Single phase media

©ZACE Services Ltd

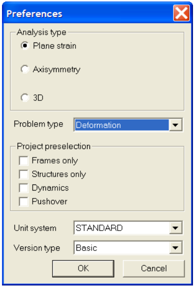
25.08.2009

Outline of this lecture.

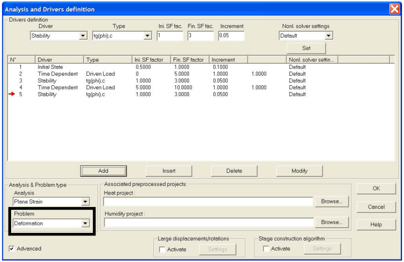
1. Project preselection menu
2. Obligatory and optional model components
3. Drivers
4. Material models for soils
5. Material models for rocks
6. Handling in situ effective stress state
7. Handling gravity load
8. Effective vs total stress analysis

Setting options in preselection menu.

- **Deformation** mode can be set in the preselection dialog just on start or by invoking it from menu *Control/Project preselection*



- **Deformation** mode can be modified in the **Analysis & Drivers dialog** invoked from main menu *Control/Analysis & Drivers*



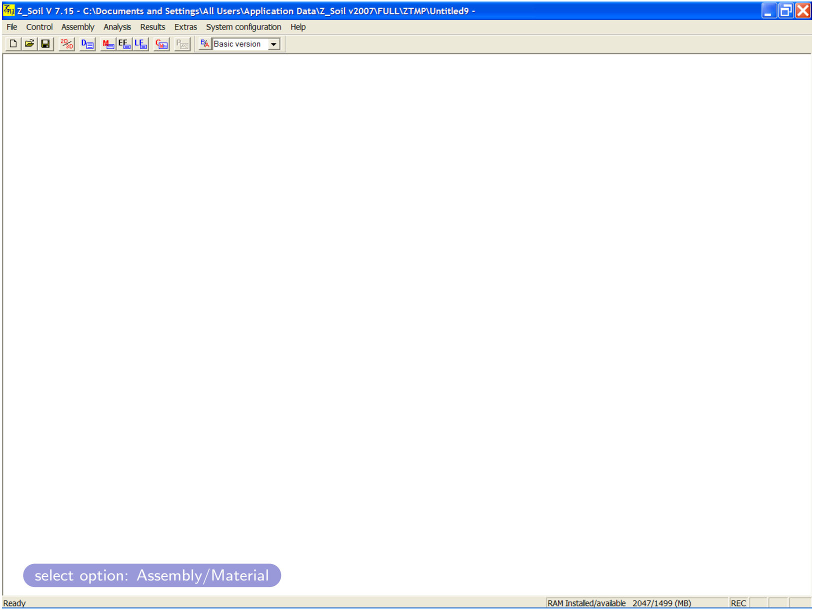
Obligatory and optional components of the model.

1. **Obligatory components:**
   - **Geometrical model** (mesh) **video**
   - **Boundary conditions** (fixities+<loads>) **video**
   - **Driver(s)** **video**
   - **Material definition** (models+properties) **video**
2. **Optional components:**
   - **Load time functions** **video**
     to handle evolution of loads/imposed displacements but also material properties in time
   - **Existence functions** **video**
     (associated with elements/boundary conditions) to handle excavation/stage construction
   - **Initial conditions** **video**
     to set in situ effective stress state in explicit manner
   - **Data super-elements** **video**
     to handle spatial variation of the data e.g. E modulus with depth

Drivers: General information.

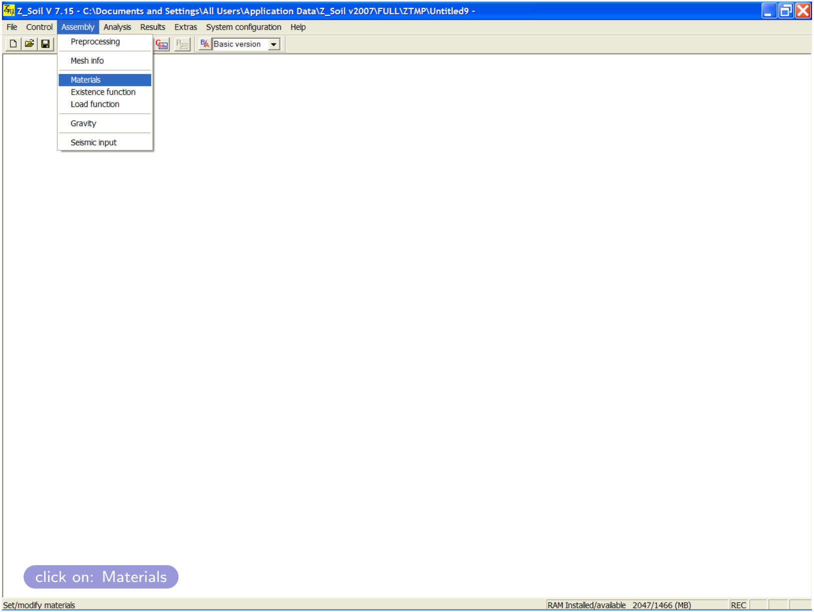
1. Drivers give an instruction to the program what is to be computed and in which order
2. By default the **Initial State** driver is set up as the first and the only one in the list for execution
   (it always yields zero deformation !)
3. Available drivers:
   - **Initial state**
   - **Stability**
   - **Time Dependent**
   - **Pushover** ()
   - **Dynamics** ()

Structure types/Material formulations

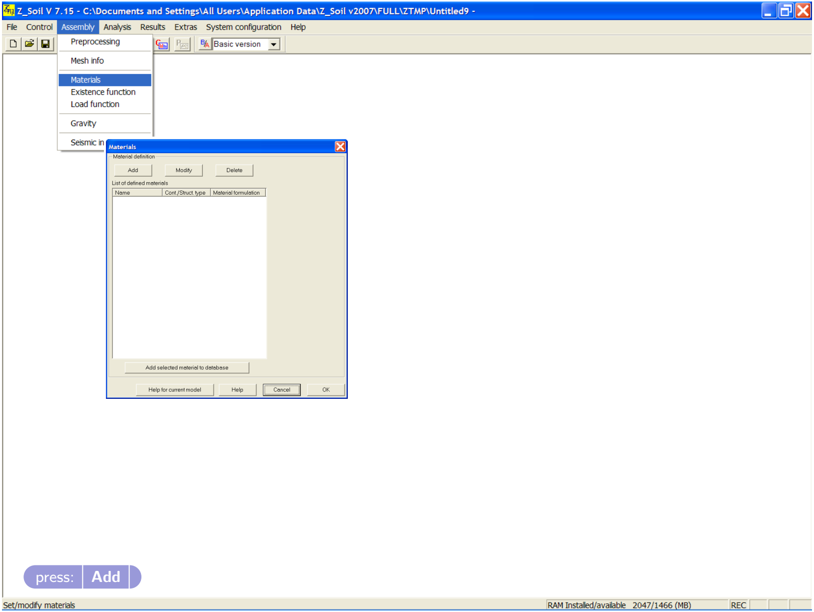


select option: Assembly/Material

Structure types/Material formulations

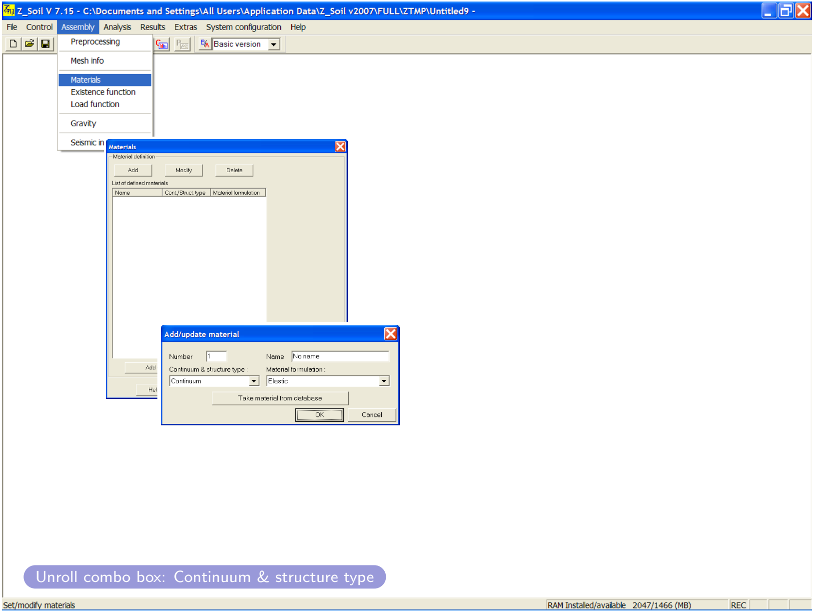


Structure types/Material formulations

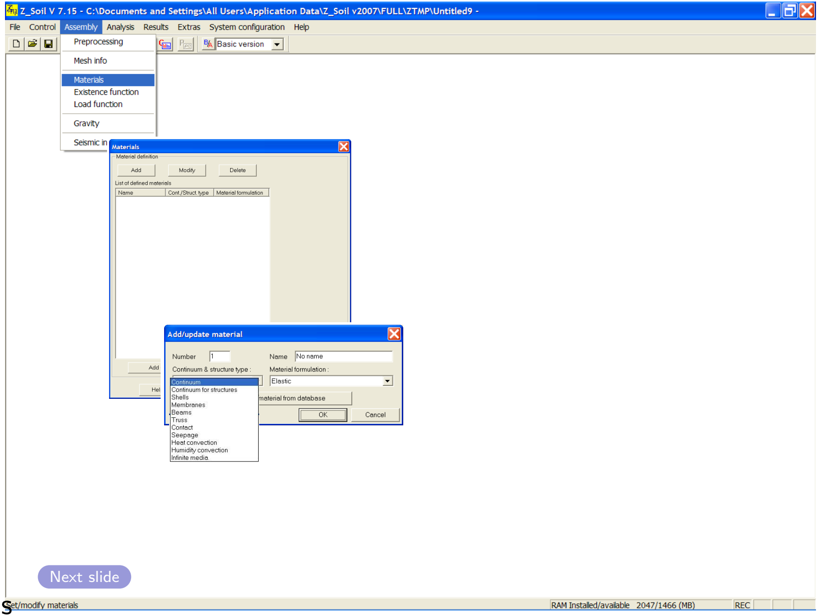


press: **Add**

Structure types/Material formulations



Structure types/Material formulations

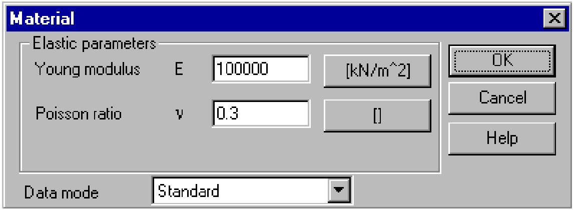


Next slide

Materials for continuum: common data groups

1. Elastic
2. Unit weight
3. Flow/2D
   Flow/3D
4. Creep
5. Init. state Ko (2D)
   Init. state Ko (3D)
6. Heat
7. Humidity

Obligatory



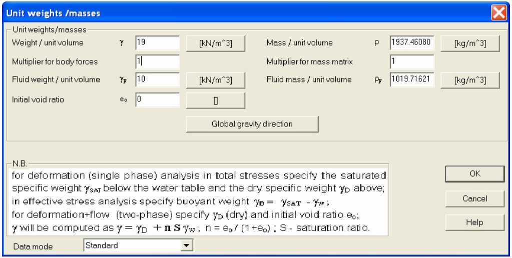
Next slide

Materials for continuum: common data groups

1. Elastic
2. **Unit weight**
3. Flow/2D
   Flow/3D
4. Creep
5. Init. state Ko (2D)
   Init. state Ko (3D)
6. Heat
7. Humidity

Next slide

Obligatory



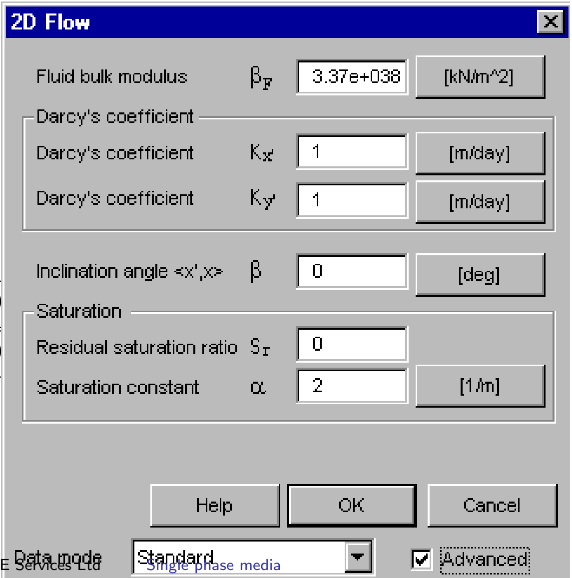
NB. Mass and body force multipliers visible only if Dynamics is activated in the project preselection

Materials for continuum: common data groups

1. Elastic
2. Unit weight
3. **Flow/2D**
   Flow/3D
4. Creep
5. Init. state Ko (2D
   Init. state Ko (3D
6. Heat
7. Humidity

Next slide

Not meaningful for single-phase



Materials for continuum: common data groups

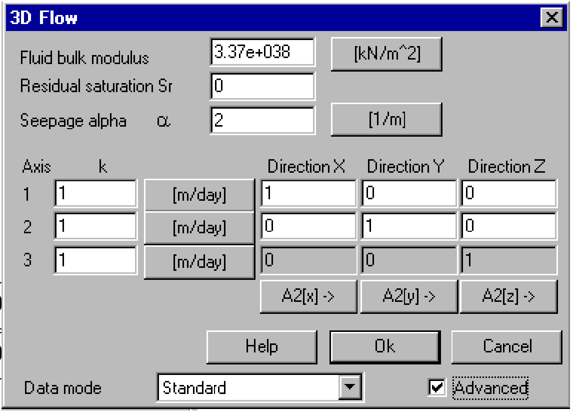
1. Elastic
2. Unit weight
3. Flow/2D

   Flow/3D
4. Creep
5. Init. state Ko (2D

   Init. state Ko (3D
6. Heat
7. Humidity

Next slide

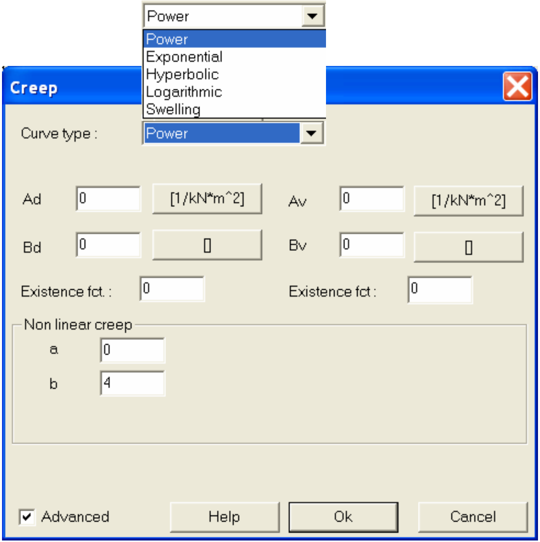
Not meaningful for single-phase



Materials for continuum: common data groups

Optional

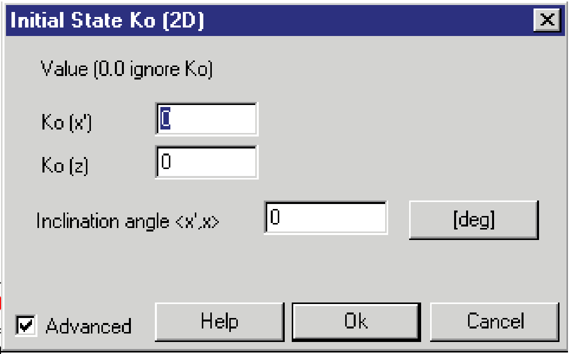
1. Elastic
2. Unit weight
3. Flow/2D
   Flow/3D
4. Creep
5. Init. state Ko (2D
   Init. state Ko (3D
6. Heat
7. Humidity



Next slide

Optional

1. Elastic
2. Unit weight
3. Flow/2D
   Flow/3D
4. Creep
5. Init. state Ko (2D
   Init. state Ko (3D)
6. Heat
7. Humidity



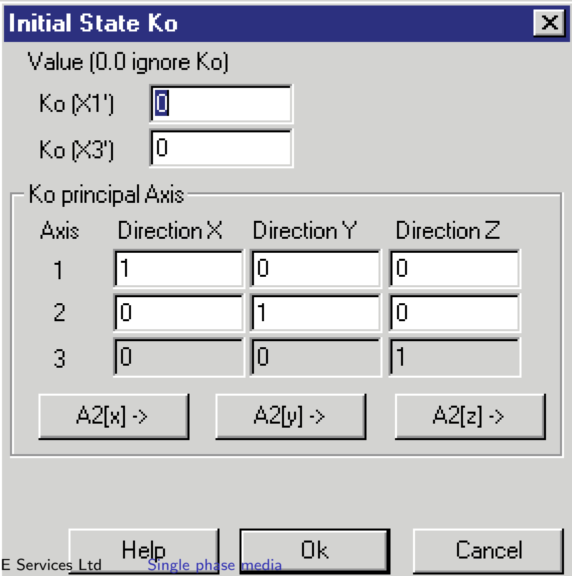
Next slide

Materials for continuum: common data groups

1. Elastic
2. Unit weight
3. Flow/2D
   Flow/3D
4. Creep
5. Init. state Ko (2D
   Init. state Ko (3D
6. Heat
7. Humidity

Next slide

Optional

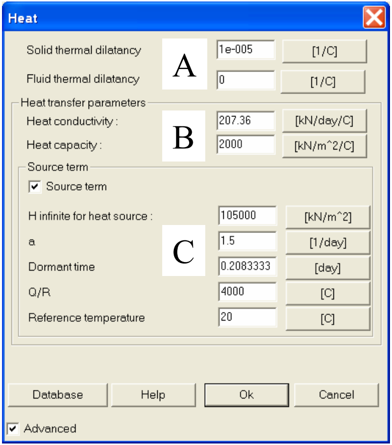


Materials for continuum: common data groups

1. Elastic
2. Unit weight
3. Flow/2D
   Flow/3D
4. Creep
5. Init. state Ko (2D
   Init. state Ko (3D
6. **Heat**
7. Humidity

Next slide

Optional



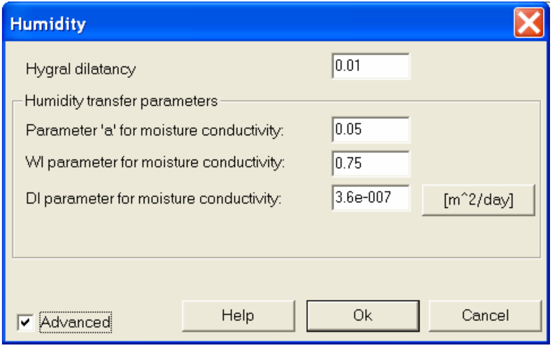
NB. For all models except **Aging concrete** only **Heat dilatancy** is meaningful for single-phase

Materials for continuum: common data groups

1. Elastic
2. Unit weight
3. Flow/2D
   Flow/3D
4. Creep
5. Init. state Ko (2D
   Init. state Ko (3D
6. Heat
7. **Humidity**

Next slide

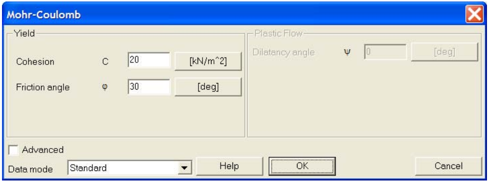
Optional



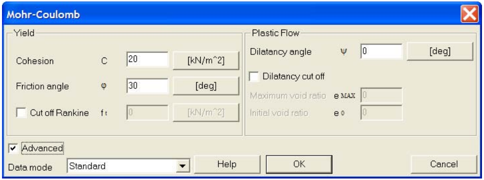
NB. For all models only **Hygral dilatancy** is meaningful for single-phase

Materials for continuum: soils (basic)

1. **Elastic**
2. **Mohr-Coulomb**
   - Basic setup



   - Advanced setup



Remark: dilatancy cut-off ()

Materials for continuum: soils (advanced)

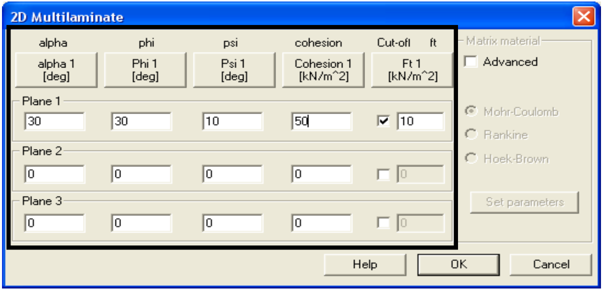
1. **Drucker-Prager**
2. **Cap**
3. **Cam-Clay**
4. **Duncan-Chang**
5. **ECP Hujeux** (only monotonic kernel)()

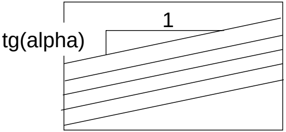
Remark: For plane-strain applications **Drucker-Prager** model will always yield same safety factors and bearing capacity as the standard Mohr-Coulomb model (plane-strain matching must be used)

Materials for continuum: concrete and rock

1. **Elastic**
2. **Aging concrete**
3. **Rankine** (through standard M-C model)
4. **Multilaminate**
5. **Hoek-Brown (M-W)**
6. **Mohr-Coulomb (M-W)**
7. **Rankine (M-W)**

Materials for continuum: Multilaminate model

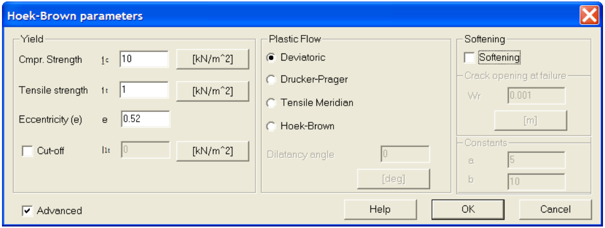




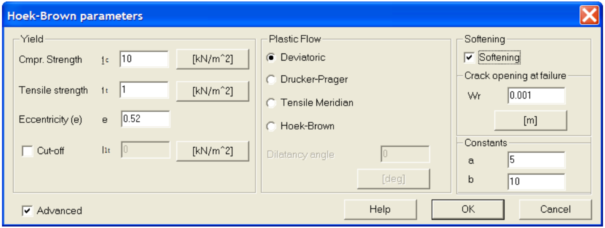
1. To deactivate weakness plane put zeros to all data
2. Switch to advanced mode to activate intact rock model

Materials for continuum: Hoek-Brown model

- Basic setup

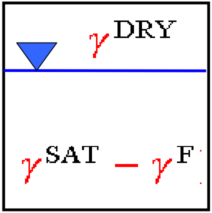


- Advanced setup



Effective stress analysis: Simplified treatment of water

- Equilibrium: $\sigma'_{ij,j} + f_i = 0$
- A different treatment of the gravity is necessary above and below water table
- $f_i = \gamma^{\mathbf{DRY}}\ b_i$ above water table
- $f_i = \left(\gamma^{\mathbf{SAT}} - \gamma^{\mathbf{F}}\right)\ b_i$ under water table
- Hence two material zones must be distinguished

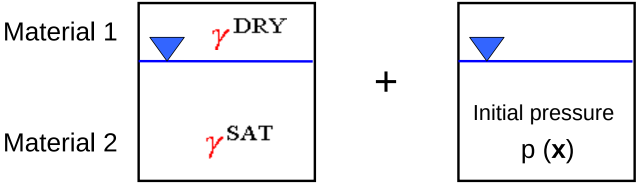


Material 1

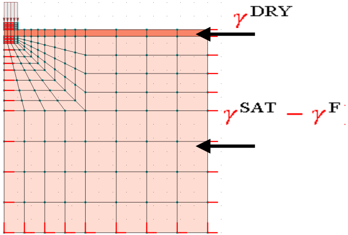
Material 2

Total stress analysis: Simplified treatment of water

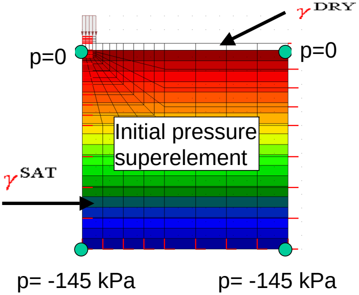
- Equilibrium: $\left(\sigma'_{ij} + \delta_{ij}\, p\right)_{,j} + f_i = 0$
- A different treatment of the gravity is necessary above and below water table
- $f_i = \gamma^{\textbf{DRY}}\, b_i$ above water table
- $f_i = \gamma^{\textbf{SAT}}\, b_i$ under water table
- Hence again two material zones must be distinguished



Effective stress: footes.inp



Total stress: footwt.inp



video

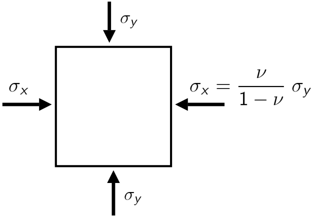
- Remark: Both approaches are equivalent

Handling in situ stress: General

Two different approaches can be used

1. **(A)** Running the **Initial state** driver with optionally additional constraint put on K_o^{insitu}
   - **(A1)** with explicit setting of $\boldsymbol{\sigma}_o$ as the initial guess
   - **(A2)** without explicit setting of $\boldsymbol{\sigma}_o$
2. **(B)** Explicit setting of $\boldsymbol{\sigma}_o$

Note that the initial state computation without prescribed value of K_o^{insitu} will yield

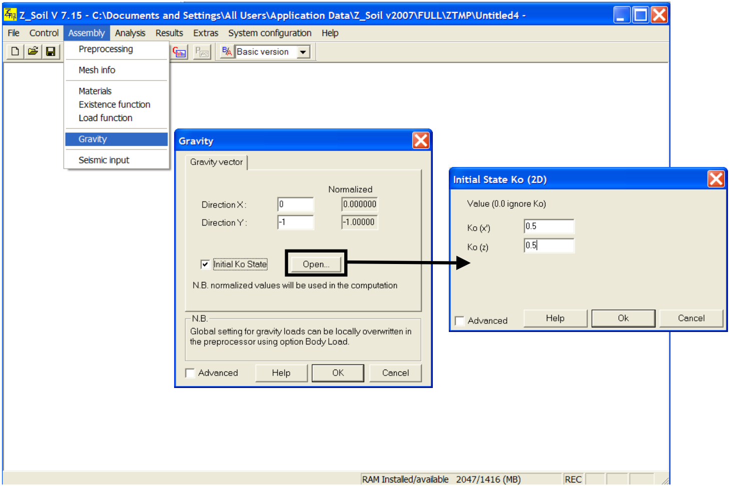


ex. $\nu = 0.2$ will yield
$K_o^{insitu} = 0.25$
which can be far below the real value

Handling in situ stress: Handling K_o^{insitu} constraint

The K_o^{insitu} constraint is meaningful only for approach **(A)**

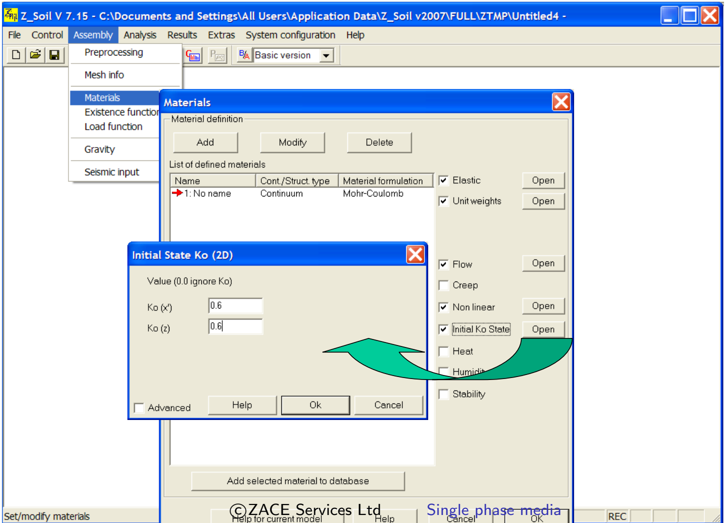
1. K_o^{insitu} can be set globally (for all continuum materials)



Handling in situ stress: Handling K_o^{insitu} constraint (cont..)

The K_o^{insitu} constraint is meaningful only for approach **(A)**

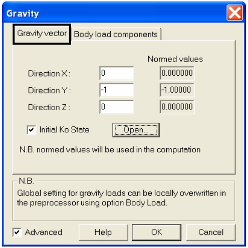
1. K_o^{insitu} can also be set locally (for all continuum materials)

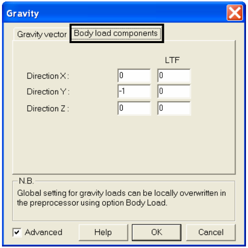


Handling in situ stress: K_o^{insitu} constraint: Remarks

1. For normally consolidated soils or sands we can make use of simplified Jaky formula $K_o^{insitu} = K_o^{NC} = 1 - \sin(\phi)$
2. This setting is important in soil structure interaction problems
3. Global setting for K_o^{insitu} applies this constraint to all continuum materials
4. Local setting for K_o^{insitu} overwrites the global one for the specific material (can be done only at the material level);
5. Setting $K_{ox}^{insitu} = 0.0$ and $K_{oz}^{insitu} = 0.0$ cancels the constraint
6. In approach **(A)** final initial stresses may differ from the ones specified by the user

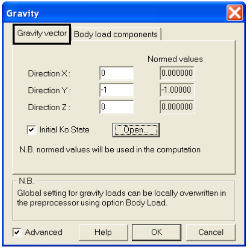
Handling gravity loads: option **Gravity vector**

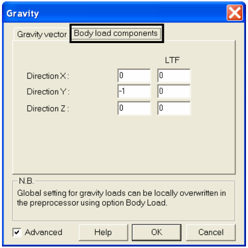




- Using this definition user may declare gravity vector $\bar{\mathbf{b}}$ (default is (0,-1,0))
- This vector $\bar{\mathbf{b}}$ is used to compute current body load vector $\mathbf{b} = \gamma\,\bar{\mathbf{b}}$ at each finite element

Handling gravity loads: option **Body load components**





- Option visible in the ☒ **Advanced** mode
- Using this definition user may declare gravity vector $\bar{\mathbf{b}}$ (default is (0,-1,0)) plus load time functions independently for all directions
- In this case gravity direction vector is not normalized
- Vector $\bar{\mathbf{b}}$ is used to compute current body load components $b_i = \gamma \; \bar{b}_i \; LTF_i(t)$ at each finite element

Handling gravity loads: Remarks

1. Body load definition may be overwritten, at the element level, during preprocessing session, using option `Body load`
2. For pseudo-seismic stability analysis the option `Body load components` has to be used (for instance in 2D models one may assume $\overline{\mathbf{b}} = (0.1, -1.0)$)