

# Finite Element Modelling using PLAXIS 2D



Krishna Kumar  
Email: [kks32@cam.ac.uk](mailto:kks32@cam.ac.uk)

Lent Term 2017

# What is PLAXIS?

A finite element program that simulates the behaviour of soil and its interaction with structures

Introduction to Plaxis 2D

Running a simulation

- Plaxis Input
- Plaxis Calculations
- Plaxis Output
- Plaxis Curves

Discussion of Homework #2



## IMPORTANT WARNING AND DISCLAIMER

PLAXIS is a finite element program for geotechnical applications in which soil models are used to simulate the soil behaviour. The PLAXIS code and its soil models have been developed with great care. Although a lot of testing and validation have been performed, it cannot be guaranteed that the PLAXIS code is free of errors.

Moreover, the simulation of geotechnical problems by means of the finite element method implicitly involves some inevitable numerical and modelling errors. 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 models and their limitations, the selection of model parameters, and the ability to judge the reliability of the computational results. Hence, PLAXIS may only be used by professionals that possess the aforementioned expertise.

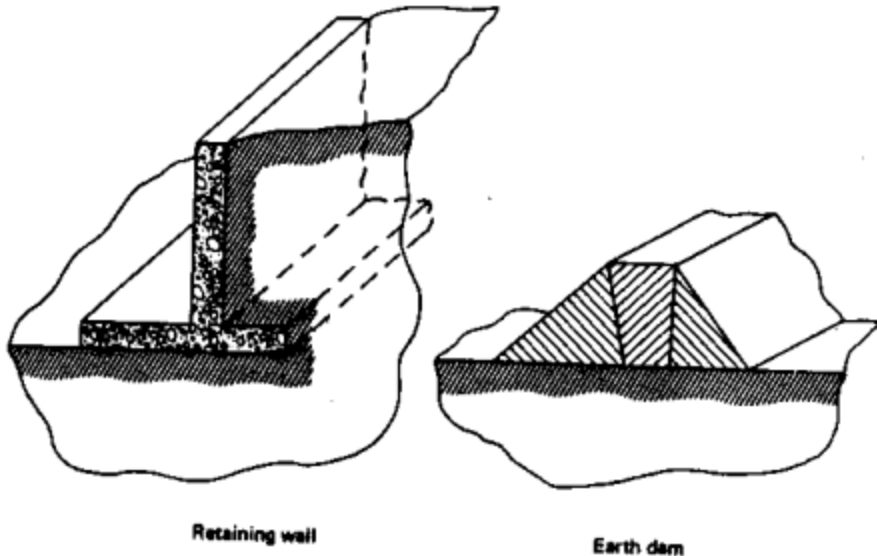
The user must be aware of his/her responsibility when he/she uses the computational results for geotechnical design purposes. The PLAXIS organization cannot be held responsible or liable for design errors that are based on the output of PLAXIS calculations.

# Consistent System of Units

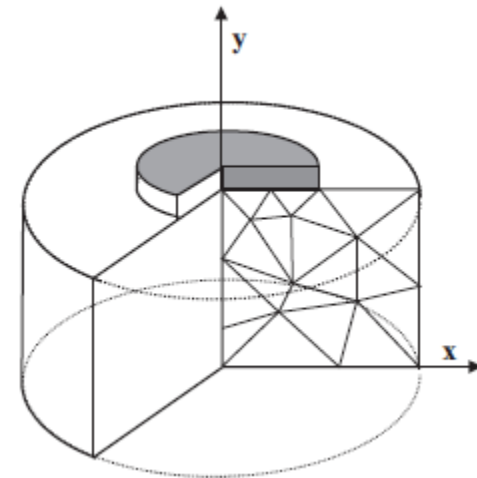
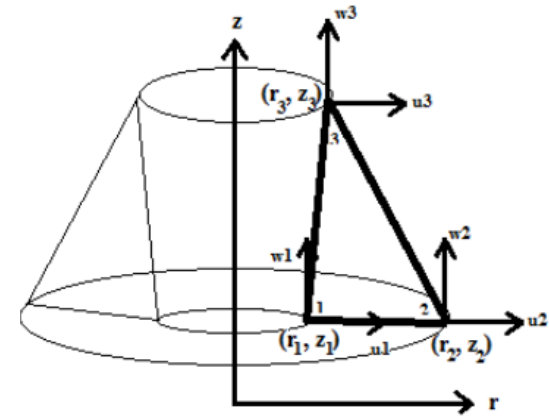
SI				
Length	m	m	m	cm
Density	$\text{kg/m}^3$	$10^3 \text{ kg/m}^3$	$10^6 \text{ kg/m}^3$	$10^6 \text{ g/cm}^3$
Force	N	kN	MN	Mdynes
Stress	Pa	kPa	MPa	bar
Gravity	$\text{m/sec}^2$	$\text{m/sec}^2$	$\text{m/sec}^2$	$\text{cm/s}^2$
Stiffness*	$\text{Pa/m}$	$\text{kPa/m}$	$\text{MPa/m}$	$\text{bar/cm}$

# General Settings

**Plane Strain:** Strain normal to x-y plane is zero  $\varepsilon_z = 0$  and shear strains  $\gamma_{zy}$  and  $\gamma_{zx}$



**Axisymmetric:**



# General Settings (cont...)

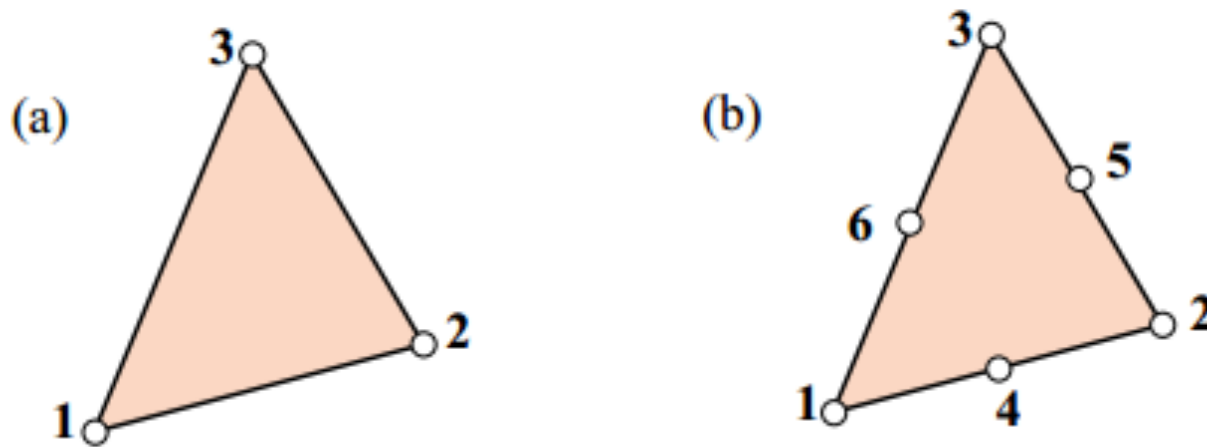


Figure 29.3. Nodal configurations for: (a) linear interpolation of  $w$  by three values  $w_i$ , at corners  $i = 1, 2, 3$ ; (b) quadratic interpolation of  $w$  by six values  $w_i$  at corners  $i = 1, 2, 3$  and midpoints  $i = 4, 5, 6$ .

General options

Model

Plane strain

Elements

6-Node

Acceleration

Gravity angle

-90°

1.0 G

x-acceleration

0.000

G

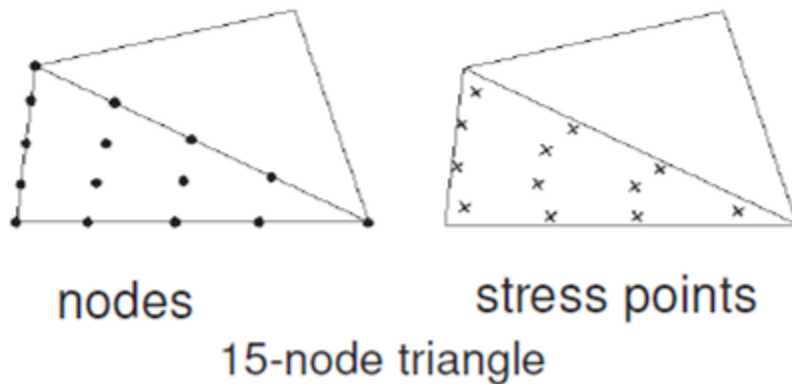
y-acceleration

0.000

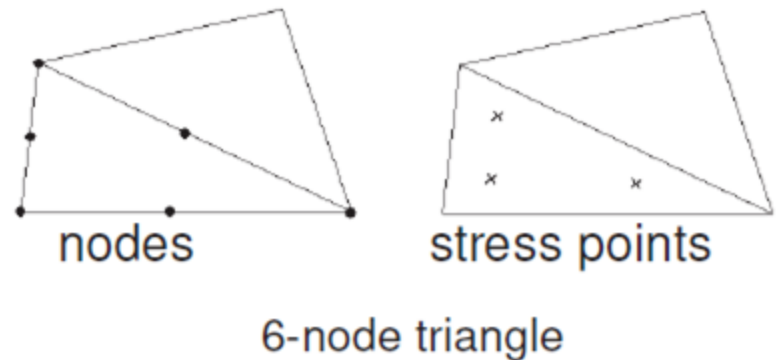
G

# 6 and 15 Node Elements

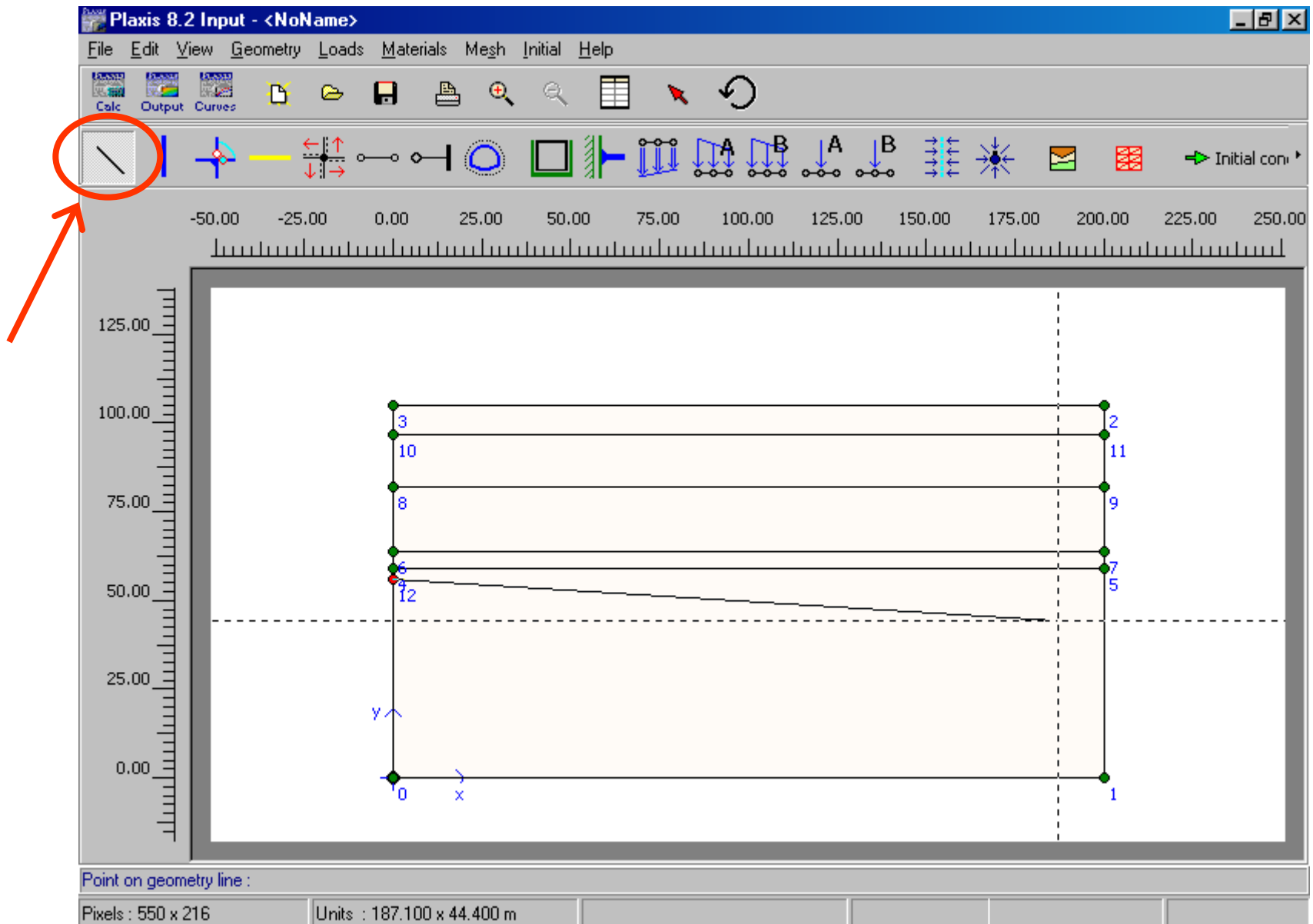
- 12 Gauss points
- 4<sup>th</sup> order interpolation
- Most accurate in PLAXIS 2D
- Better than four 6-noded triangle elements



- 3 Gauss points
- 2<sup>nd</sup> order interpolation
- Care should be taken while using for Axisymmetric and Failure problems



# Setting up the geometry





# Plate Elements

- Plates are structural Objects used to **model slender structures** in the ground with **a significant flexural rigidity (or bending stiffness) and a normal stiffness.**
- Plates can be used to simulate influence of walls, plates, shells or linings extending in z-direction
- 3-node and 5-node plate element, with 4 and 16 stress points.

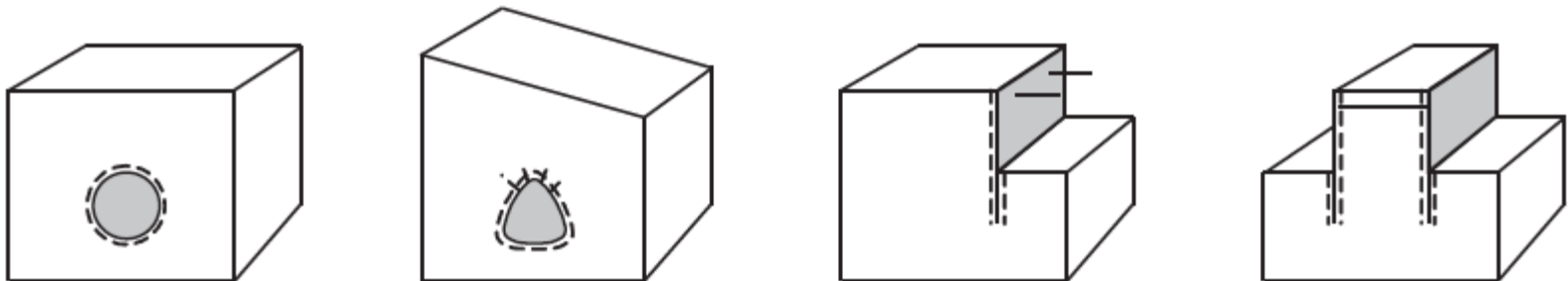
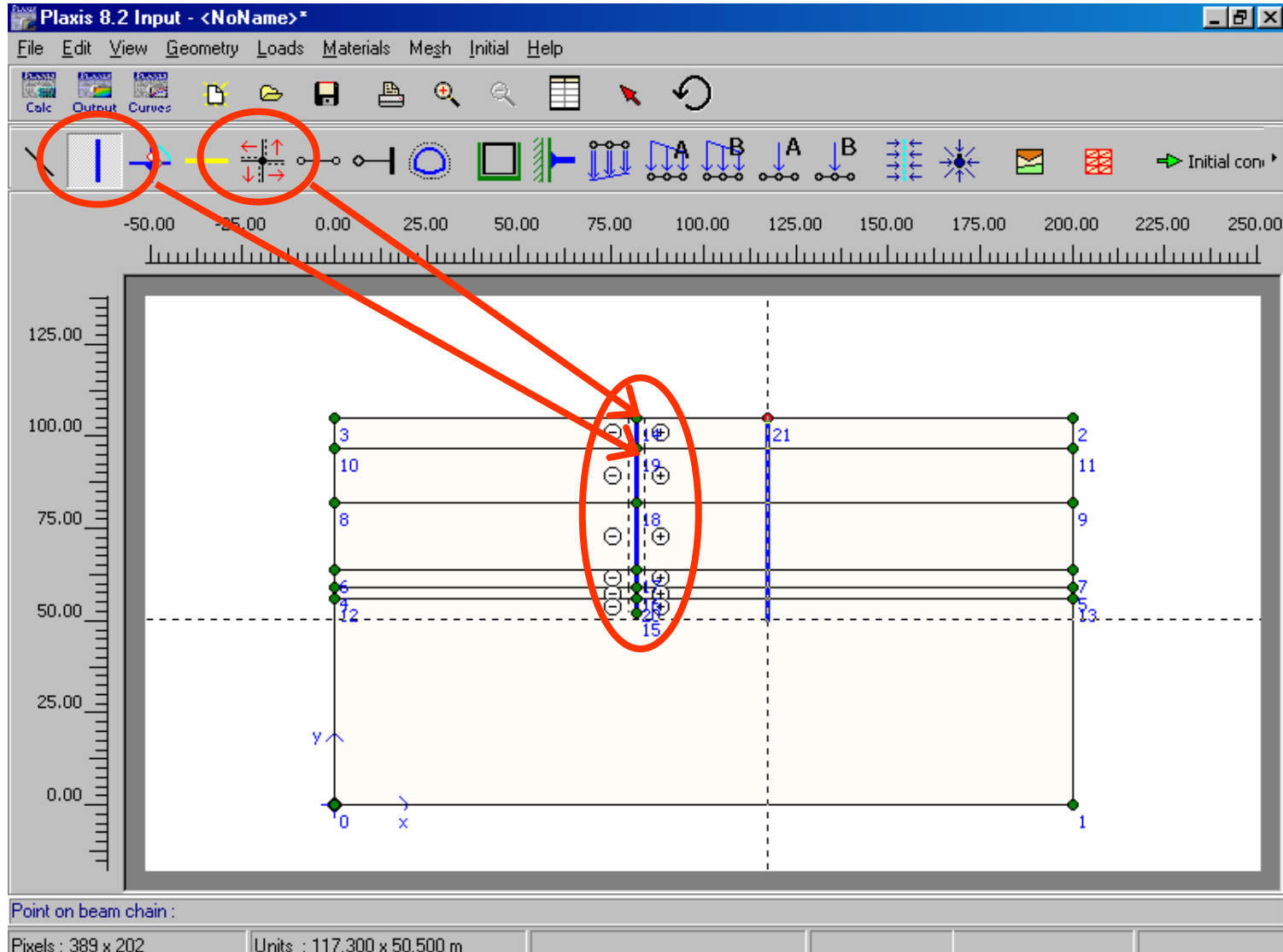


Figure 3.9 Applications in which plates, anchors and interfaces are used

# Modelling walls

- Use plate elements AND interface elements



# Interface Elements

- Problems involving Soil-Structure Interaction

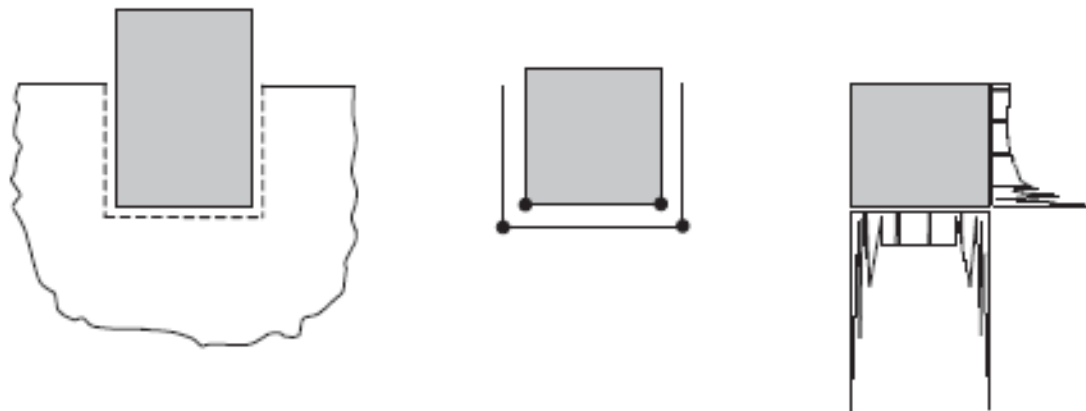


Figure 3.14 Inflexible corner point, causing poor quality stress results

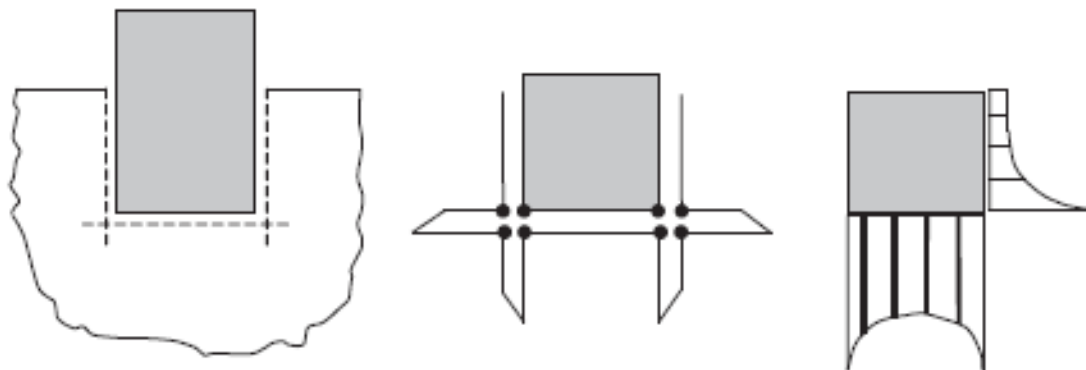
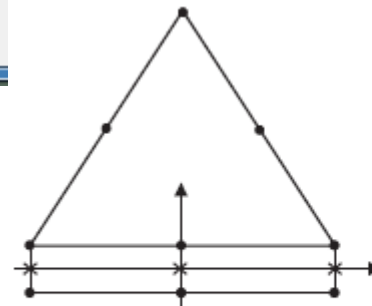
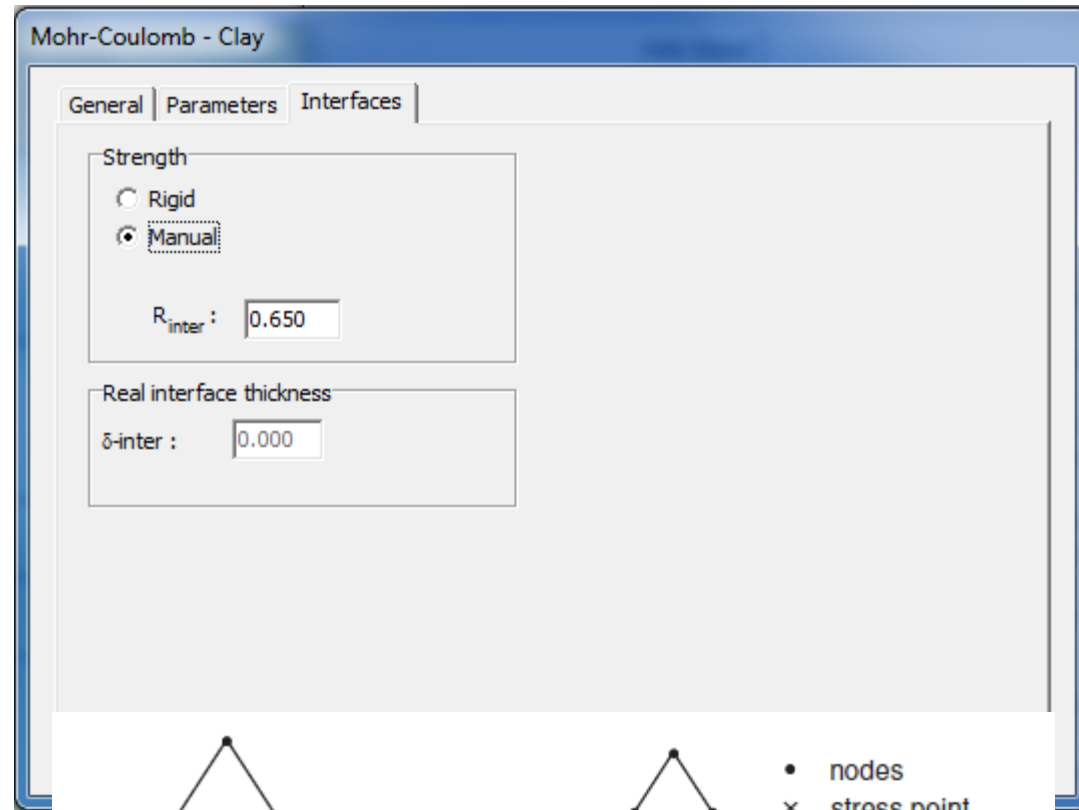


Figure 3.15 Flexible corner point with improved stress results

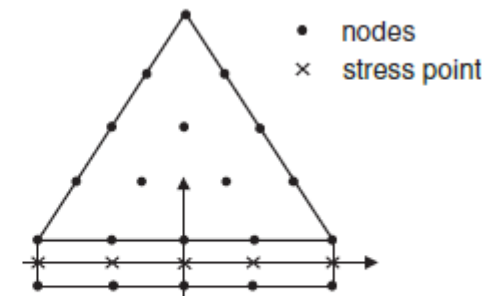
# Interface Elements (cont...)

- Properties of interface are associated with material data set.
- **Virtual thickness** of interface element is 0.1, depends on the coarseness of the mesh.
- **Rigid and Strength reduction** (Rinter) interfaces.
- In a consolidation analysis or groundwater flow analysis, **interface elements can be used to block the flow perpendicular** to the interface as an impermeable screen.
- **Should not be used to create unrealistic strength loss in soil.**

# Interface Elements



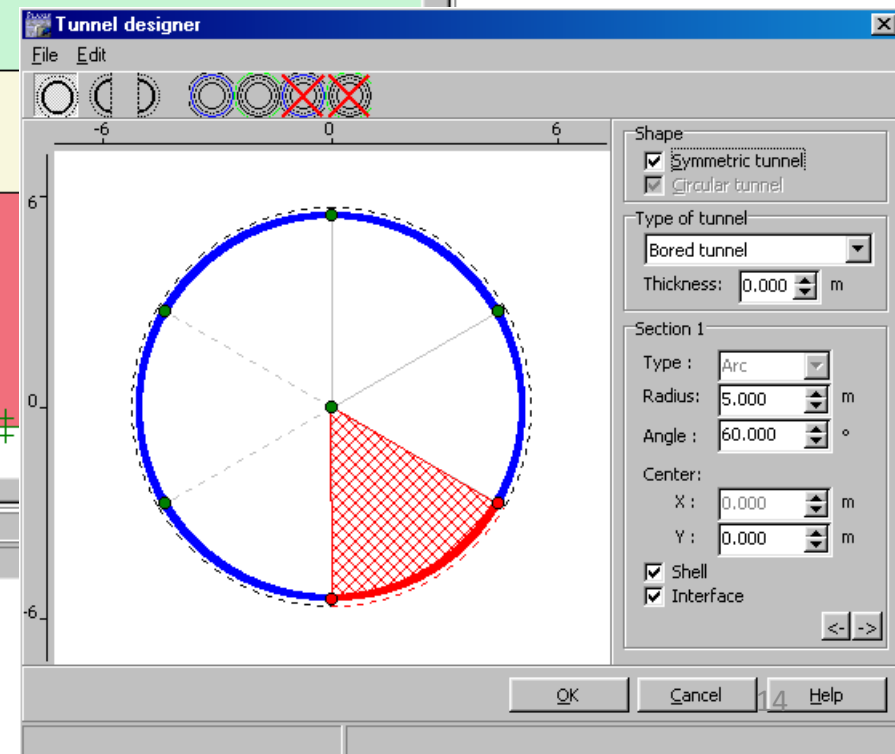
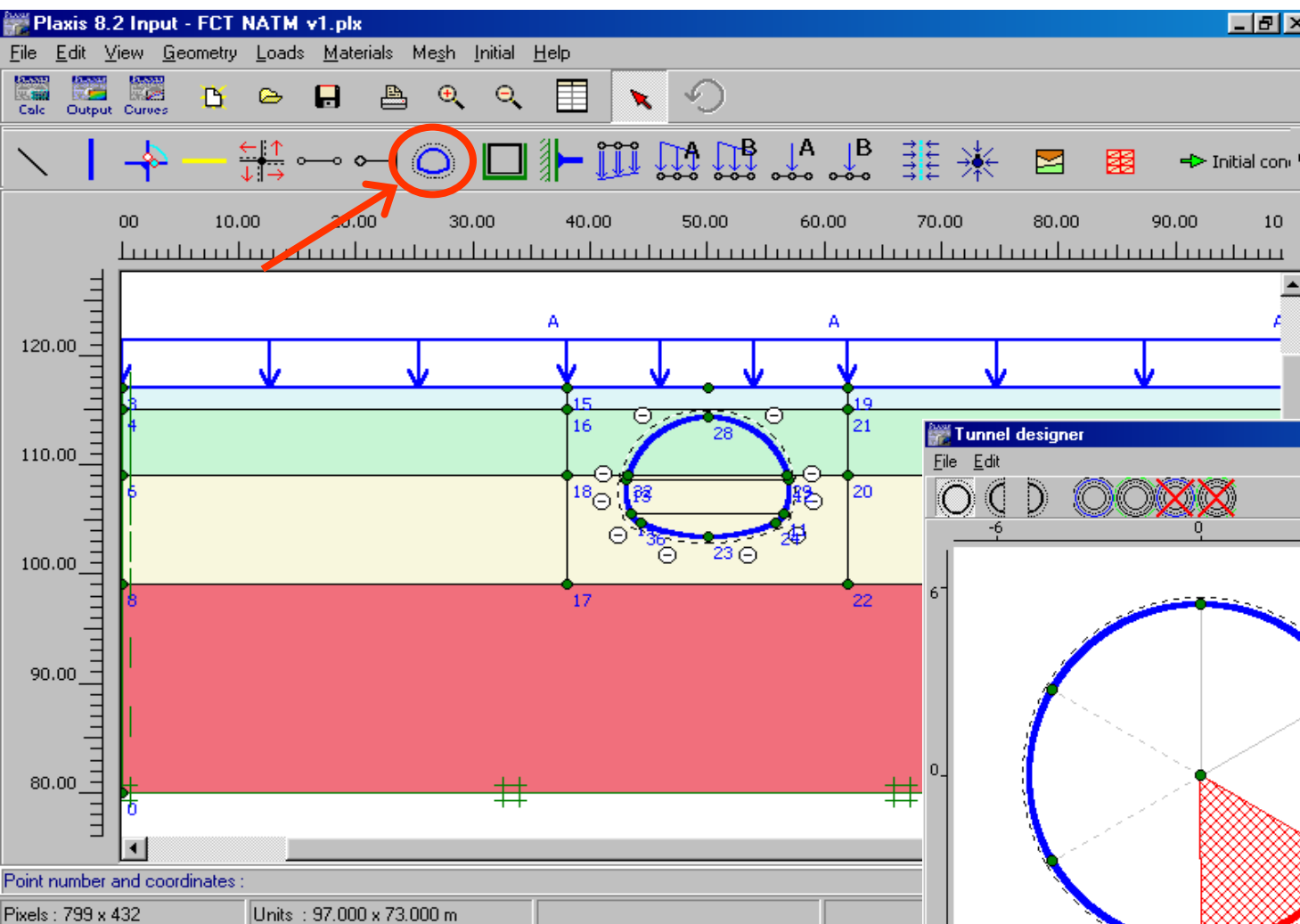
a. 6-node soil element



b. 15-node soil element

13 Distribution of nodes and stress points in interface elements and their connection to soil elements

# Modelling tunnels

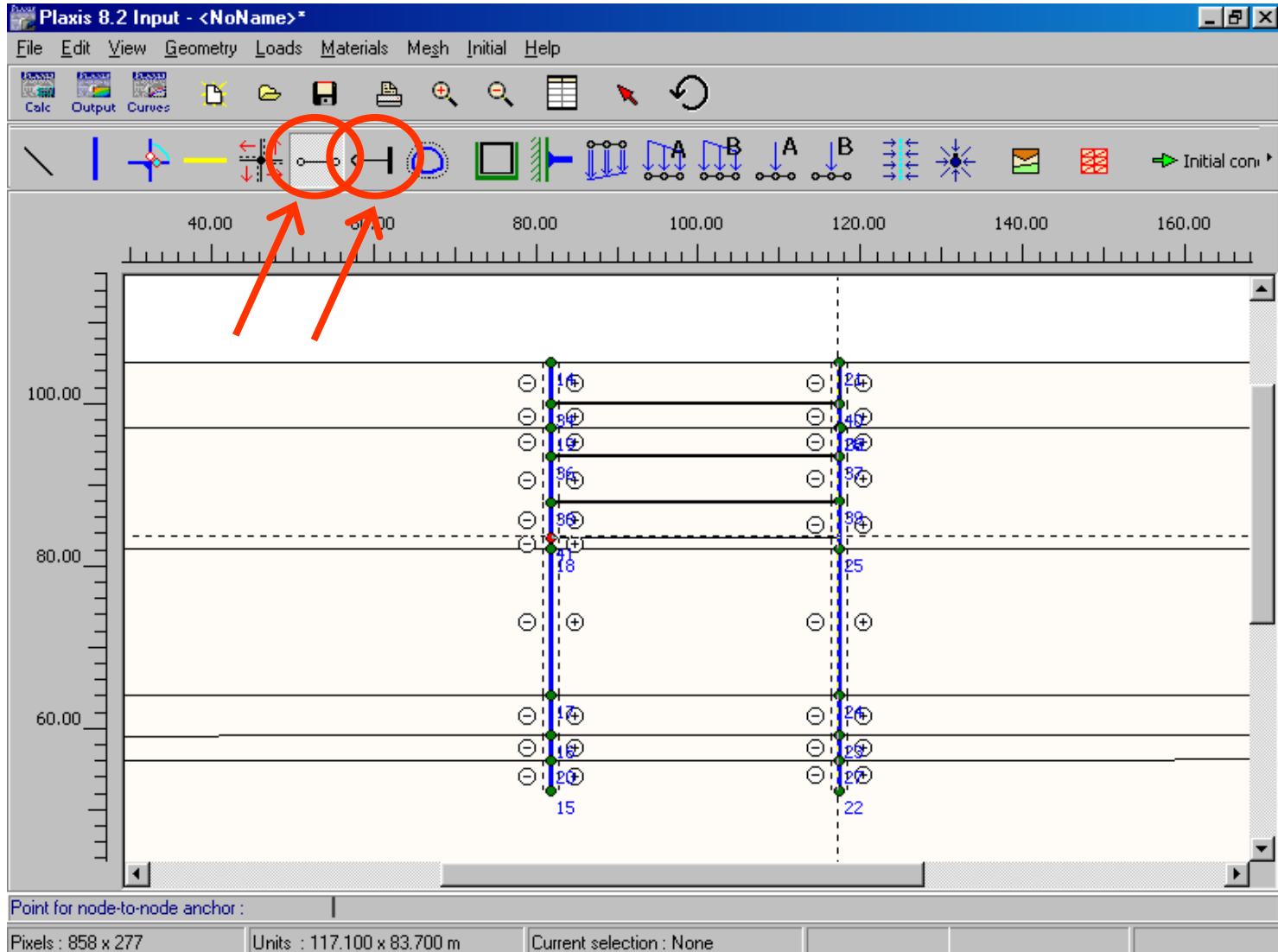


# Anchors

- **Node to Node:** are springs that are used to model ties between two points.
- It's not recommended to draw geometry line at position where node-to-node anchor is to be placed.
- It's a **2 node elastic spring element** with normal stiffness (Spring constant)
- Element can sustain both tensile forces (anchors) as well as compressive forces (struts).
- **Fixed-End anchors:** Modelling of struts or props to sheet pile walls.

# Modelling struts

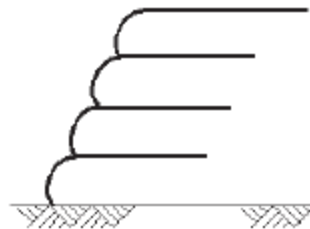
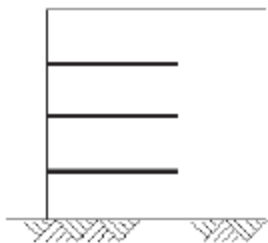
- Use node to node anchors OR fixed end anchors



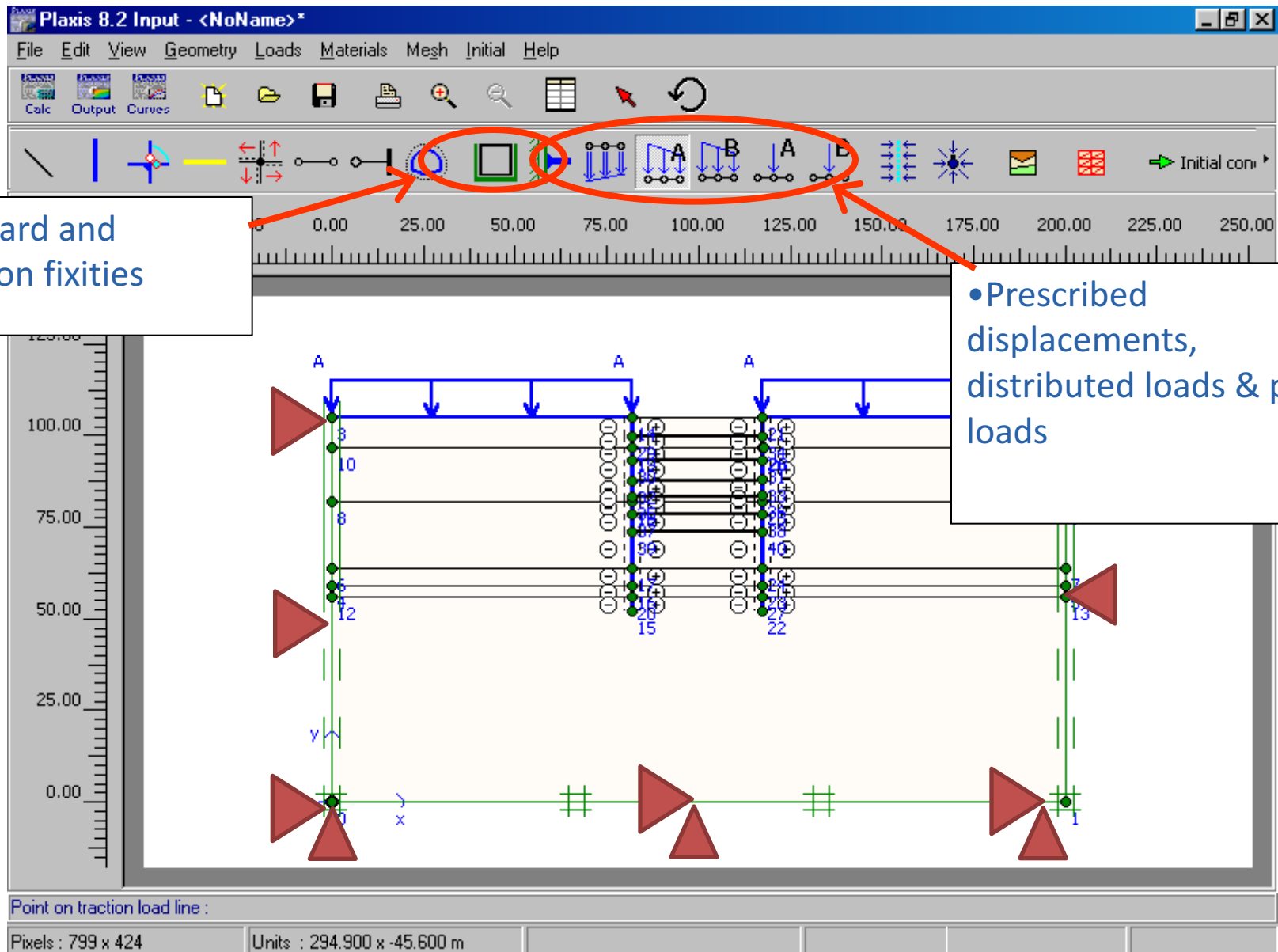


# Geogrids

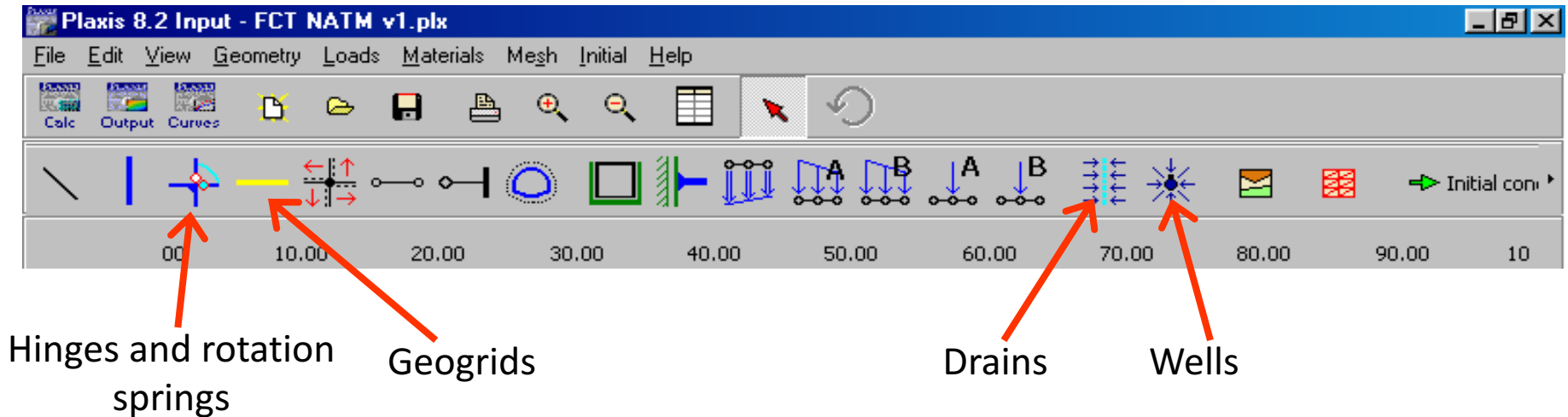
- Geogrids are slender structures with a normal stiffness but no bending stiffness.
- Geogrids can only sustain \*Tensile forces\* and no compression!
- Structures involving geotextiles.



# Defining load & boundary conditions

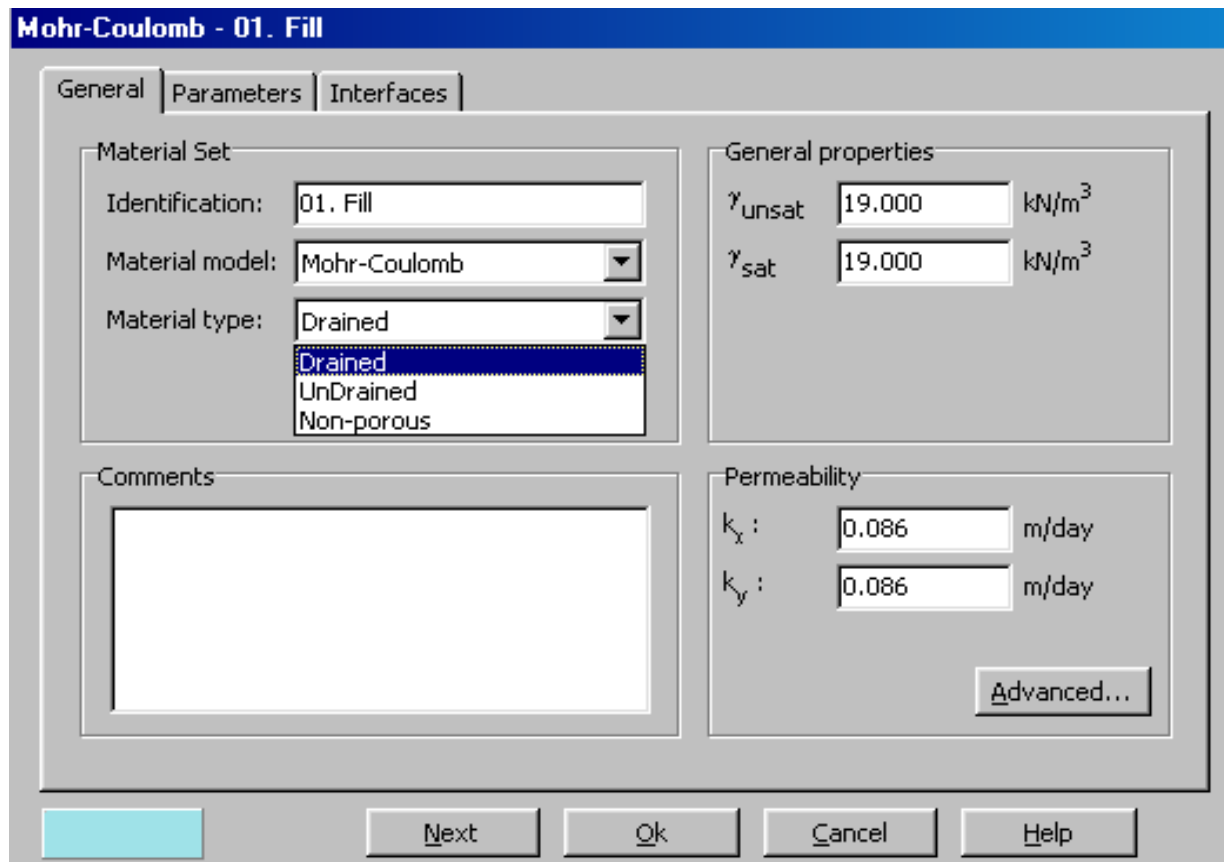
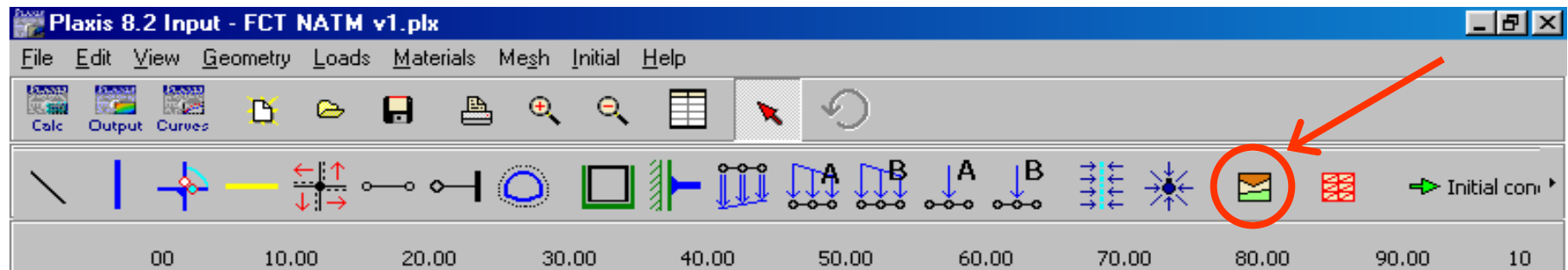


# Other modelling capabilities



- Drains – lines that set pore pressures to zero
- Wells – points where a specific discharge is subtracted from or added to the soil

# Defining material properties - soil



# Material models

1. Linear elastic – primarily used for stiff structures
2. Mohr-coulomb – very simple and popular model with 5 parameters:  $E$ ,  $\nu$ ,  $c$ ,  $\phi$  and  $\psi$
3. Soft-soil – a Cam Clay model for NC clays and peat
4. Hardening soil – an advanced elastoplastic hyperbolic model
5. Soft soil creep – similar to the soft-soil model, but can consider secondary compression
6. Jointed-rock – used for stratified or jointed rock.
7. User-defined– it is possible to use your own model.

# The Mohr Coulomb model

**Mohr-Coulomb - 01. Fill**

General Parameters Interfaces

**Stiffness**

$E_{ref}$  : 2231.000 kN/m<sup>2</sup>

$\nu$  (nu) : 0.200

**Strength**

$c_{ref}$  : 0.100 kN/m

$\phi$  (phi) : 30.000 °

$\psi$  (psi) : 0.000 °

**Alternatives**

$G_{ref}$  : 3846.154 kN/m<sup>2</sup>

$E_{oed}$  : 1.026E+04 kN/m<sup>2</sup>

**Velocities**

$V_s$  : 44.540 m/s

$V_p$  : 83.330 m/s

Advanced...

Next Ok Cancel Help

Stiffness & strength parameters

Variation of stiffness and strength with depth

**Advanced parameters Mohr-Coulomb**

**Stiffness**

$E_{increment}$  : 0.000 kN/m<sup>2</sup>/m

$\gamma_{ref}$  : 0.000 m

**Strength**

$c_{increment}$  : 0.000 kN/m<sup>2</sup>/m

$\gamma_{ref}$  : 0.000 m

☒ Tension cut off

Tensile strength : 0.000 kN/m<sup>2</sup>

**Undrained behaviour**

☒ Standard settings

☐ Manual settings

Skempton-B : 0.987

$\nu_u$  : 0.495

$K_{w,ref}/n$  : 3.782E+05 kN/m<sup>2</sup>

**Consolidation**

$C_{v,ref}$  : N/A m<sup>2</sup>/day

$C_{v,ref} = \frac{k_y \cdot E_{oed}}{\gamma_w}$

OK Cancel Default Help

# Defining material properties – plates and anchors

## Plates

**Plate properties**

Material set  
Identification: 800mm Dwall  
Material type: Elastic

Comments

Properties

EA :	2.240E+07	kN/m
EI :	8.360E+05	kNm <sup>2</sup> /m
d :	0.669	m
w :	6.400	kN/m/m
$\nu$ :	0.150	
$M_p$ :	1.000E+15	kNm/m
$N_p$ :	1.000E+15	kN/m
Rayleigh $\alpha$ :	0.000	
Rayleigh $\beta$ :	0.000	

Ok Cancel Help

**Anchor properties**

Material set  
Identification: S1- H350x350  
Material type: Elastoplastic

Comments

Properties

EA :	9.023E+05	kN
$L_{\text{spacing}}$ :	1.000	m
$ F_{\text{max,tens}} $ :	0.000	kN
$ F_{\text{max,comp}} $ :	920.150	kN

Values per anchor !

OK Cancel Help

## Anchors

# Assigning materials

Plaxis 8.2 Input - Excavation v1.plx\*

File Edit View Geometry Loads Materials Mesh Initial Help

Calc Output Curves

Material sets

Global >>>

Project Database

Set type: Soil & Interfaces

Group order: None

- 01. Fill
- 02. Estuarine
- 03. Upper Marine Clay
- 04. Fluvial Clay F2
- 05. Lower Marine Clay
- 06. Fluvial Sand F1
- 07. OA SW1
- 08. OA-CZ
- 09. JGP

New... Edit... Copy... Del...

OK Apply Help

Point number and coordinates :

Pixels : 409 x 8 Units : 123.830 x 123.320 m Current selection : Clusters

The screenshot displays the Plaxis 8.2 software interface. The main window shows a cross-section of an excavation with a soil profile. The soil profile is divided into several layers, each represented by a different color. The layers are labeled with point numbers: 01 (Fill), 02 (Estuarine), 03 (Upper Marine Clay), 04 (Fluvial Clay F2), 05 (Lower Marine Clay), 06 (Fluvial Sand F1), 07 (OA SW1), 08 (OA-CZ), and 09 (JGP). The 'Material sets' dialog box is open, showing the 'Project Database' with the 'Set type' set to 'Soil & Interfaces' and 'Group order' set to 'None'. The list of materials is shown, with '05. Lower Marine Clay' highlighted. The background shows a cross-section of an excavation with a soil profile. The soil profile is divided into several layers, each represented by a different color. The layers are labeled with point numbers: 01 (Fill), 02 (Estuarine), 03 (Upper Marine Clay), 04 (Fluvial Clay F2), 05 (Lower Marine Clay), 06 (Fluvial Sand F1), 07 (OA SW1), 08 (OA-CZ), and 09 (JGP). The 'Material sets' dialog box is open, showing the 'Project Database' with the 'Set type' set to 'Soil & Interfaces' and 'Group order' set to 'None'. The list of materials is shown, with '05. Lower Marine Clay' highlighted. The background shows a cross-section of an excavation with a soil profile. The soil profile is divided into several layers, each represented by a different color. The layers are labeled with point numbers: 01 (Fill), 02 (Estuarine), 03 (Upper Marine Clay), 04 (Fluvial Clay F2), 05 (Lower Marine Clay), 06 (Fluvial Sand F1), 07 (OA SW1), 08 (OA-CZ), and 09 (JGP).

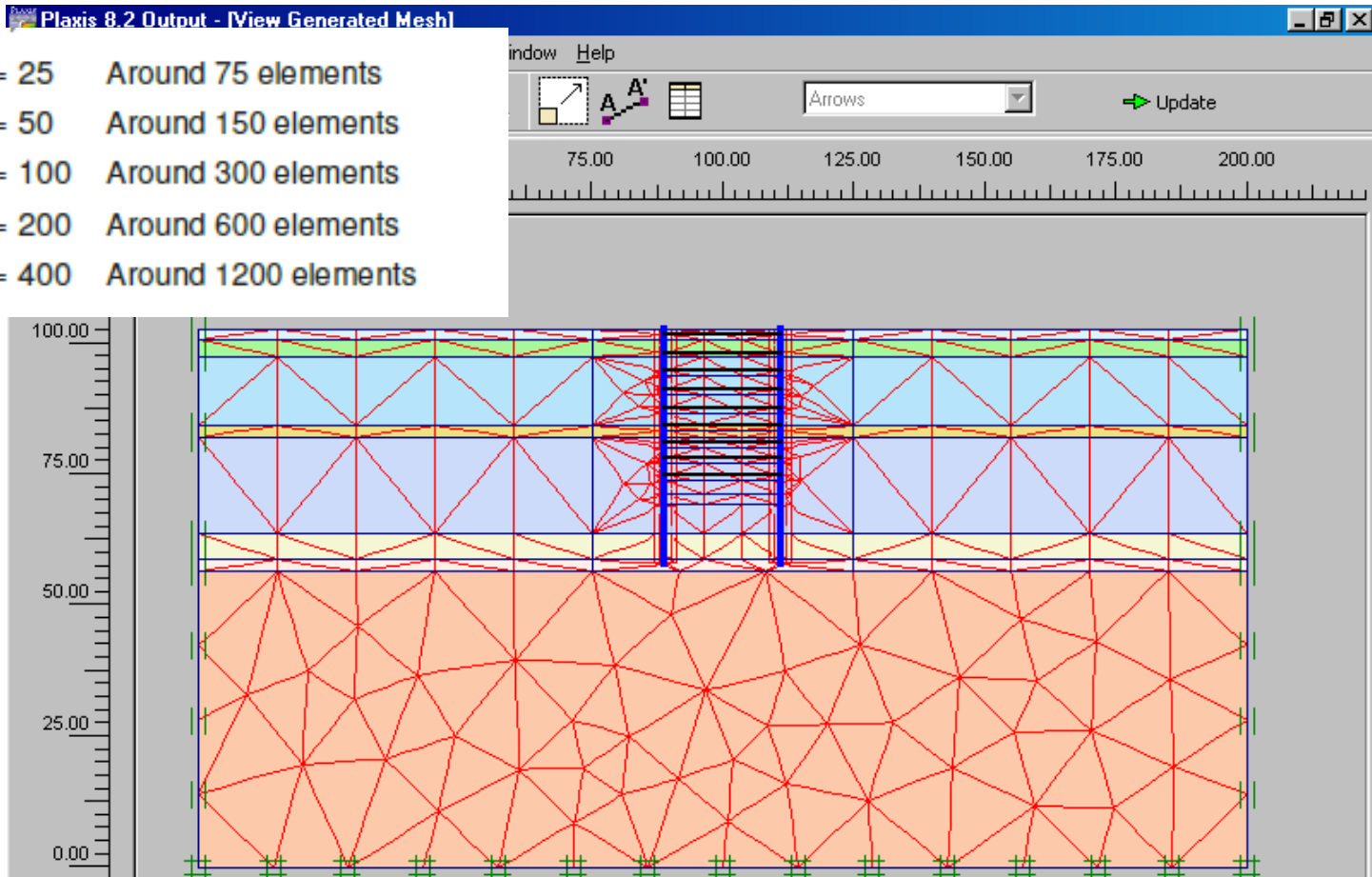


# Generating the mesh

Specify global coarseness and local refinement: Element size:

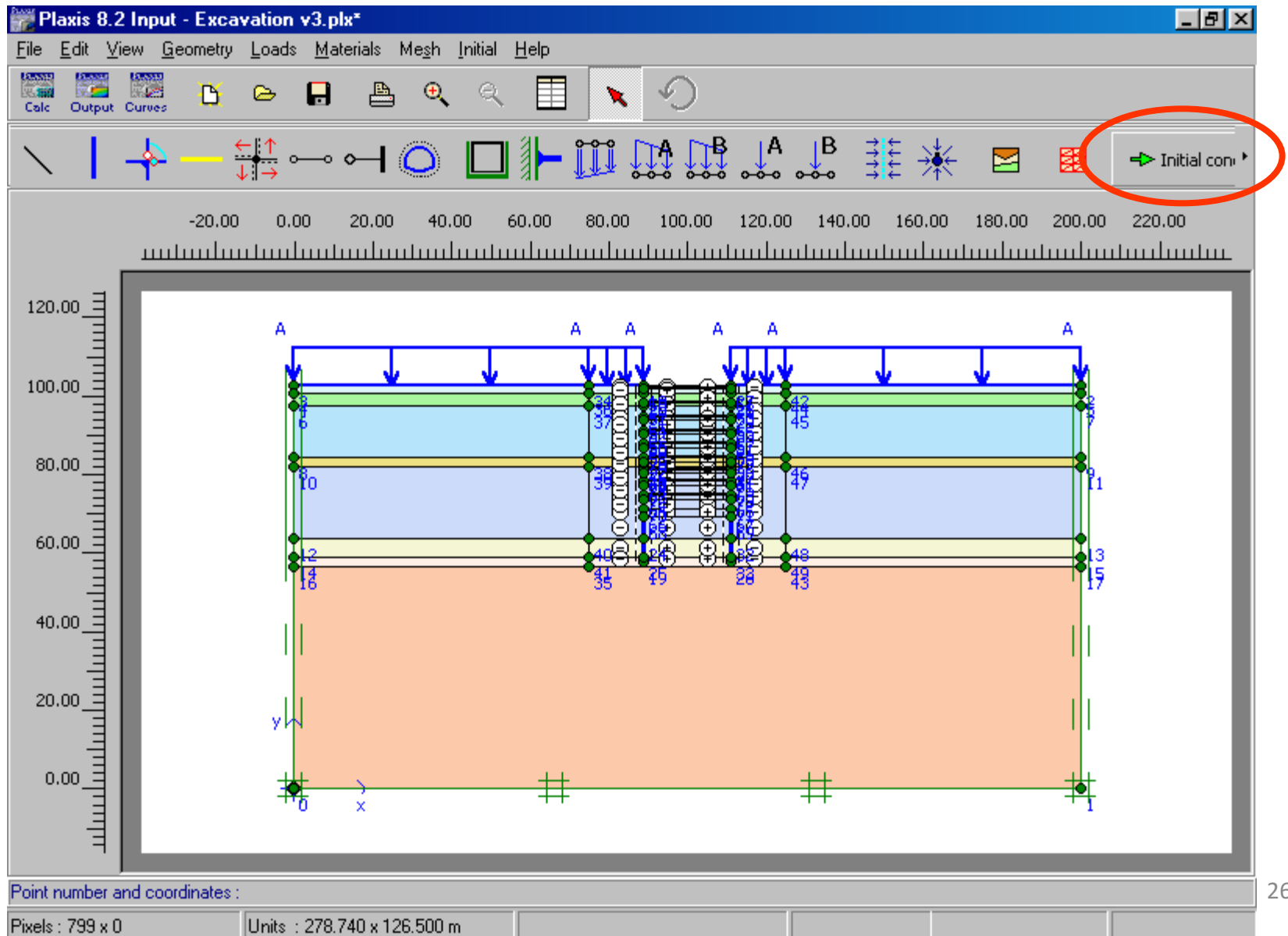
$$l_e = \sqrt{\frac{(x_{\max} - x_{\min})(y_{\max} - y_{\min})}{n_c}}$$

*Very coarse:*  $n_c = 25$  Around 75 elements  
*Coarse:*  $n_c = 50$  Around 150 elements  
*Medium:*  $n_c = 100$  Around 300 elements  
*Fine:*  $n_c = 200$  Around 600 elements  
*Very fine:*  $n_c = 400$  Around 1200 elements

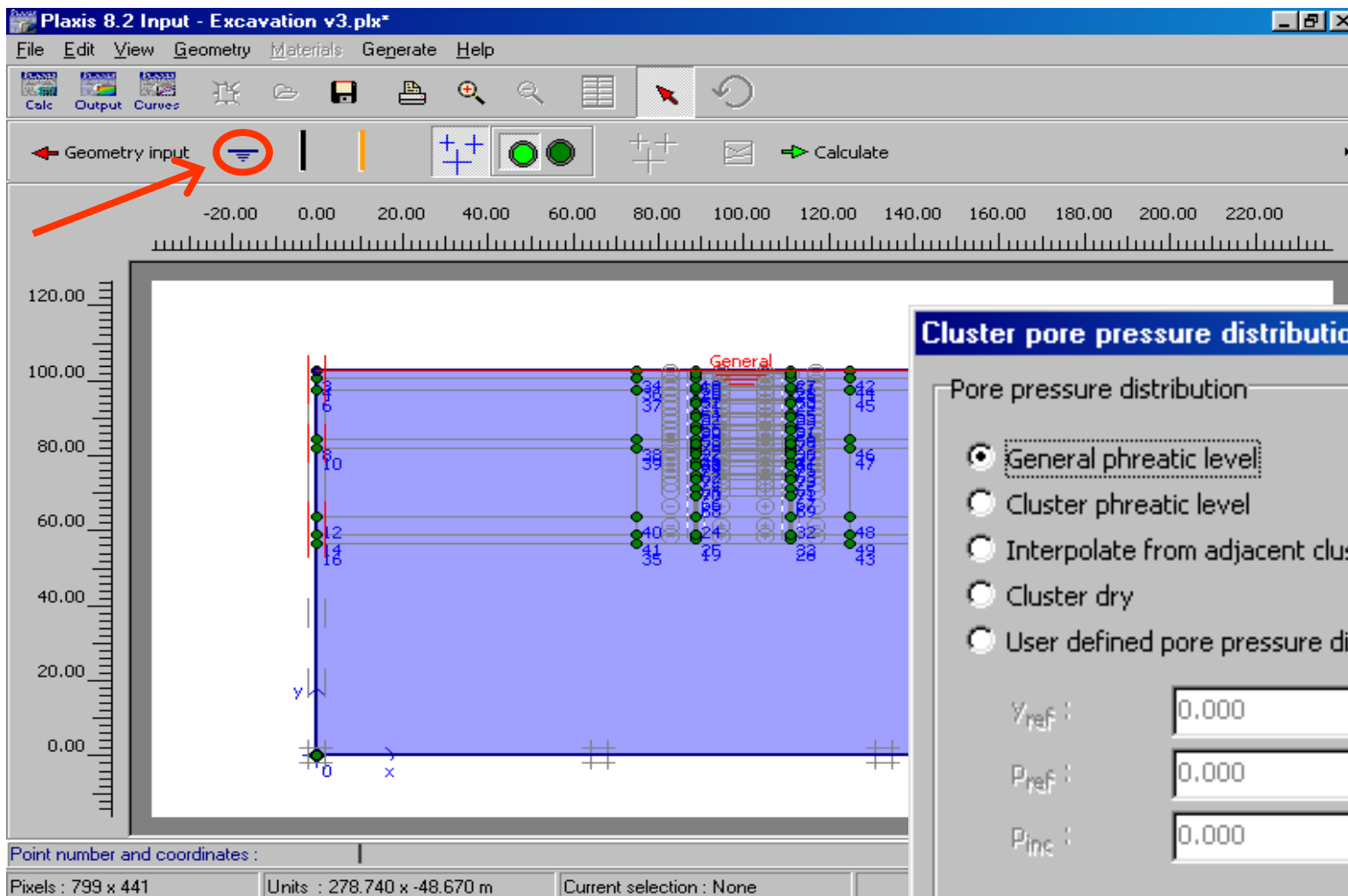


Local element size is 1.0 at all geometries and to reduce the element size by half set the local size factor to 0.5. Values between 0.05 to 5.0 is acceptable for local refinement

# Defining initial conditions



# Defining initial pore pressures



### Cluster pore pressure distribution

Pore pressure distribution

- ☒ General phreatic level
- ☐ Cluster phreatic level
- ☐ Interpolate from adjacent clusters or lines
- ☐ Cluster dry
- ☐ User defined pore pressure distribution

$V_{ref}$  :  m

$P_{ref}$  :   $\text{kN/m}^2$

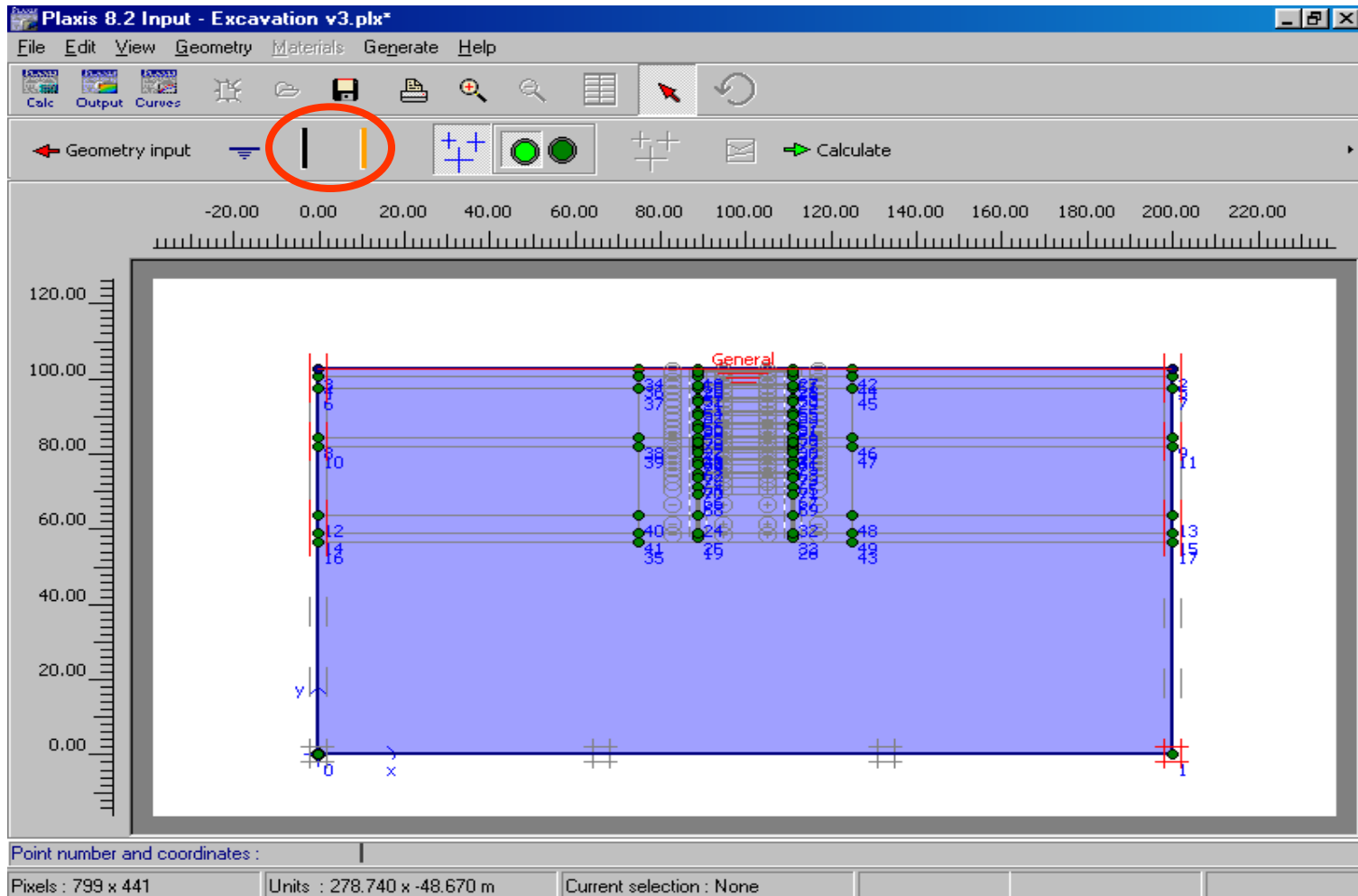
$P_{inc}$  :   $\text{kN/m}^2/\text{m}$

Note: - Pressures are negative  
- Increment defined per metre depth

OK Cancel Help

# Defining initial pore pressures

- Can specify closed flow and/or closed consolidation boundaries if necessary



# Calculating pore pressures

Plaxis 8.2 Input - Excavation v3.plx\*

File Edit View Geometry Materials Generate Help

Calc Output Curves

Geometry input

Calculate

General

Water pressure generation

Generate by

- ☒ Phreatic level
- ☐ Groundwater calculation (steady state) [Change configuration...](#)

Groundwater calculation

- ☐ Standard settings
- ☐ Manual settings [Define...](#)

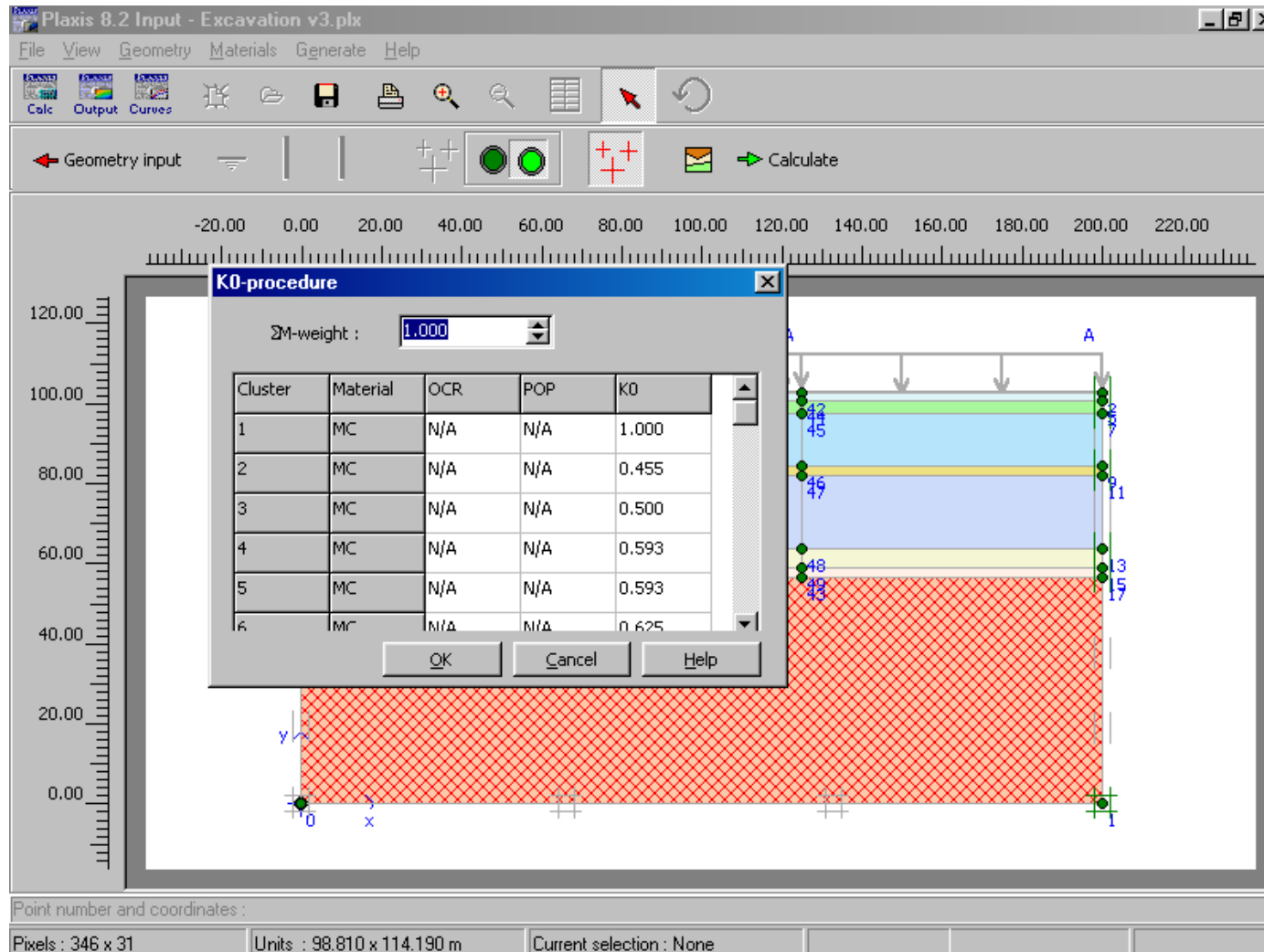
OK Cancel Help

Point number and coordinates :

Pixels : 799 x 441 Units : 278.740 x -48.670 m Current selection :

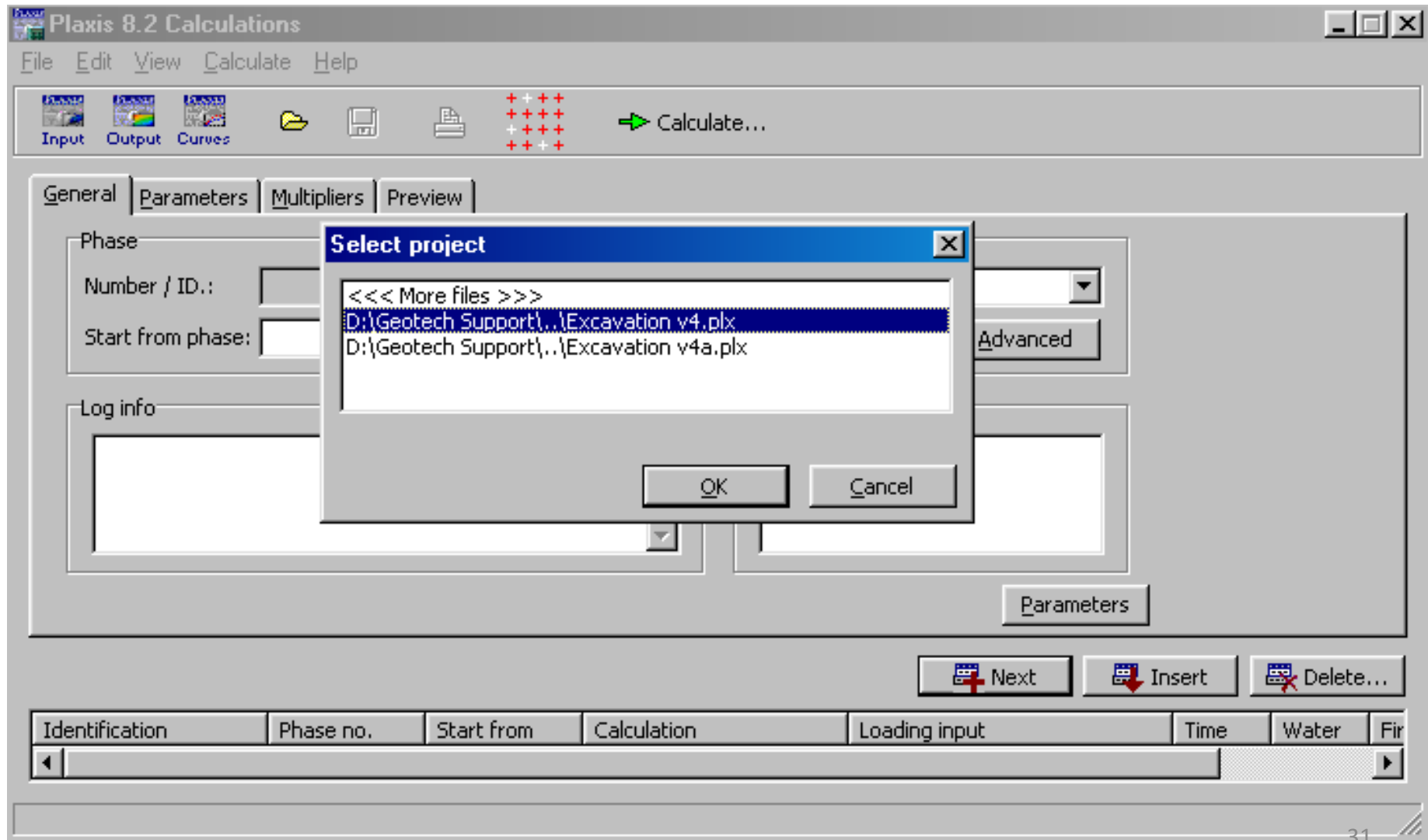
The screenshot displays the Plaxis 8.2 software interface for an excavation model. The main window shows a cross-section with a blue shaded area representing the excavation. A red line indicates the phreatic level. A grid of points is visible, with some points labeled with numbers (e.g., 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16, 17, 18, 19, 20, 21, 22, 23, 24, 25, 26, 27, 28, 29, 30, 31, 32, 33, 34, 35, 36, 37, 38, 39, 40, 41, 42, 43, 44, 45, 46, 47, 48, 49, 50, 51, 52, 53, 54, 55, 56, 57, 58, 59, 60, 61, 62, 63, 64, 65, 66, 67, 68, 69, 70, 71, 72, 73, 74, 75, 76, 77, 78, 79, 80, 81, 82, 83, 84, 85, 86, 87, 88, 89, 90, 91, 92, 93, 94, 95, 96, 97, 98, 99, 100). A dialog box titled 'Water pressure generation' is open, showing options for generating water pressure. The 'Phreatic level' option is selected under 'Generate by'. The 'Groundwater calculation' section shows 'Standard settings' selected. The 'OK', 'Cancel', and 'Help' buttons are at the bottom of the dialog box.

# Generate initial soil stresses



- Note: default  $K_0$  value is  $1 - \sin \phi$  in Plaxis
- Can also use gravity loading method

# PLAXIS CALCULATIONS



# Type of Calculations

Plaxis 8.2 Calculations - Excavation v3.plx

File Edit View Calculate Help

Input Output Curves [Icons] Calculate...

General Parameters Multipliers Preview

Phase

Number / ID.: 1 <Phase 1>

Start from phase: 0 - Initial phase

Calculation type

Plastic

Advanced

Log info

Comments

Identification	Phase no.	Start from	Calculation	Loading input	Time	Water	Fir
Initial phase	0	0	N/A	N/A	0.00 ...	0	0
→ <Phase 1>	1	0	Plastic	Staged construction	0.00 ...	0	

- Plastic calculations
- Consolidation analysis
- Phi-c reduction



# Defining construction stages

- Define modelling phase, Insert next phase

Plaxis 8.2 Calculations - Excavation v1.plx

File Edit View Calculate Help

Input Output Curves Calculate...

General Parameters Multipliers Preview

Control parameters

Additional Steps: 250

☒ Reset displacements to zero  
☐ Ignore undrained behaviour  
☒ Delete intermediate steps

Iterative procedure

☒ Standard setting  
☐ Manual setting

Define...

Loading input

☒ Staged construction  
☐ Total multipliers  
☐ Incremental multipliers

Advanced...

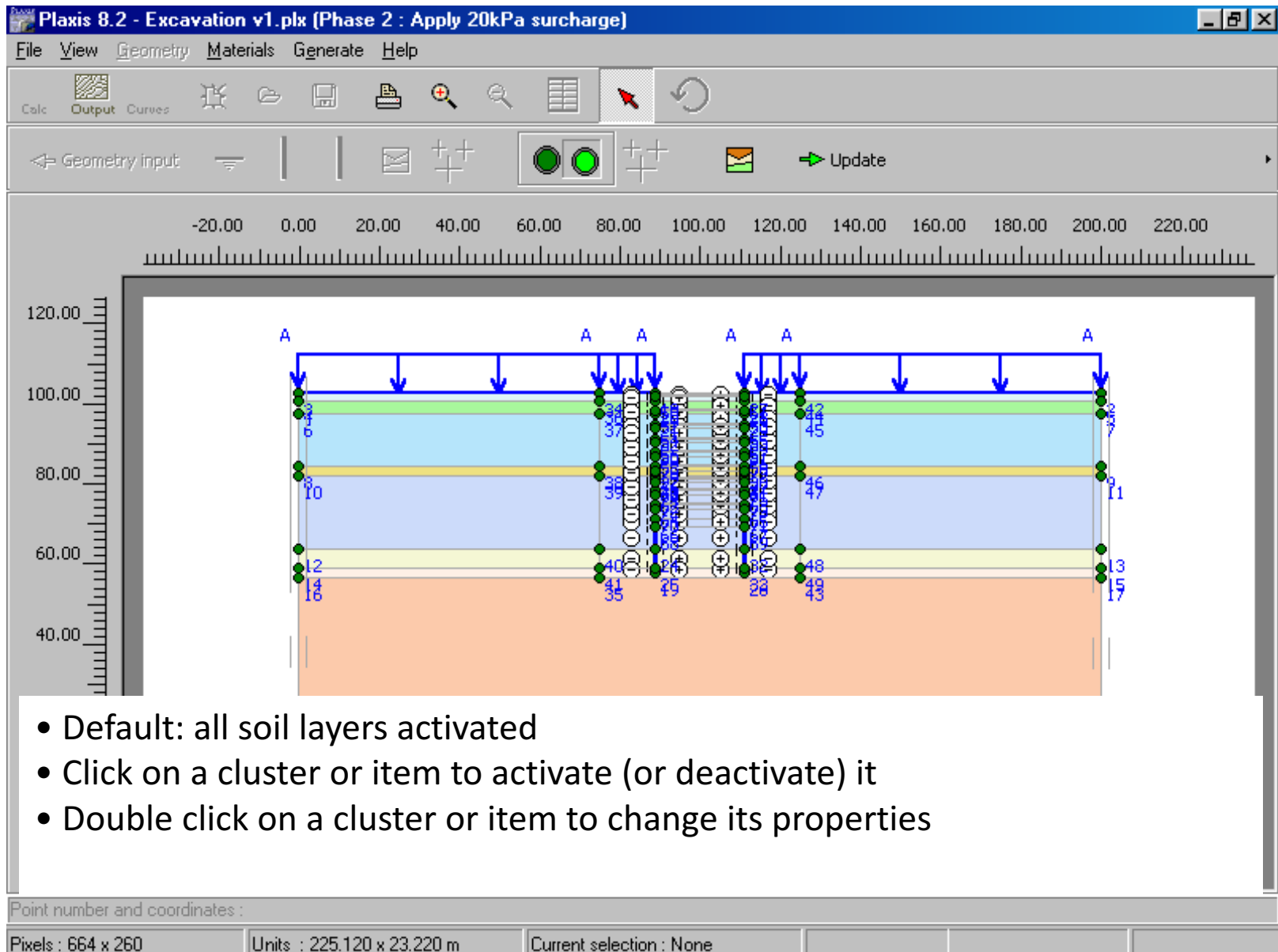
Time interval : 0.0000 day  
Estimated end time : 0.0000 day

Define...

Next Insert Delete...

Identification	Phase no.	Start from	Calculation	Loading input	Time	Water
Initial phase	0	0	N/A	N/A	0.00 ...	0
Install Dwall	1	0	Plastic	Staged construction	0.00 ...	1
Apply 20kPa surc...	2	1	Plastic	Staged construction	0.00 ...	2

# Activating elements for calculation



Plaxis 8.2 - Excavation v1.plx (Phase 2 : Apply 20kPa surcharge)

File View Geometry Materials Generate Help

Calc Output Curves

Geometry input

Update

Point number and coordinates :

Pixels : 664 x 260 Units : 225.120 x 23.220 m Current selection : None

- Default: all soil layers activated
- Click on a cluster or item to activate (or deactivate) it
- Double click on a cluster or item to change its properties

# Changing groundwater conditions

Plaxis 8.2 - Excavation v1.plx (Phase 10 : 4th excavate)\*

File View Geometry Materials Generate Help

Calc Output Curves

Geometry input

Water pressure generation

Generate by

- ☐ Phreatic level
- ☒ Groundwater calculation (steady state) Change configuration...

Groundwater calculation

- ☒ Standard settings Define...
- ☐ Manual settings

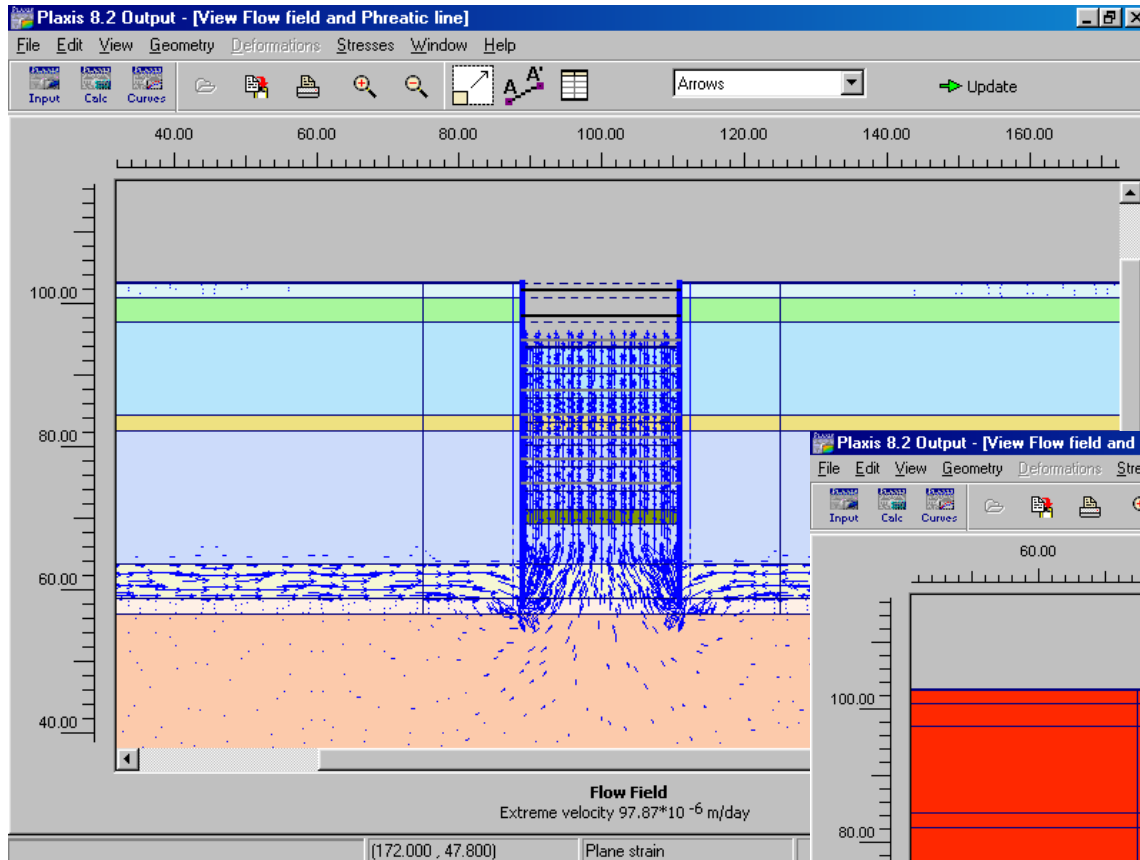
OK Cancel Help

Point on phreatic level :

Pixels : 1439 x 175 Units : 93.210 x 110.470 m Current selection : None

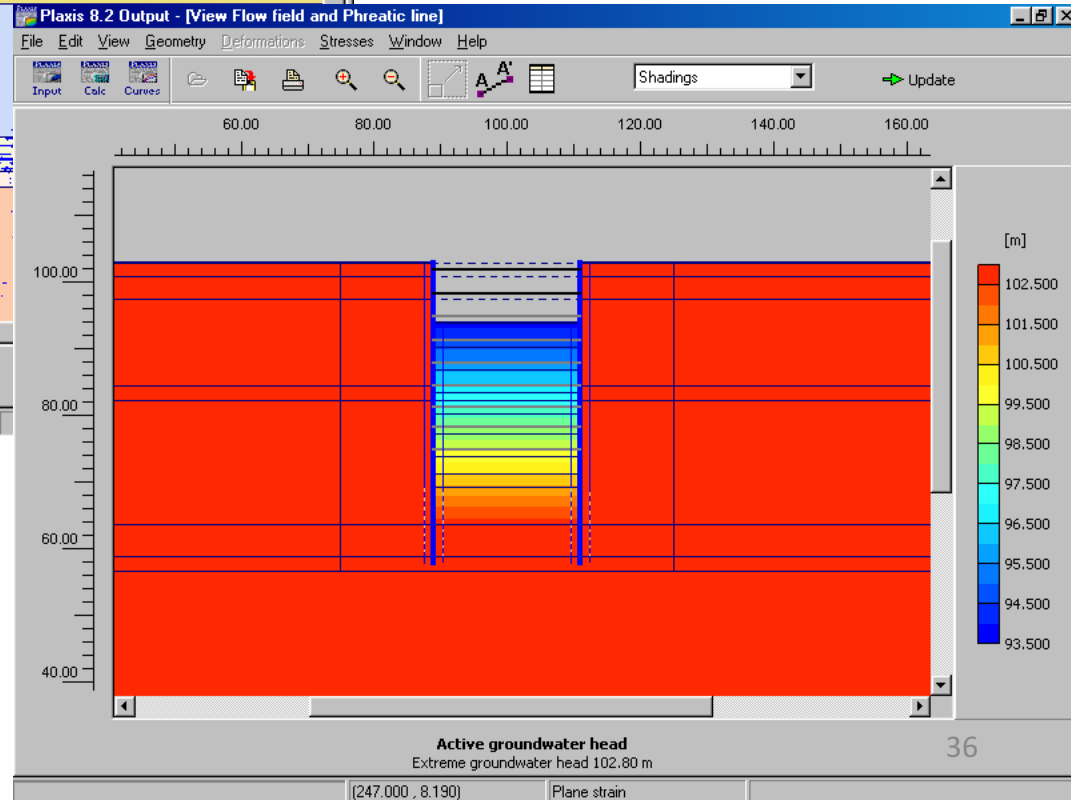
- To de-water, redefine water conditions

# Groundwater calculations

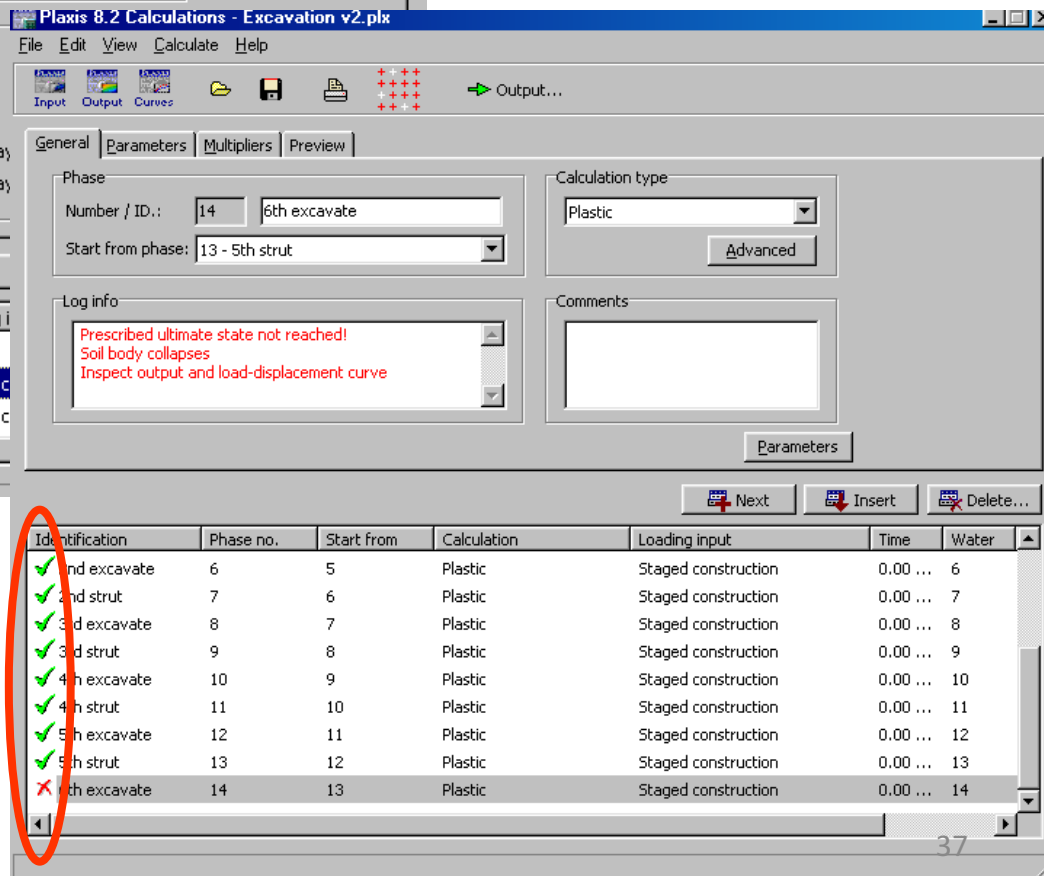
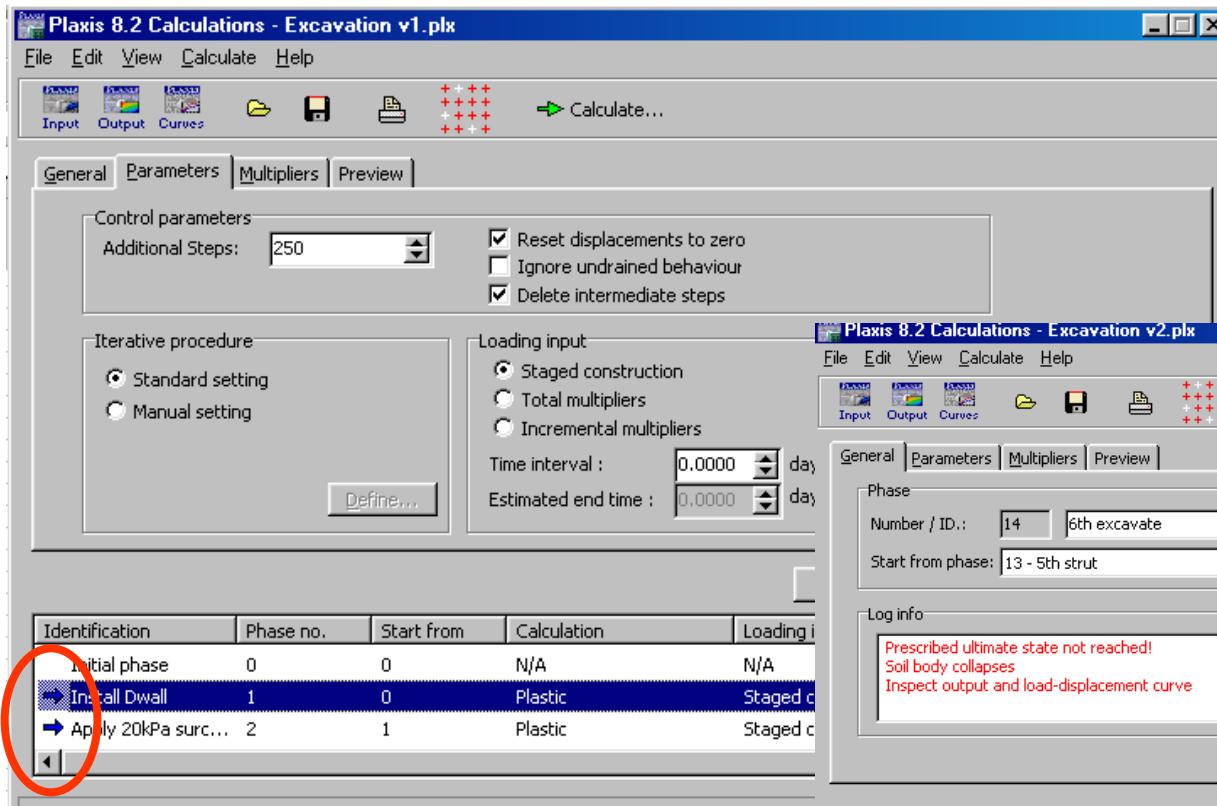


Flow field arrows

Active groundwater level  
contours



# Selecting calculation phase



Arrow -- Check -- Cross

# Achieving convergence

Plaxis 8.2 Calculations - Excavation v2.plx

File Edit View Calculate Help

Input Output Curves

General Parameters Multipliers Preview

Phase

Number / ID.: 14 6th excavate

Start from phase: 13 5th strut

Calculation type

Plastic

Advanced

Log info

Prescribed ultimate state not reached!  
Soil body collapses  
Inspect output and load-displacement curve

Comments

Parameters

Next Ins

Identification	Phase no.	Start from	Calculation	Loading input
✓ 2nd excavate	6	5	Plastic	Staged construction
✓ 2nd strut	7	6	Plastic	Staged construction
✓ 3rd excavate	8	7	Plastic	Staged construction
✓ 3rd strut	9	8	Plastic	Staged construction
✓ 4th excavate	10	9	Plastic	Staged construction
✓ 4th strut	11	10	Plastic	Staged construction
✓ 5th excavate	12	11	Plastic	Staged construction
✓ 5th strut	13	12	Plastic	Staged construction
✗ 6th excavate	14	13	Plastic	Staged construction

Plaxis 8.2 Calculations - Excavation v2.plx

File Edit View Calculate Help

Input Output Curves

General Parameters Multipliers Preview

Control parameters

Additional Steps: 250

Reset displacements to zero  
Ignore undrained behaviour

Iterative procedure

☐ Standard setting  
☒ Manual setting

Define...

Manual Setting

Parameters

Tolerated error: 0.03

Over relaxation: 1.2000

Maximum iterations: 60

Desired minimum: 6

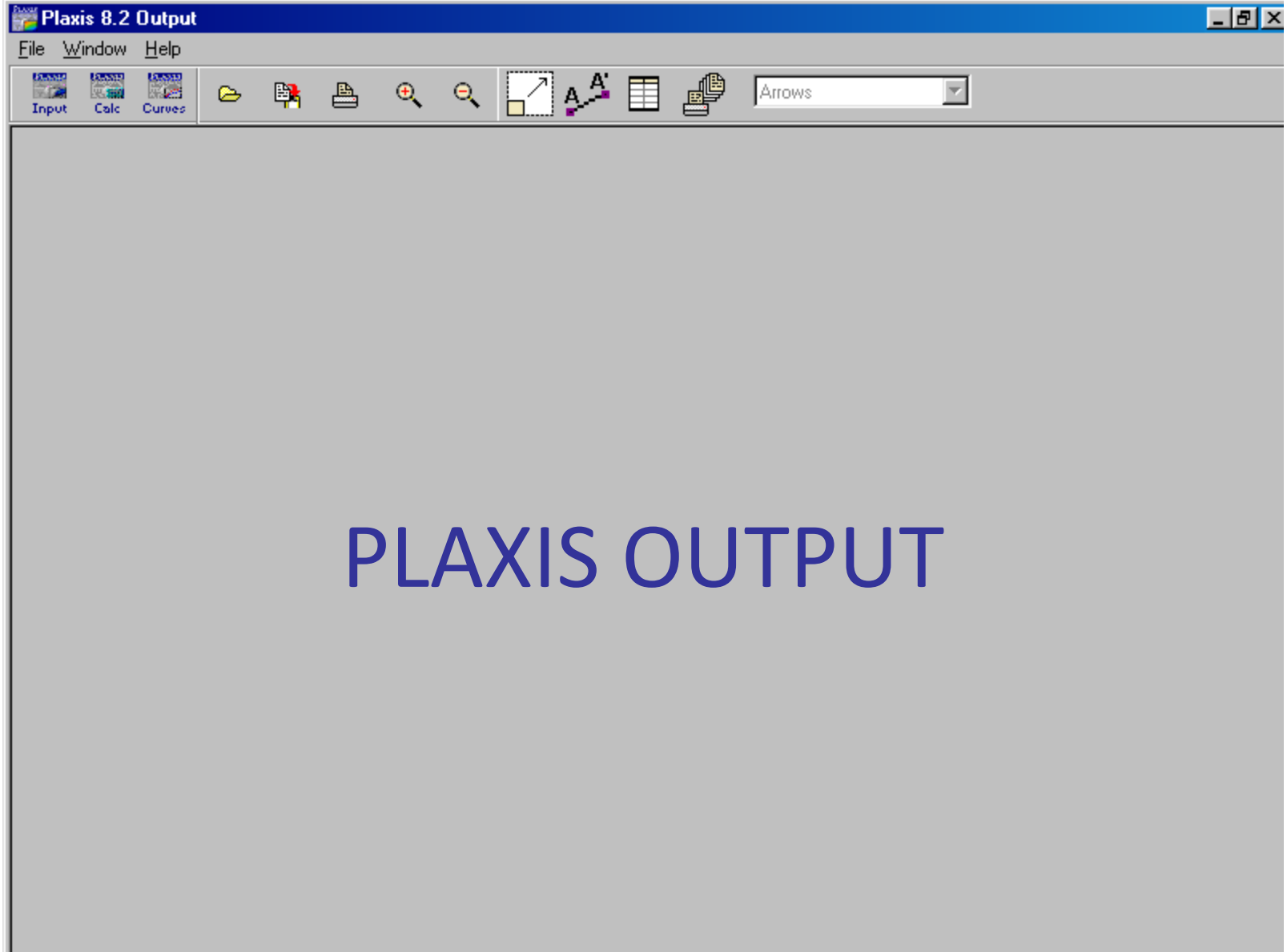
Desired maximum: 15

☒ Arc-length control

OK Cancel Help

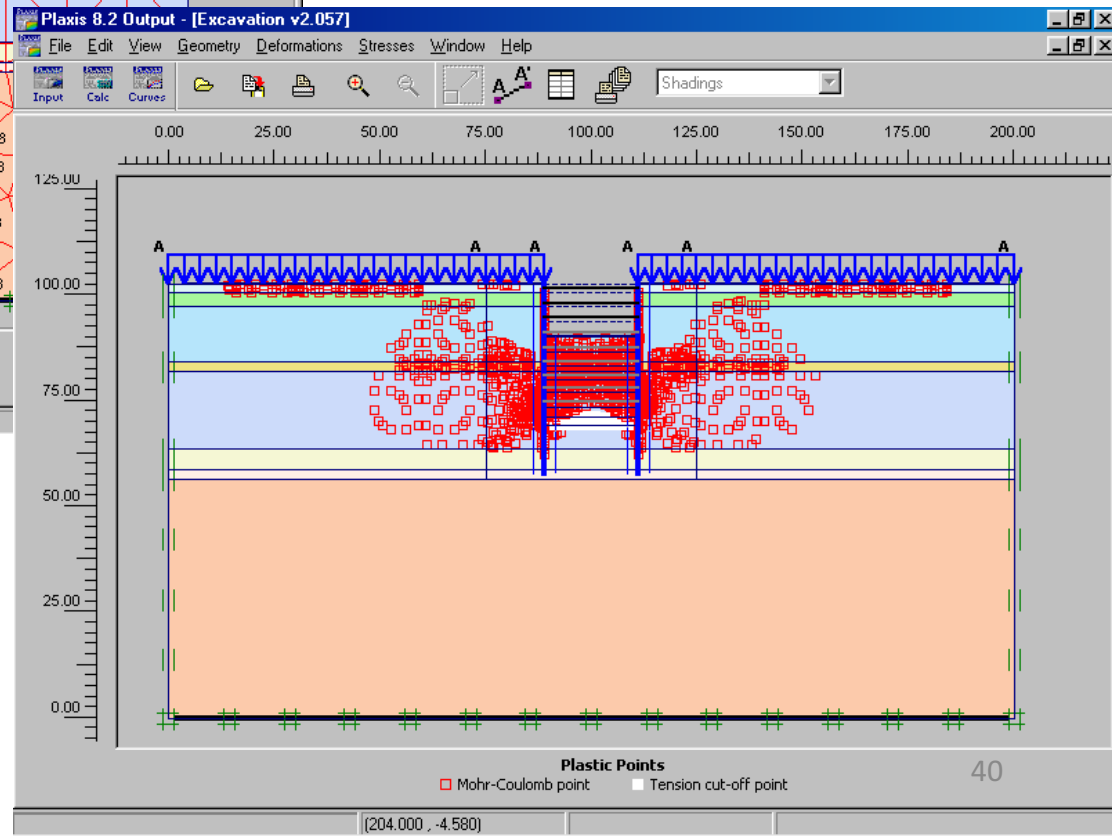
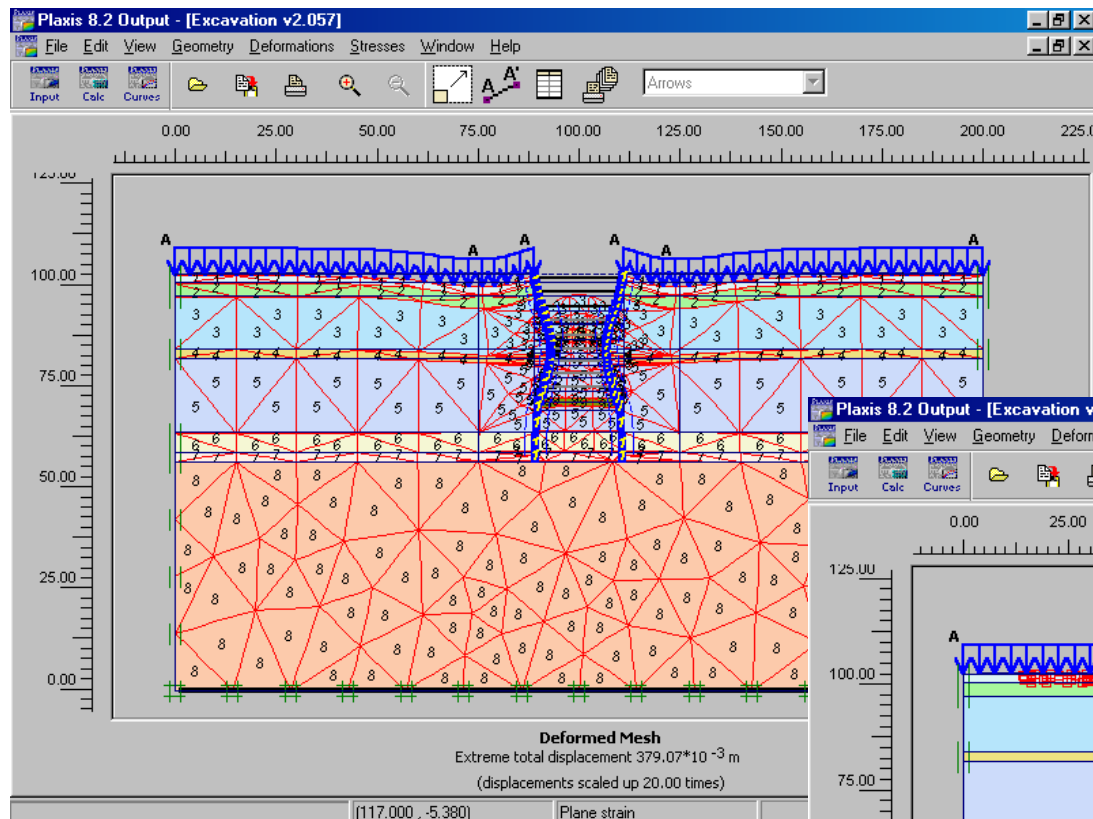
Identification	Phase no.	Start from	Calculation	Loading input	Time	Water
✓ 2nd excavate	6	5			0.00 ...	6
✓ 2nd strut	7	6			0.00 ...	7
✓ 3rd excavate	8	7			0.00 ...	8
✓ 3rd strut	9	8			0.00 ...	9
✓ 4th excavate	10	9	Plastic	Staged construction	0.00 ...	10
✓ 4th strut	11	10	Plastic	Staged construction	0.00 ...	11
✓ 5th excavate	12	11	Plastic	Staged construction	0.00 ...	12
✓ 5th strut	13	12	Plastic	Staged construction	0.00 ...	13
✗ 6th excavate	14	13	Plastic	Staged construction	0.00 ...	14

38



# Deformed mesh & plastic points

Deformed mesh



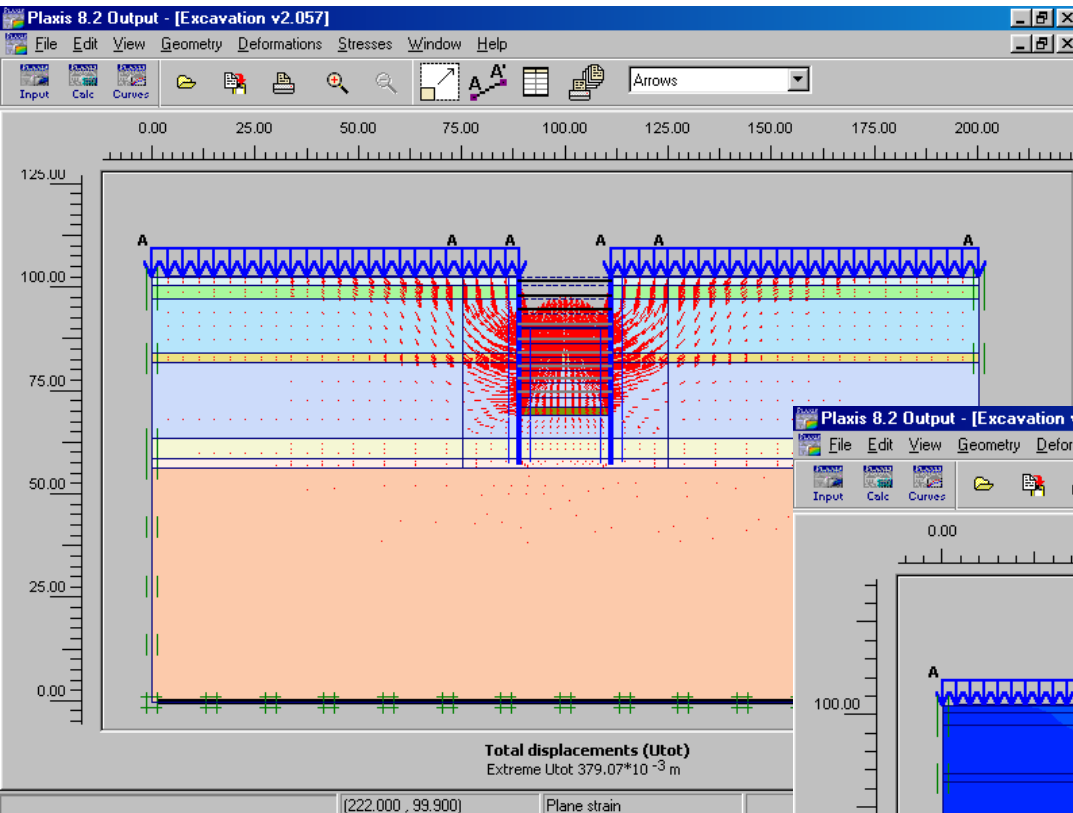
Red - Yielded Points

White - Tension cut-off

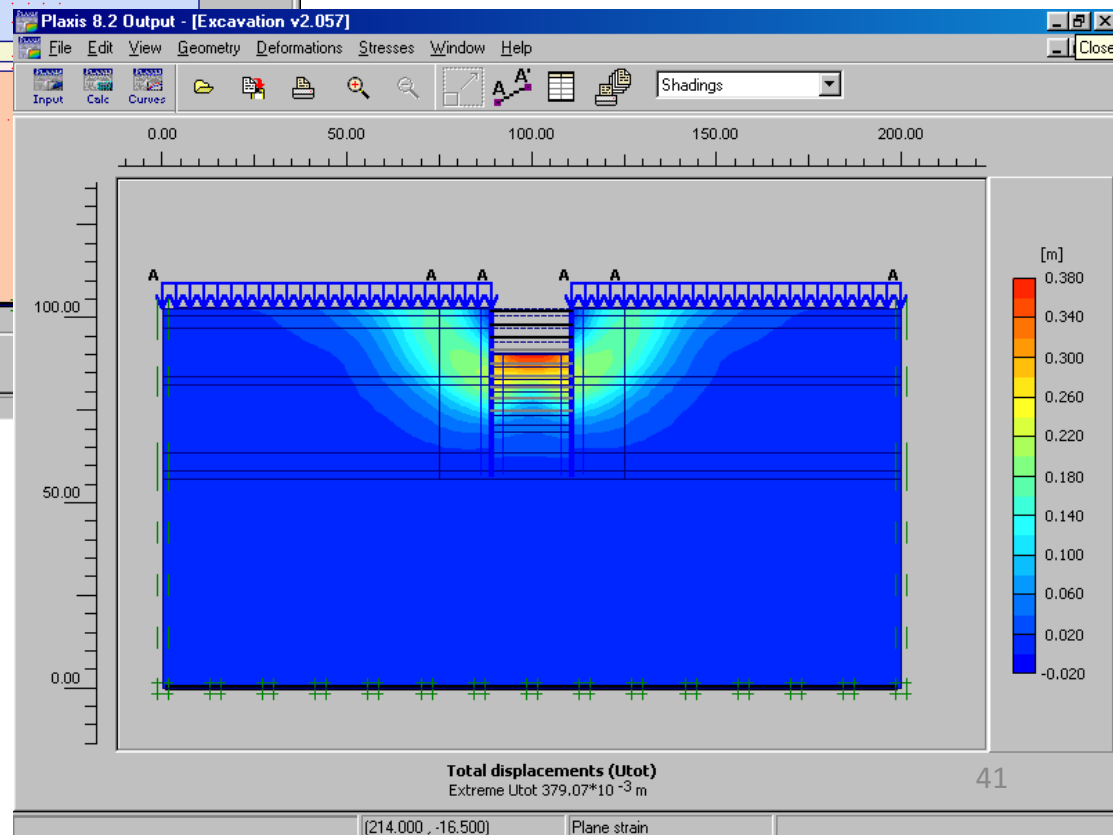


# Soil Displacement Plots

Arrows

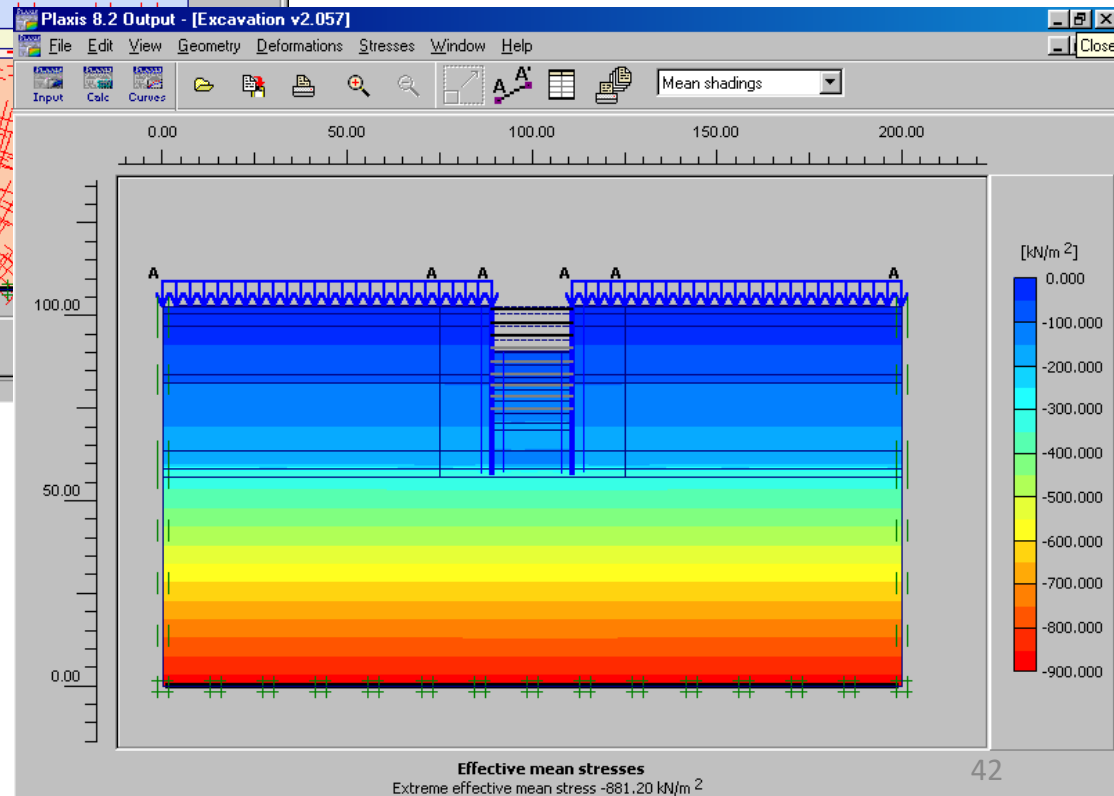
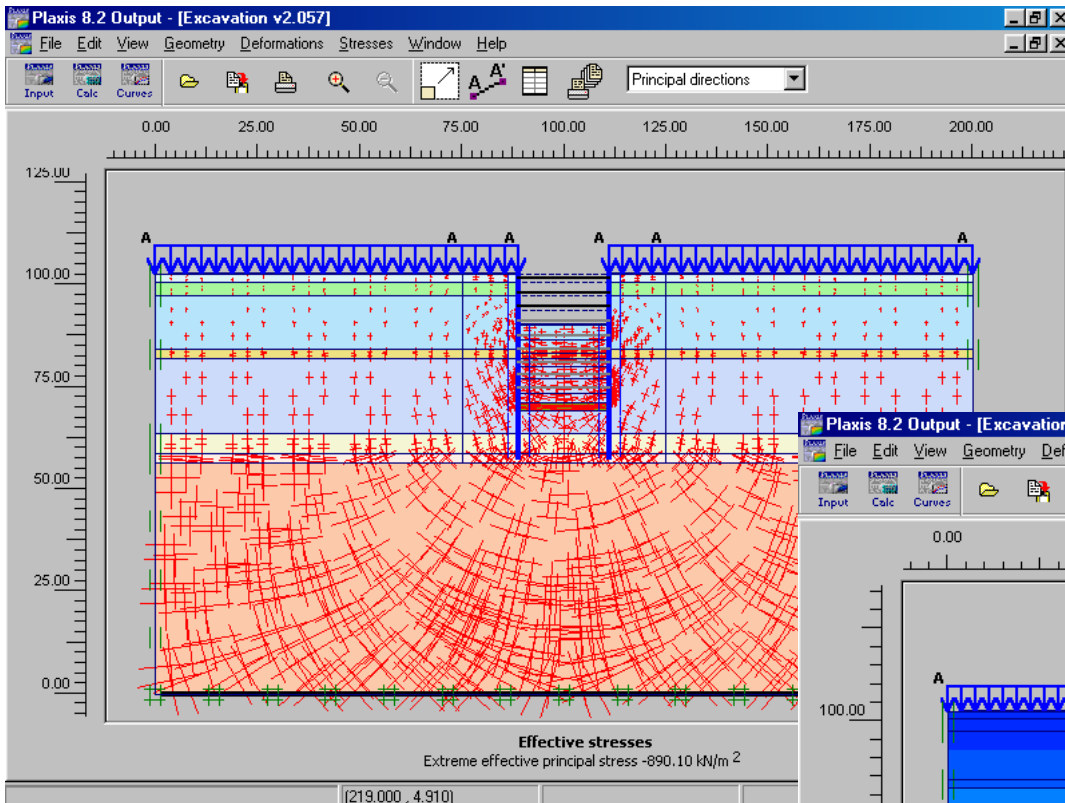


Contours



# Soil Stress Plots

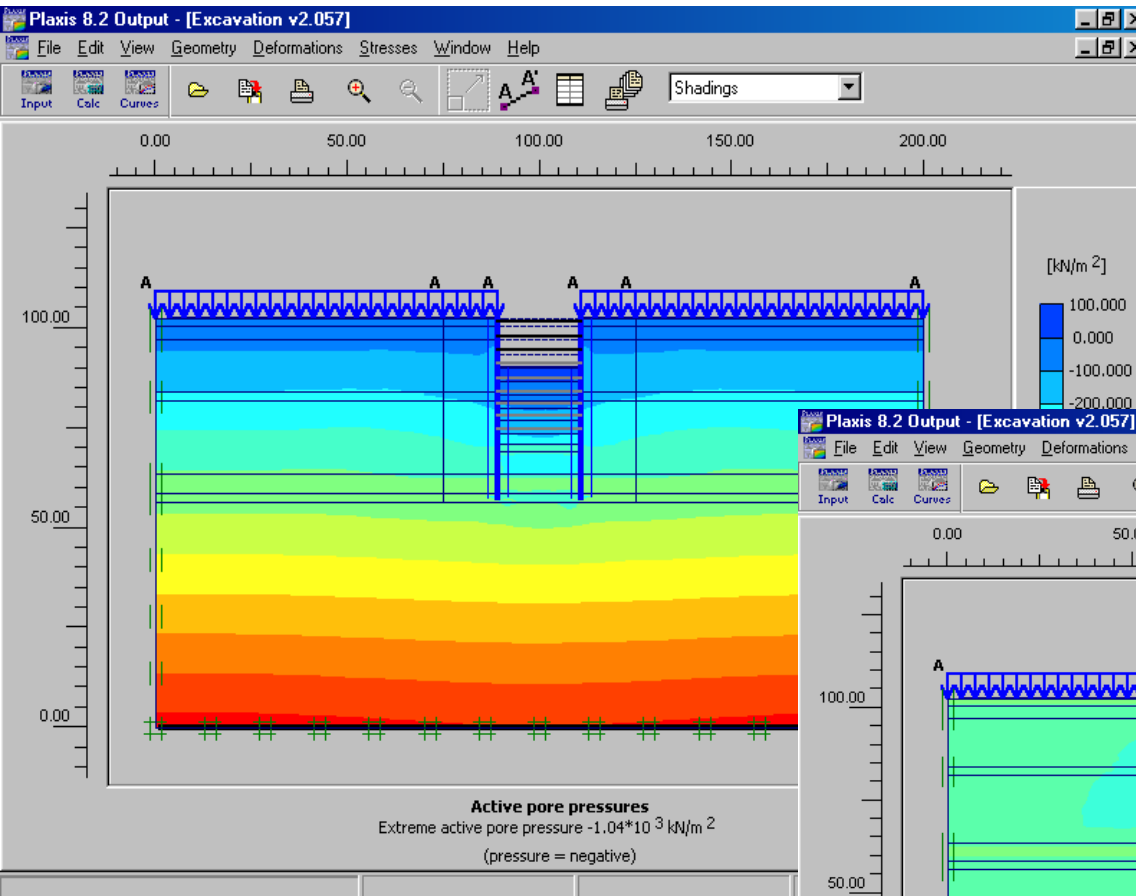
Principal directions



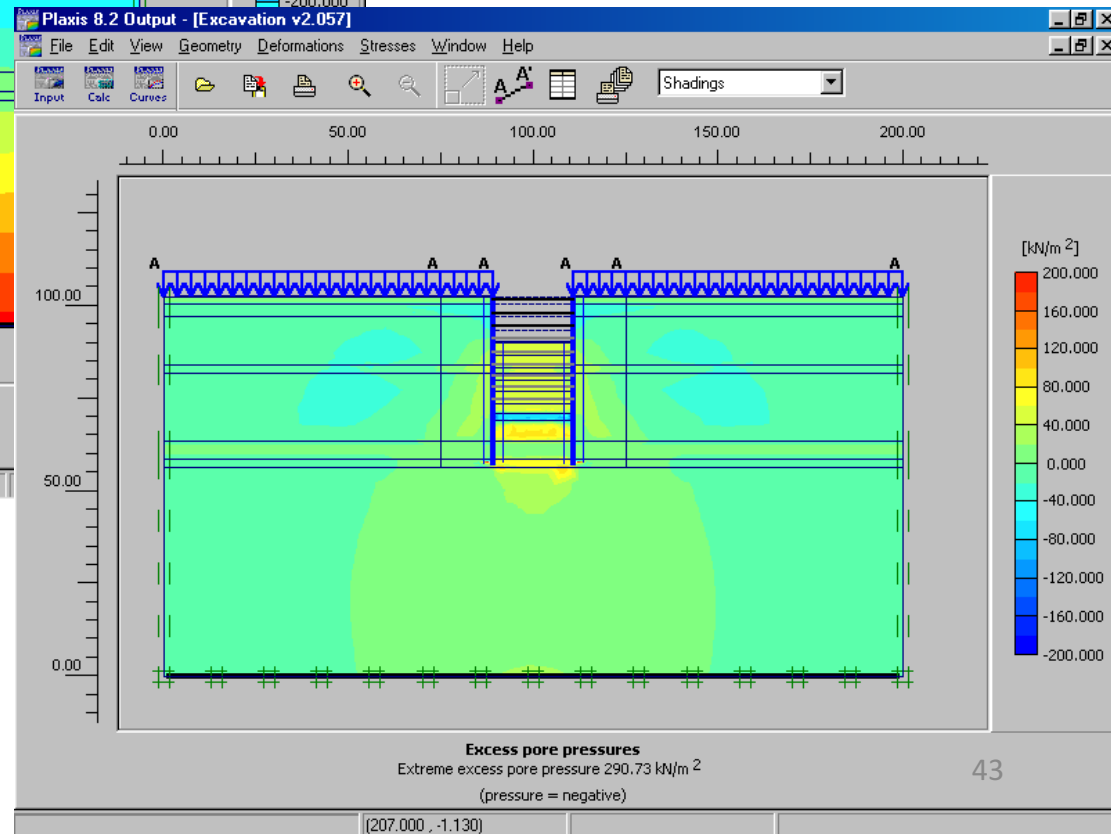
Contours

# Pore Water Pressures

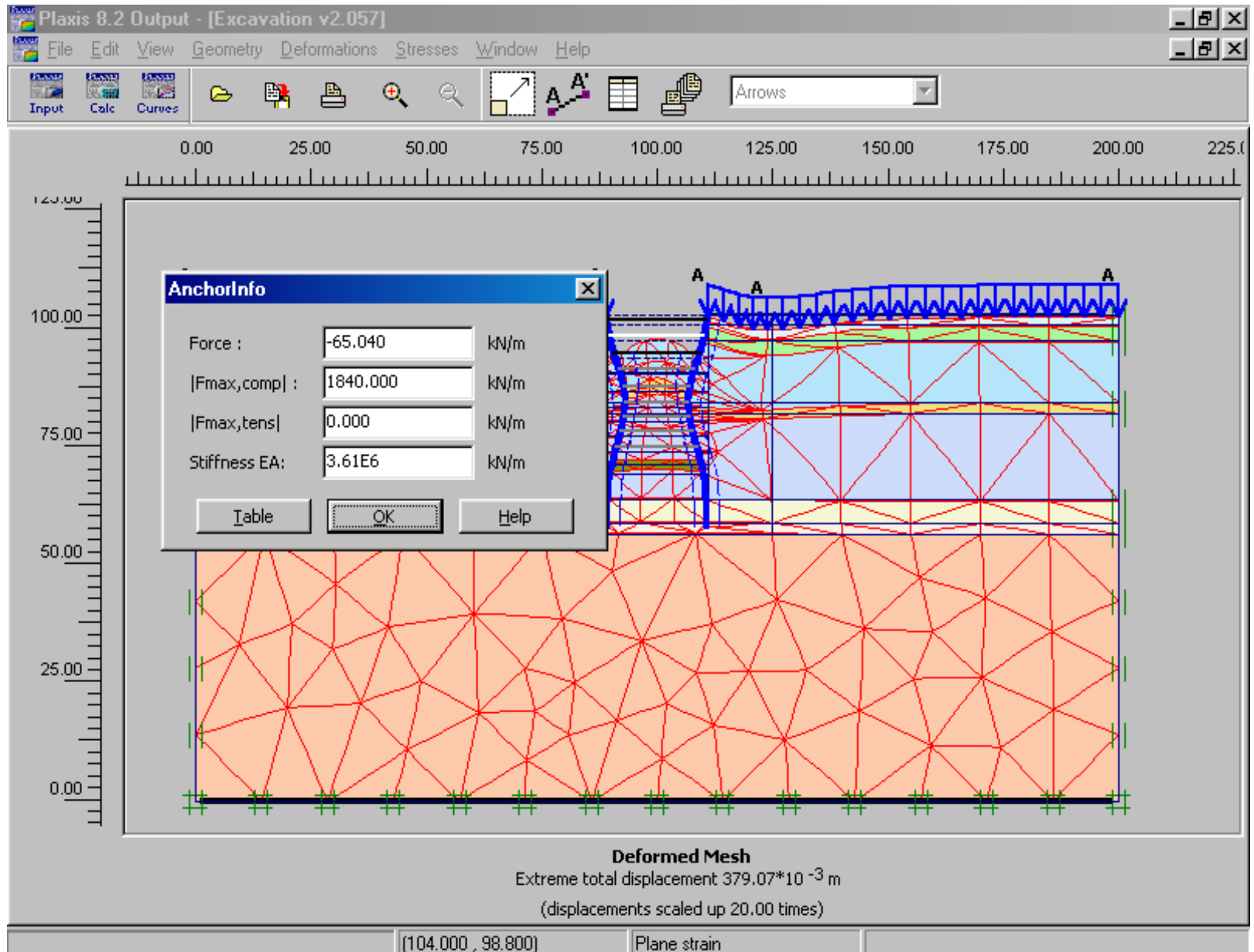
Active pore Pressures



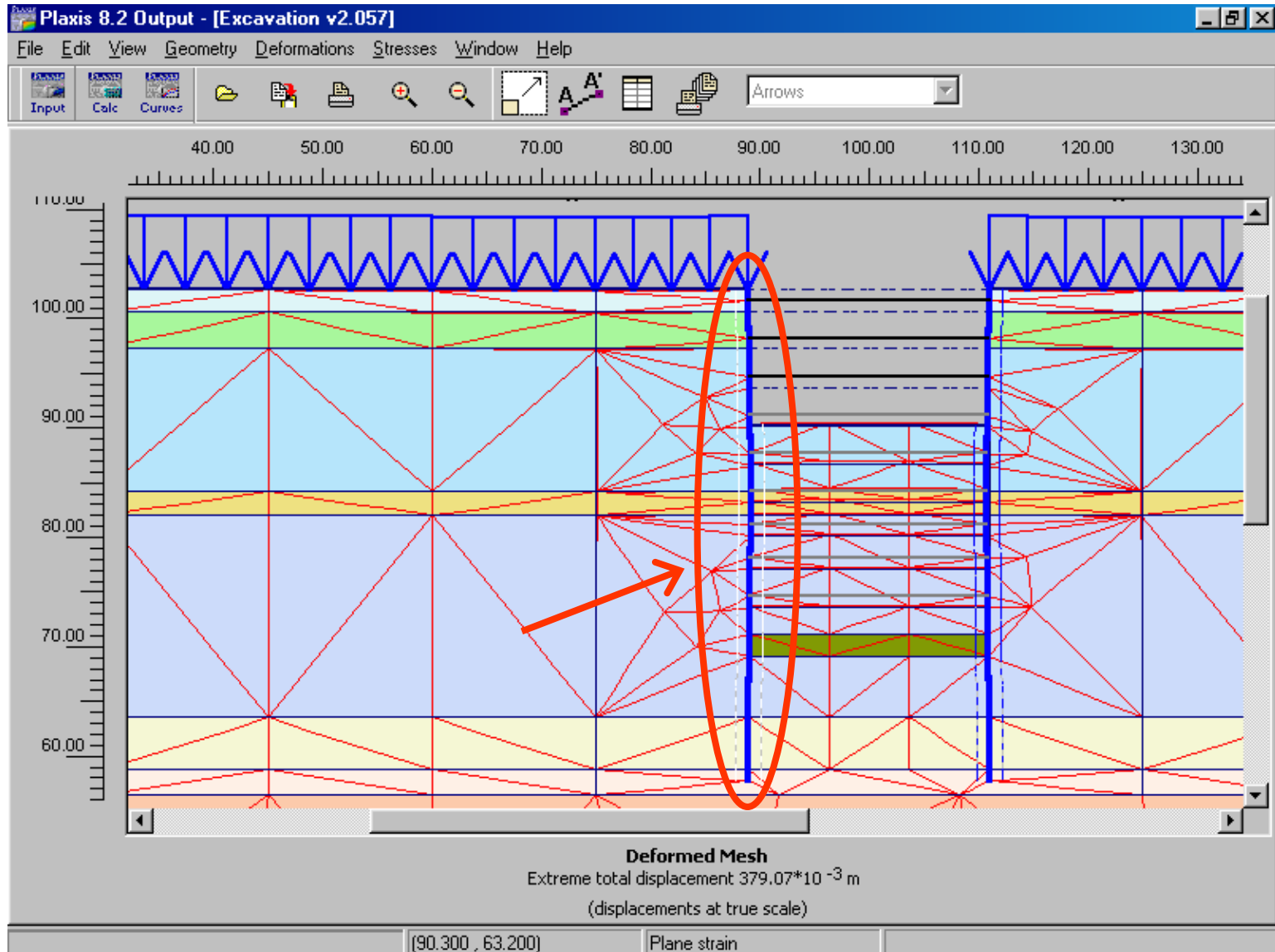
Excess pore pressures



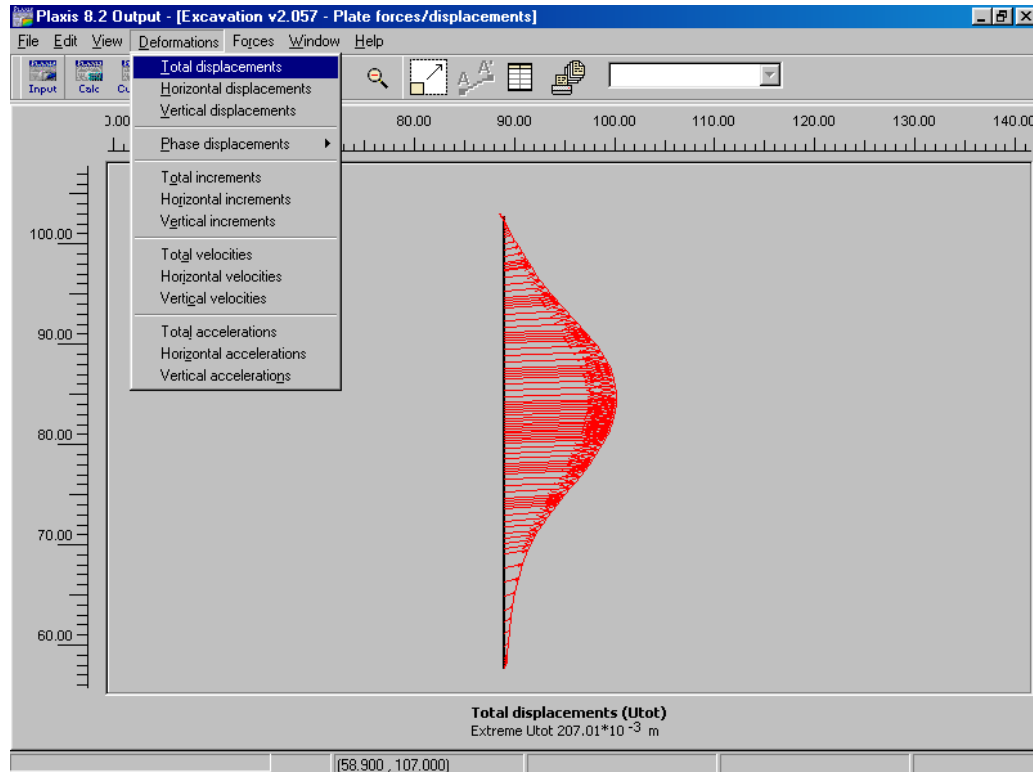
# Strut loads



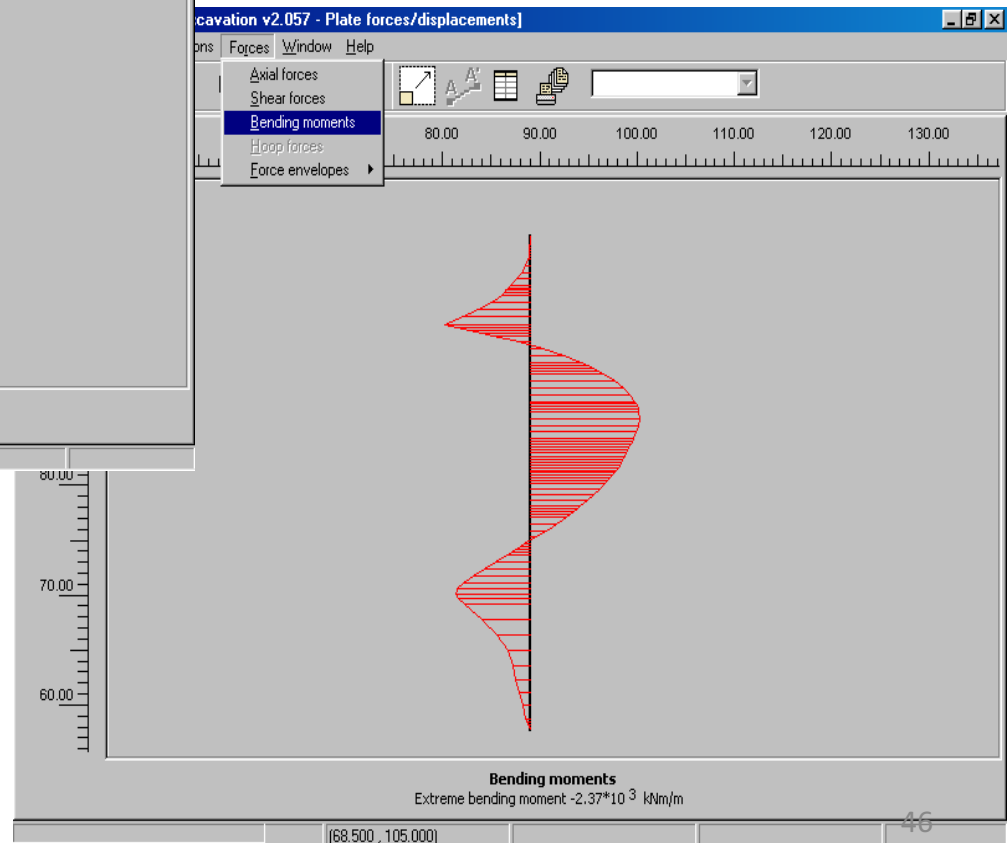
# Plate element plots



# Plate element plots

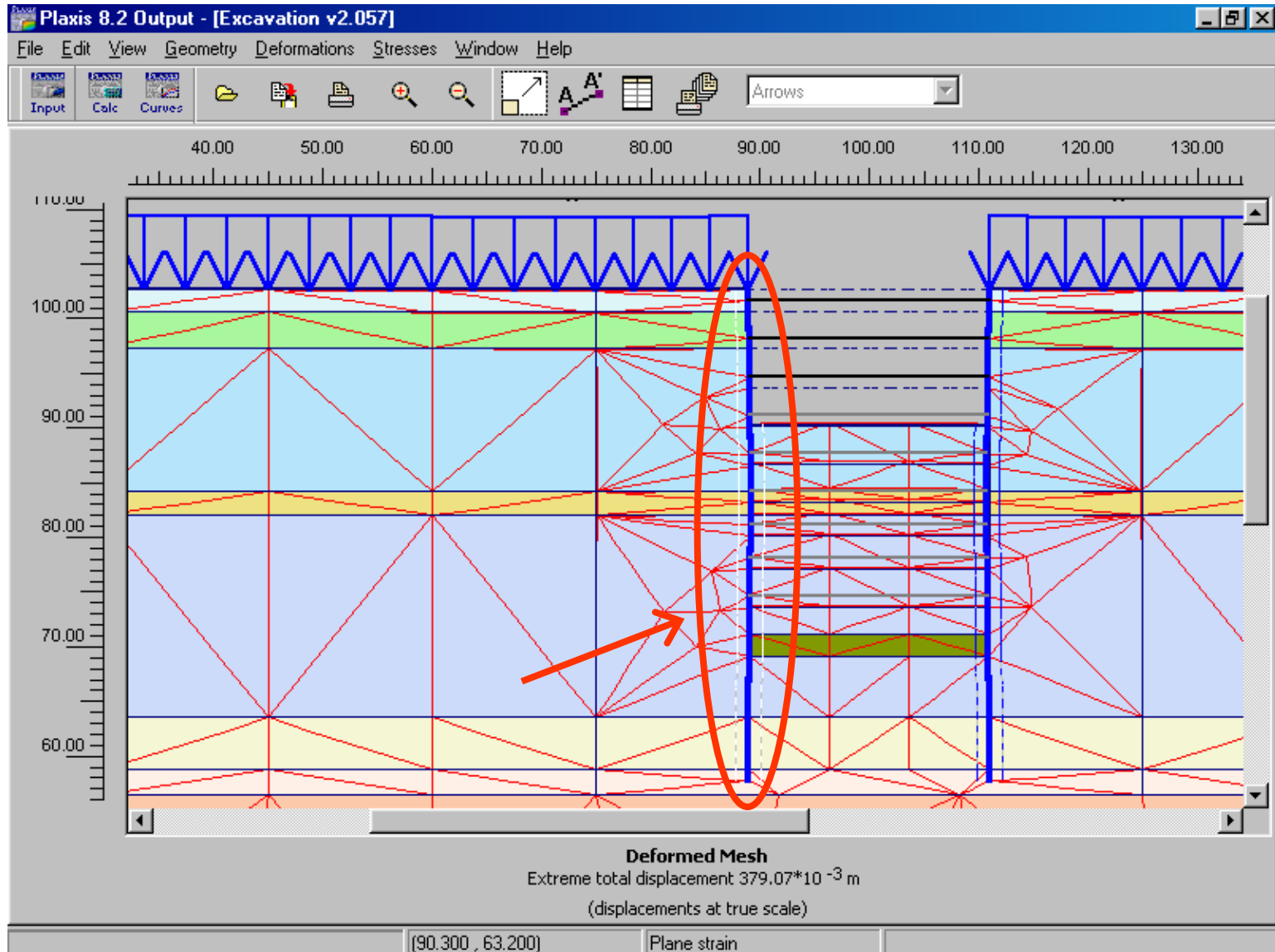


Total displacements

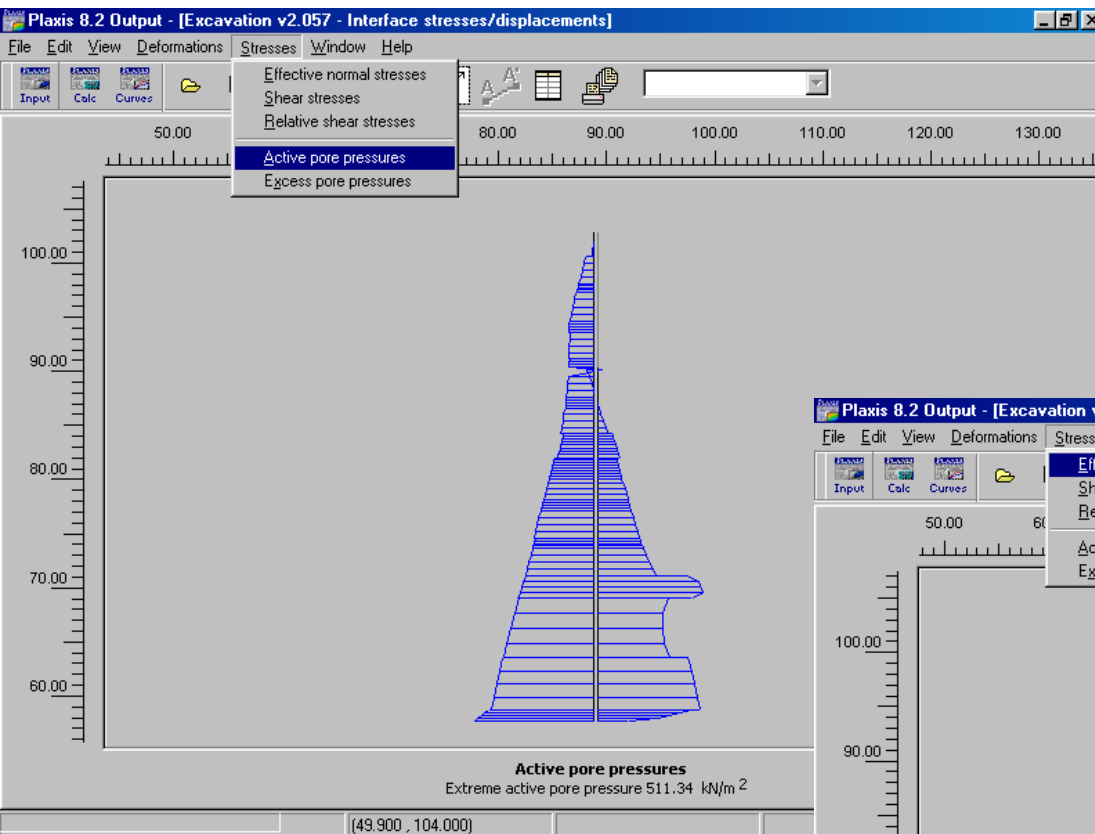


Bending moments

# Interface element plots

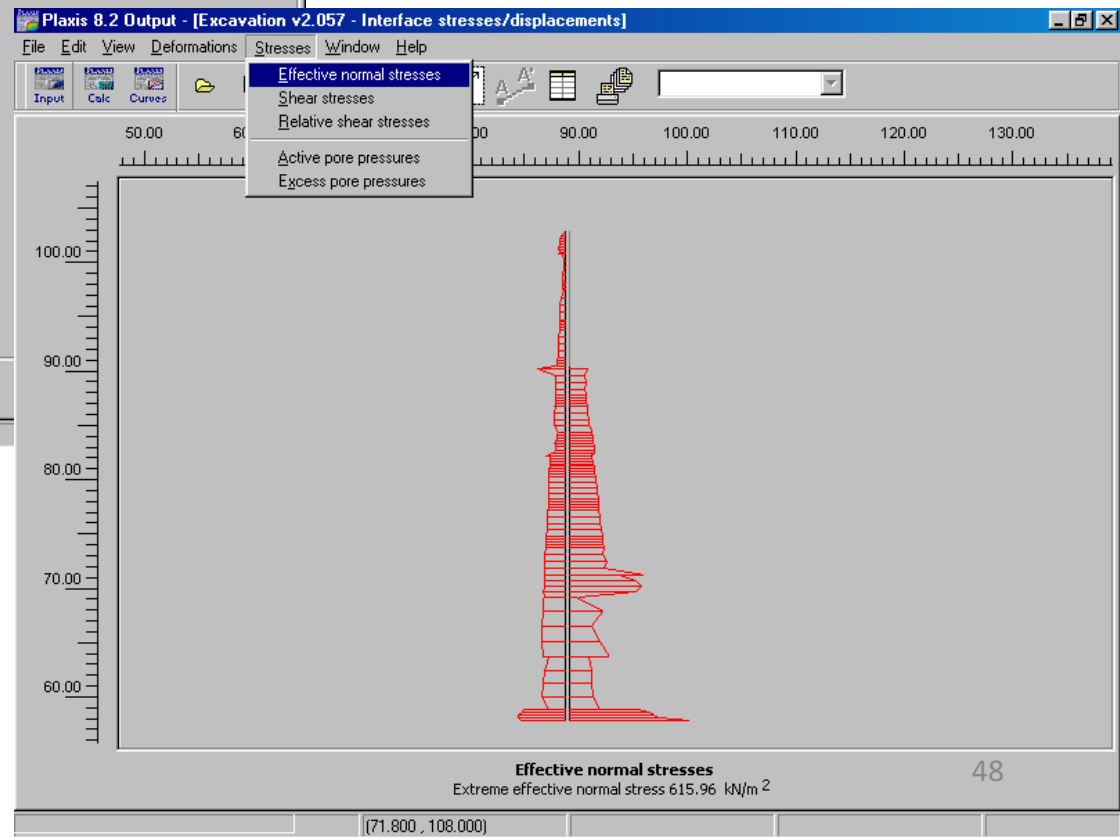


# Interface element plots

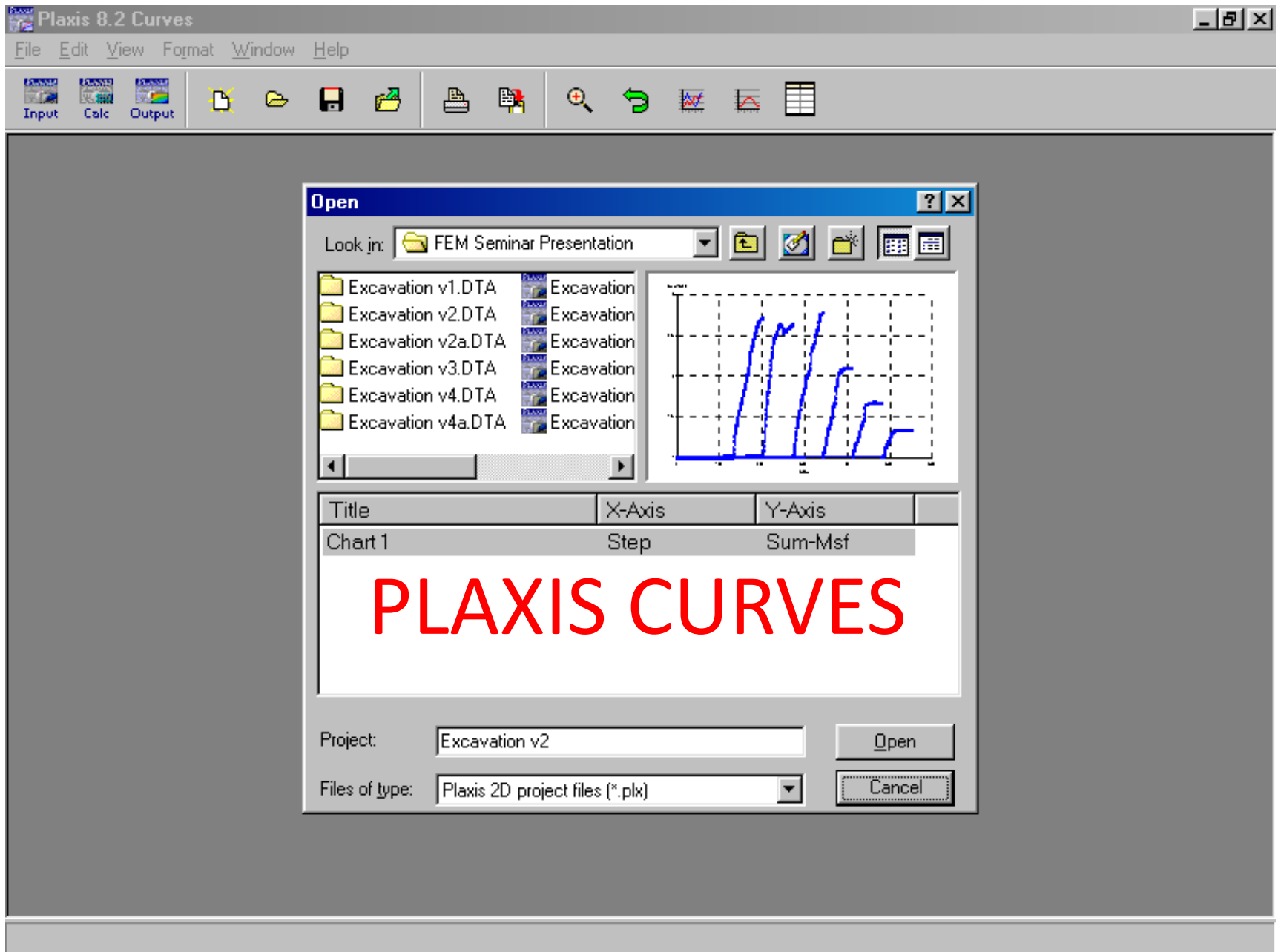


Active pore Pressures

Excess pore  
pressures







# Selecting Nodes & Stress Points

Plaxis 8.2 Output - [Select points]

File Edit View Geometry Deformations Stresses Window Help

Input Calc Curves

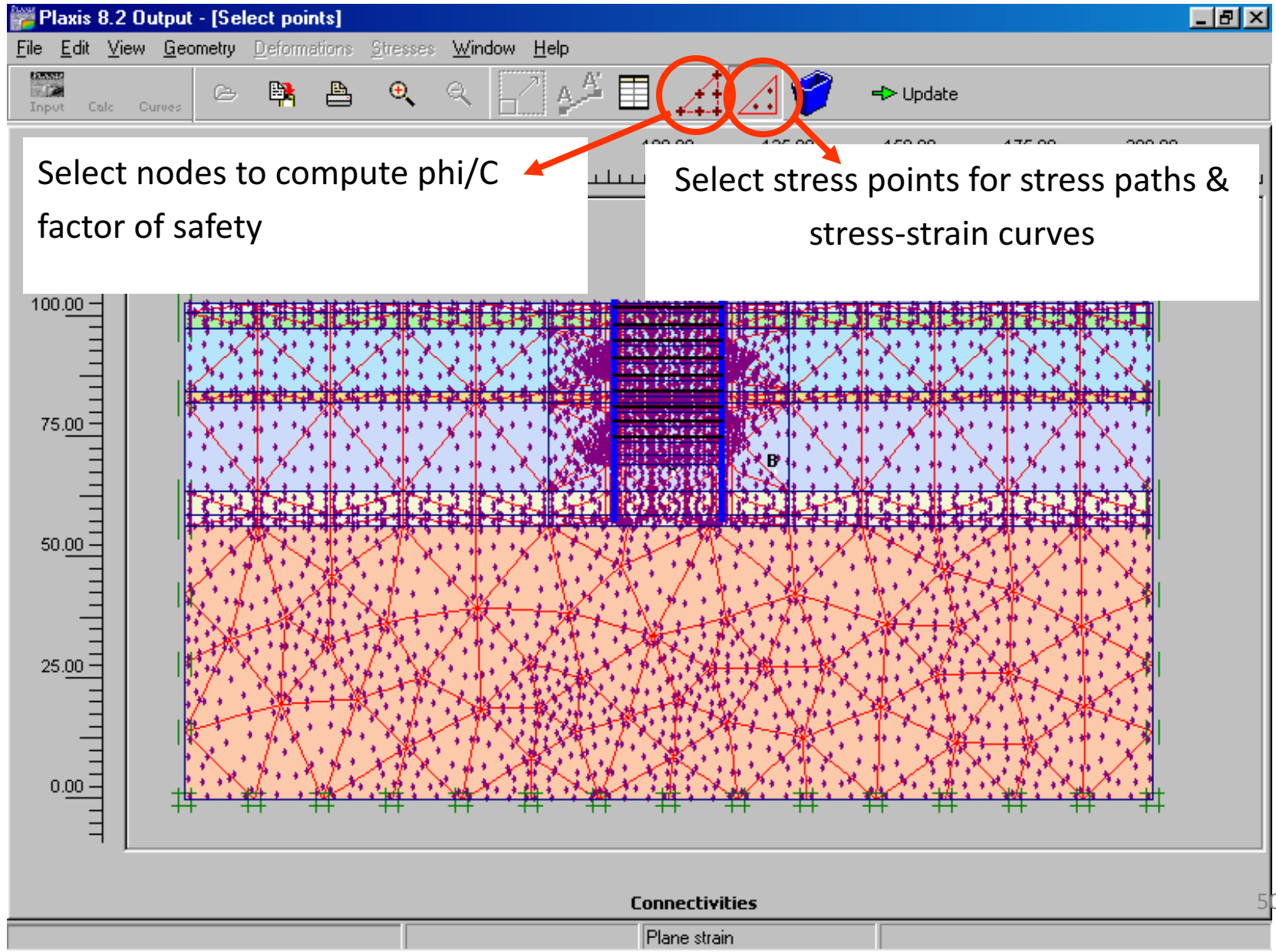
Select nodes to compute  $\phi/C$  factor of safety

Select stress points for stress paths & stress-strain curves

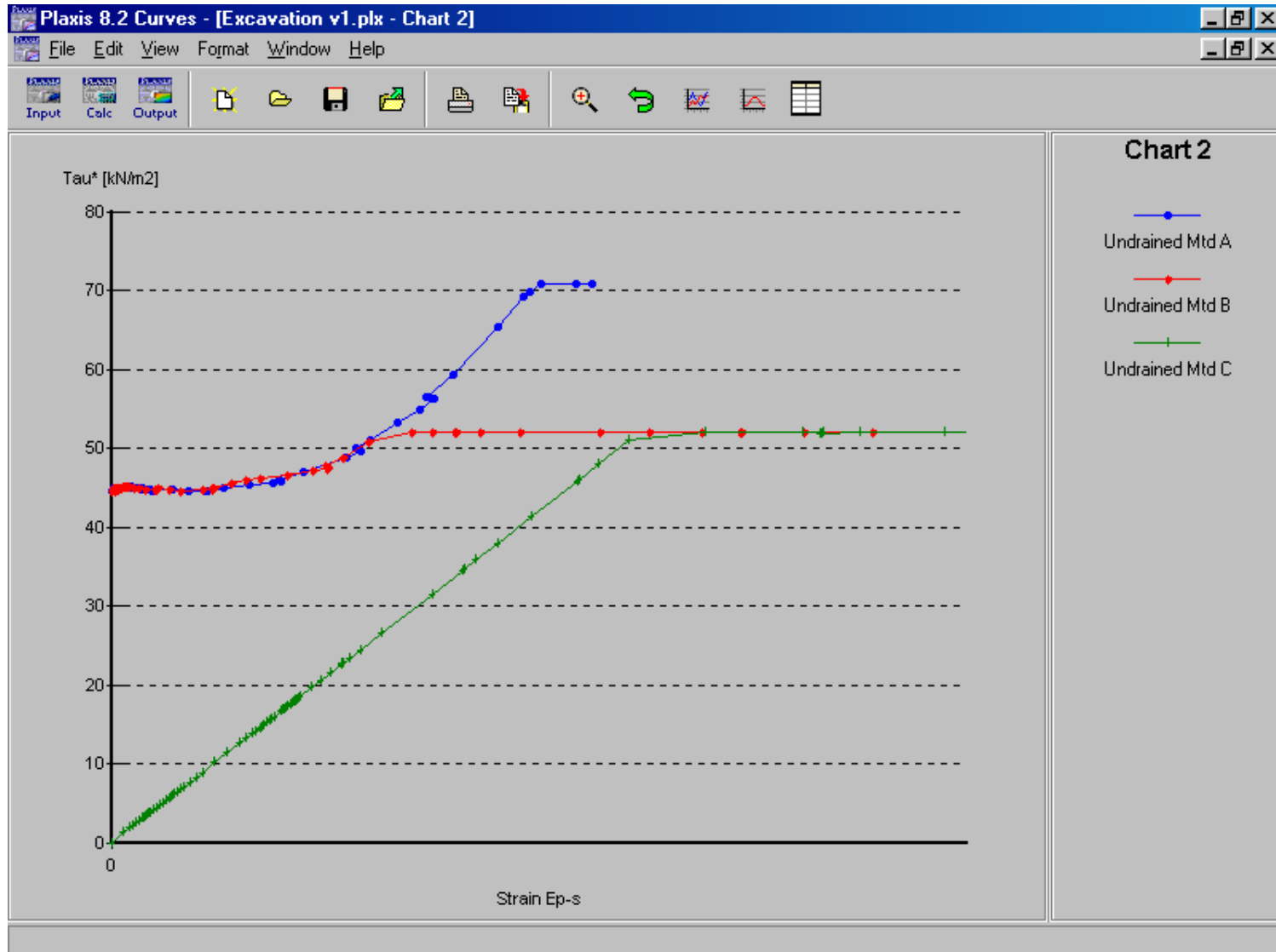
100.00  
75.00  
50.00  
25.00  
0.00

Connectivities

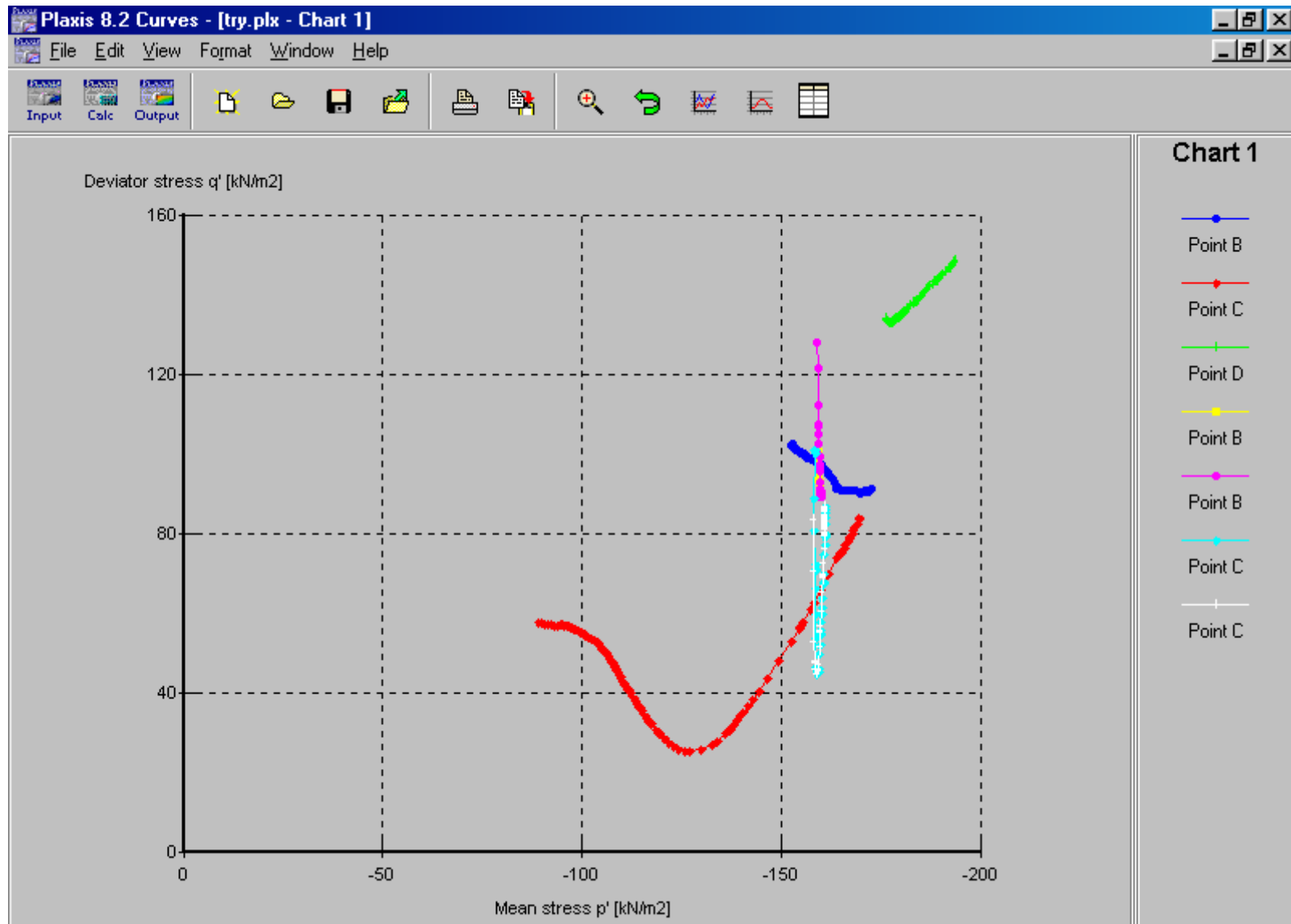
Plane strain



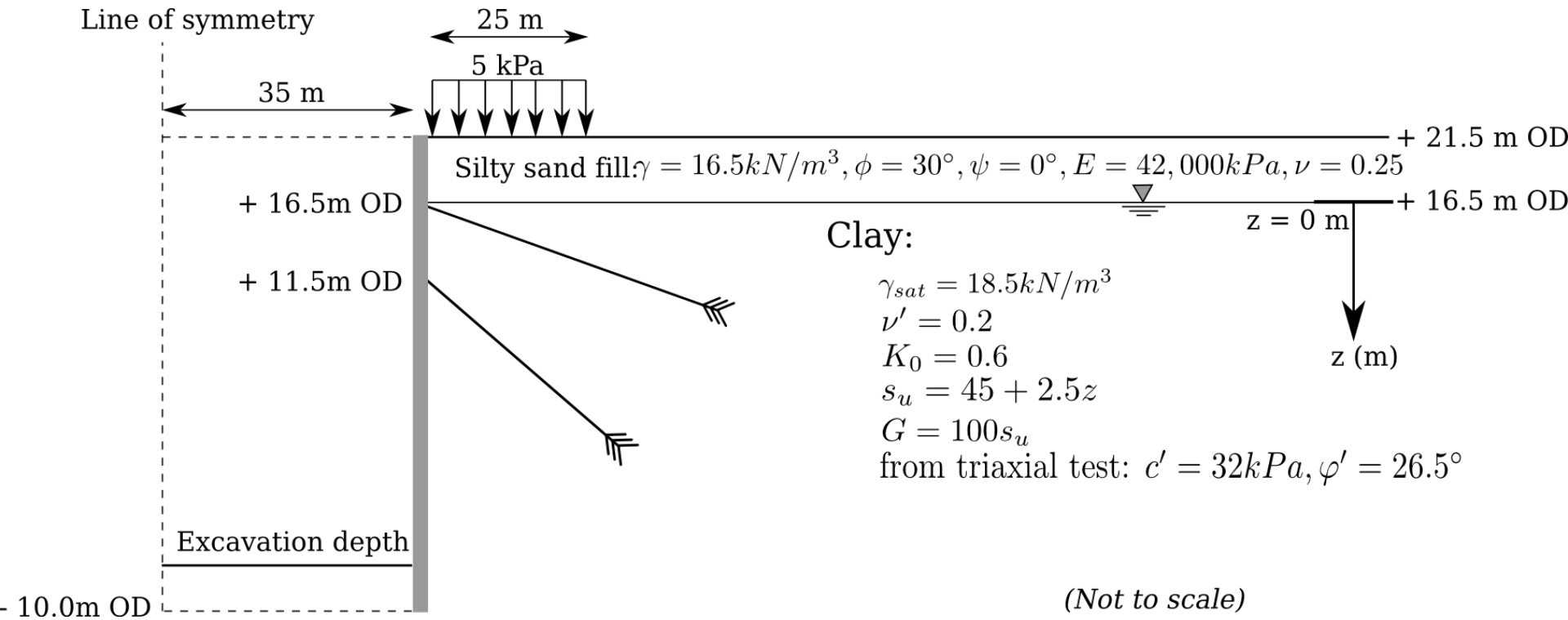
# Plaxis Curves - Stress-Strain plots



# Plaxis Curves - Stress Path plots



# Homework #2–FE Modelling of a Tie-Back Excavation in Clay



# Total Stress Approach

- Stiffness is modelled using an undrained Young's modulus  $E_u$  and Poisson ratio.
- Pore pressures are not generated
- Undrained shear strength  $S_u$  is an input parameter.
- Consolidation analysis has no effect and should not be performed.
- $C_u$  ( $S_u$ ) and undrained friction  $\varphi_u=0$
- Drained simulation, with Poisson ratio of 0.495 (0.5 can't be used as it results in singularity).

# Effective Stress Approach or Undrained (A)

- Effective stiffness and **effective strength parameters** are used.
- **Pore pressures are generated**, but may be **inaccurate**, depending on model.
- Undrained shear strength is not an input parameter but an outcome of the constitutive model . The resulting shear strength must be checked against known data.
- Consolidation analysis can be performed after the undrained calculation, which affect the shear strength

# Equivalent Effective Stress or Undrained (B)

- Effective stiffness parameters and **undrained strength parameters** are used.
- Pore pressures are generated, but may be **highly inaccurate**.
- **Undrained shear strength  $S_u$**  is an input parameter.
- Consolidation analysis should not be performed after undrained calculation.  $S_u$  must be updated, if consolidation is performed anyway.



# Methods of undrained analysis for Mohr-Coulomb clay

Methods of Undrained Analysis	Material Type	Deformation Parameters	Strength Parameters	Initial Conditions
Total Stress	Non-porous/ Drained	$E_u, \nu_u$	$c_u, \phi_u=0$	$K_{0,u}$
Effective Stress (triaxial parameters)	Undrained	$E', \nu'$	$c', \phi'$	$K_0$
Effective stress (strength profile)	Undrained	$E', \nu'$	$c', \phi'$	$K_0$

# Hints

## Total Stress Method

- Use  $v_u = 0.495$  because 0.5 won't compute
- $K_{0,u}$  may not be uniform throughout the clay
- There should be no water in the clay

## Effective Stress Methods

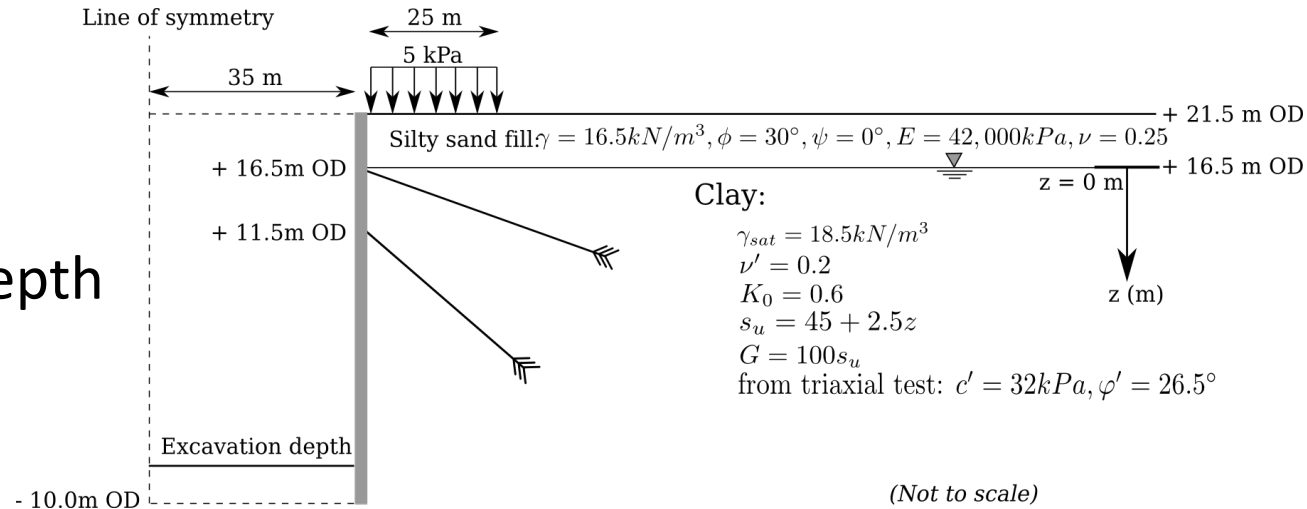
- Use Mohr's circles to determine  $s_u$  from  $c'$  and  $\phi'$  and vice versa
- Remember to dewater the excavation

## General:

- Assume no dilation (i.e.  $\psi=0$ )
- Excavate in stages
- Design the mesh carefully

# Total Stress Approach

- Use  $V = 0.495$
- $K_0$  Varies with depth
- $\phi' = 0$

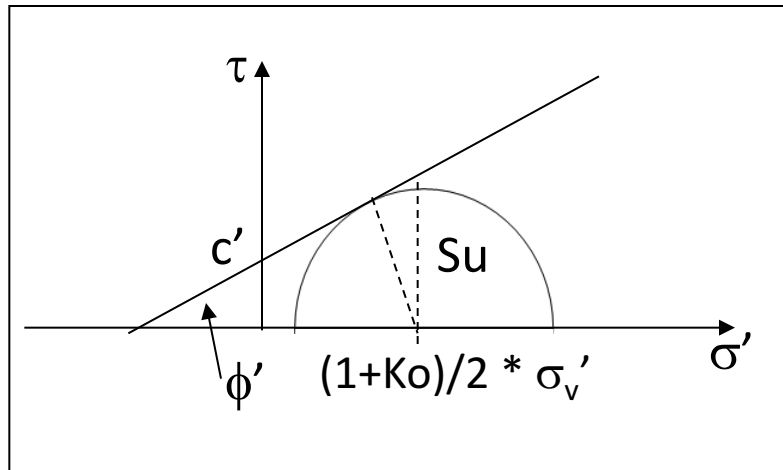


Depth	$\sigma_v$	u	$\sigma_v'$	$k_0 \sigma_v' = \sigma_h'$	$\sigma_h$	$K_0$	$K_0$ Layer
0	82.5	0	82.5	49.5	49.5	0.6	
6.5	202.75	65	137.75	82.65	147.65	0.728	0.664
11.5	295.25	115	180.25	108.15	223.15	0.756	0.742
16.5	387.75	165	222.75	133.65	298.65	0.77	0.763

# Undrained analysis using Effective stress method

## Effective stress A

- Define strength  $c'$ - $\phi'$  in terms of the real effective stress parameters, assuming zero dilation
- $v' = 0.2$
- $E$  and  $K_o$  should be 'effective stress' based



Mohr-Coulomb failure criteria  
 $\tau = c' + \sigma' \tan \phi'$

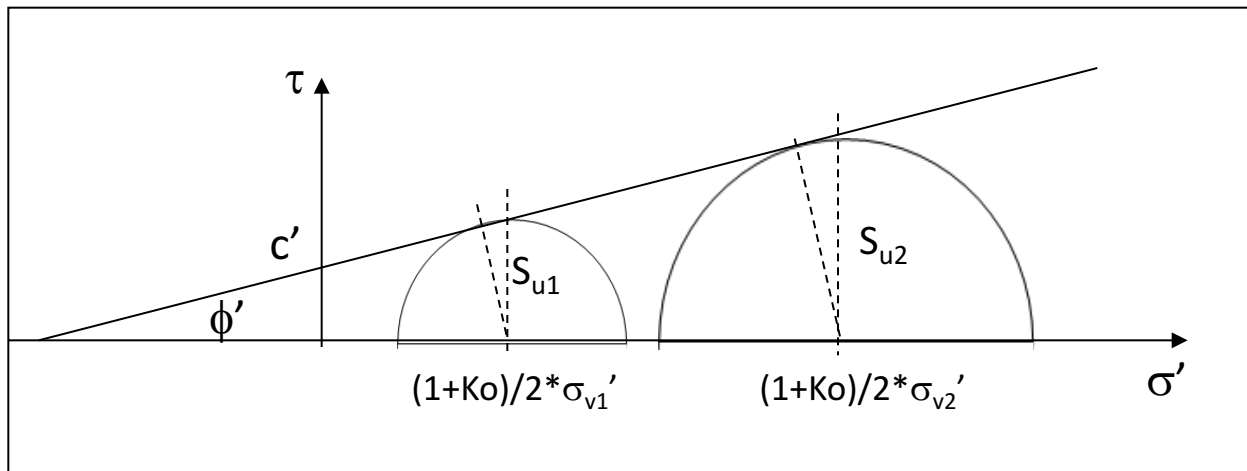
$$S_u = c' \cos \phi + (1+K_o) \sin \phi' / 2 * \sigma_{v_o}'$$

Thus when  $z = 0$ ,  $S_u = c'$

# Undrained analysis using Effective stress method

## Effective stress B

- Define strength  $c' - \phi'$  in terms of the “equivalent” effective stress parameters, with zero dilation
- $v' = 0.2$
- $E$  and  $K_o$  should be ‘effective stress’ based



Simulations are believed by no one except those who conducted them.

Experimental results are believed by everyone except those who conducted them.

Thank You!