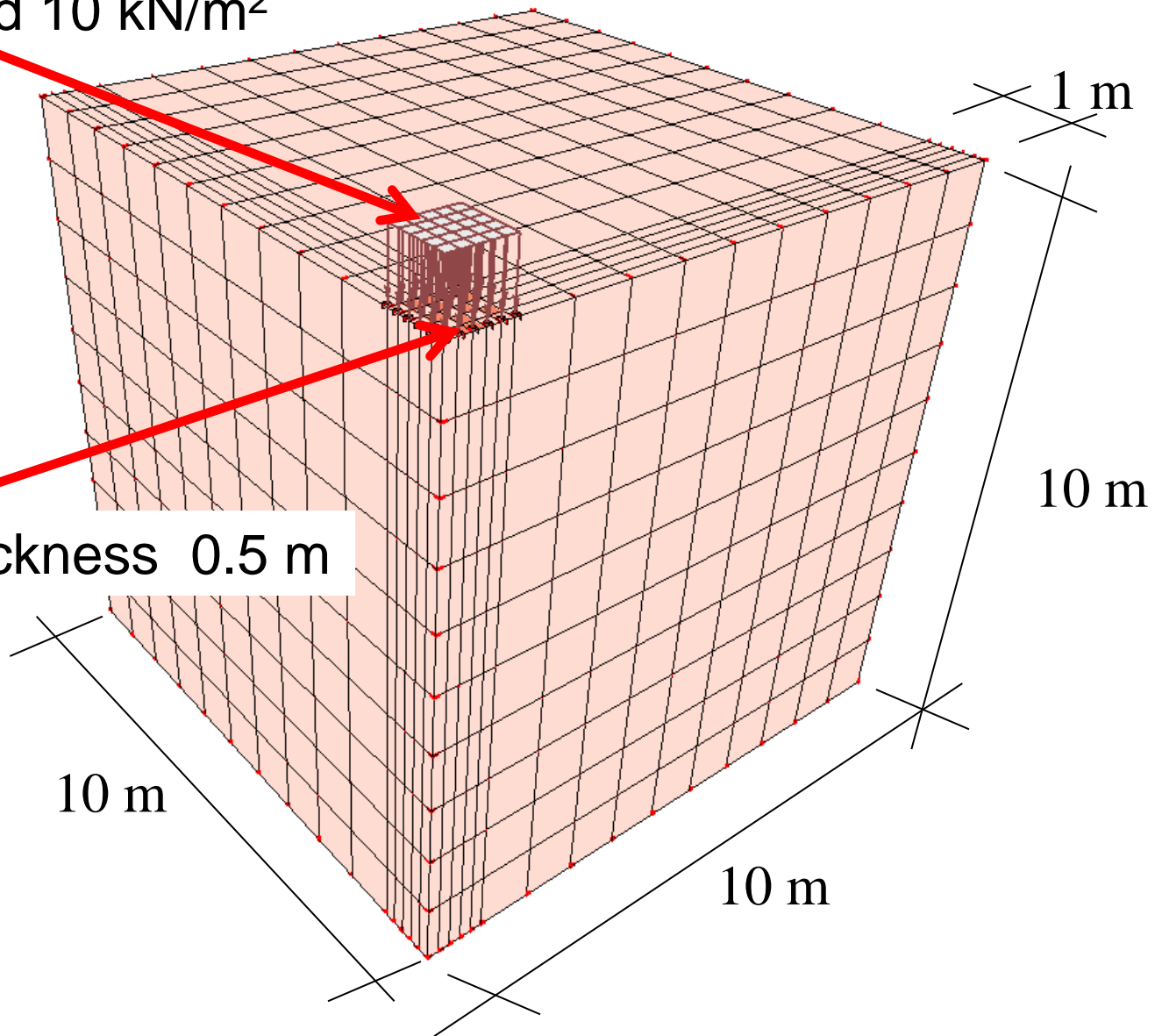


# Lecture 1 - Foundation

# Foundation

Surface load  $10 \text{ kN/m}^2$

Foundation thickness  $0.5 \text{ m}$

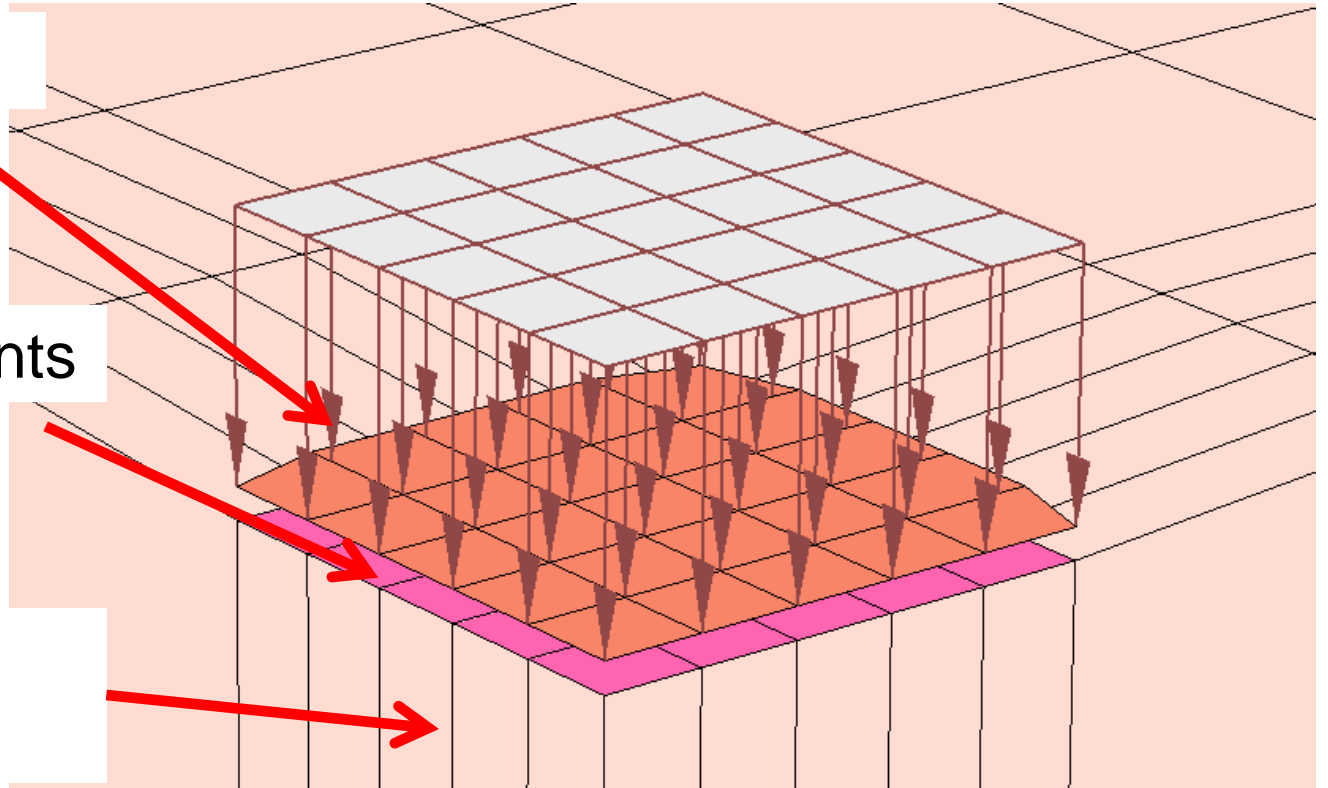


# Foundation

Shell elements

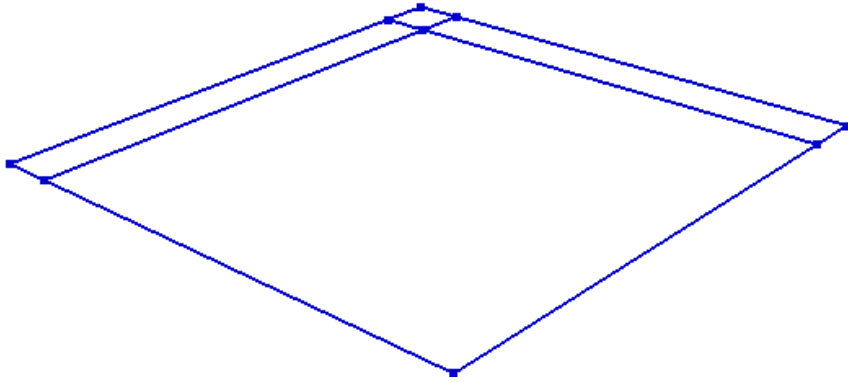
Interface elements

Continuum 3D  
elements

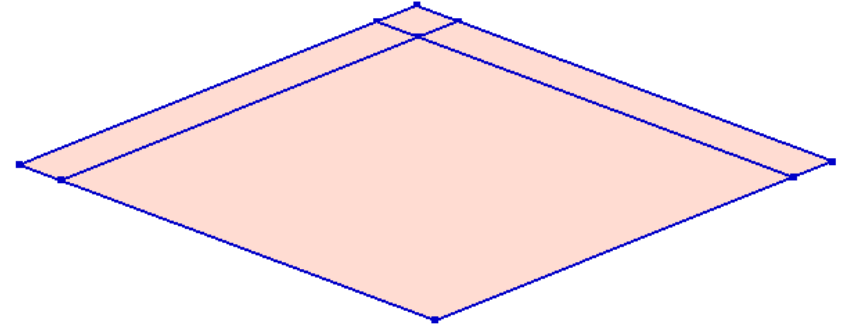


# Overview

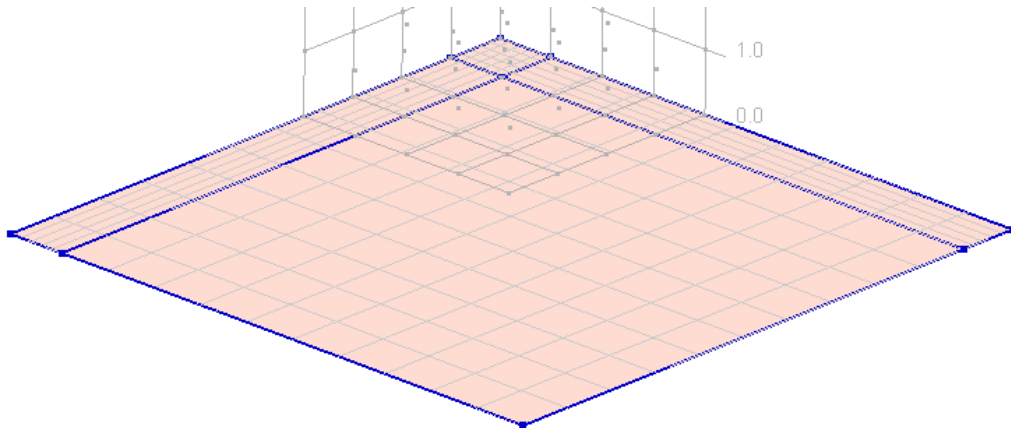
## 1. Lines



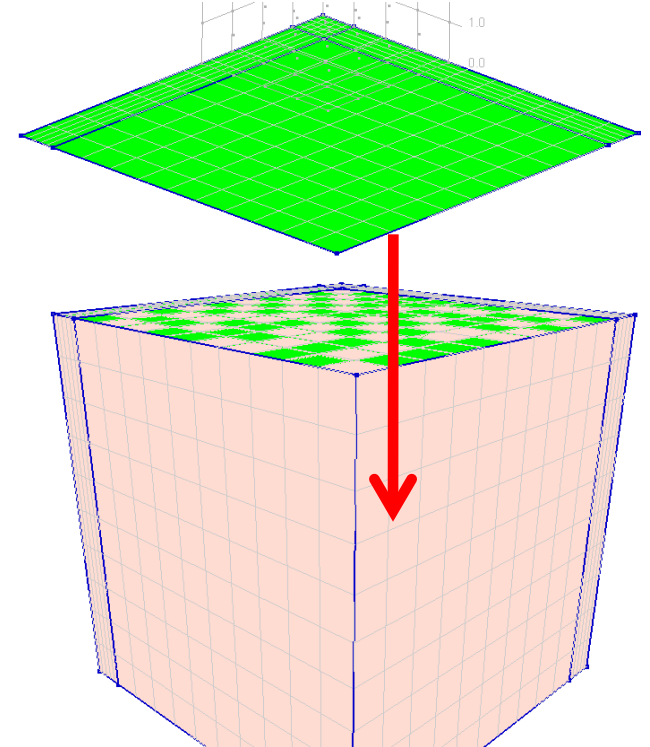
## 2. Continuum 2D subdomain



## 3. Virtual mesh for Continuum 2D Subdomains

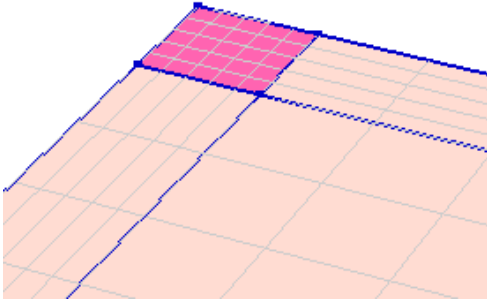


## 4. Extrude 2D subdomain

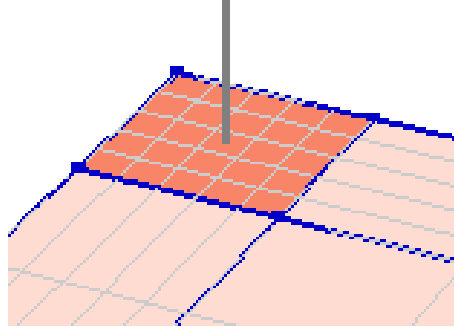


# Overview

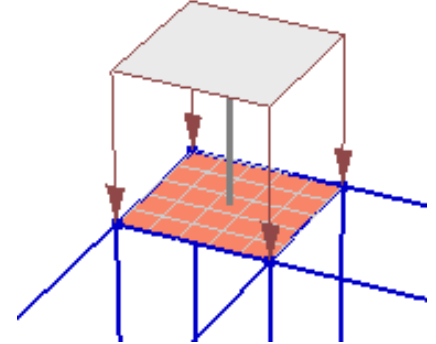
5. Interface element on subdomain face



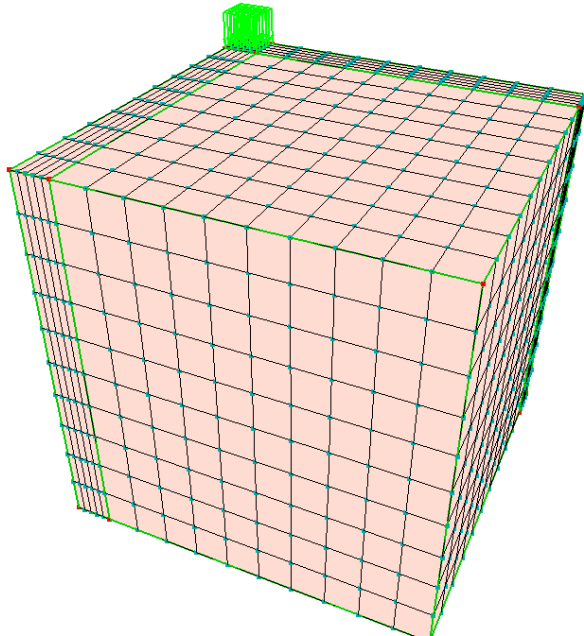
6. Shell element on subdomain face



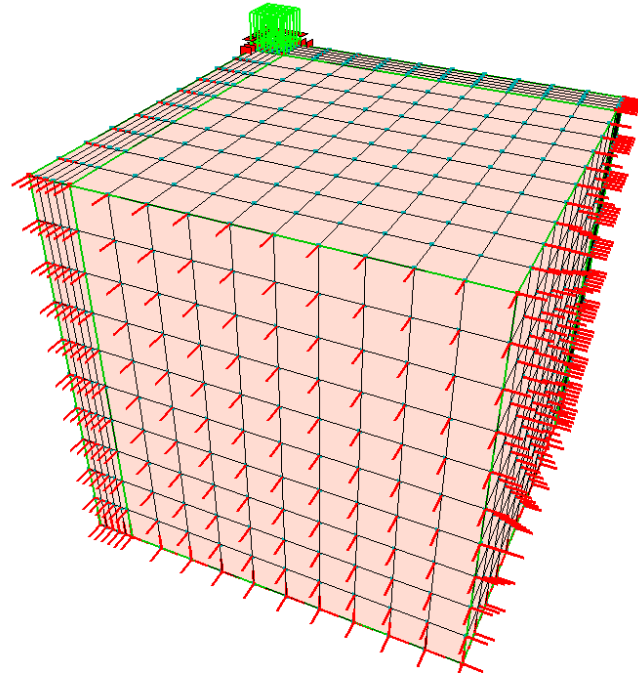
7. Surface load on subdomain face



8. Virtual to real mesh



9. Solid boundary conditions on box

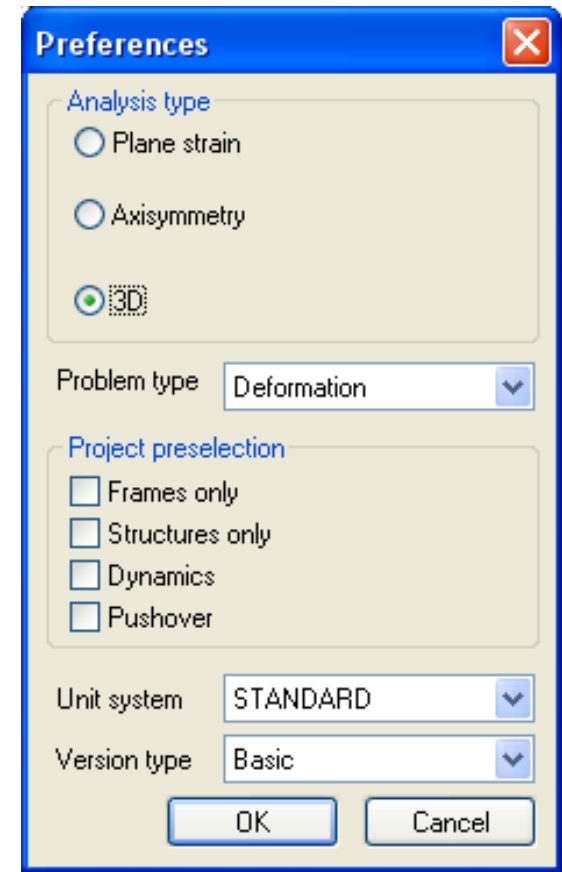
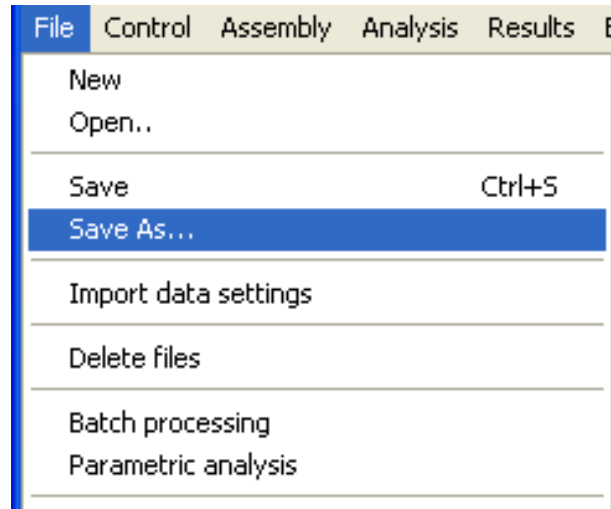


# Run Zsoil

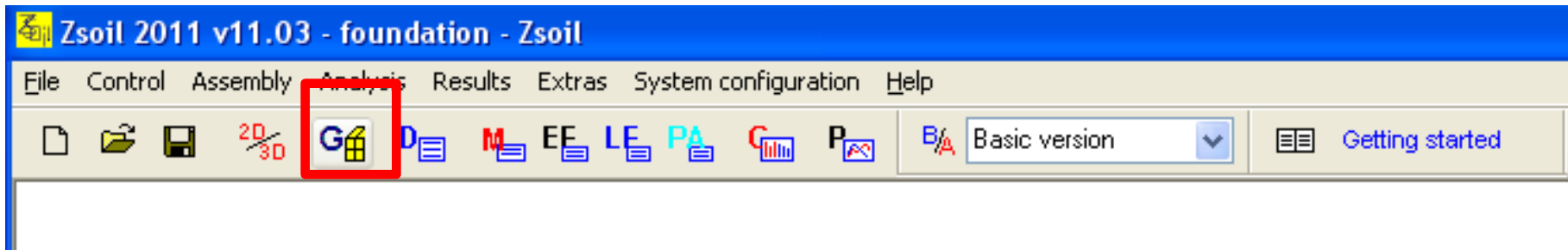
In Preferences select :

- Analysis type as 3D
- Problem type as Deformation

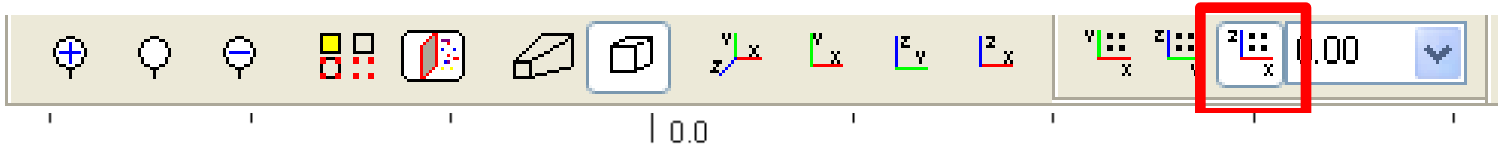
Save project as:  
Foundation.inp



Run preprocessing choosing Assembly\Preprocessing from top menu or icon from top toolbar

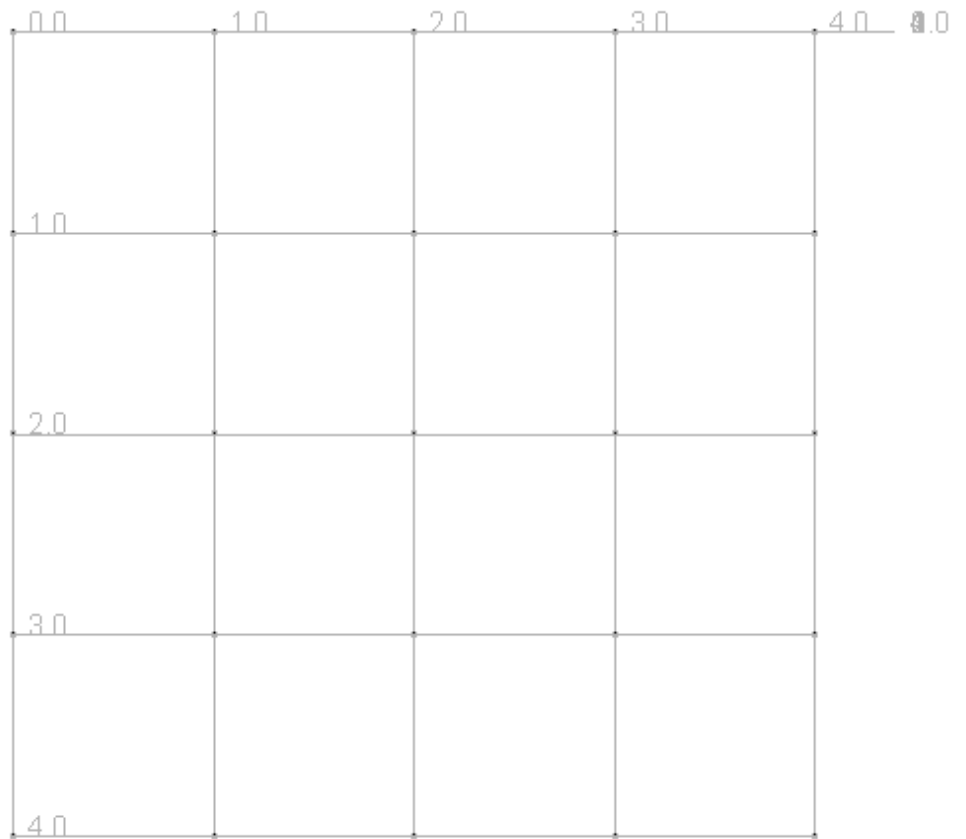


# Set XZ projection



5.0

0.0

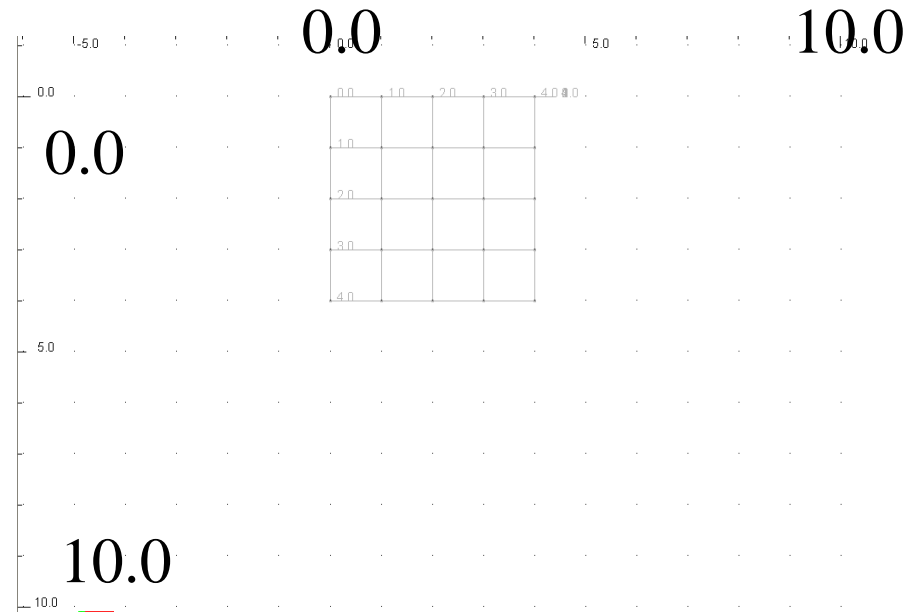
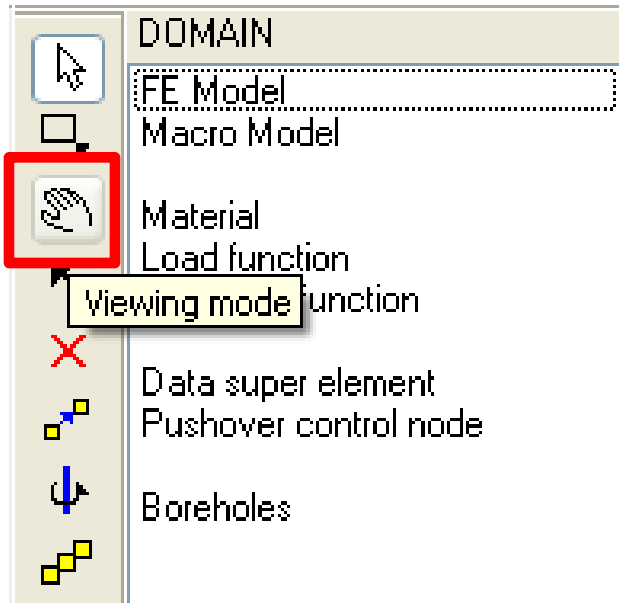


# Change camera position

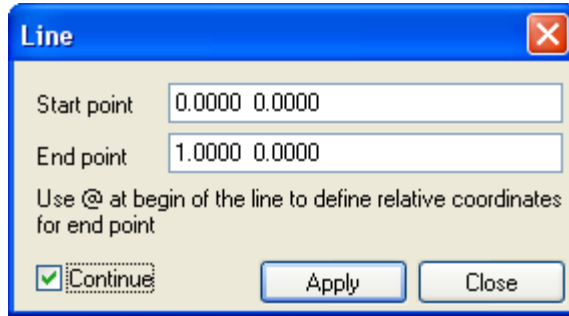
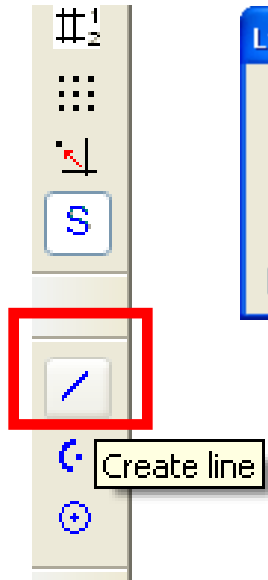


Pick **Zoom out** icon several time or use mouse wheel to change camera distance from the objects

Pick **Viewing mode** icon and change camera position with the mouse keeping left button down

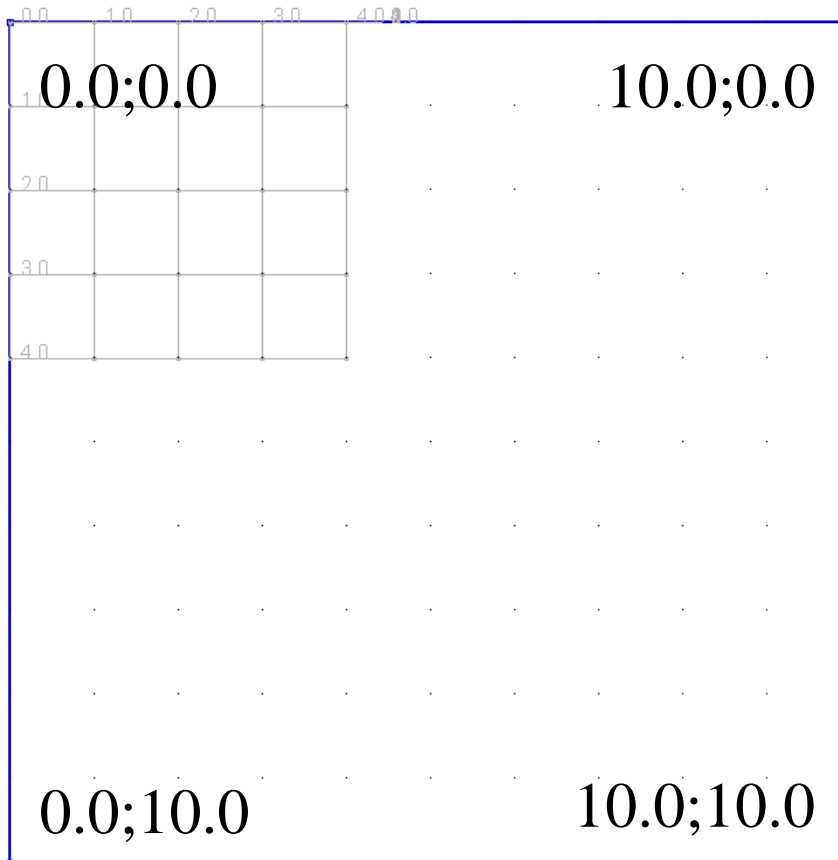


# Create lines

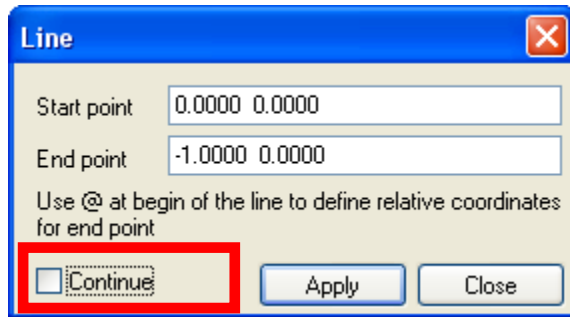


Create 4 lines clicking grid points

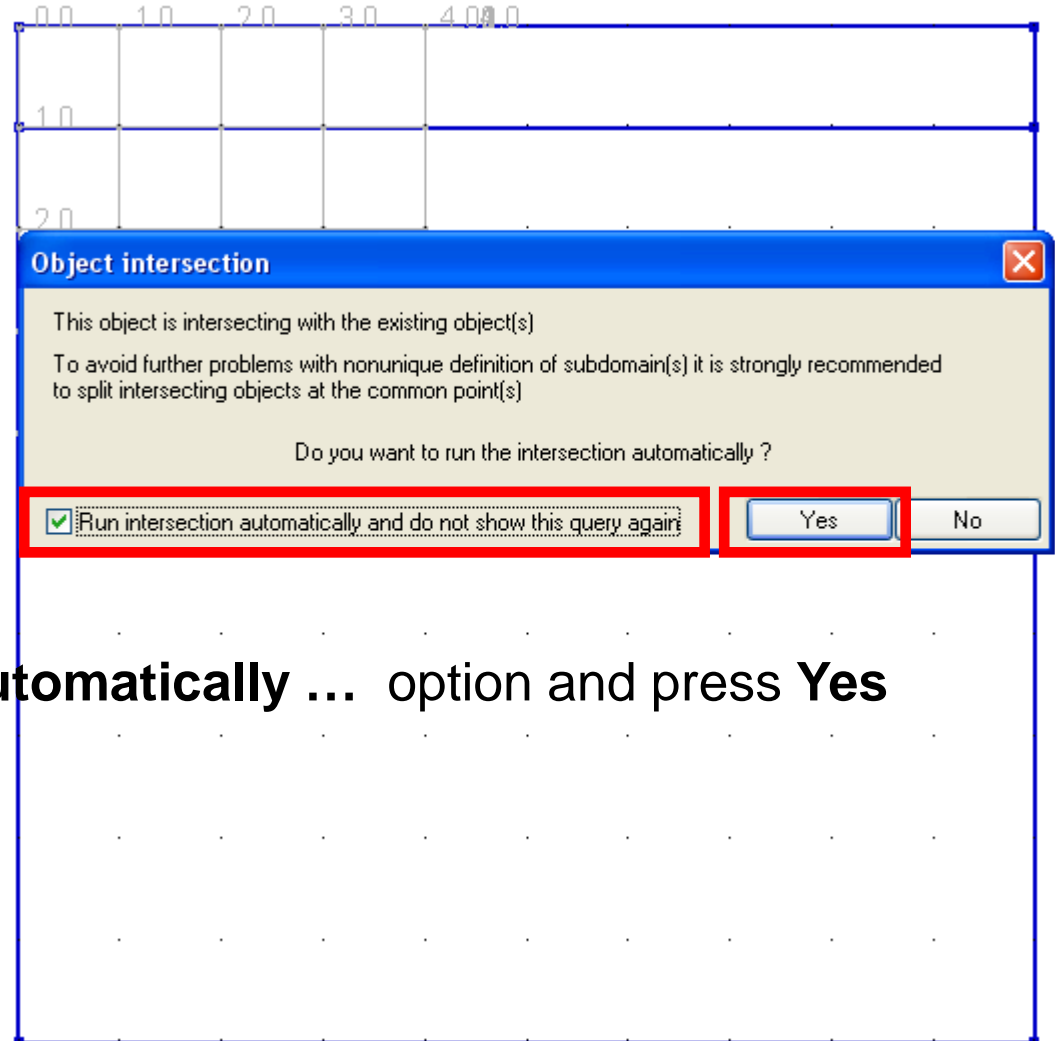
Use rules on top and left side to find cursor position



# Create lines



Turn off **Continue** check box  
Create line

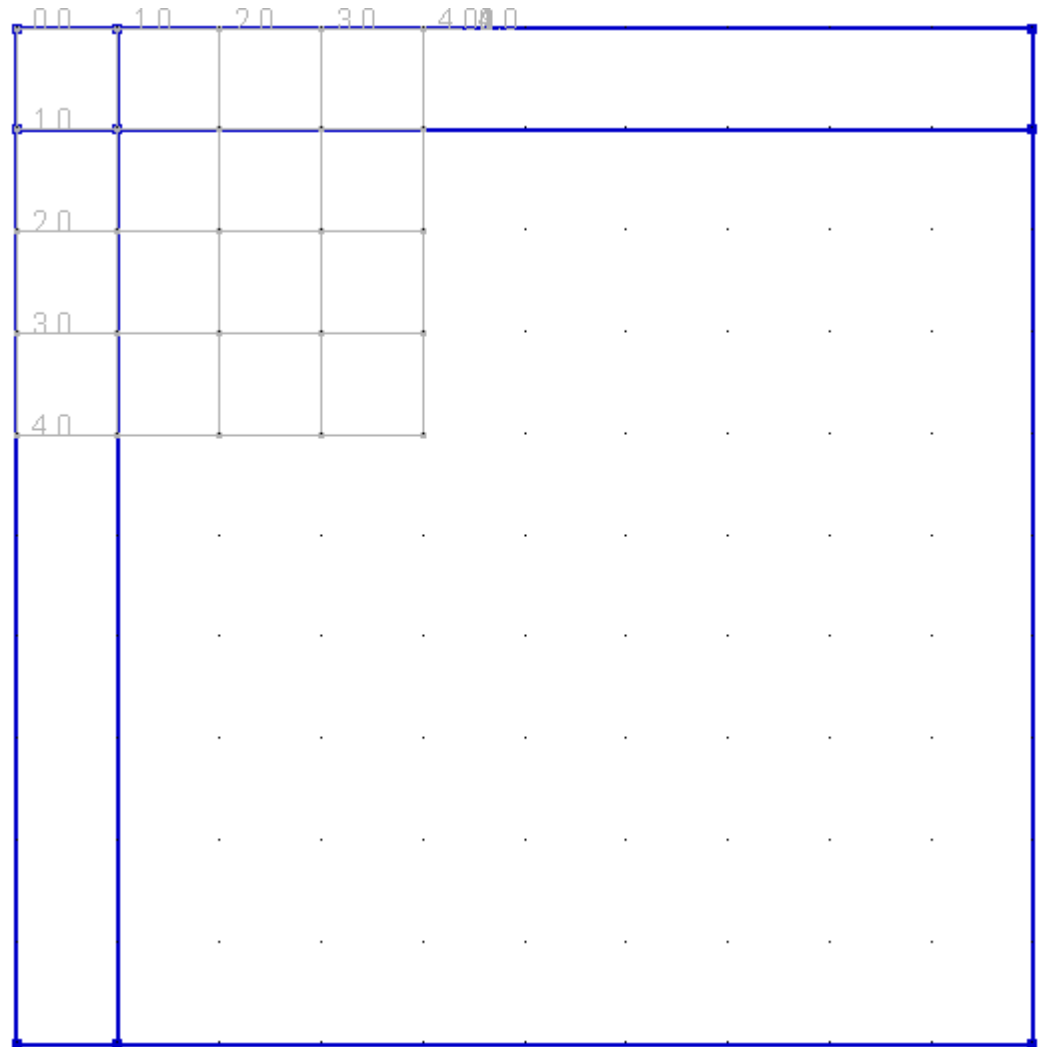
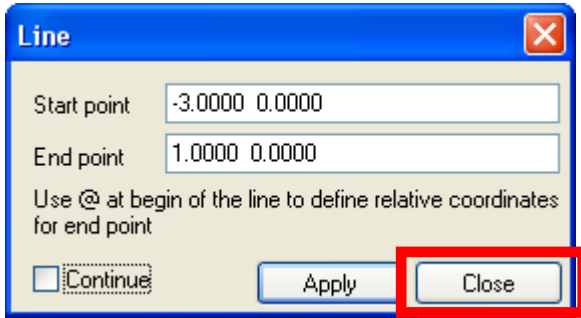


Activate **Run intersection automatically ...** option and press **Yes**

# Create lines

Create line

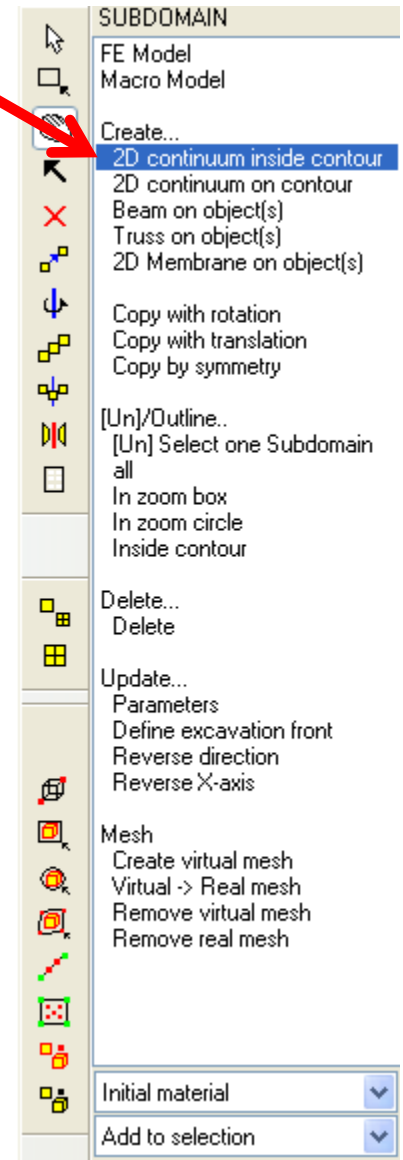
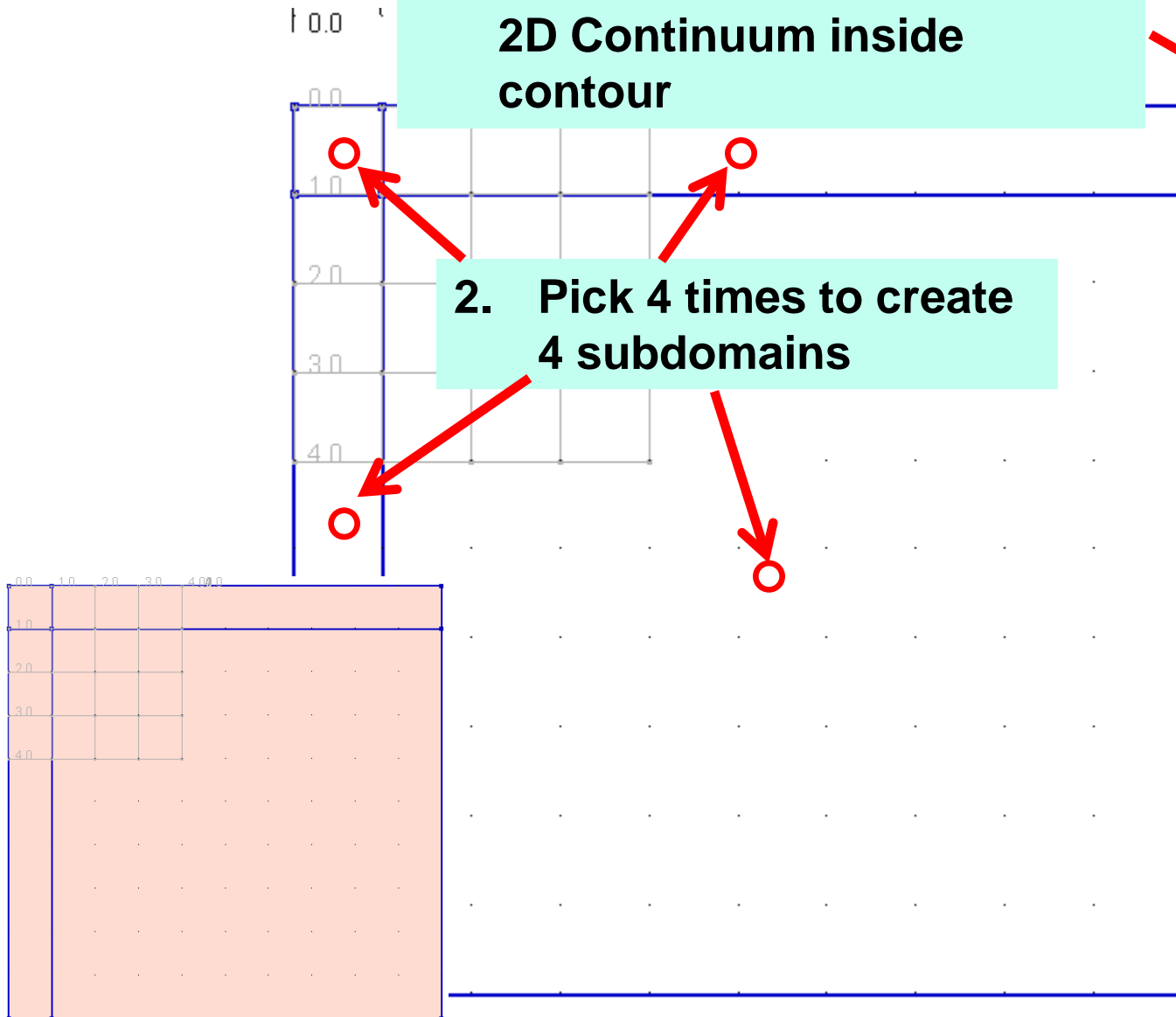
Close Line dialog box



# Create Continuum 2D inside contour

1. Macro Model\Subdomain\  
2D Continuum inside  
contour

2. Pick 4 times to create  
4 subdomains



# Create virtual mesh

1. Macro Model\Subdomain\ Create virtual mesh
2. Pick Subdomain discontinuous
3. Set Split for Edge 1-2 and 1-4
4. Press button Create virtual mesh

The image shows a software interface for creating a virtual mesh. On the left is the 'Meshing parameters' dialog box, and on the right is a 2D mesh visualization.

**Meshing parameters dialog box:**

- Mesh type:** Structured (selected)
- Mapping template:** Quadilateral (selected)
- Control points:** A table with 3 columns and 4 rows of numerical values.
- Unstructured:** Approximate element size on the boundary (0.1 [m]), Division of each subdomain segment (1).
- Through morphing:** Mapping template (Quadilateral), Control nodes (empty).
- Adjust split to existing meshed Subdomains:** Checked.
- Buttons:** Pick, Remove all, Apply, Create virtual mesh (highlighted with a red box), Close.

**Mesh visualization:**

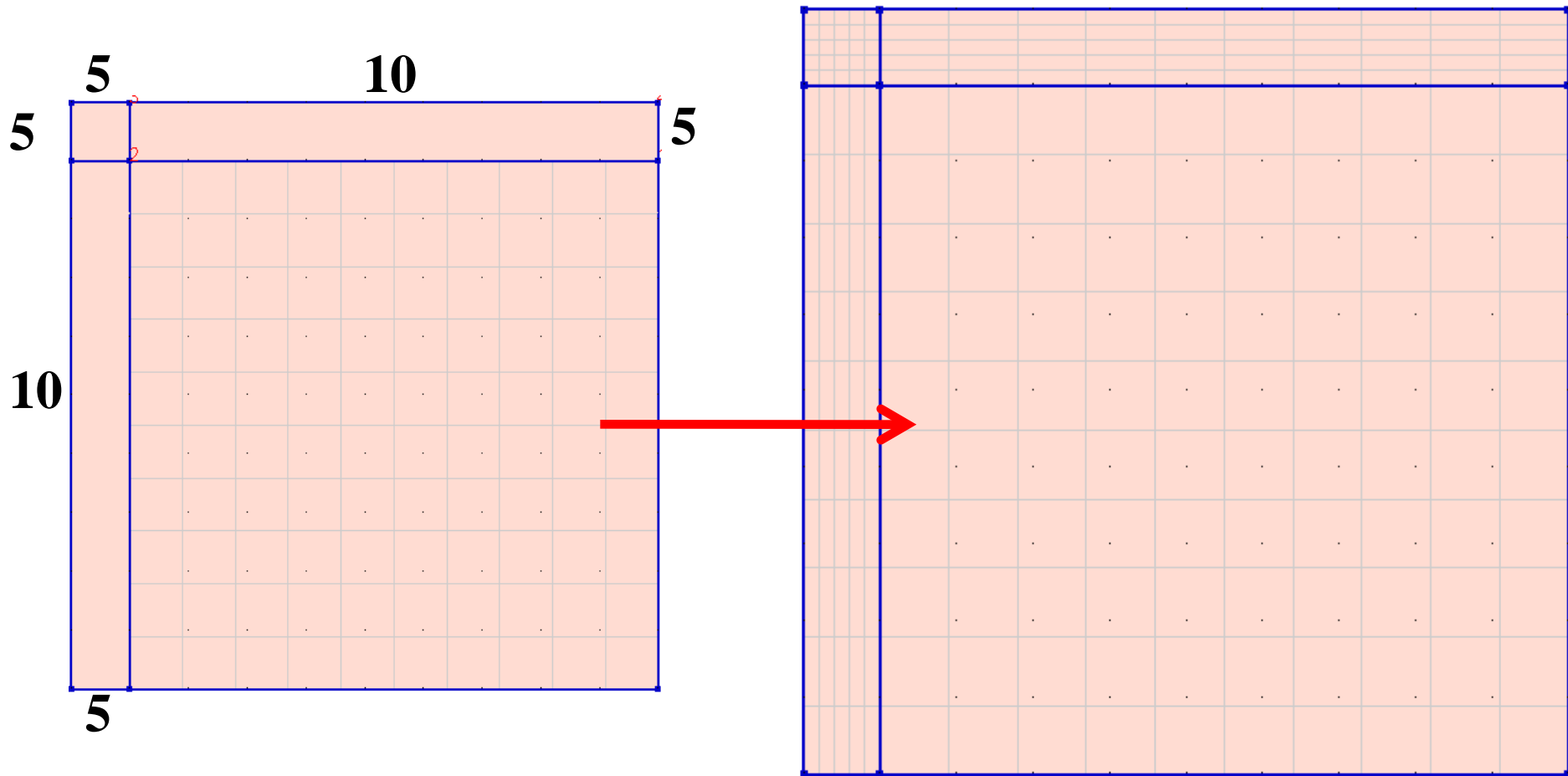
- A 2D rectangular mesh with a grid of nodes and edges.
- The mesh is divided into four quadrants by a vertical line (Edge 1-2) and a horizontal line (Edge 1-4).
- The edges are labeled 1, 2, 3, and 4.
- The 'Create virtual mesh' button in the dialog box is highlighted with a red arrow pointing to the mesh.

**Right-hand menu (SUBDOMAIN):**

- FE Model
- Macro Model
- Create...
  - 2D continuum inside contour
  - 2D continuum on contour
  - Beam on object(s)
  - Truss on object(s)
  - 2D Membrane on object(s)
- Copy with rotation
- Copy with translation
- Copy by symmetry
- [Un]/Outline..
- [Un] Select one Subdomain all
- In zoom box
- In zoom circle
- Inside contour
- Delete...
  - Delete
- Update...
  - Parameters
  - Define excavation front
  - Reverse direction
  - Reverse X-axis
- Mesh
  - Create virtual mesh (highlighted with a blue bar)
  - Virtual -> Real mesh
  - Remove virtual mesh
  - Remove real mesh

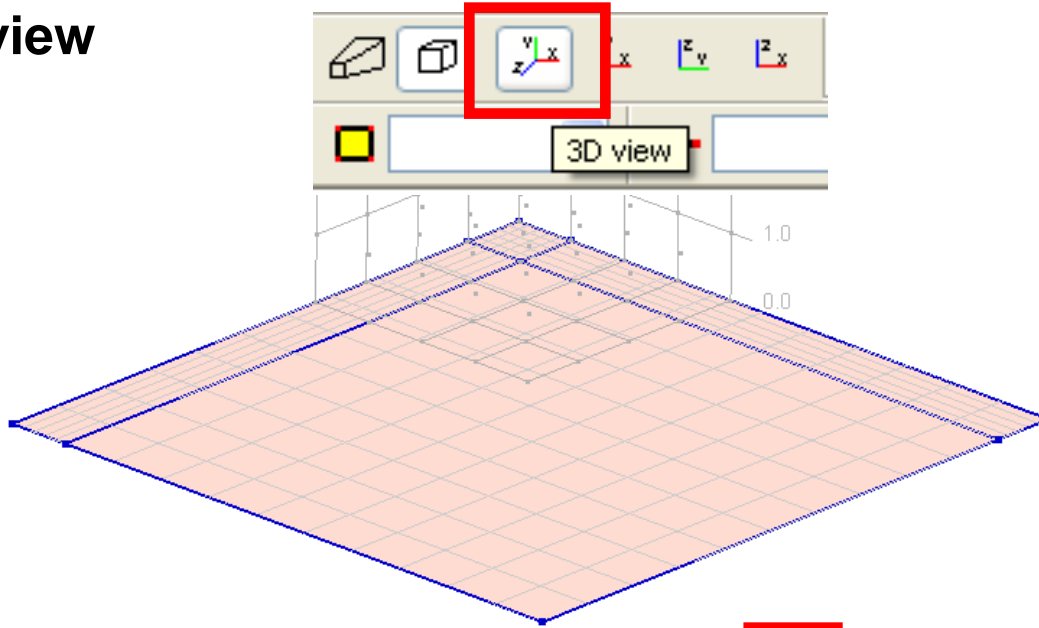
# Create virtual mesh

1. Pick next Subdomain
2. Set Split for edges
3. Press button Create virtual mesh
4. Repeat steps 1-4 for next three subdomains

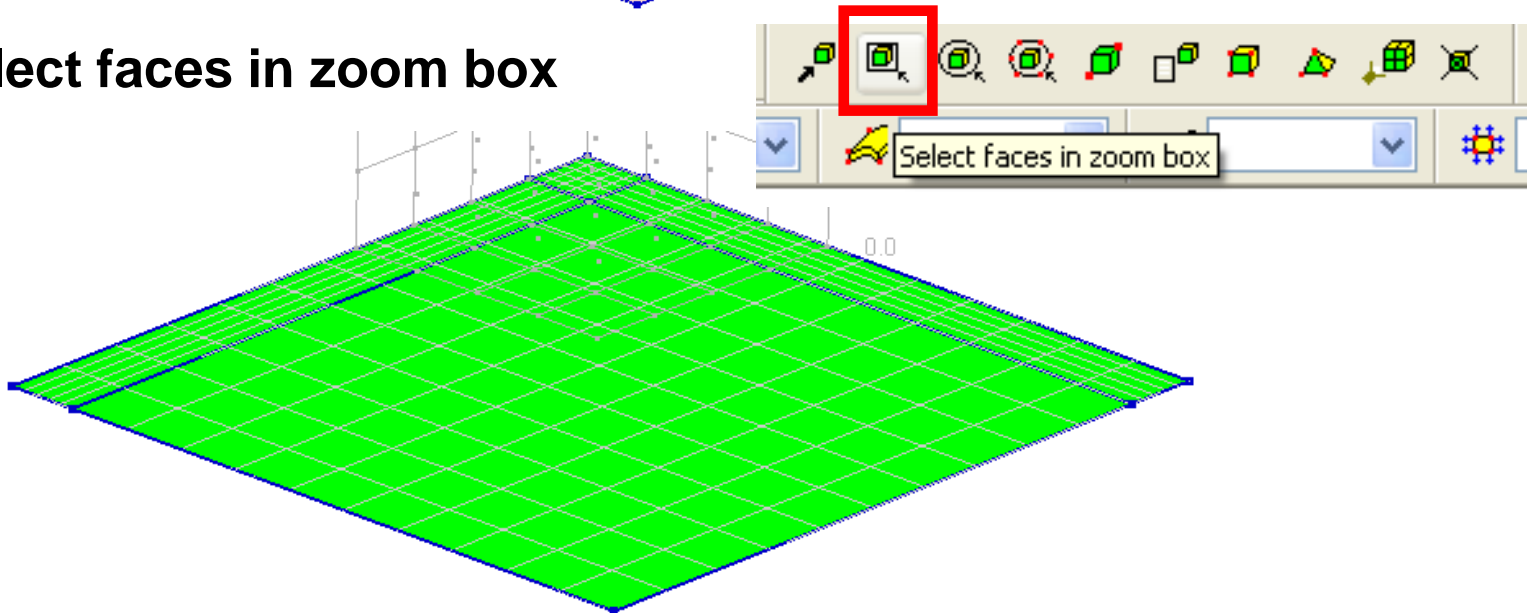


# Create Continuum 3D by faces extrusion

## 1. Set 3D view

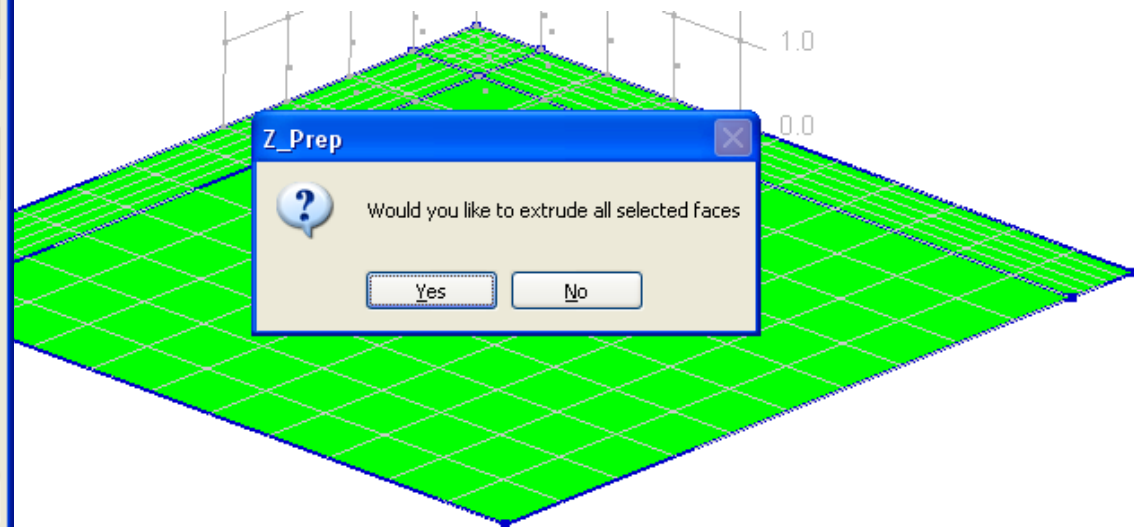
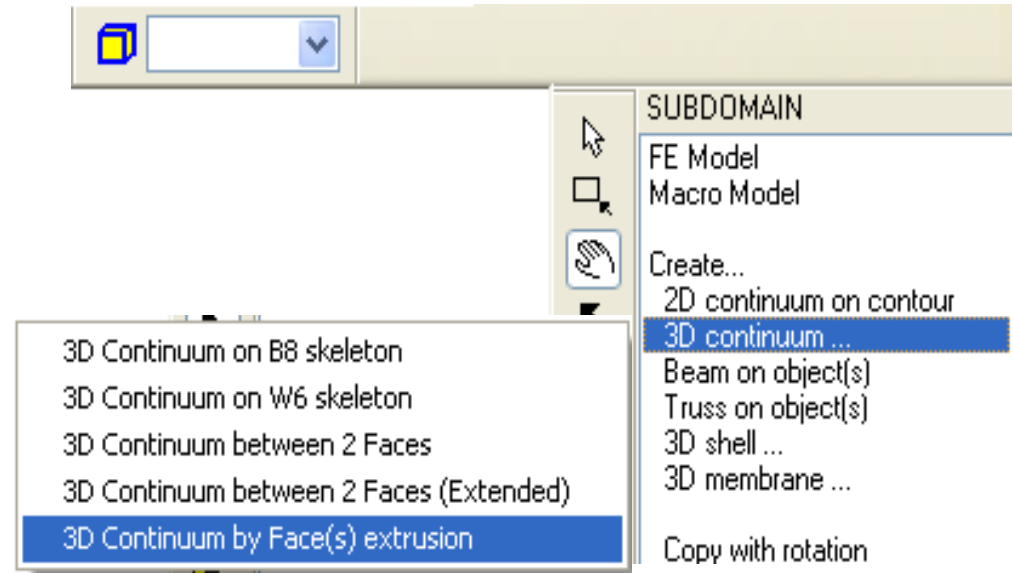
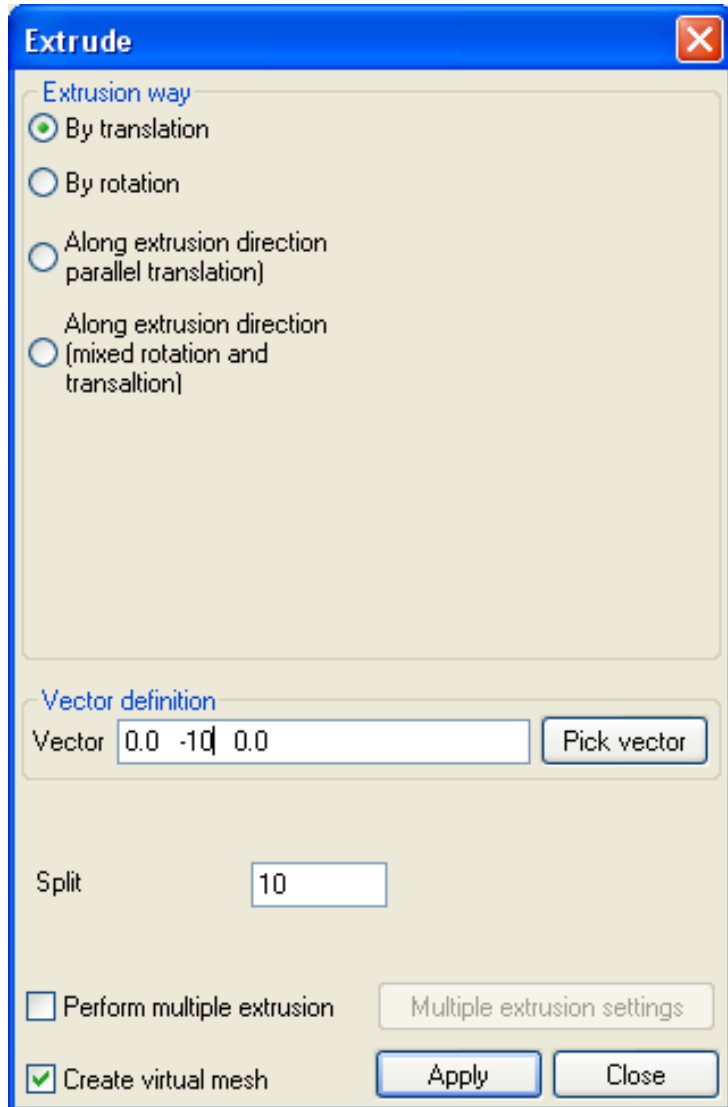


## 2. Select faces in zoom box



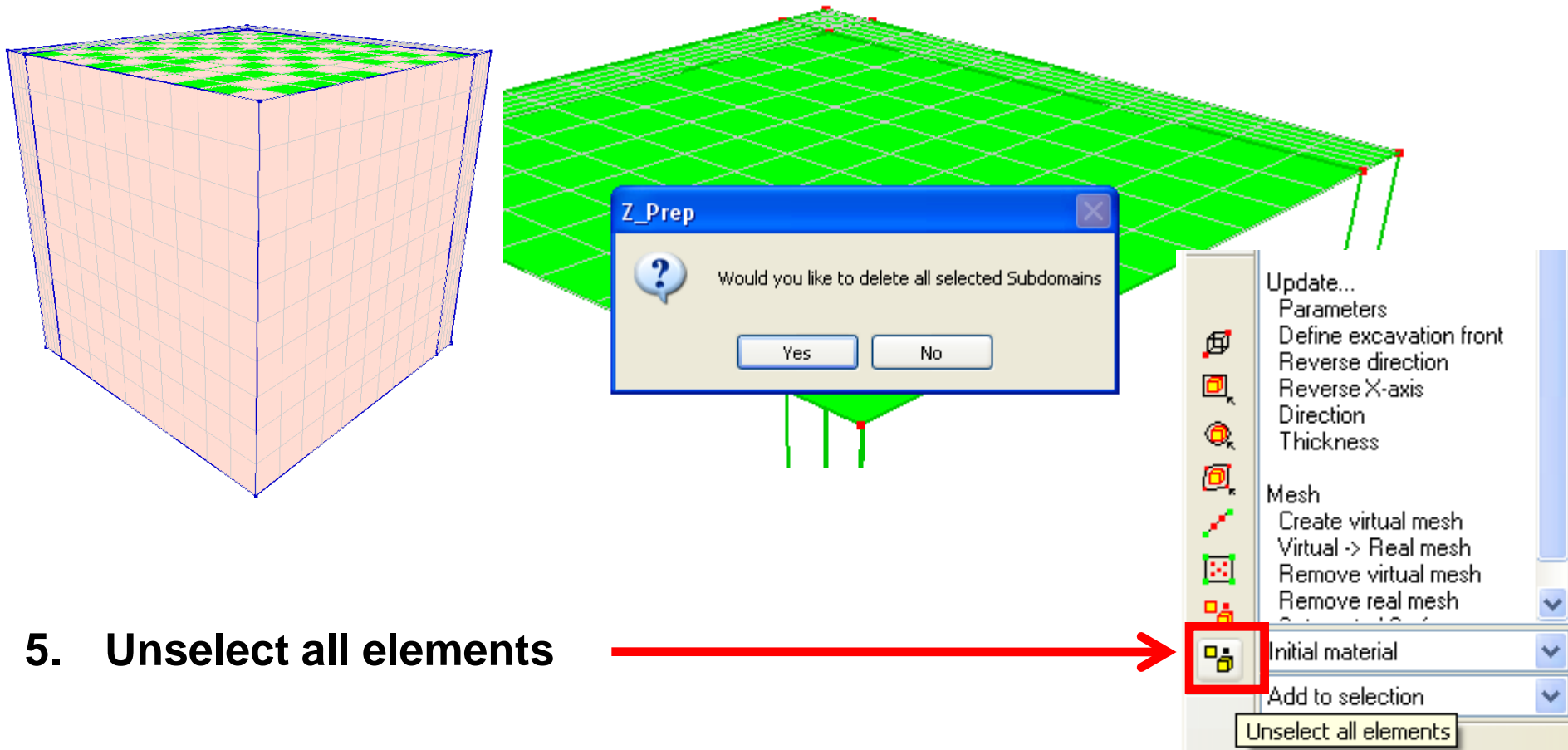
# Create Continuum 3D by faces extrusion

## 3. Macro Model\Subdomain\ 3D Continuum\ 3D Continuum by Face(s) extrusion



# Delete Continuum 2D subdomains

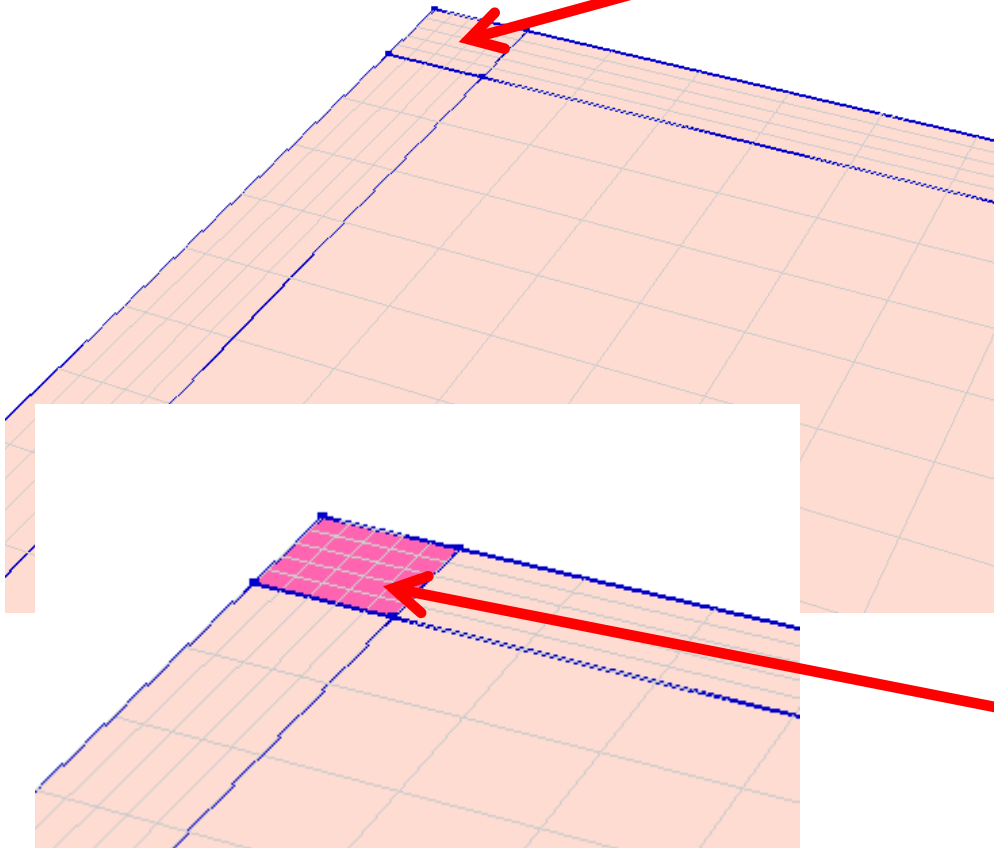
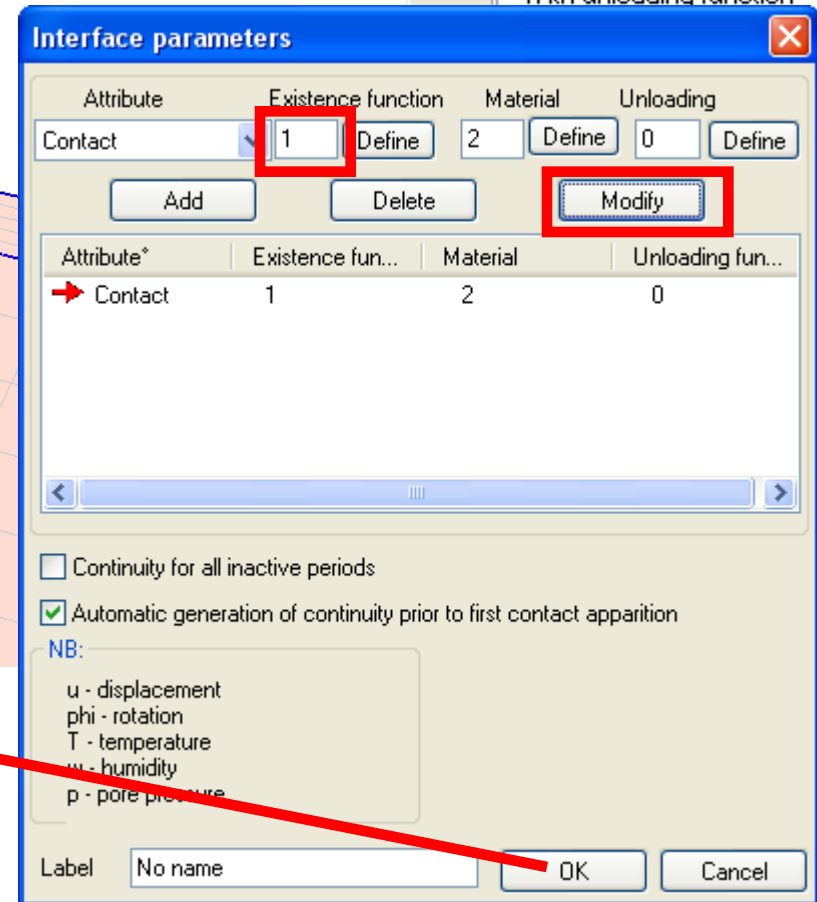
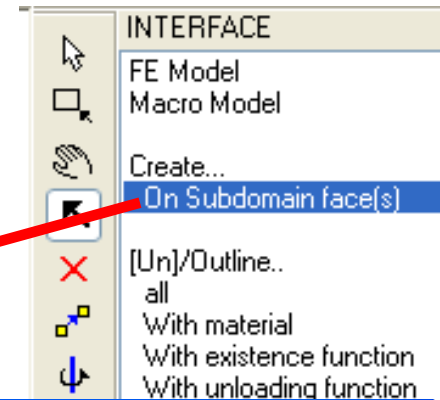
1. CTRL-H – hide Continuum 3D subdomains
2. CTRL-A – select all
3. Delete from menu SUBDOMAIN => Continuum 2D subdomains will be deleted
4. CTRL-H – show Continuum 3D subdomains



5. Unselect all elements

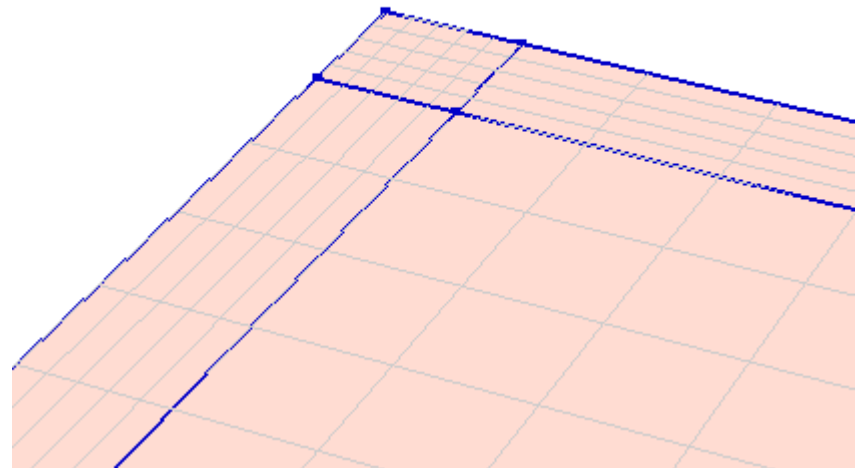
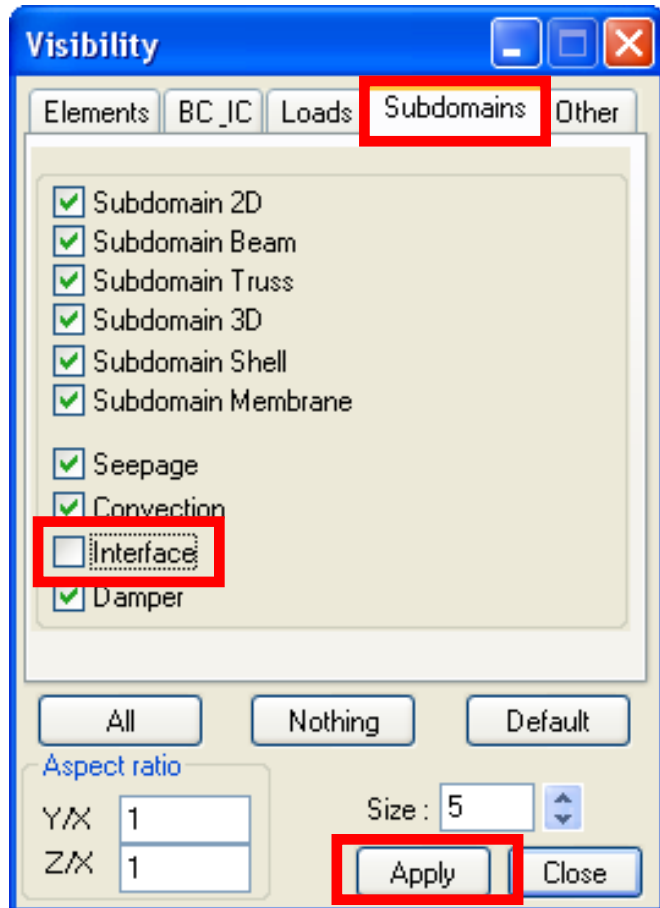
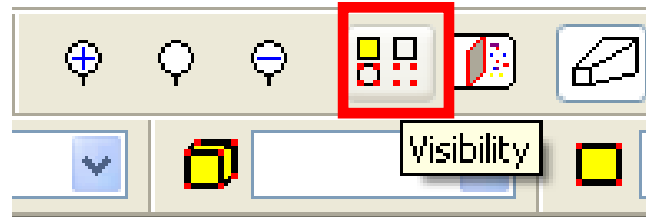
# Definition of Interface elements

1. Change camera orientation and position
2. Macro Model\ Interface\On Subdomain face(s)
3. Pick Subdomain face



# Definition of foundation

1. Pick visibility icon
2. Hide interface elements



# Definition of foundation

1. Macro Model\Subdomain\ 3D shell\3D shell on face(s)
2. Pick Subdomain face

The image shows a software interface for defining a foundation. A 3D model of a structure is shown with a red arrow pointing to a highlighted face. The 'Set parameters' dialog box is open, showing the 'Materials' section with 'Initial material' set to 3, and the 'Functions' section with 'Existence Function' set to 1. The 'Thickness' dialog box is also open, showing the 'Thickness' section with 'Uniform' selected and 'Th = 0.5' [m]. The 'SUBDOMAIN' menu is open, showing the '3D shell on Face(s)' option selected.

**Set parameters**

Materials

Initial material: 3

1-st replacement mat.: 0

2-nd replacement mat.: 0

Functions

Existence Function: 1

Load Function: 0

Cancel

**Thickness**

Thickness

Uniform Th = 0.5 [m]

2 points

3 points

4 points

Label

	Point coordinates (x y z)				Thickness	
P1	0	0	0	[m]	0	[m]
P2	0	0	0	[m]	0	[m]
P3	0	0	0	[m]	0	[m]
P4	0	0	0	[m]	0	[m]

**SUBDOMAIN**

FE Model

Macro Model

Create...

2D continuum on contour

3D continuum ...

Beam on object(s)

Truss on object(s)

3D shell ...

3D membrane ...

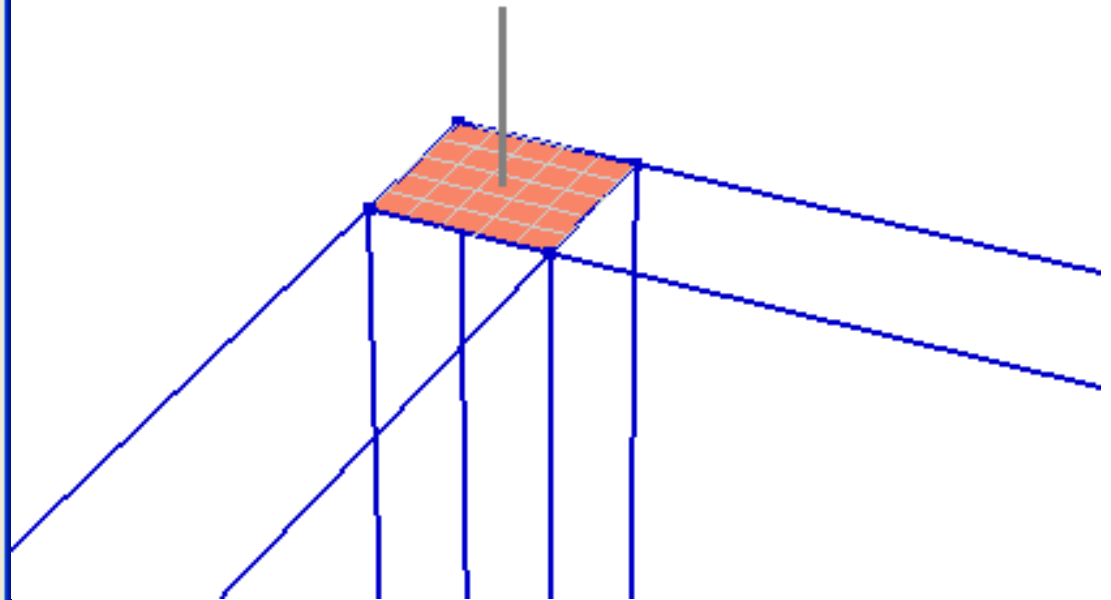
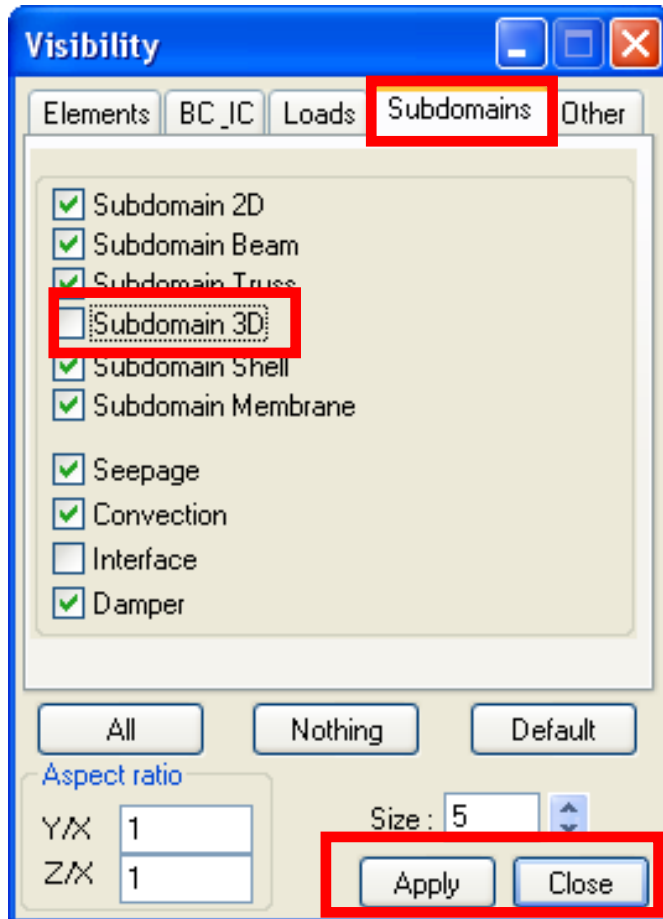
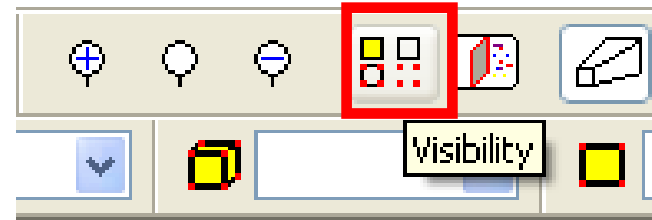
Copy with rotation

Copy with translation

Copy by symmetry

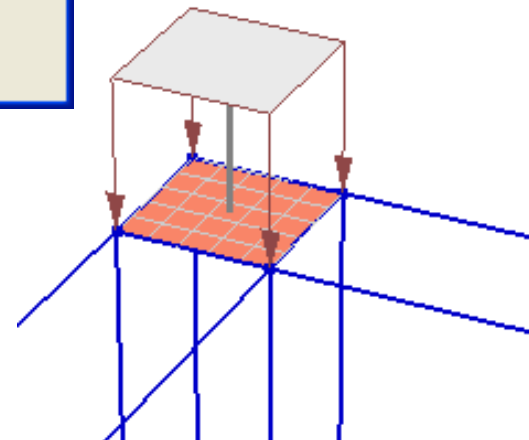
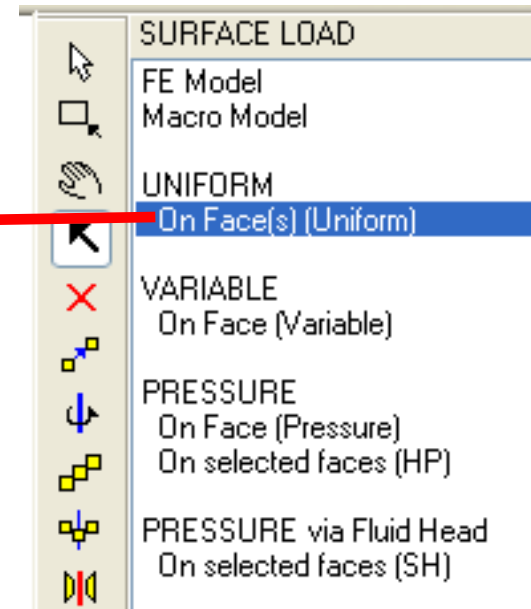
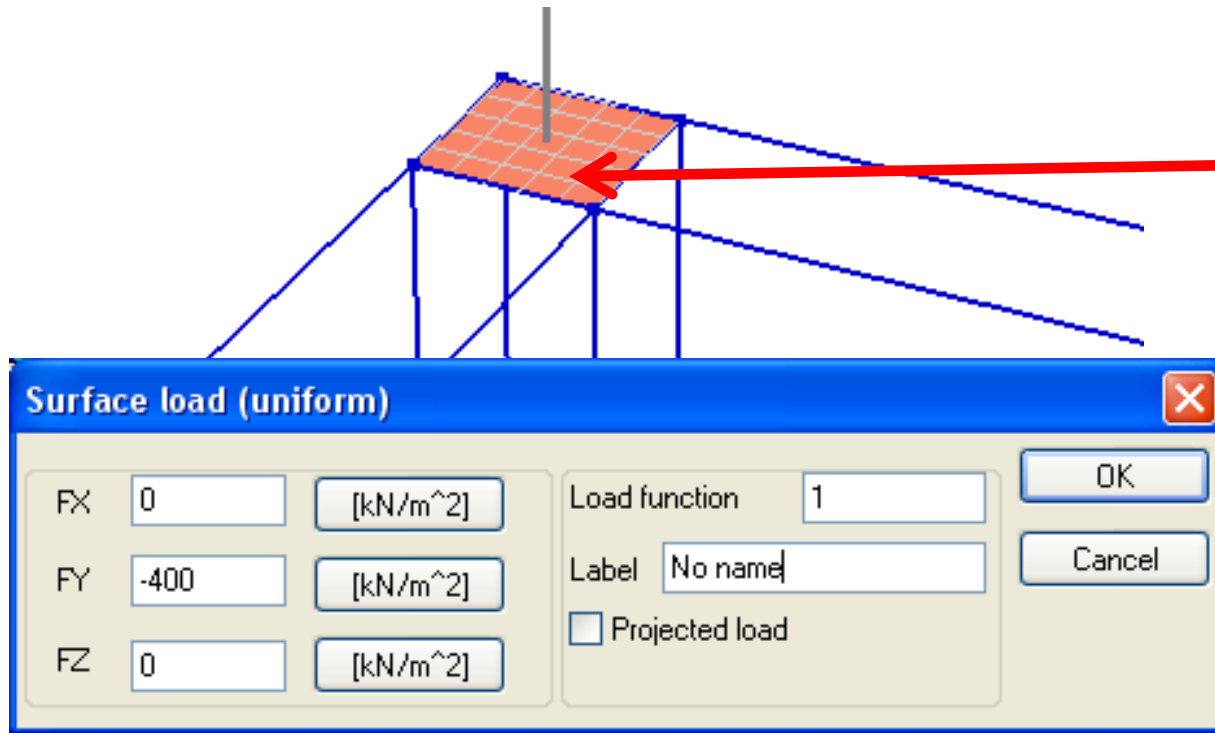
# Definition of surface load

1. Pick visibility icon
2. Hide Subdomain 3D



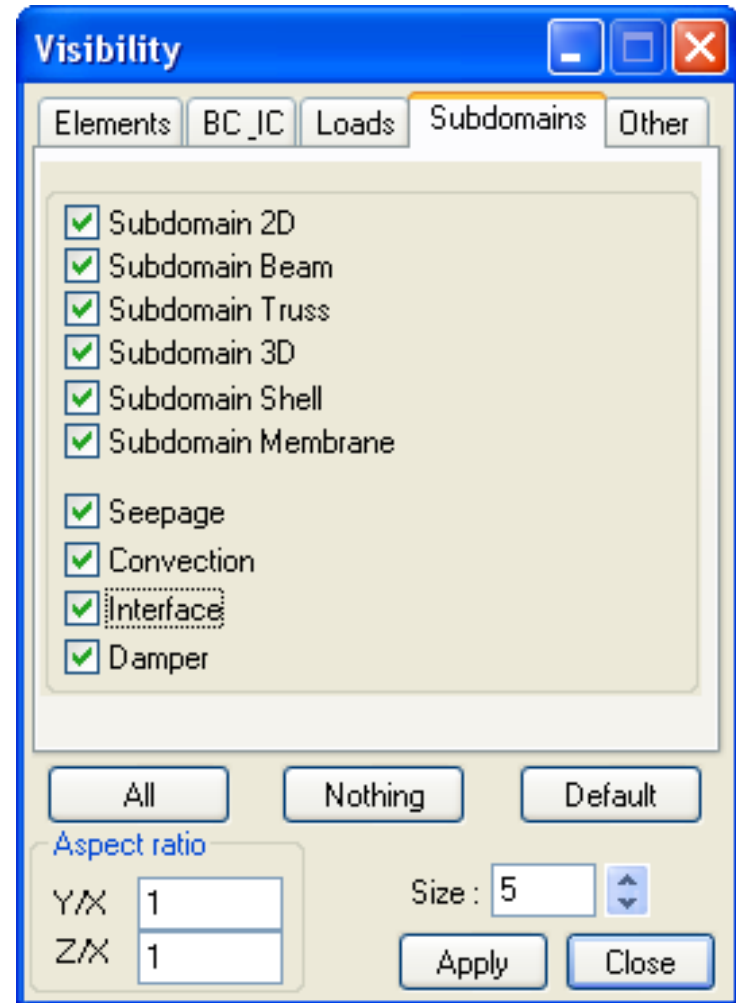
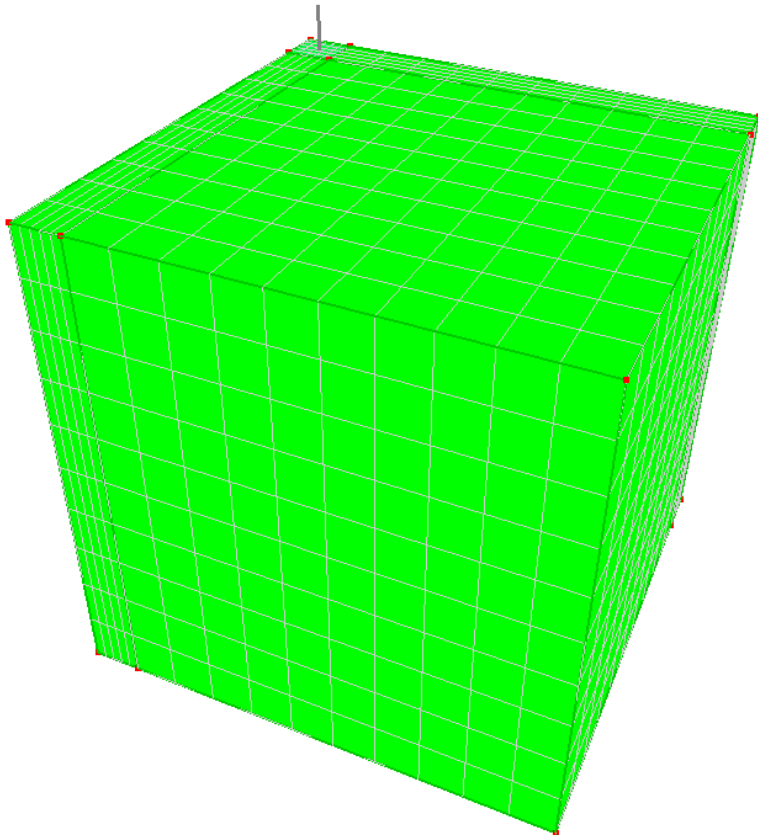
# Definition of surface load

1. Macro Model\Surface load
2. Hide Subdomain 3D



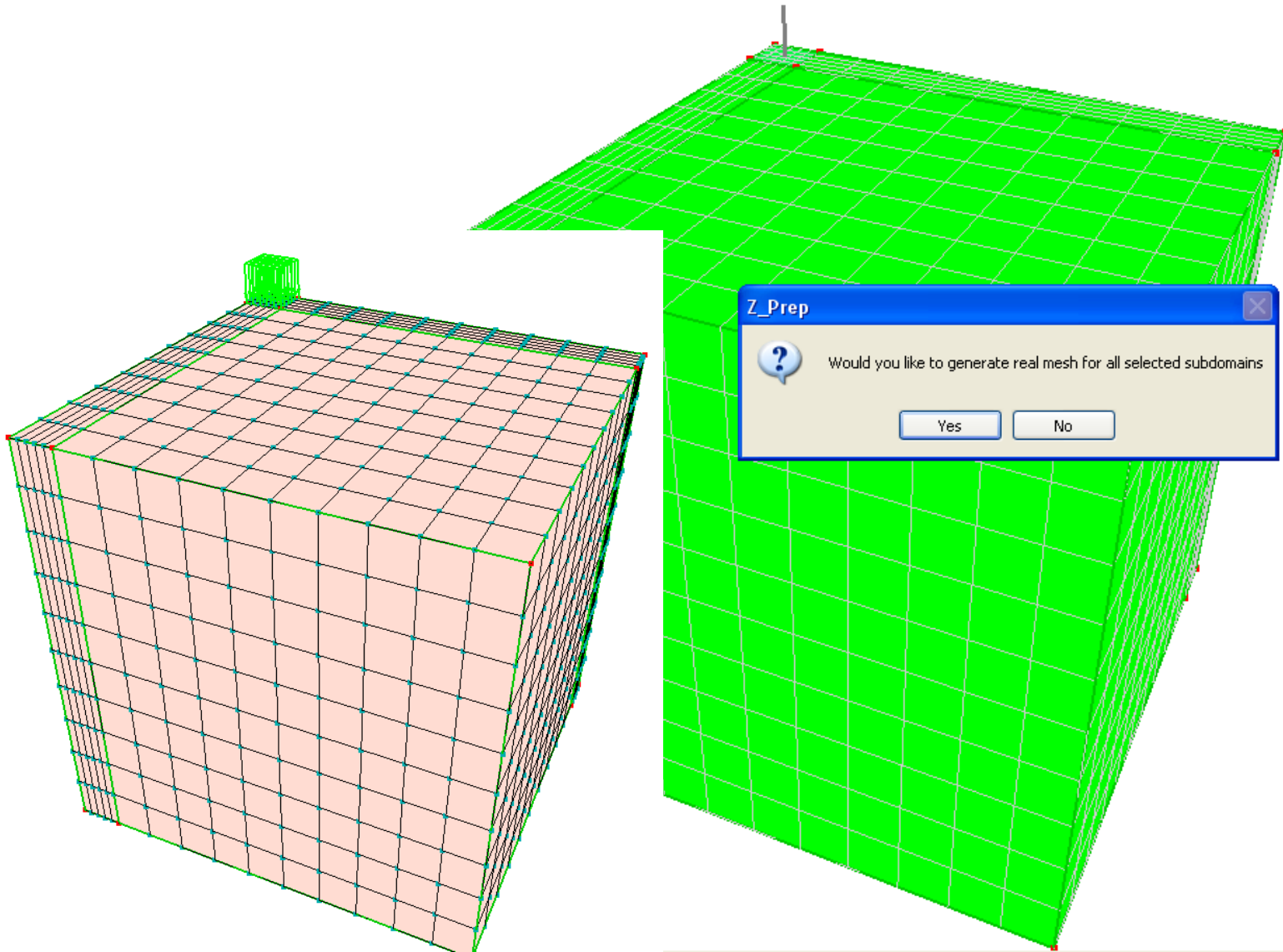
# Real mesh generation

1. Pick visibility icon
2. Show Subdomain 3D
3. Show Interface
4. Ctrl-A – Select all



# Real mesh generation

Macro Model\Subdomain\Virtual->Real mesh



SUBDOMAIN

FE Model  
Macro Model

Create...  
2D continuum on contour  
3D continuum ...  
Beam on object(s)  
Truss on object(s)  
3D shell ...  
3D membrane ...

Copy with rotation  
Copy with translation  
Copy by symmetry

[Un]/Outline..  
[Un] Select one Subdomain  
all  
In zoom box  
In zoom circle  
Inside contour

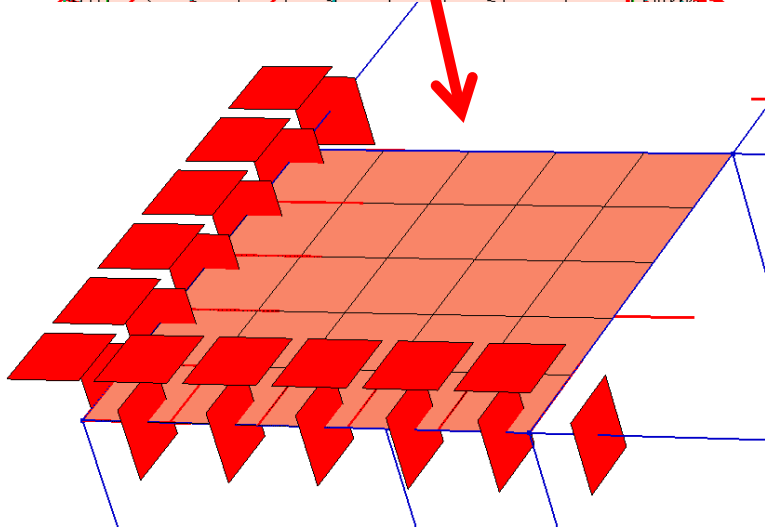
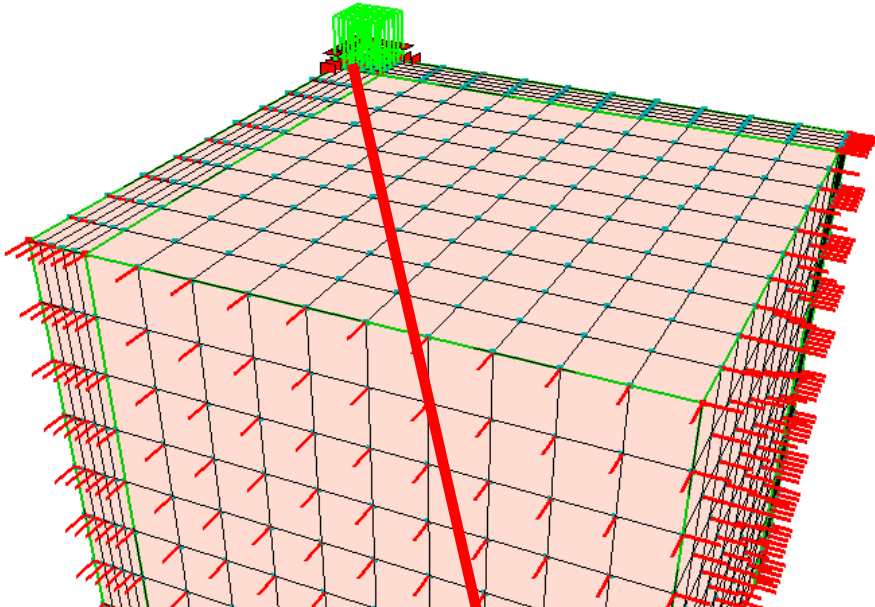
Delete...  
Delete

Update...  
Parameters  
Define excavation front  
Reverse direction  
Reverse X-axis  
Direction  
Thickness

Mesh  
Create virtual mesh  
Virtual -> Real mesh  
Remove virtual mesh  
Remove real mesh  
Set control Surface

# Create Solid boundary conditions

## FE model\Boundary Conditions\Solid BC\On box



SOLID BC

FE Model  
Macro Model

Create...

- On node
- On element node

Add BC

**On box**

[Un]/Outline..

- all
- between 2 Nodes
- between 4 Nodes
- Box with 2 Nodes
- In zoom box
- In zoom circle
- Inside contour
- With load function
- With existence function
- With label

FE MODEL

DOMAIN

Macro Model

Node

- Beam
- Anchor
- Continuum 2D
- Continuum 3D
- Shell
- Shell one layer
- Membrane
- Infinite element

Interface

- Interface (large deformation)
- Seepage
- Convection
- Viscous damper
- Shell Hinge

**Boundary Conditions**

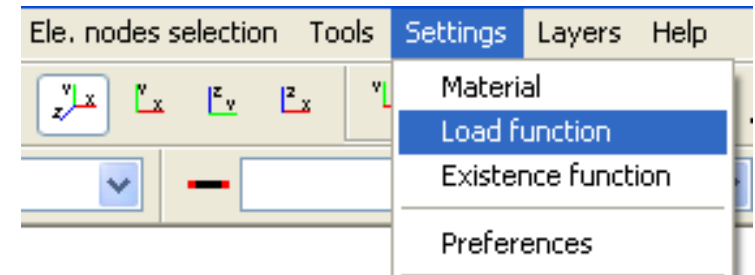
- Loads
- Added masses
- Distributed Fluxes
- Initial Conditions
- DRM domains

**Solid BC**

- Temperature BC
- Humidity BC
- Pressure BC

Periodic BC

# Load function



### Load function

Function definition

Number  Name

Function

	Time	Value
1	0.0000	0.0000
2	2.0000	0.0000
3	3.0000	1.0000

1.00

Value

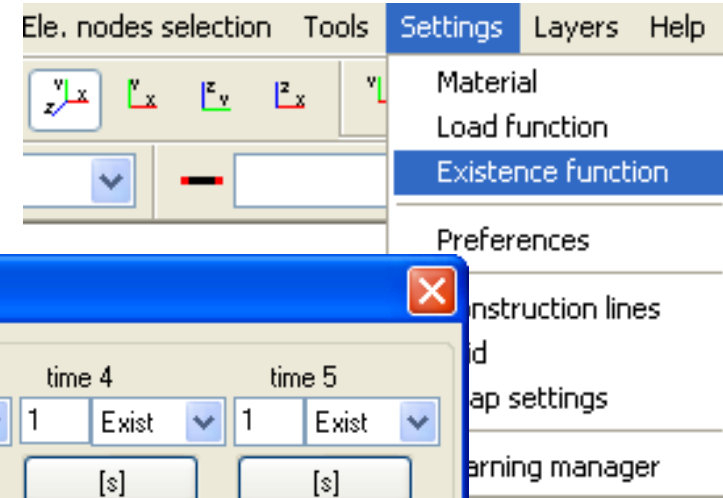
0.00

0.00

3.00

Time

# Existence function



### Existence functions

Function definition

Number	Name	time 1	time 2	time 3	time 4	time 5
1	No name	1 Exist [s]	1 Exist [s]	1 Exist [s]	1 Exist [s]	1 Exist [s]

Buttons: Add, Replace, Delete

N*:name	T 1	Exist...	T 2	Exist...	T 3	Exist...	T 4	Exist...	T 5	Exist...
→ 1 : No name	1.0000	exist	1.0000	exist	1.0000	exist	1.0000	exist	1.0000	exist

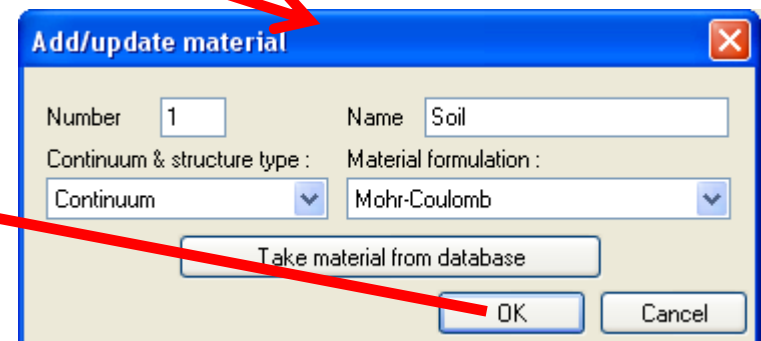
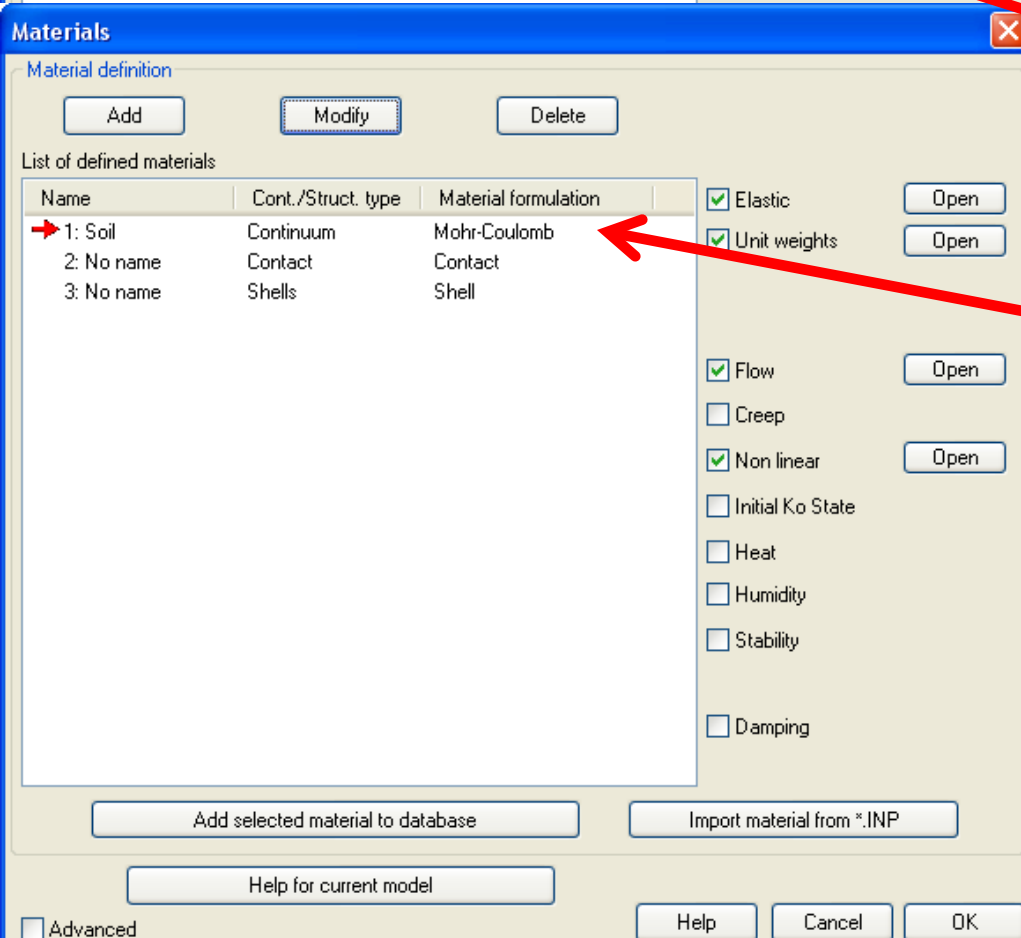
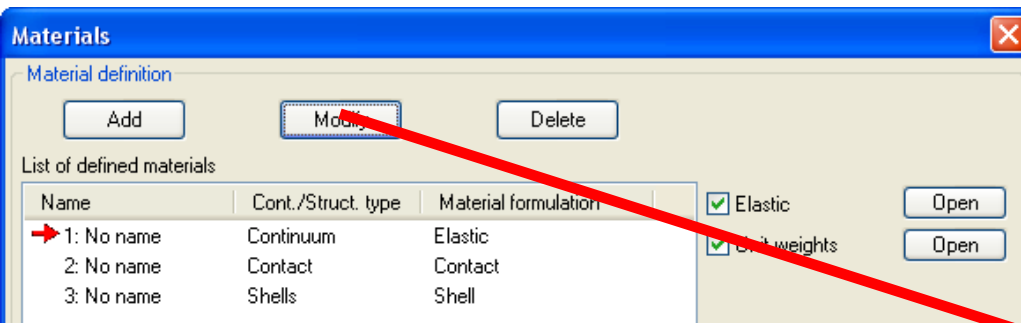
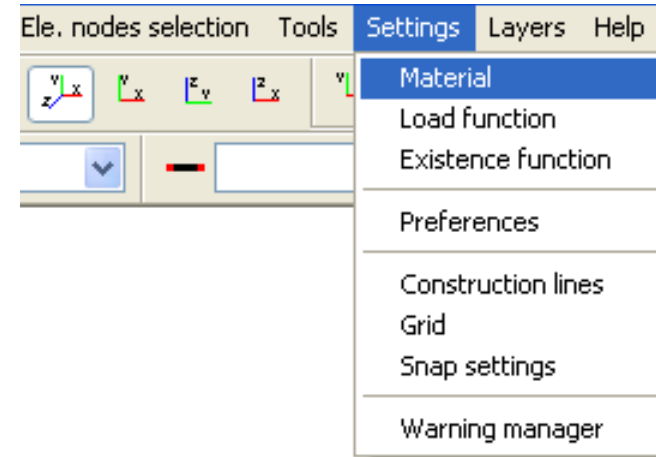
Value

Time

0.00 1.00

Buttons: Export, Import, Automatic generation, Help, OK, Cancel

# Define Materials for soil



# Define material parameters for soil

The image shows a sequence of dialog boxes for defining material parameters for soil. The main dialog is 'Materials', which lists defined materials and has checkboxes for various material models. A red arrow points from the 'Soil' entry in the 'Materials' dialog to the 'Material' dialog. Another red arrow points from the 'Unit weights / masses' dialog to the 'Material' dialog. A third red arrow points from the 'Mohr-Coulomb' dialog to the 'Material' dialog.

**Materials** (Material definition)

Buttons: Add, Modify, Delete

List of defined materials:

Name	Cont./Struct. type	Material formulation
1: Soil	Continuum	Mohr-Coulomb
2: No name	Contact	Contact
3: No name	Shells	Shell

Checkboxes:

- Elastic
- Unit weights
- Flow
- Creep
- Non linear
- Initial Ko State
- Heat
- Humidity
- Stability
- Damping

Buttons: Add selected material to database, Import material from \*.INP, Help for current model, Help, Cancel, OK

**Material**

Elastic parameters

Young modulus E: 60000 [kN/m<sup>2</sup>]

Poisson ratio  $\nu$ : 0.3

Data mode: Standard

Buttons: OK, Cancel, Help

**Unit weights / masses**

Unit weights/masses

Weight / unit volume  $\gamma$ : 20 [kN/m<sup>3</sup>]

Mass / unit volume  $\rho$ : 2039.4324 [kg/m<sup>3</sup>]

Fluid weight / unit volume  $\gamma_F$ : 10 [kN/m<sup>3</sup>]

Fluid mass / unit volume  $\rho_F$ : 1019.7162 [kg/m<sup>3</sup>]

Initial void ratio  $e_0$ : 0

Global gravity direction

Buttons: OK, Cancel, Help

(Single phase) analysis in total stresses specify the saturated weight  $\gamma_{SAT}$  below the water table and the dry specific weight  $\gamma_D$  above; for stress analysis specify buoyant weight  $\gamma_B = \gamma_{SAT} - \gamma_W$ ; for flow (two-phase) specify  $\gamma_D$  (dry) and initial void ratio  $e_0$ ; defined as  $\gamma = \gamma_D + n S \gamma_w$ ;  $n = e_0 / (1 + e_0)$ ;  $S$  - saturation ratio.

Data mode: Standard

Buttons: OK, Cancel, Help

**Mohr-Coulomb**

Yield

Cohesion  $c$ : 10 [kN/m<sup>2</sup>]

Friction angle  $\phi$ : 20 [deg]

Plastic Flow

Dilatancy angle  $\psi$ : 5 [deg]

Buttons: Help, OK, Cancel

Data mode: Standard

Buttons: Help, OK, Cancel

# Define materials for interface

**Materials**

Material definition

Add Modify Delete

List of defined materials

Name	Cont./Struct. type	Material formulation
1: Soil	Continuum	Mohr-Coulomb
→ 2: No name	Contact	Contact
3: No name	Shells	Shell

Elastic  
 Unit weights  
 Flow  
 Non linear

Open Open Open Open Open

**Nonlinear Contact**

Direct input  
 Friction angle  [deg]

Cohesion  [kN/m<sup>2</sup>]

Inherit data from adjacent continuum materials

Setup

Advanced Data mode

OK Cancel Help

**Contact properties**

	Material	Model	Original continuum strength parameters			Standard Interface data			Piles interface data			Tensile bearing capacity qt [kN/m <sup>2</sup> ]		Compressive bearing capacity qc [kN/m <sup>2</sup> ]		Nails/Fixed anchor zone interface data
			Phi [deg]	c [kN/m <sup>2</sup> ]	Psi [deg]	tg(Phi) mult.	c mult.	tg(Psi) mult.	tg(Phi) mult.	c mult.	tg(Psi) mult.					Ultimate shear stress tau [kN/m <sup>2</sup> ]
1	Soil	Mohr-Coulom	30	20	10	0.6	0	0	0.6	0	0	0	1e+038		1e+038	

Advanced

OK Cancel

# Define materials for shell

**Materials**

Material definition

Add Modify Delete

List of defined materials

Name	Cont./Struct. type	Material formulation
1: Soil	Continuum	Mohr-Coulomb
2: No name	Contact	Contact
→ 3: No name	Shells	Shell

Elastic  
 Unit weights  
 Flow  
 Non linear

Open Open Open

**Material**

Elastic parameters

Young modulus E 27e6 [kN/m<sup>2</sup>]  
Poisson ratio  $\nu$  0.2 []

Data mode Standard

OK Cancel Help

**Unit weights /masses**

Unit weights/masses

Weight / unit volume  $\gamma$  24 [kN/m<sup>3</sup>]  
 $\gamma$

Fluid weight / unit volume  $\gamma_F$  10 [kN/m<sup>3</sup>]  
Initial void ratio  $e_0$  0 []

Mass / unit volume  $\rho$  0 [kg/m<sup>3</sup>]  
 $\rho$

Fluid mass / unit volume  $\rho_F$  1019.7162 [kg/m<sup>3</sup>]

Global gravity direction

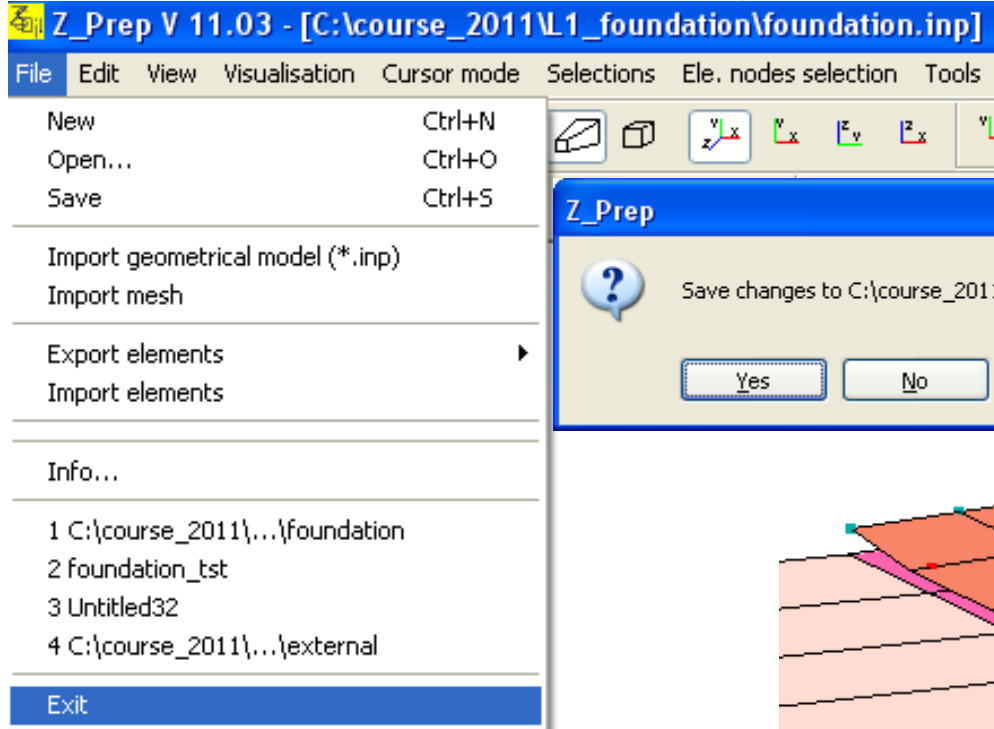
**N.B.**  
for deformation (single phase) analysis in total stresses specify the saturated specific weight  $\gamma_{SAT}$  below the water table and the dry specific weight  $\gamma_D$  above;  
in effective stress analysis specify buoyant weight  $\gamma_B = \gamma_{SAT} - \gamma_w$ ;  
for deformation+flow (two-phase) specify  $\gamma_D$  (dry) and initial void ratio  $e_0$ ;  
 $\gamma$  will be computed as  $\gamma = \gamma_D + n S \gamma_w$ ;  $n = e_0 / (1 + e_0)$ ; S - saturation ratio.

Data mode Standard

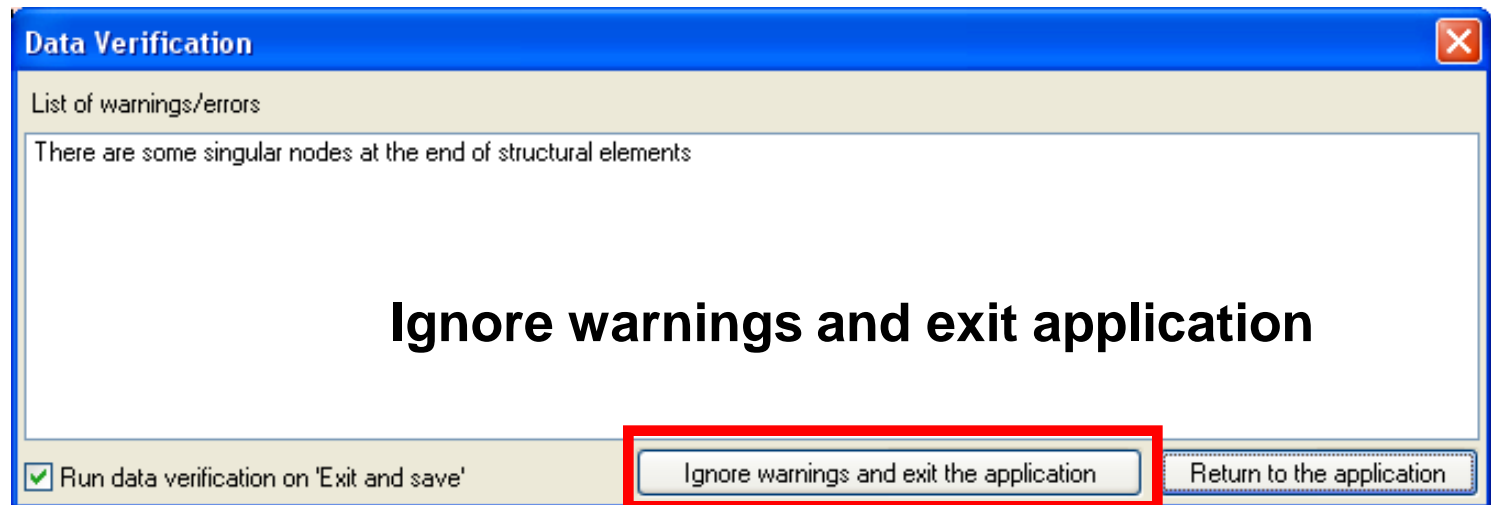
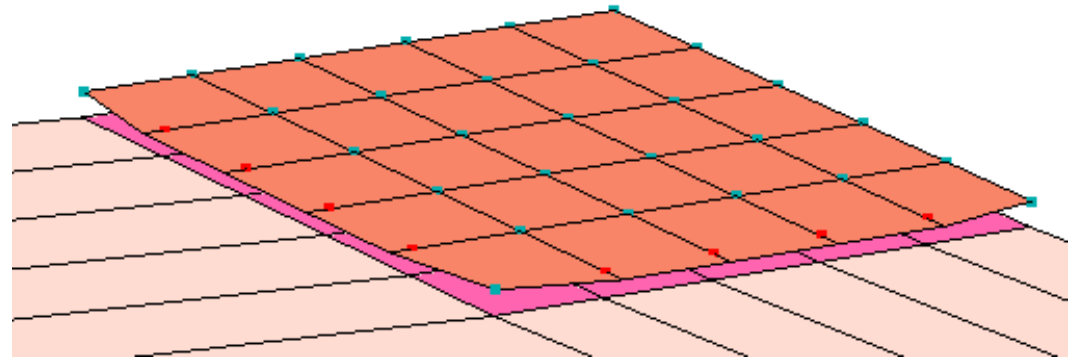
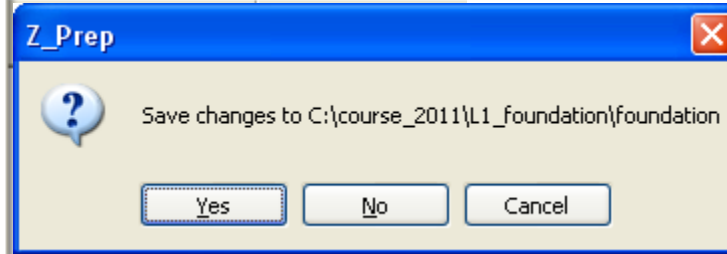
Add selected material to database  
Help for current model  
 Advanced

OK Cancel Help

# Close preprocessing



**FILE\Exit -  
with saving changes**



**Ignore warnings and exit application**

# Define Drivers

**Analysis and Drivers definition**

Drivers definition

Driver: Time Dependent | Type: Driven Load | Time start: 2 | Time end: 3 | Time incr.: 0.1 | Multiplier: 1 | Nonl. solver settings: Default

[day] Set

N°	Driver	Type	Time start	Time end	Increment	Multiplier	Nonl. solver se...	Dyn. anal. setti...
1	Initial State		0.5000	1.0000	0.5000		Default	
2	Time Dependent	Driven Load	0	2.0000	1.0000	1.0000	Default	
3	Time Dependent	Driven Load	2.0000	3.0000	0.1000	1.0000	Default	

Add Insert Delete Modify

Analysis & Problem type

Analysis: 3D | Problem: Deformation

Associated preprocessed projects:

Heat project: [ ] Browse...

Humidity project: [ ] Browse...

Domain Reduction for transient dynamics

Free field motion project: [ ] Browse...

Advanced

OK Cancel Help

# Postprocessing – Results maps

The image shows a software interface for postprocessing results. The main window is titled "Maps" and contains several panels and a 3D visualization.

**Graph Option Panel (Left):**

- View data
- Deformed mesh
- Displacement vectors
- Plastic zones
- Principal stresses
- Principal moments/memb. forces
- Fluid velocities
- Maps** (selected)
- Sectional quantities
- Element time history
- Nodal time history
- MNT for continuum 3D
- Results in Beams/Trusses/Piles/Nails/Ar
- Eigenmodes
- Push over
- Save settings
- Read settings
- Default settings
- Parametric analysis

**Settings Panel (Middle):**

- Graph contents
- Local coord. system for
- Averaging
- T1 axis for sections 3D
- Legend
- Units
- Reset MIN/MAX to visible results

**Maps Panel (Right):**

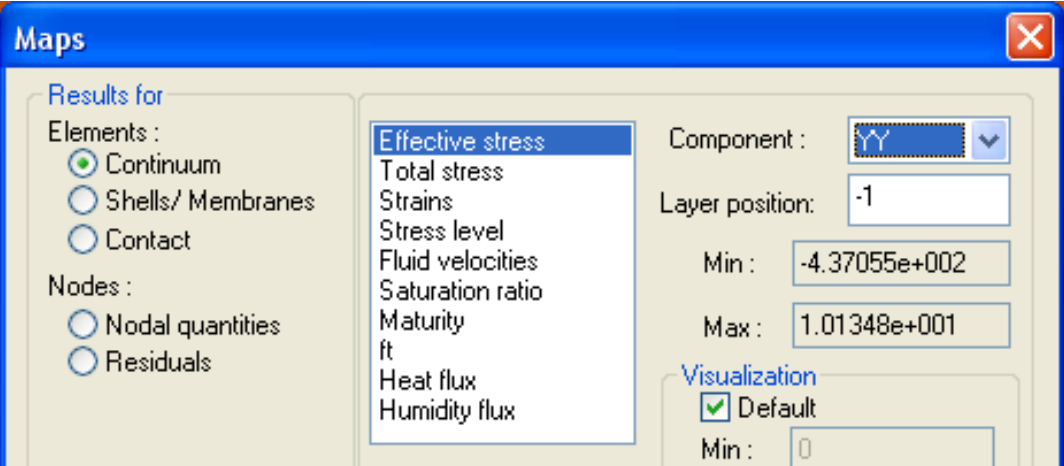
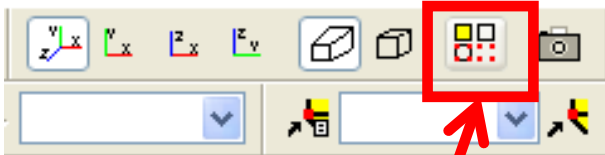
- Results for
- Elements:
  - Continuum
  - Shells/ Membranes
  - Contact
- Nodes:
  - Nodal quantities
  - Residuals
- Effective stress (selected)
- Total stress
- Strains
- Stress level
- Fluid velocities
- Saturation ratio
- Maturity
- ft
- Heat flux
- Humidity flux

- Component: YY
- Layer position: YY
- Min: -4.3
- Max: 1.01
- Visualization:
- Default
- Min: 0
- Max: 0
- Step: 20
- Deformation visibility:
- Deformed mesh
- Displ. vectors
- Settings
- Cancel
- OK

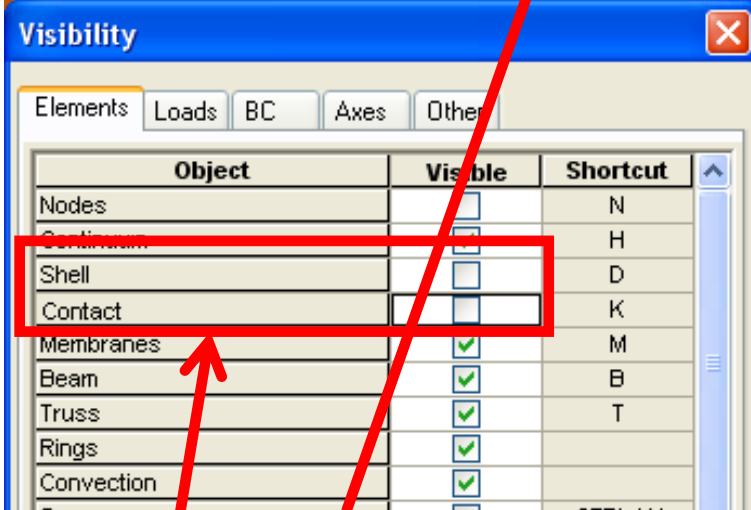
**3D Visualization:**

The 3D visualization shows a stress map of a structure, likely a bridge pier or similar. The structure is colored according to the stress distribution, with a color scale ranging from blue (low stress) to red (high stress). The highest stress is concentrated in the central vertical column, with a peak value of 1.01. The stress decreases towards the top and bottom of the column, and is also lower in the surrounding structure. The color scale is: Blue, Green, Yellow, Orange, Red.

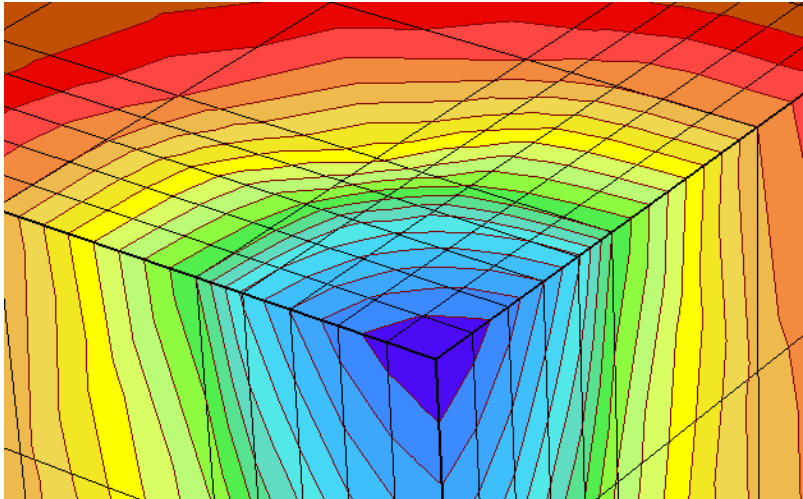
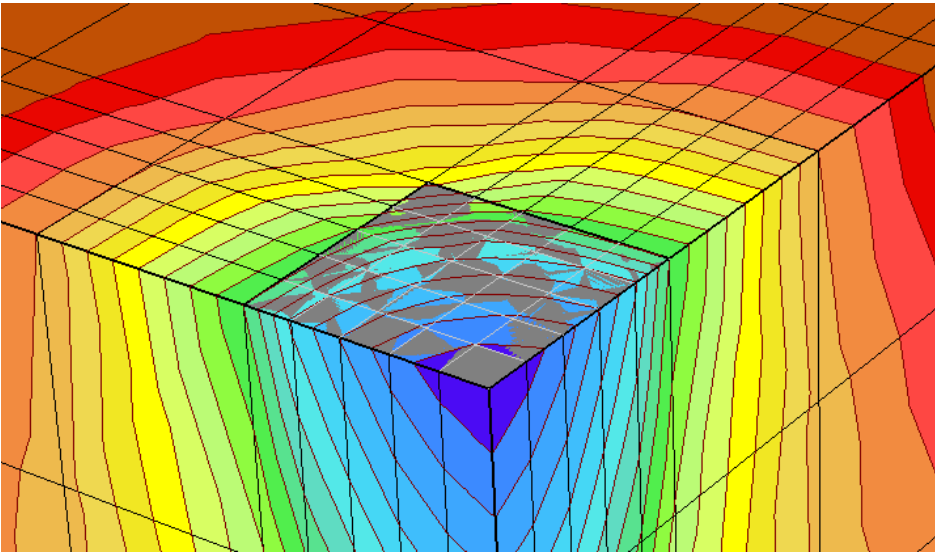
# Postprocessing – Results maps



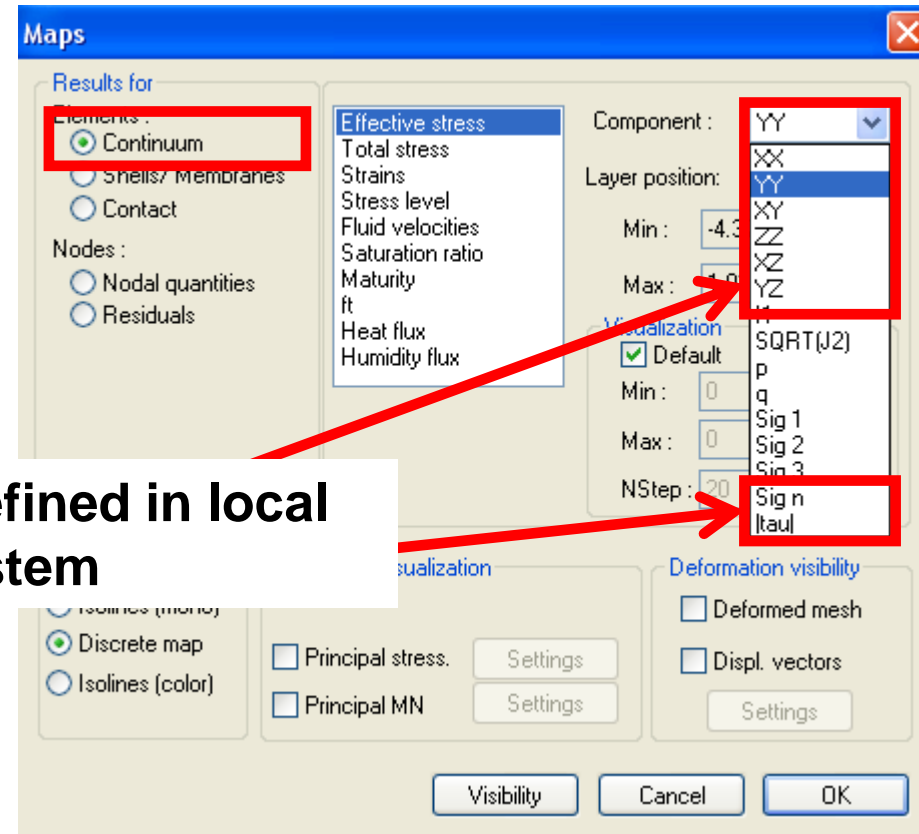
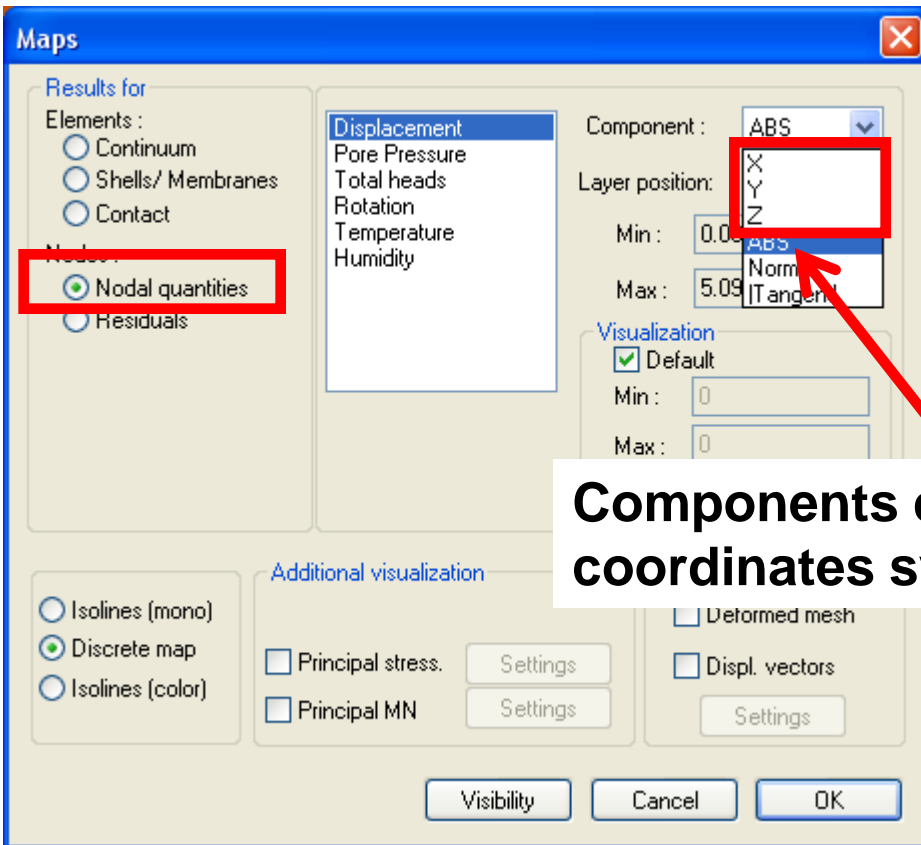
**Results for Continuum selected  
No results for shells and contact  
elements**



**Open Visibility dialog box  
Turn off elements with no  
active results**

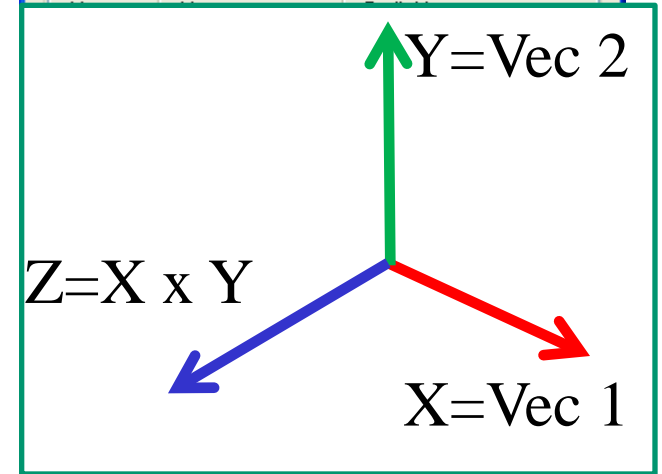
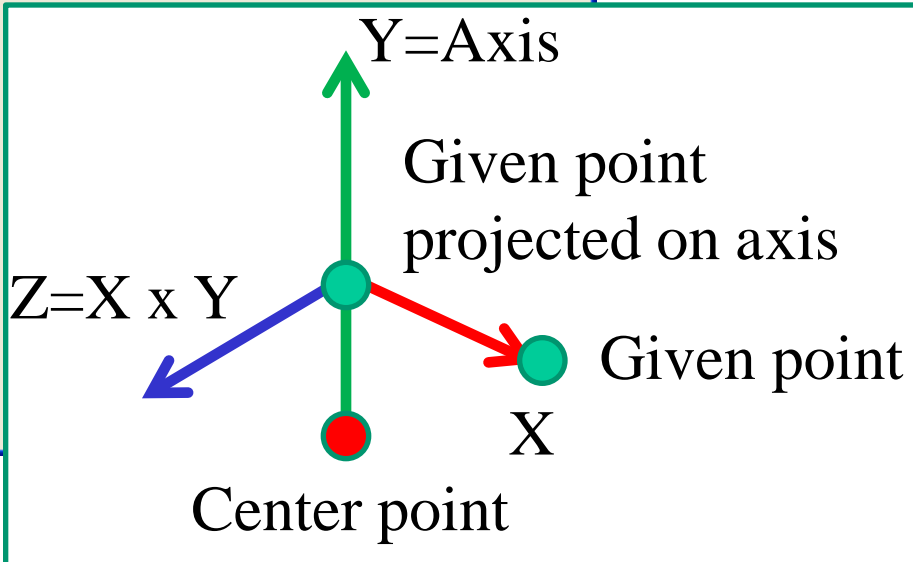
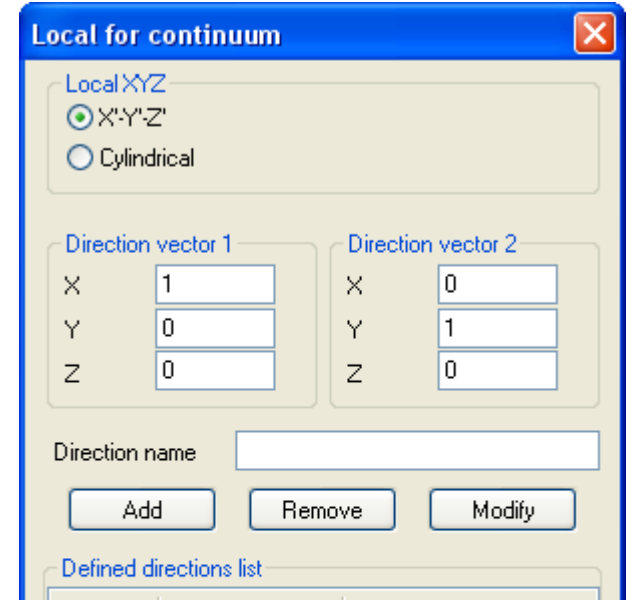
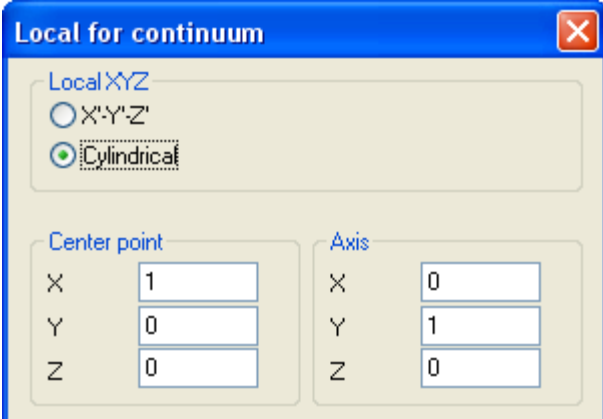
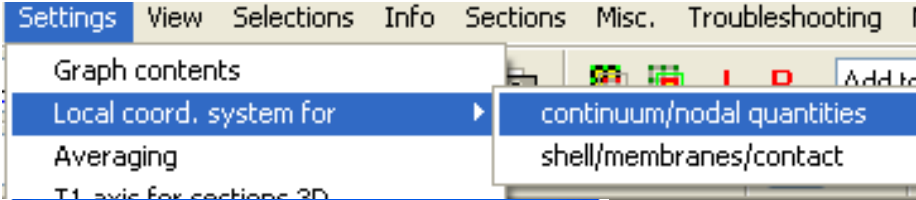


# Results maps – local coordinates system for continuum/nodal quantities



**Components defined in local coordinates system**

# Results maps – local coordinates system for continuum/nodal quantities



# Results maps – local coordinates system for shell/membranes/contact

The image displays two side-by-side screenshots of the 'Maps' dialog box in a software application, illustrating the configuration for results maps in a local coordinate system.

**Left Screenshot:**

- Results for Elements:**  Shells/ Membranes (highlighted with a red box and arrow).
- Component:**  Moment (highlighted with a red box).
- Layer position:**  M1 (highlighted with a red box).
- Visualization:**  Default.

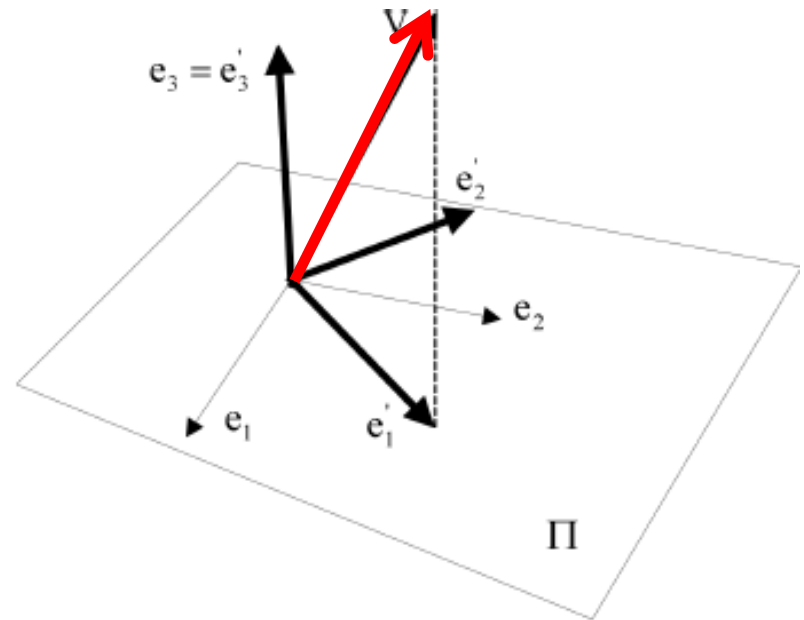
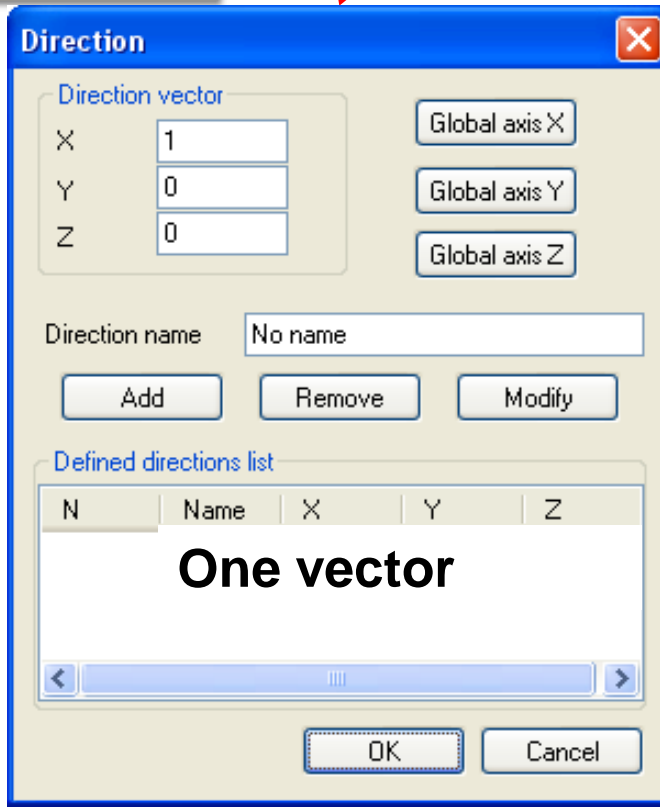
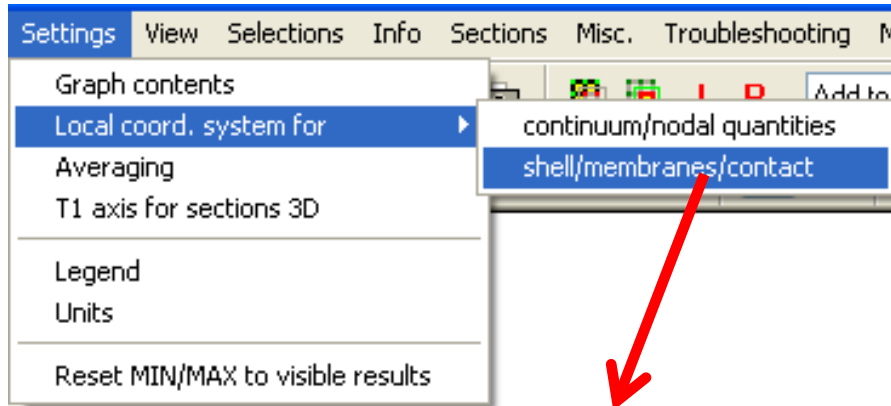
**Right Screenshot:**

- Results for Elements:**  Contact (highlighted with a red box and arrow).
- Component:**  Eps n (highlighted with a red box).
- Layer position:** -1.
- Visualization:**  Default.

**Central Text:** Components defined in local coordinates system

Both screenshots show the 'Additional visualization' and 'Deformation visibility' sections at the bottom, with 'Discrete map' selected in the visualization options and 'Deformed mesh' selected in the deformation visibility options.

# Results maps – local coordinates system for shell/membranes/contact



# Maps – moments XX

**Maps**

Results for  
Elements :  
 Continuum  
 Shells/ Membranes  
 Contact

Nodes :  
 Nodal quantities  
 Residuals

Membrane forces  
Moment  
Shear forces  
Stress  
Wood-Armer M-bot  
Wood-Armer M-top  
Wood-Armer N(+)  
Wood-Armer N(-)

Component :  $\times\times$

Layer position: -1

Min : -1.66561e+001  
Max : -5.31535e-001

Visualization  
 Default  
Min : 0  
Max : 0  
NStep : 20

Isoline  
 Discrete  
 Isoline

visibility  
ed mesh  
ectors  
igs  
OK

**Direction**

Direction vector

X 0 Global axis X  
Y 1 Global axis Y  
Z 0 Global axis Z

Direction name No name

Add Remove Modify

Defined directions list

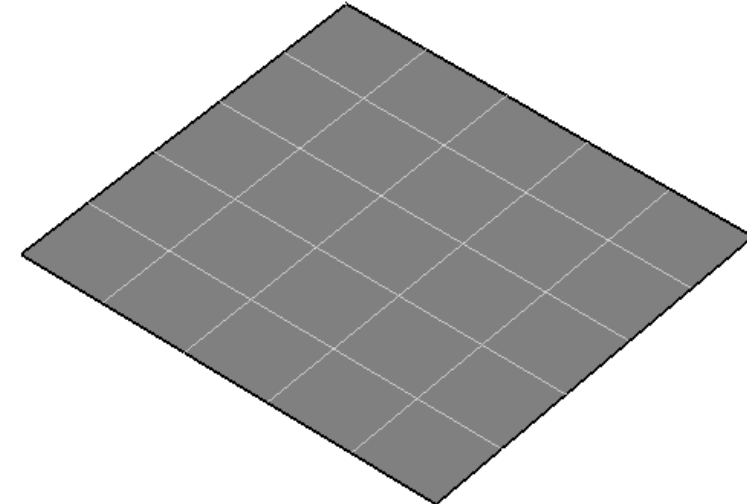
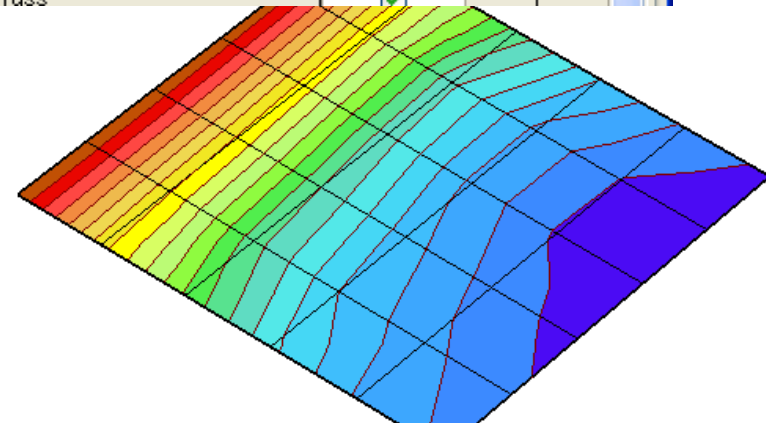
N	Name	X	Y	Z
---	------	---	---	---

OK Cancel

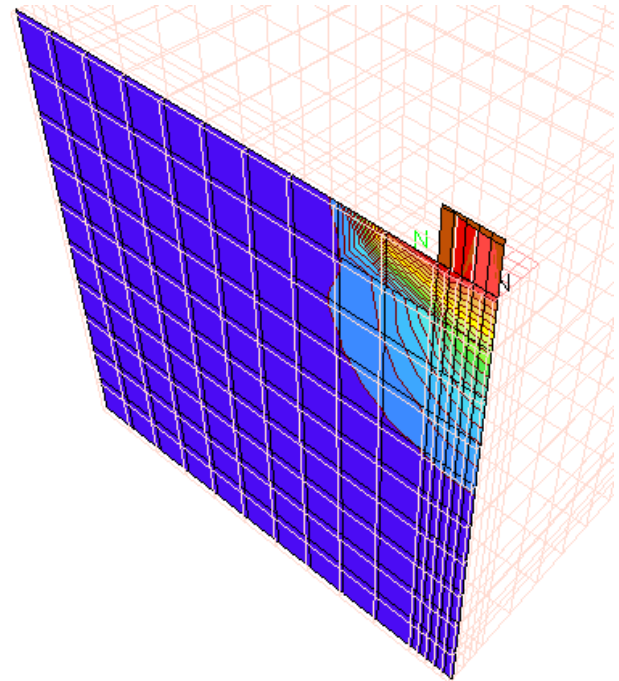
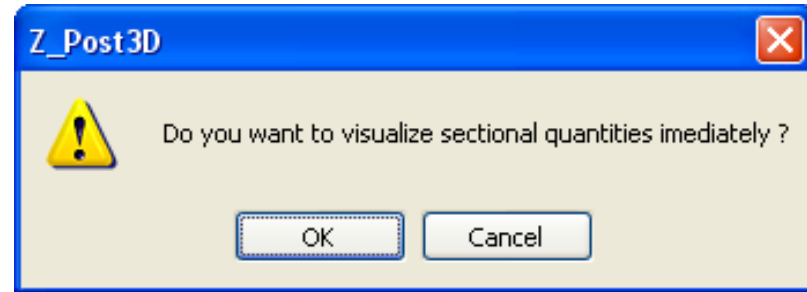
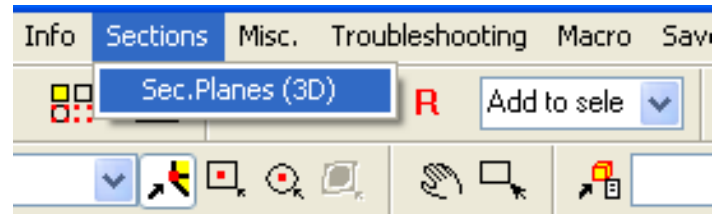
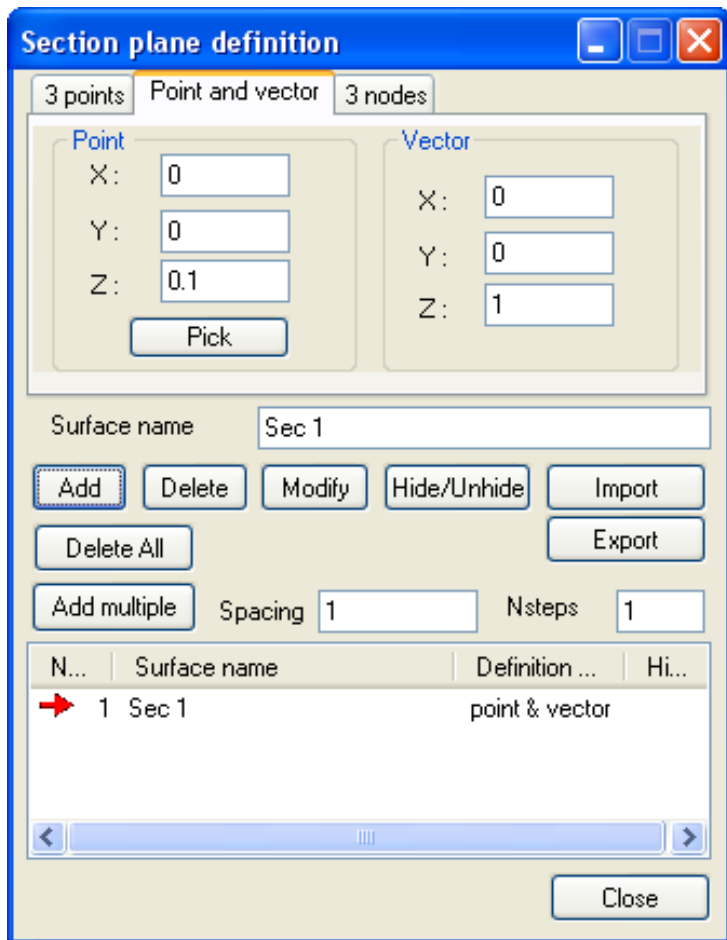
**Visibility**

Elements Loads BC Axes Other

Object	Visible	Shortcut
Nodes	<input type="checkbox"/>	N
Continuum	<input type="checkbox"/>	H
Shell	<input checked="" type="checkbox"/>	D
Contact	<input type="checkbox"/>	K
Membranes	<input checked="" type="checkbox"/>	M
Beam	<input checked="" type="checkbox"/>	B
Truss	<input checked="" type="checkbox"/>	T



# Sectional quantities



# Sectional quantities

Settings View Selections Info Sec

Graph contents

Local coord. system for

Averaging

T1 axis for sections 3D

---

Legend

Units

---

Reset MIN/MAX to visible results

### Sectional quantities

Results for

Elements :

- Continuum
- Shells/ Membranes
- Contact

Nodes :

- Nodal quantities
- Residuals

Displacement  
Pore Pressure  
Total heads  
Rotation  
Temperature  
Humidity

Component : ABS

Min : Not calculated

Max : Not calculated

Evaluate Min/Max

Visualization

- Default
- Automatic diagram scaling

Min : 0

Max : 0

NStep : 20

Scale 1

Label size 10

Additional visualization

- Fluid vel. vec. Settings

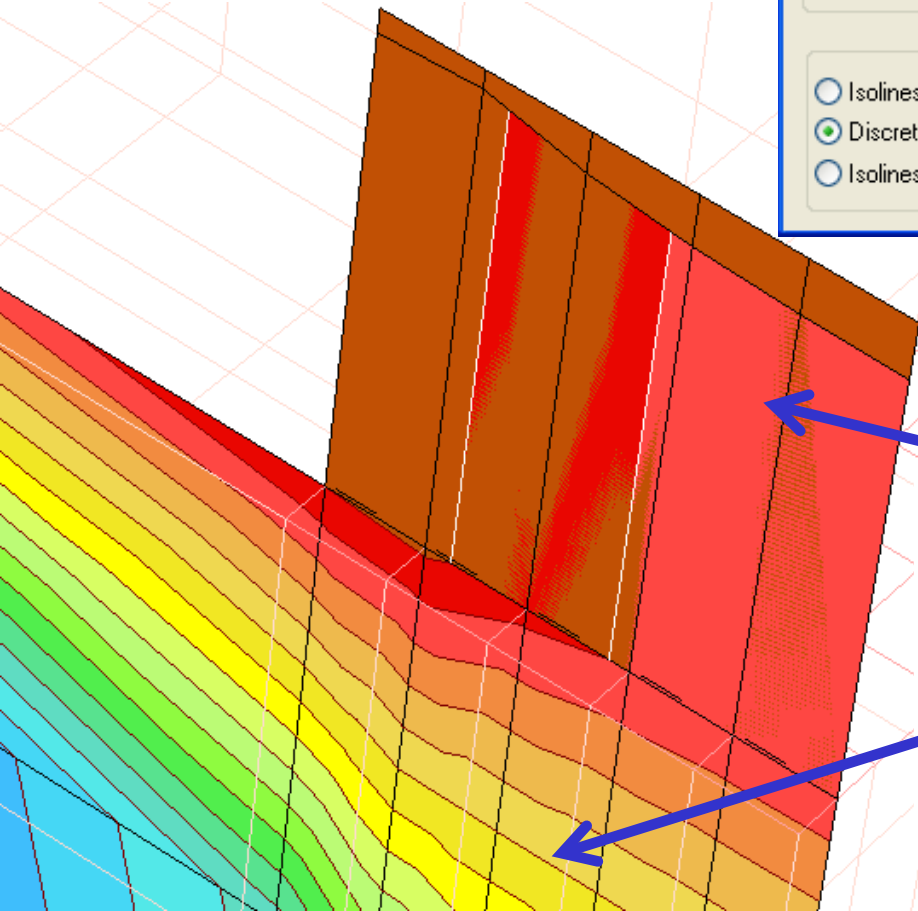
Deformation visibility

- Displ. vectors
- Deformed mesh Settings

OK

Cancel

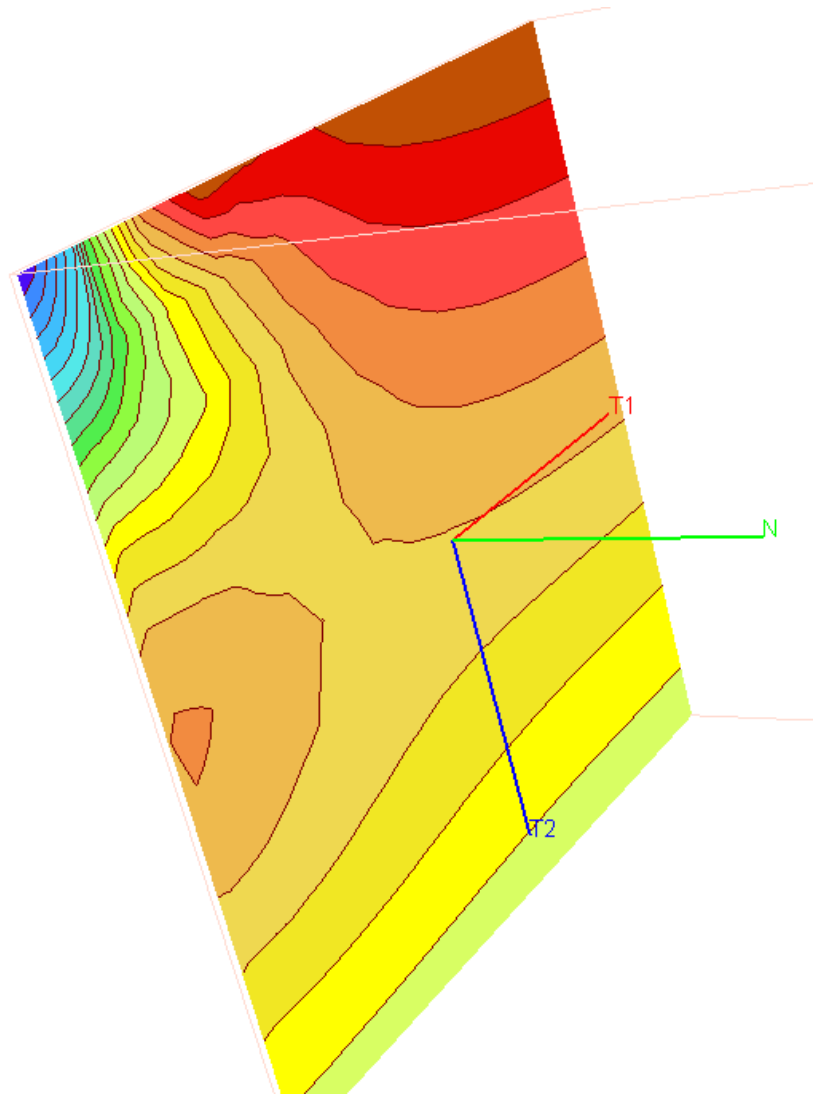
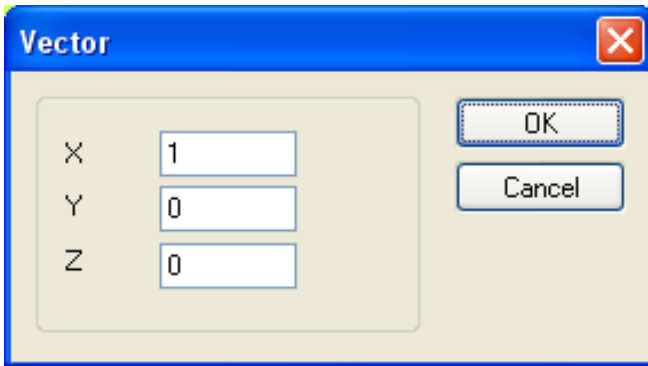
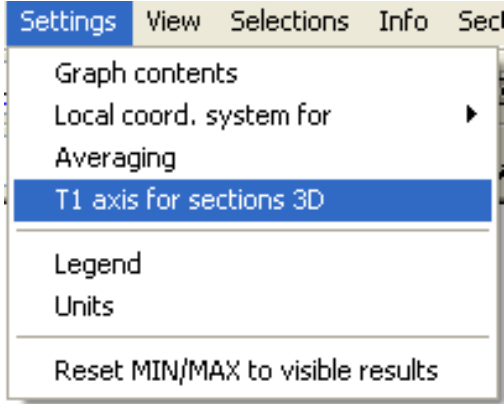
Visibility



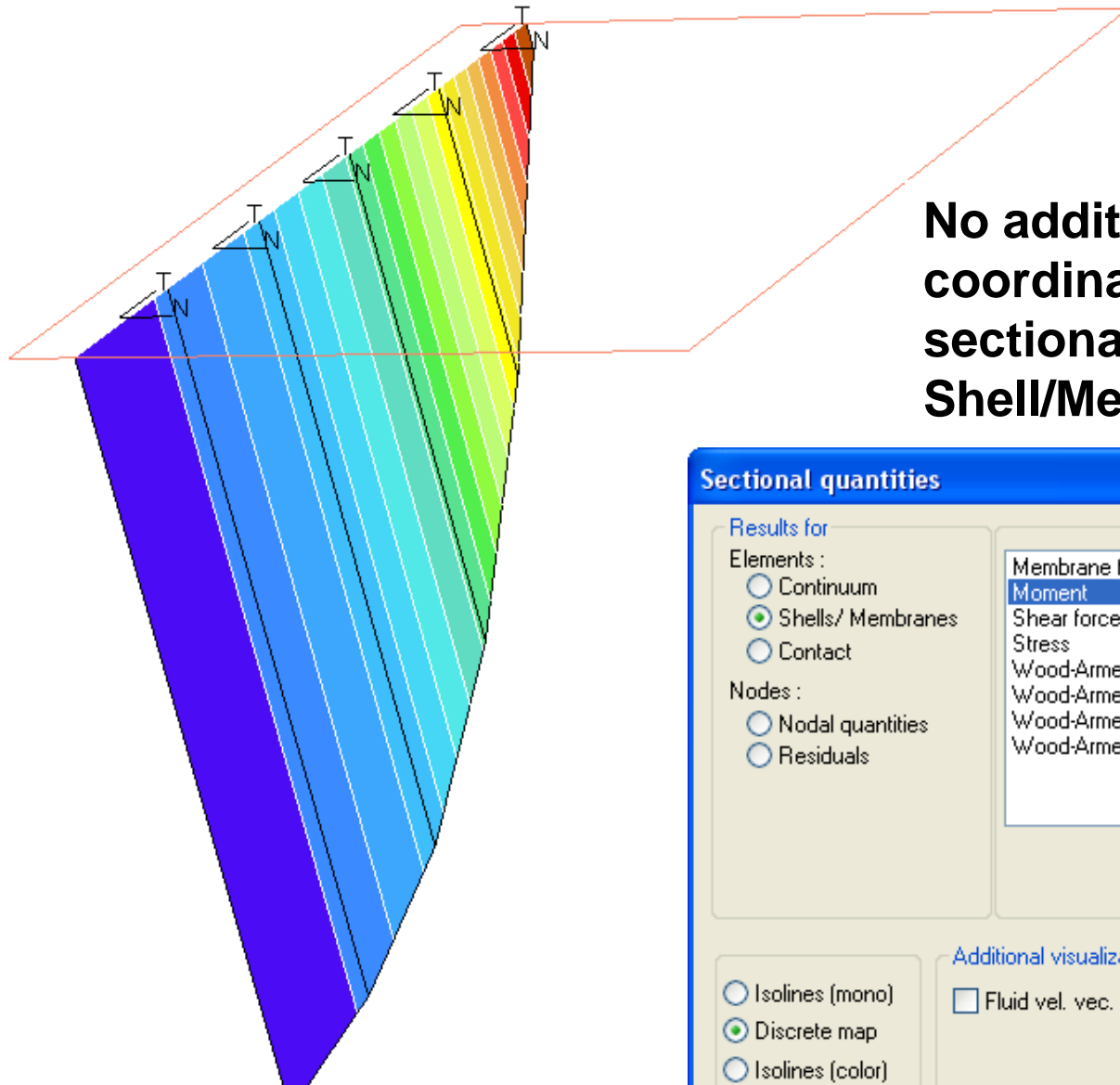
Diagrams for sections through  
Shell/Membrane/Contact

Maps for sections through Continuum  
3D

# Sectional quantities – local coordinates system for continuum



# Sectional quantities – local coordinates system for shells/membranes/contact



No additional definition of local coordinates system is required for sectional quantities through Shell/Membrane/Contact

**Sectional quantities**

Results for

Elements :

- Continuum
- Shells/ Membranes
- Contact

Nodes :

- Nodal quantities
- Residuals

Membrane forces

Moment

Shear forces

Stress

Wood-Armer M-bot

Wood-Armer M-top

Wood-Armer N(+)

Wood-Armer N(-)

Component : TT

Min : Not calculated

Max : Not calculated

Visualization

- Default
- Automatic diagram scaling

Min : 0

Max : 0

NStep : 20

Scale : 1

Label size : 10

Additional visualization

- Fluid vel. vec. Settings

Deformation visibility

- Displ. vectors
- Deformed mesh Settings

OK

Cancel

Visibility

# Nodal time history

Graph Option Settings View Selections Info Sections

View data

Deformed mesh

Displacement vectors

Plastic zones

Principal stresses

Principal moments/memb.

Fluid velocities

Maps

Sectional quantities

Element time history

✓ Nodal time history

MNT for continuum 3D

Results in Beams/Trusses

Eigenmodes

Push over

Save settings

Read settings

Default settings

Parametric analysis

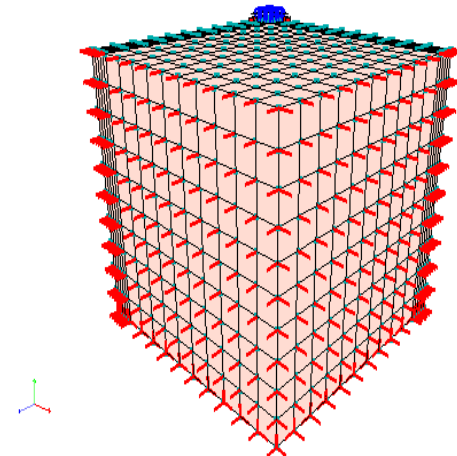
Z\_Post3D

File Time Graph Option Settings Time history Macro Save plot Help



foundation Time history

foundation



Nodes list

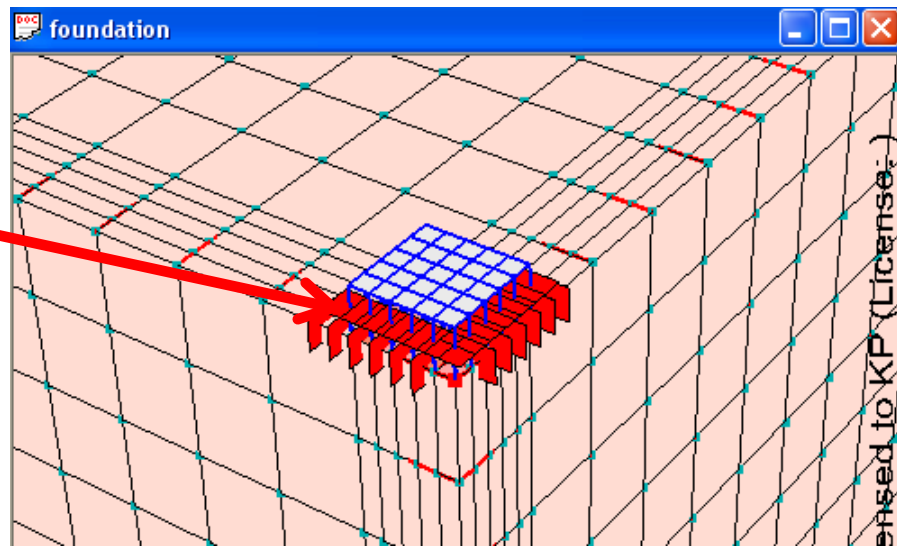
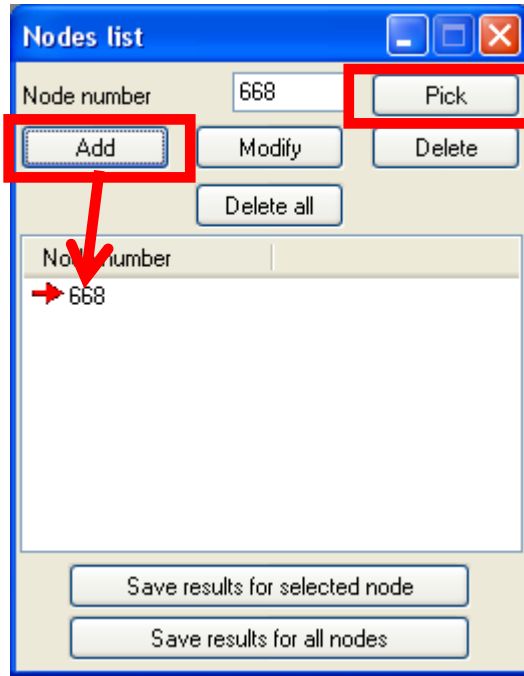
Node number 0 Pick

Add Modify Delete

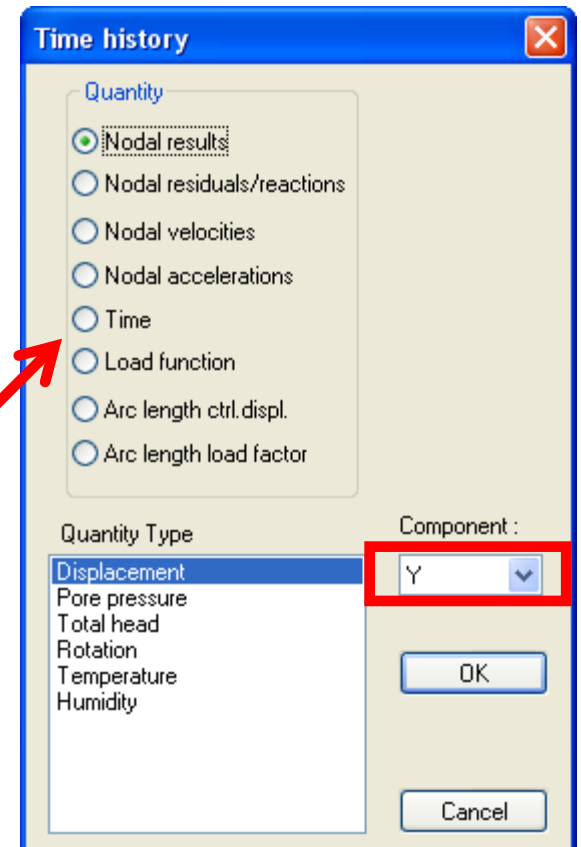
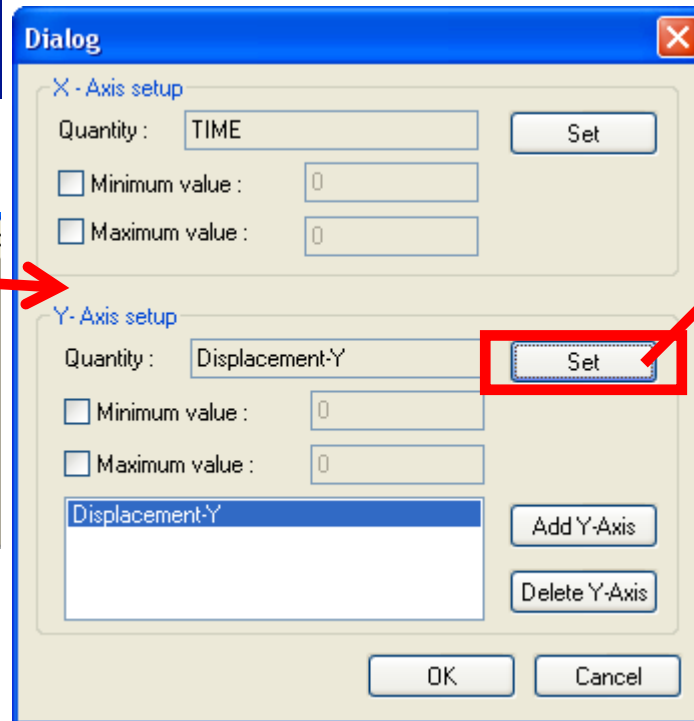
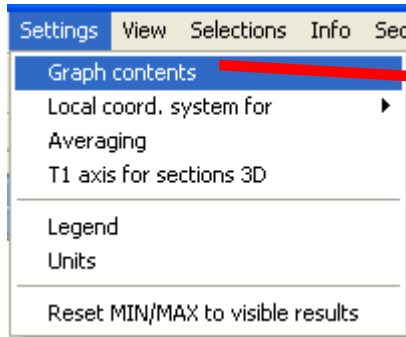
Delete all

Node number

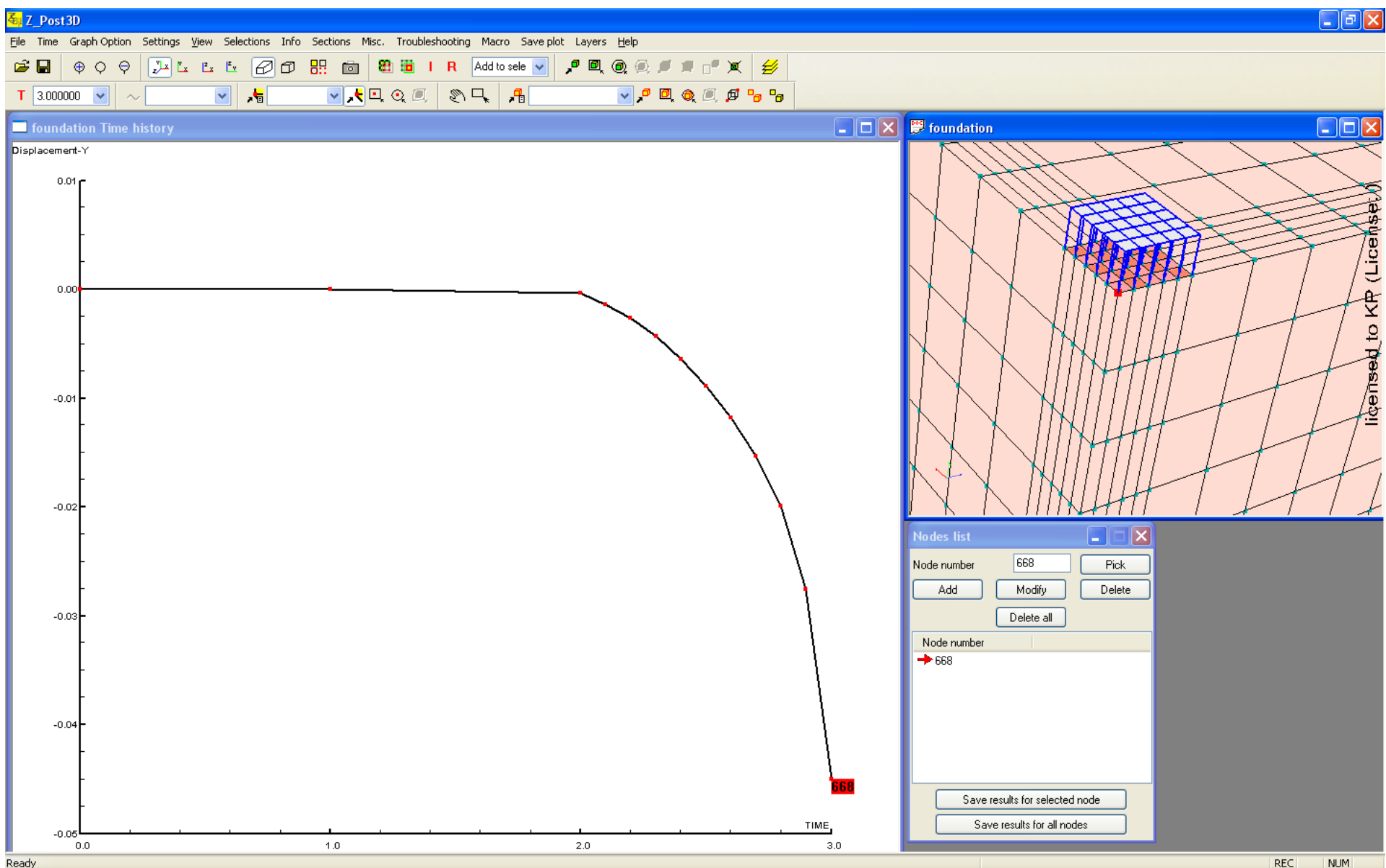
# Nodal time history



ensed to KP (License: )



# Nodal time history



# Nodal time history

**Dialog** [X]

X - Axis setup

Quantity :

Minimum value :

Maximum value :

Y - Axis setup

Quantity :

Minimum value :

Maximum value :

