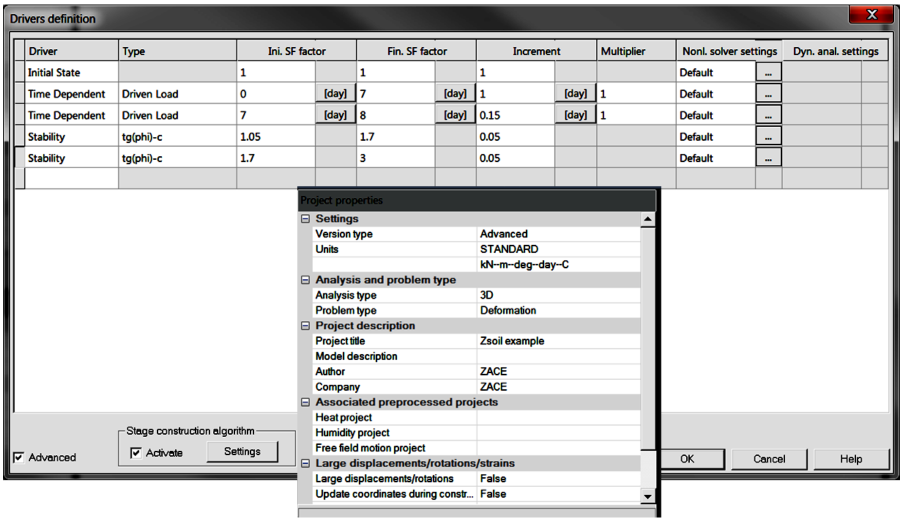
In the dialog box for project preselection (it appears automatically for option `File/New` in main Z_Soil menu) set ⊙ **3D** to ON and select `Deformation` item from the `Problem type` list. The predefined system of units for both data preparation and visualization of results can be verified in menu `Control/Units`.



- **Drivers**

The whole computational process will consist of the two drivers i.e. the `Initial state` and then `Time dependent/Driven load`. The fist driver will yield the in situ stress state at time $t = 0$ while the second one allows to trace all excavation/construction steps including trial loading.

A special incremental procedure to handle construction stage analysis is enabled by setting the option ⊠ **Stage construction algorithm** to ON. All construction steps will be solved in 3 increments corresponding to the 50%, 75% and 100% of the dead weight load of a new layer. In addition each newly added layer in the first time step (through all increments) behaves as incompressible material (by an internal setting $\nu = 0.49999$). In the next time step an original value of the Poisson's ratio is used. This incremental treatment of construction stage analysis can be setup by clicking the button **Settings** next to the ⊠ **Stage construction algorithm** checkbox. The corresponding settings are shown in the following figure.

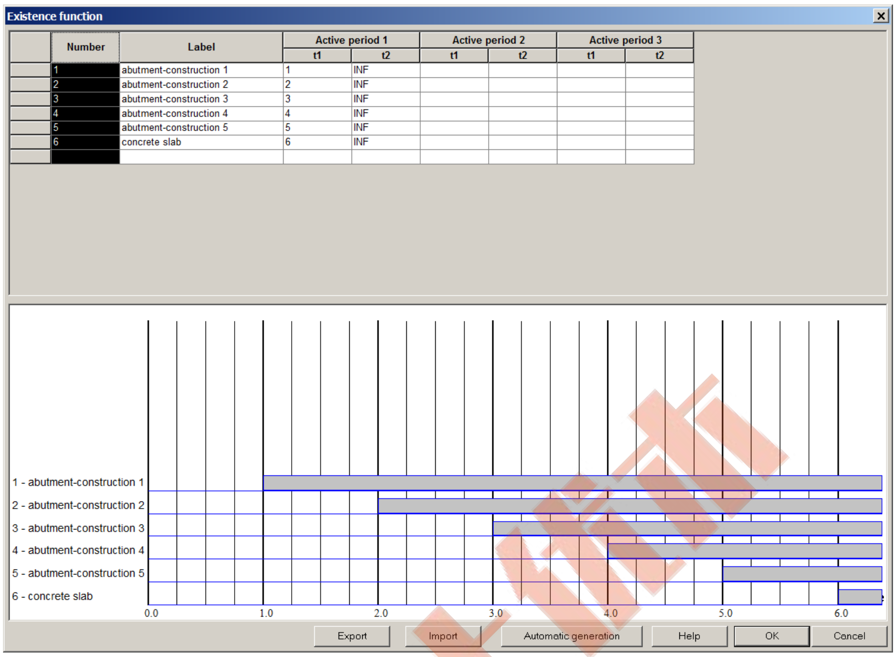


The external load defined at the FE element model level as a pair of the two external nodal forces is applied in 7 steps as indicated in the definition of the third driver.

To learn on how to set up the drivers list watch the video Set drivers

- ## Existence functions

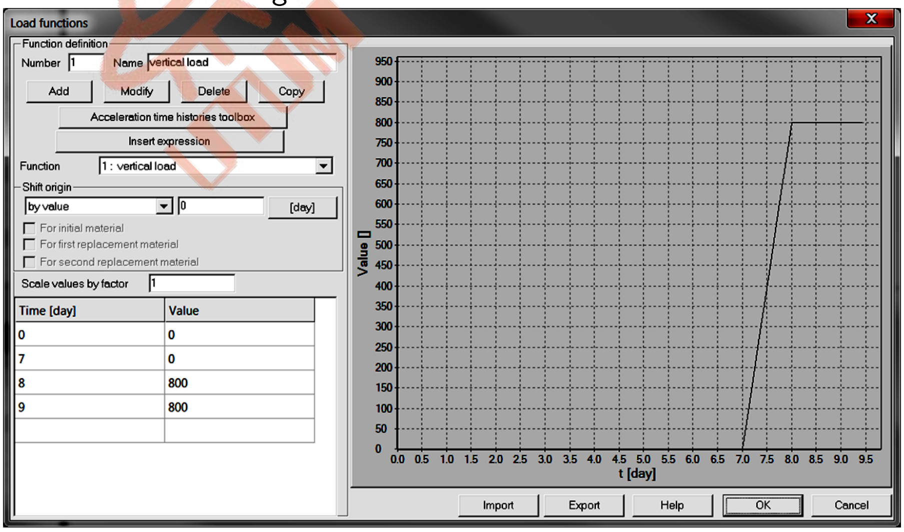
The sequence of excavation/construction events is controlled by the existence functions shown in the figure below. All these existence functions are defined according to the sequence of events specified earlier in the description of the problem. In this tutorial we will show on how to generate existence functions in an automatic manner.



To learn on how to enter existence functions in an automatic manner watch the video Edit existence functions .

- ## Load time function

The load time function associated with the vertical concentrated forces applied to the concrete slab is shown in the figure.



To learn on how to edit load time functions watch the video Edit load time function .

- **Materials**

Material properties for subsoil, soil used to construct the abutment, geotextiles, contact interface and concrete slab are given in the following table:

Material	Model	Data group	Properties	Unit	Value
1 Subsoil	Mohr-Coulomb	Elastic	E	[kN/m^2]	80000
			ν	–	0.32
		Density	γ_D	[kN/m^3]	19
			γ^F	[kN/m^3]	10
			e_o	–	0.0
		Non-linear	ϕ	[°]	20
			ψ	[°]	0
			C	[kN/m^2]	30
		Initial K_0 state	K'_{ox}	–	0.6
			K'_{oz}	–	0.6
2 Geotextile	Anisotropic membrane	Elastic	K_{xx}	[kN/m]	600
			K_{yy}	–	200
			K_{xy}	–	0
		Density	γ	[kN/m^3]	0.0
		Geometry	Direction	—	0/0/1
		Non-linear	f_{tx}	[kN/m]	15.0
			f_{ty}	[kN/m]	5.0
			f_{cx}	[kN/m]	0.3
			f_{cy}	[kN/m]	0.1
3 Interface	Contact	Non-linear	ϕ	[°]	20
			ψ	[°]	0
			C	[kN/m^2]	10.0
4 Soil abut.	Mohr-Coulomb	Elastic	E	[kN/m^2]	110000
			ν	–	0.32
		Density	γ_D	[kN/m^3]	18.5
			γ^F	[kN/m^3]	10
			e_o	–	0.0
		Non-linear	ϕ	[°]	30
			ψ	[°]	0
			C	[kN/m^2]	15
4 Concrete footing	Elastic	Elastic	E	[kN/m^2]	30000000
			ν	–	0.2
		Density	γ_D	[kN/m^3]	24.0
			γ^F	[kN/m^3]	10
			e_o	–	0.0

In the considered case initial void ratio and fluid specific weight do not play any role.

To edit material properties use menu `Assembly/Materials`.

To learn on how to enter material data watch the video Edit materials .

• Generation of the model

The computational model is built in the following steps and some of them are documented in form of video films.

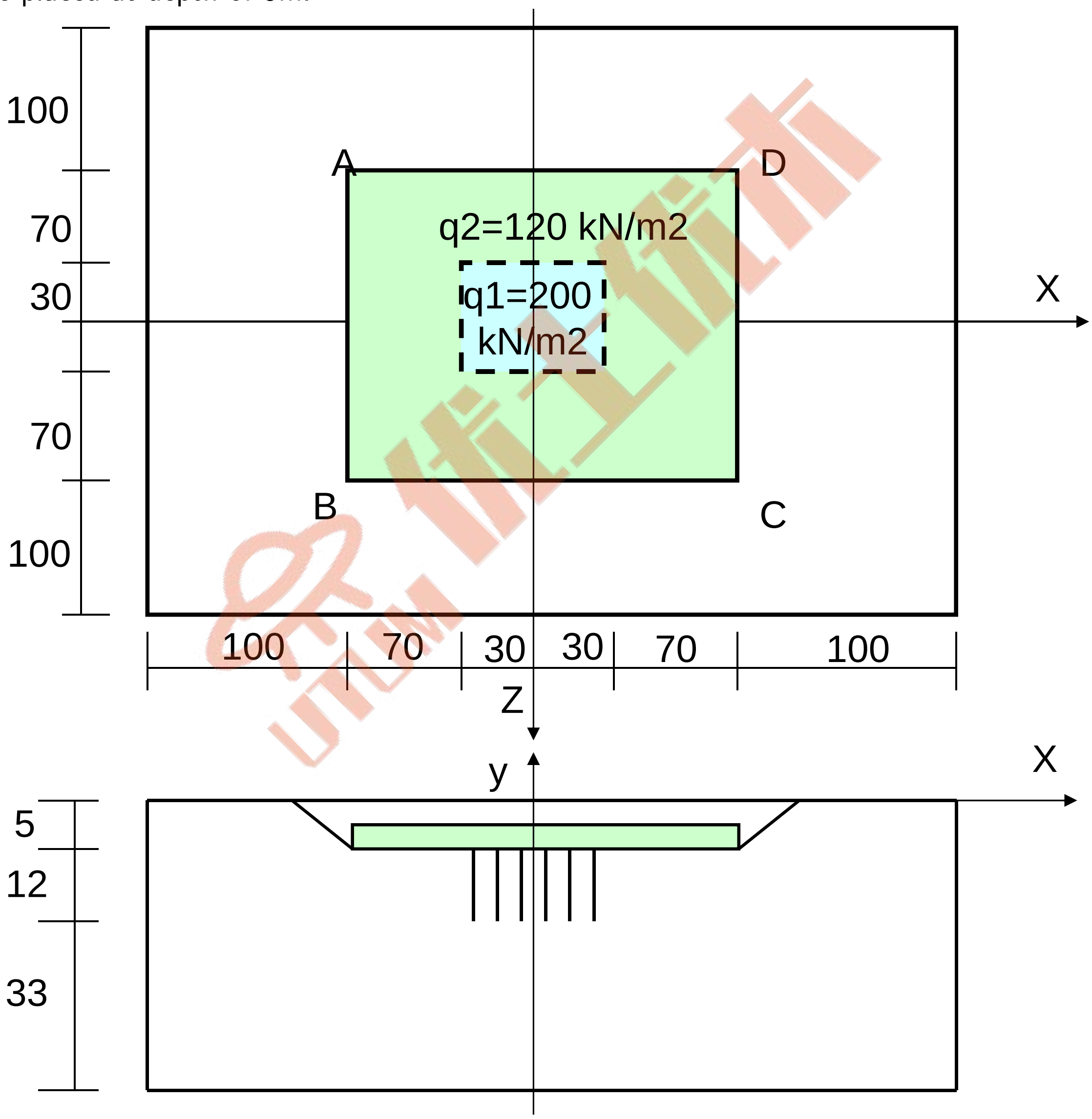
1	Create a new project under name : RSslope.inp
2	Edit existence functions
3	Edit load time function
4	Edit materials
5	Edit drivers
6	Create construction axes
7	Create macro-model
8	Create virtual mesh
9	Create real mesh, boundary conditions and nodal loads
10	Run computation
11	Visualize results

‘

2.4 FOUNDATION RAFT STRENGTHENED BY PILES

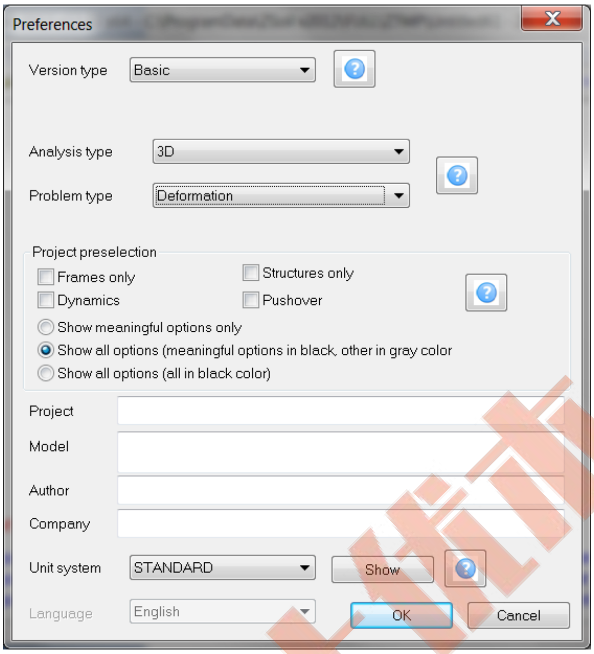
- **Data file: tutorials/foundation-raft.INP**
- **Description**

This example concerns modeling of 1.8m thick large foundation raft strengthened at some part by a large number of bored piles. Geometry of the model is shown in the following figure. Due to the dual symmetry only one quarter of the model will be analyzed. In the central part of the raft the averaged uniform loading ($q1 = 200$ kN/m^2) is higher than in the remaining part ($q = 120$ kN/m^2) and therefore larger differential settlements are foreseen at that part. To cancel excessive settlements in that part a set of 12m long bored piles of diameter $\phi = 80$cm, placed in the grid 6m x 6m, is designed. Foundation raft will be placed at depth of 5m.



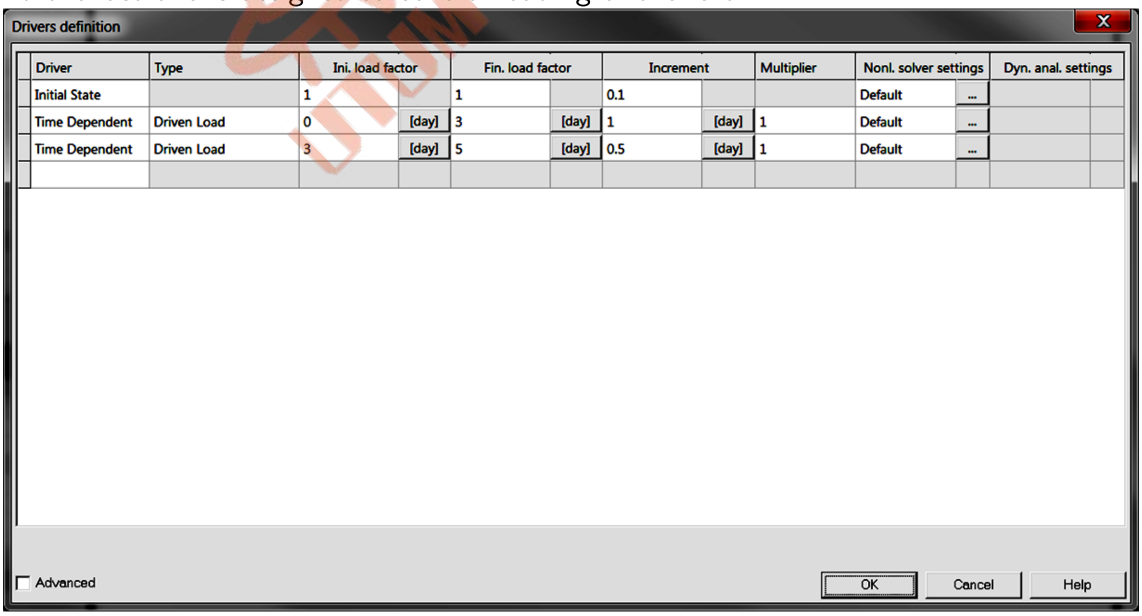
• Project preselection

In the dialog box for project preselection (it appears automatically for option `File/New` in main Z_Soil menu) set ◎ **3D** to ON and select `Deformation` item from the `Problem type` list. The predefined system of units for both data preparation and visualization of results can be verified in menu `Control/Units`.



• Drivers

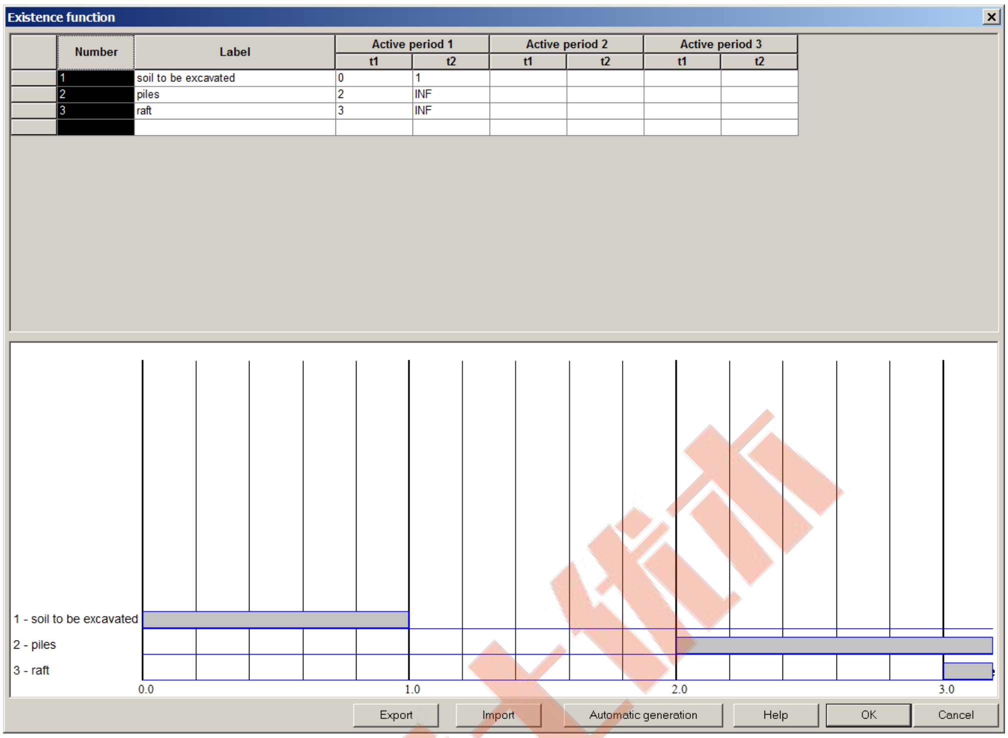
The whole computational process will consist of three drivers i.e. the `Initial state` and then two `Time dependent/Driven load` drivers. The fist driver will yield the in situ stress state at time $t = 0$, the second will allow to trace all excavation/construction steps and the last one is designed to control loading of the raft.



To learn on how to set up the drivers list watch the video Set drivers

- ## Existence functions

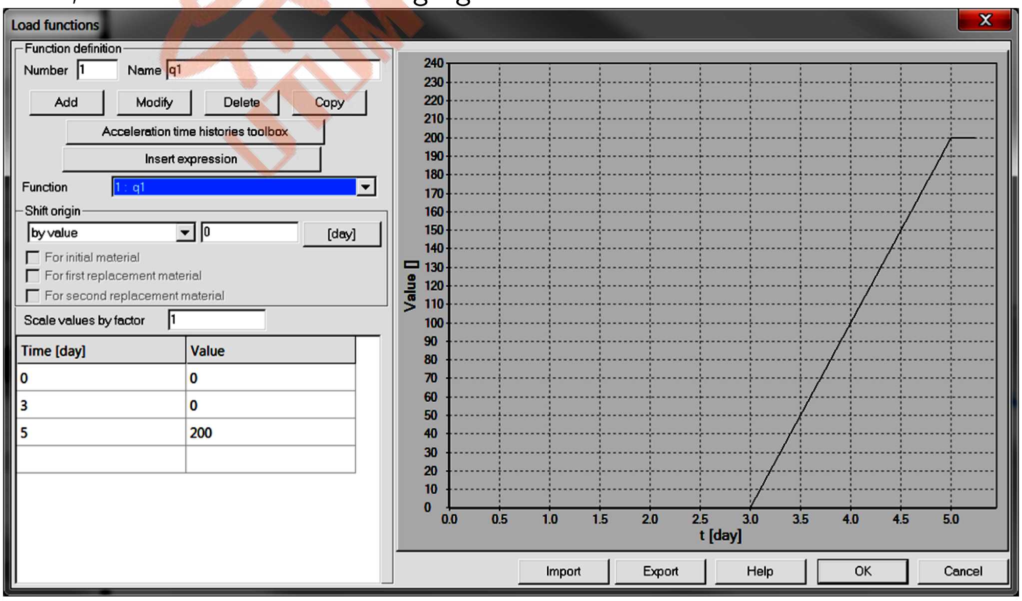
The sequence of excavation/construction events is controlled by the existence functions shown in the figure below. All these existence functions are defined according to the sequence of events specified earlier in the description of the problem.



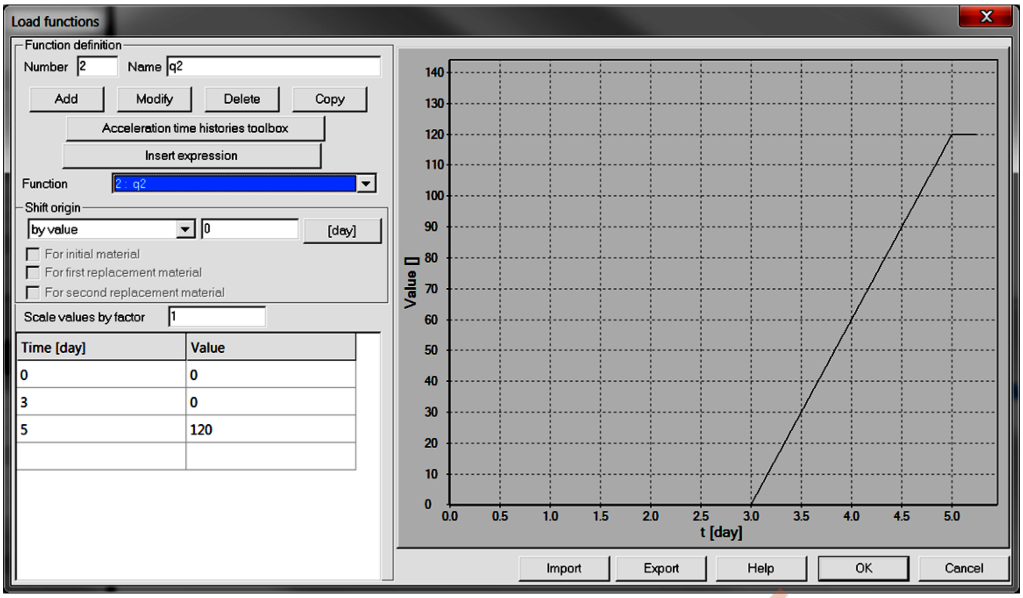
To learn on how to enter existence functions watch the video Edit existence functions .

- ## Load time function

The load time functions associated with the raft loading (defined as a uniform loading) in both zones, are shown in the following figures.



$LTF_1(t)$ Evolution of the load $q1$



$LTF_1(t)$ Evolution of the load $q2$

To learn on how to edit load time functions watch the video Edit load time function .

- **Materials**

Material properties for subsoil, concrete foundation raft, pile and pile tip interfaces are summarized in the following table:

Material	Model	Data group	Properties	Unit	Value
1 Subsoil	Mohr-Coulomb	Elastic	E	[kN/m^2]	60000
			ν	–	0.32
		Density	γ_D	[kN/m^3]	20
			γ^F	[kN/m^3]	10
			e_o	–	0.0
		Non-linear	ϕ	[°]	25
			ψ	[°]	0
			C	[kN/m^2]	15
		Initial state K_o	K_{ox}	[-]	0.7
			K_{oz}	[-]	0.7
2 Raft	Elastic shell	Elastic	E	[kN/m^2]	30000000
			ν	–	0.2
		Density	γ	[kN/m^3]	24
3 Pile interface	Pile interface	Non-linear	ϕ	[°]	20.46
			ψ	[°]	0
			C	[kN/m^2]	12
4 Pile tip int.	Pile tip int.	Non-linear	q_t	[kN/m^2]	0.0
			q_c	[kN/m^2]	2850
5 Pile	Beams	Elastic	E	[kN/m^2]	30000000
			ν	–	0.2
		Density	γ	[kN/m^3]	0
		Geometry	Diameter	[m]	0.8

NB. Thickness of the concrete container is assumed to be 1.8m and is defined in the preprocessor.

- **Generation of the model**

The computational model is built in the following steps and some of them are documented in form of video films.

1	Create a new project under name : foundation-raft.inp
2	Edit materials
3	Edit existence functions
4	Edit load time function
5	Edit drivers
6	Create construction axes
7	Create macro-model
8	Create raft loading
9	Create real mesh and boundary conditions
10	Link pile heads with raft
11	Run computation
12	Visualize results

Index