

Structures: Membranes

©ZACE Services Ltd

21.08.2007

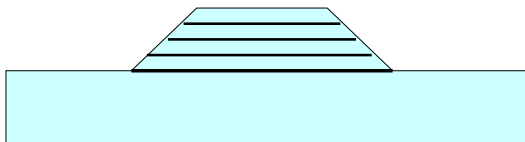


Soil-structure interaction: Example

- **Example**

Geotextiles/geogrids

- Stabilization of steep slopes
- Retaining walls and abutments
- Road construction





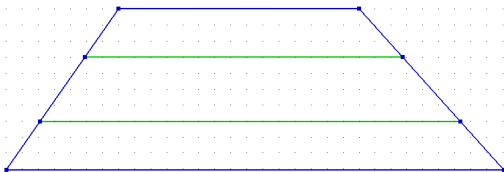
Geometrical model: Macro level

- **Create membrane on objects**

Step:A Select objects:
Macro-model/Objects/Select/....

Step:B Create membs :
Macro-model/Subdomain/Create/Membrane(s) on objects

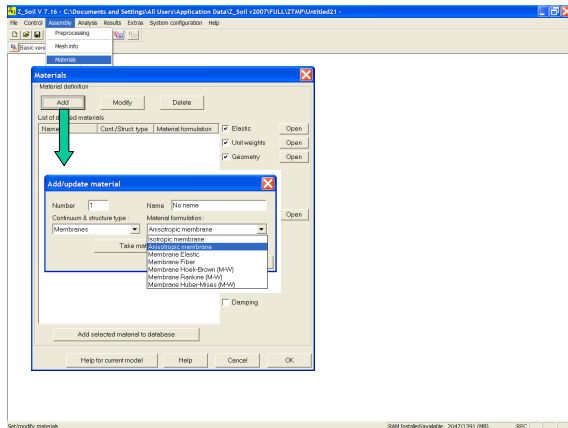
NB. Split preserves the initial curved membrane geometry
All methods at the FE level are similar to the ones designed for Macro-model level but geometry is restricted only to the straight line segment





Add new material for membranes

1 Add new membrane material





Elastic membrane: data groups

- 1 **Elastic**
- 2 Unit weight
- 3 Geometry
- 4 Heat
- 5 Humidity
- 6 Damping

Obligatory

Material

Elastic parameters

Young modulus E 100000 [kN/m²]

Poisson ratio ν 0.3 []

OK

Cancel

Help

Data mode Standard

Next slide





Elastic membrane: data groups

- 1 Elastic
- 2 **Unit weight**
- 3 Geometry
- 4 Heat
- 5 Humidity
- 6 Damping

Optional

Unit weight/mass

Unit weight/mass
Weight/Unit volume

γ 0 [kN/m³]

γ multiplier for body forces 1

Mass/Unit volume

ρ 0 [kg/m³]

ρ multiplier for mass matrix 1

Global gravity direction

Data mode Standard

OK
Cancel
Help

Next slide

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





Elastic membrane: data groups

Obligatory

- 1 Elastic
- 2 Unit weight
- 3 **Geometry**
- 4 Heat
- 5 Humidity
- 6 Damping

A screenshot of a software dialog box titled "Material" with a blue header and a red close button. The dialog has a light beige background. Inside, there is a label "Thickness" followed by a text input field containing the value "0.1" and a unit selection button labeled "[m]". At the bottom of the dialog, there are two buttons: "OK" and "Cancel".

Next slide

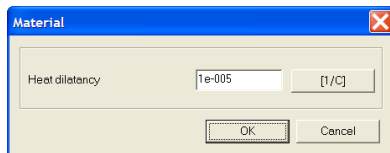




Elastic membrane: data groups

- 1 Elastic
- 2 Unit weight
- 3 Geometry
- 4 **Heat**
- 5 Humidity
- 6 Damping

Optional



NB. Preprocessed thermal strains (by Heat project) are handled by membrane elements

Next slide

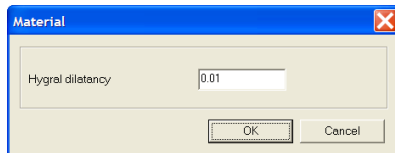




Elastic membrane: data groups

Optional

- 1 Elastic
- 2 Unit weight
- 3 Geometry
- 4 Heat
- 5 **Humidity**
- 6 Damping



NB. Preprocessed hygral strains (by Humidity project) are handled by membrane elements

Next slide





Elastic membrane: data groups

Optional

- 1 Elastic
- 2 Unit weight
- 3 Geometry
- 4 Heat
- 5 Humidity
- 6 **Damping**

Damping

α_0 0 [1/day]

β_0 0 [day]

☐ Evaluate damping coefficients from imposed values

OK Cancel

Evaluate damping coefficients from imposed values

ω_1 0 [1/day] ξ_1 0 []

ω_2 0 [1/day] ξ_2 0 []

Calculate

α_0 0 [1/day] β_0 0 [day]

OK Cancel

Next slide

NB. Meaningful only for Dynamics



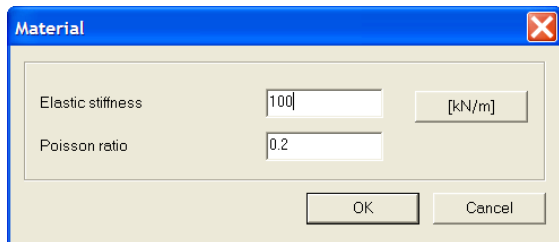


Isotropic membrane: data groups

Obligatory

- 1 **Elastic**
- 2 **Unit weight**
- 3 **Nonlinear**

Next slide



The image shows a 'Material' dialog box with a blue title bar and a close button (X) in the top right corner. The dialog contains two input fields: 'Elastic stiffness' with the value '100' and a unit box '[kN/m]', and 'Poisson ratio' with the value '0.2'. At the bottom, there are 'OK' and 'Cancel' buttons.

NB. Thickness is already included in stiffness parameter

Properties in Heat, Humidity and Damping data groups are the same as for the Elastic membrane model.



Isotropic membrane: data groups

1

Elastic

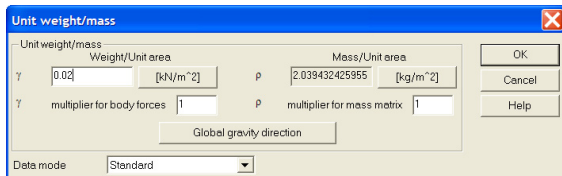
2

Unit weight

3

Nonlinear

Next slide

Optional

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

Properties in Heat, Humidity and Damping data groups are the same as for the Elastic membrane model.

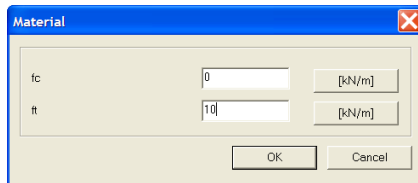




Isotropic membrane: data groups

- 1 Elastic
- 2 Unit weight
- 3 **Nonlinear**

Obligatory



The Material dialog box has a blue title bar with the text 'Material' and a close button. It contains two rows of input fields. The first row is for 'fc' with a value of '0' and a unit of '[kN/m]'. The second row is for 'ft' with a value of '10' and a unit of '[kN/m]'. At the bottom are 'OK' and 'Cancel' buttons.

Property	Value	Unit
fc	0	[kN/m]
ft	10	[kN/m]

Next slide

NB. Compressive force should be set to $f_c = 0$ to avoid parasitic compressive forces; a plane-stress elasto-plastic model with cut-off conditions for the tensile and compressive principal membrane forces is applied

Properties in Heat, Humidity and Damping data groups are the same as for the Elastic membrane model.

 Anisotropic membrane: data groups

①

Elastic

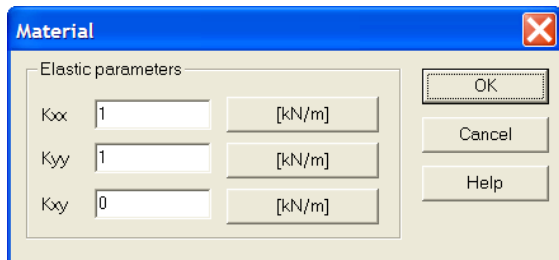
②

Unit weight

③

Nonlinear

Next slide

Obligatory

The image shows a 'Material' dialog box with a blue title bar and a red close button. Inside, the 'Elastic parameters' section contains three rows of input fields and units:

Parameter	Value	Unit
K _{xx}	1	[kN/m]
K _{yy}	1	[kN/m]
K _{xy}	0	[kN/m]

On the right side of the dialog box, there are three buttons: 'OK', 'Cancel', and 'Help'.

NB. Thickness is already included in stiffness parameters

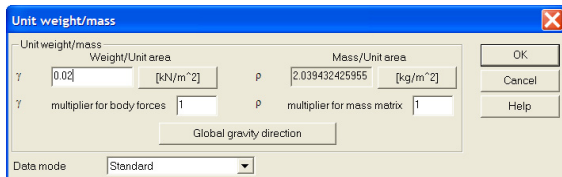
Properties in Heat, Humidity and Damping data groups are the same as for the Isotropic membrane model



 Anisotropic membrane: data groups

Optional

- 1 Elastic
- 2 **Unit weight**
- 3 Nonlinear



Next slide

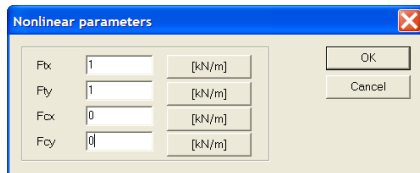
NB. Mass and body force multipliers visible only if Dynamics is activated in the project preselection

Properties in Heat, Humidity and Damping data groups are the same as for the Isotropic membrane model



 Anisotropic membrane: data groups

- 1 Elastic
- 2 Unit weight
- 3 **Nonlinear**

Obligatory

Parameter	Value	Unit
Ftx	1	[kN/m]
Fty	1	[kN/m]
Fcx	0	[kN/m]
Fcy	0	[kN/m]

[Next slide](#)

NB. Compressive limit forces should be set to $f_{cx} = 0$, $f_{cy} = 0$ to avoid parasitic compressive forces; a plane-stress elasto-plastic model with cut-off conditions for the tensile and compressive principal membrane forces is applied

Properties in Heat, Humidity and Damping data groups are the same as for the Isotropic membrane model



Fiber membrane: data groups

- 1 **Elastic**
- 2 Unit weight
- 3 Geometry
- 4 Nonlinear

Obligatory

The image shows a 'Material' dialog box with a blue title bar and a red close button. Inside, there is a text input field for 'Young modulus' containing the value '100000'. To the right of the input field is a unit label '[kN/m^2]'. At the bottom of the dialog are 'OK' and 'Cancel' buttons. The 'OK' button is highlighted with a dashed border.

Next slide

Fiber model generates stiffness only in one direction !

Properties in Heat, Humidity and Damping data groups are the same as for the Isotropic membrane model





Fiber membrane: data groups

- 1 Elastic
- 2 **Unit weight**
- 3 Geometry
- 4 Nonlinear

Optional

Unit weight/mass

Unit weight/mass

Weight/Unit volume

γ 0 [kN/m³]

Mass/Unit volume

ρ 0 [kg/m³]

γ [kg/m³]

Global gravity direction

Data mode Standard

OK Cancel Help

Next slide

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

Properties in Heat, Humidity and Damping data groups are the same as for the Isotropic membrane model

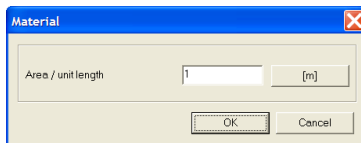




Fiber membrane: data groups

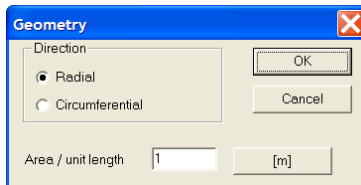
- 1 Elastic
- 2 Unit weight
- 3 **Geometry**
- 4 Nonlinear

Obligatory
2D/3D



Material dialog box showing the input field for Area / unit length set to 1 [m]. The OK and Cancel buttons are visible at the bottom.

Axisymmetry



Geometry dialog box showing the Direction section with Radio buttons for Radial (selected) and Circumferential. The Area / unit length input field is set to 1 [m]. The OK and Cancel buttons are visible at the bottom.

Next slide

Properties in Heat, Humidity and Damping data groups are the same as for the Isotropic membrane model





Fiber membrane: data groups

- 1 Elastic
- 2 Unit weight
- 3 Geometry
- 4 **Nonlinear**

Obligatory

Non Linear Truss

fc: 0 [kN/m²]

ft: 10 [kN/m²]

OK Cancel

NB. Compressive strength should be set to $f_c = 0$ to avoid parasitic compressive stresses

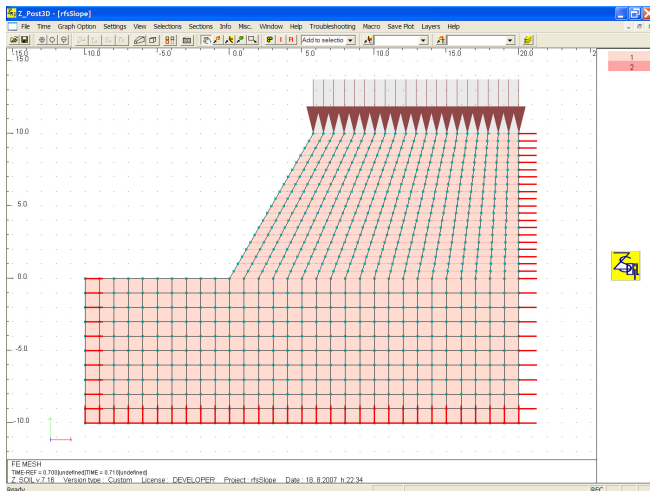
Next slide

Properties in Heat, Humidity and Damping data groups are the same as for the Isotropic membrane model





Example of limit state analysis of reinforced slope: Mesh



NB.21 layers of membrane fibers are added





Example of limit state analysis of reinforced slope: Failure mode

