Finite Element Modelling using PLAXIS 2D

Krishna Kumar
Email: 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

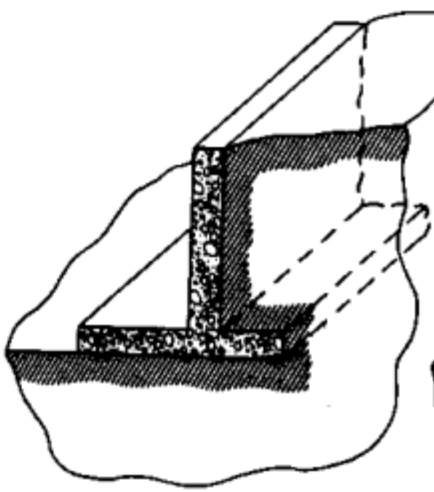
PLAXIS is a finite element program for geotechnical applications in which soil models are used to simulate the soil behaviour. The PLAXIS code and its soil models have been developed with great care. Although a lot of testing and validation have been performed, it cannot be guaranteed that the PLAXIS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

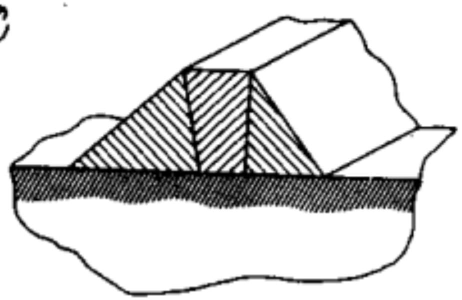
	SI			
Length	m	m	m	cm
Density	kg/m^3	$10^3\ kg/m^3$	$10^6\ kg/m^3$	$10^6\ g/cm^3$
Force	N	kN	MN	Mdynes
Stress	Pa	kPa	MPa	bar
Gravity	m/sec^2	m/sec^2	m/sec^2	cm/s^2
Stiffness*	Pa / m	kPa / m	MPa / m	bar / cm

General Settings

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

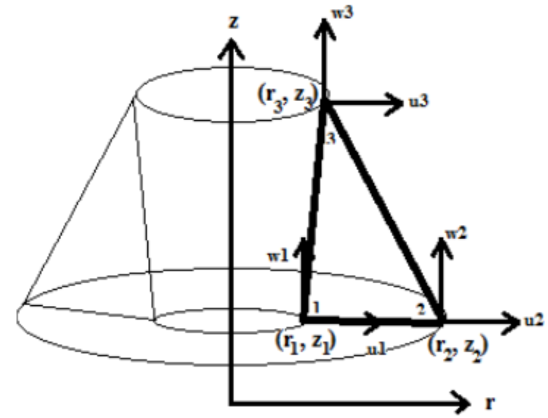


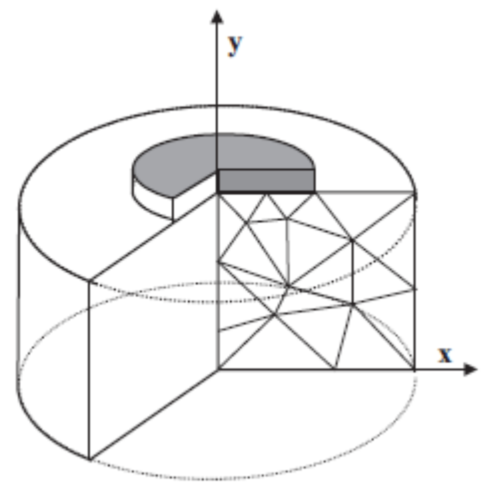
Retaining wall



Earth dam

Axisymmetric:





General Settings (cont…)

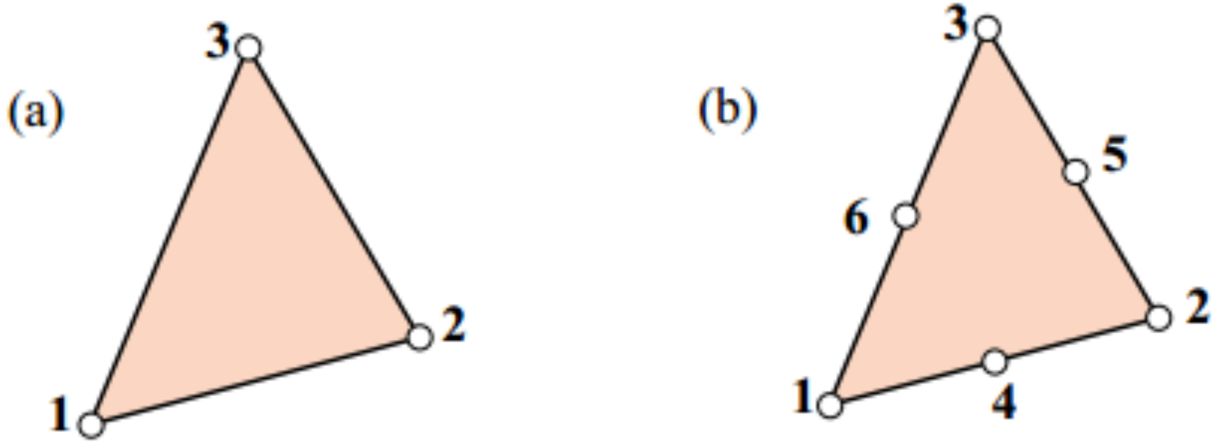
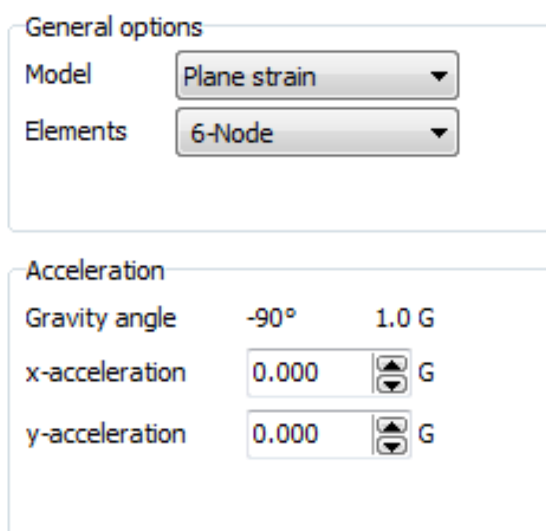


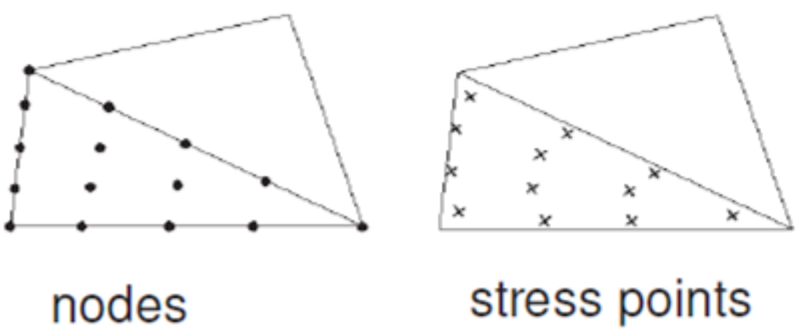
Figure 29.3. Nodal configurations for: (a) linear interpolation of w by three values w_i, at corners $i = 1, 2, 3$; (b) quadratic interpolation of w by six values w_i at corners $i = 1, 2, 3$ and midpoints $i = 4, 5, 6$.



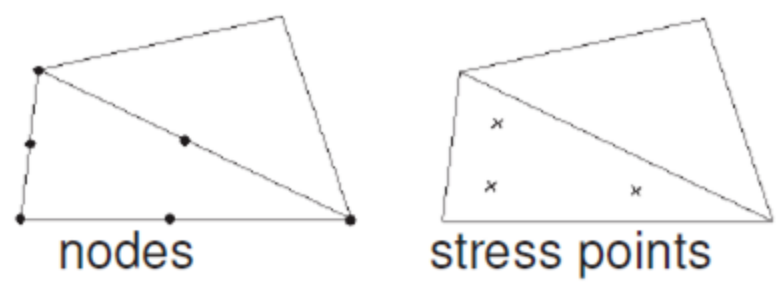
6 and 15 Node Elements

- 12 Gauss points
- 4^{th} order interpolation
- Most accurate in PLAXIS 2D
- Better than four 6-noded triangle elements

- 3 Gauss points
- 2^{nd} order interpolation
- Care should be taken while using for Axisymmetric and Failure problems



15-node triangle



6-node triangle

Setting up the geometry

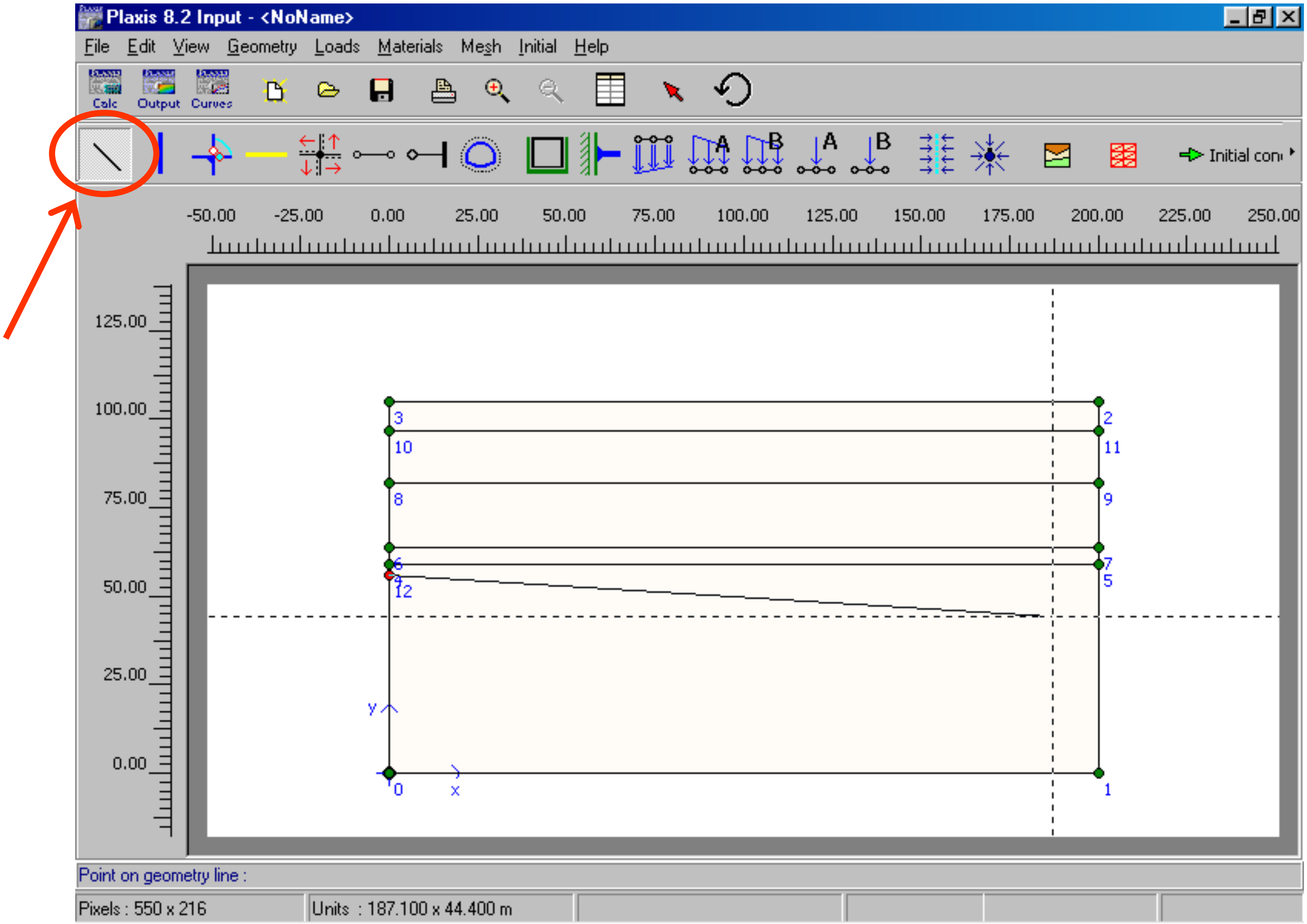


Plate Elements

- 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.

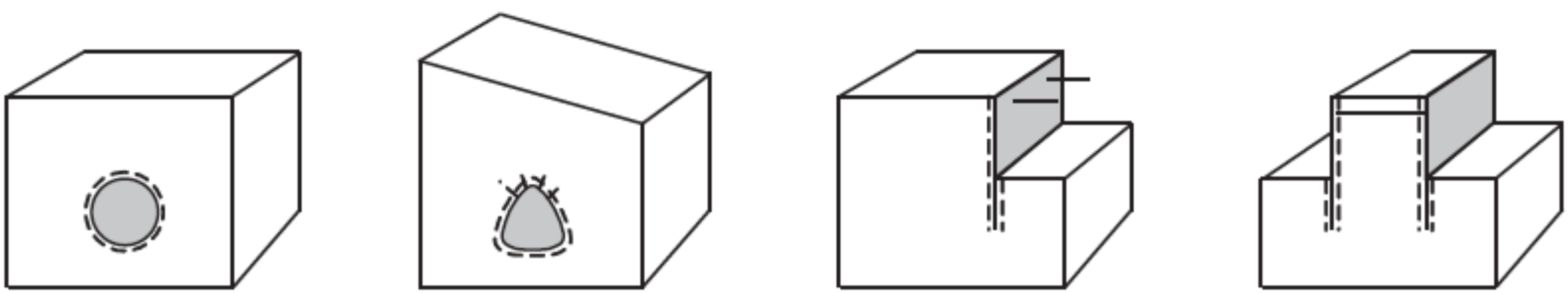
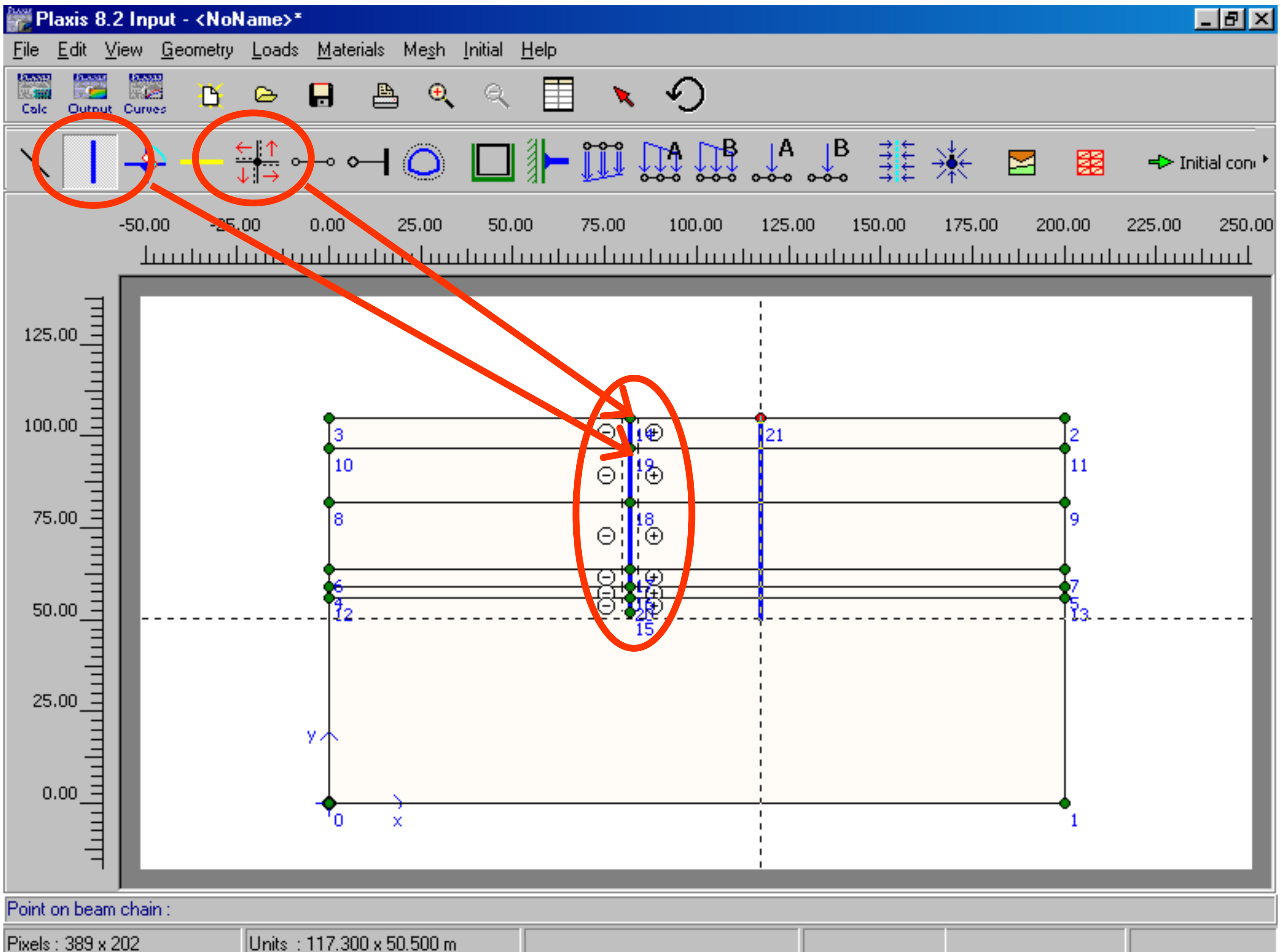


Figure 3.9 Applications in which plates, anchors and interfaces are used

Modelling walls

- Use plate elements AND interface elements



Interface Elements

- Problems involving Soil-Structure Interaction

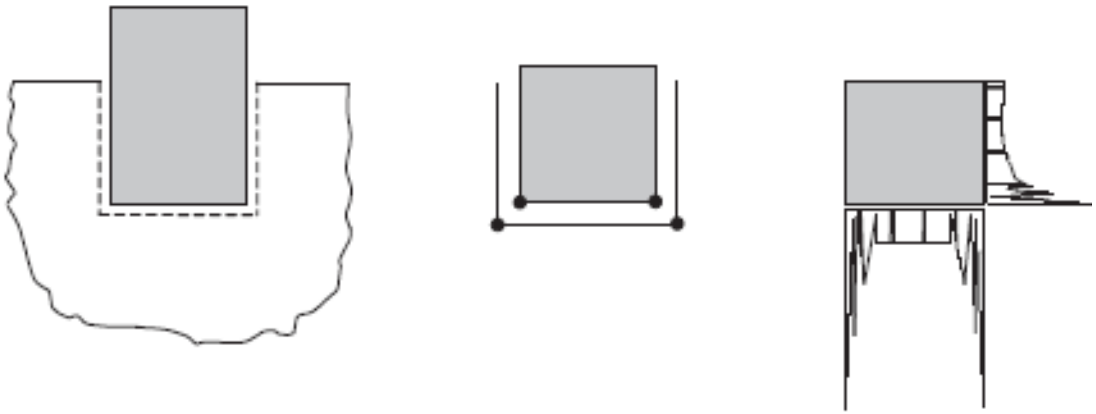


Figure 3.14 Inflexible corner point, causing poor quality stress results

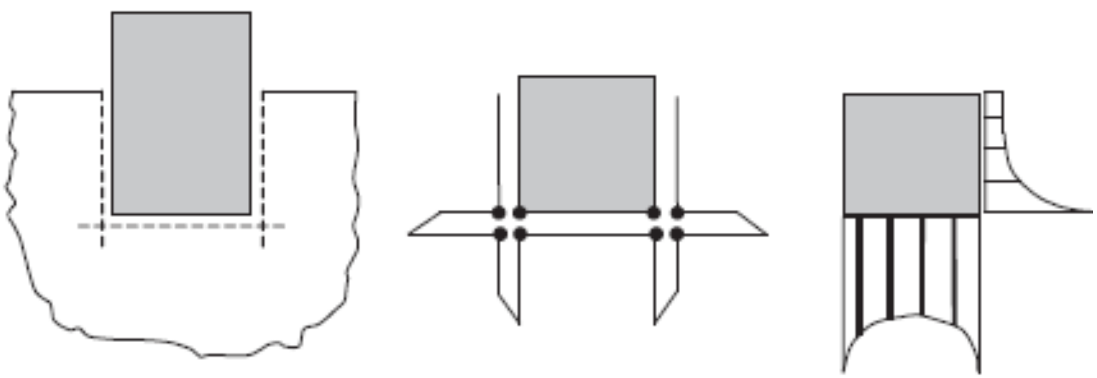
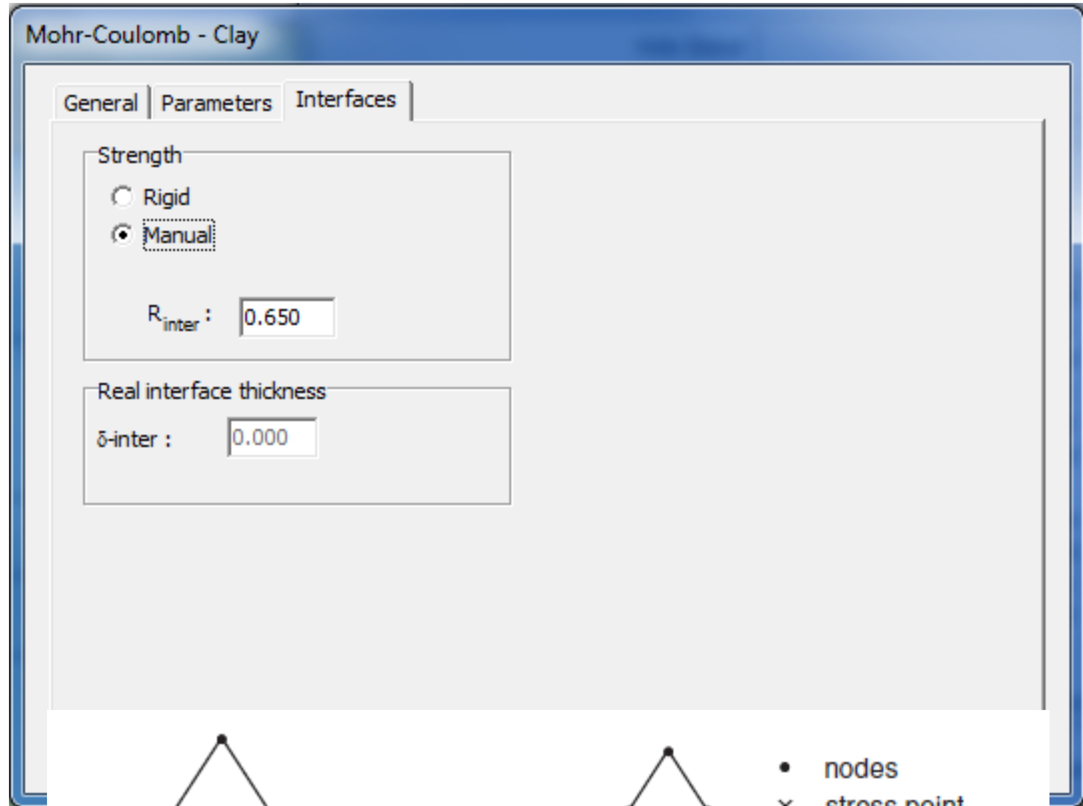


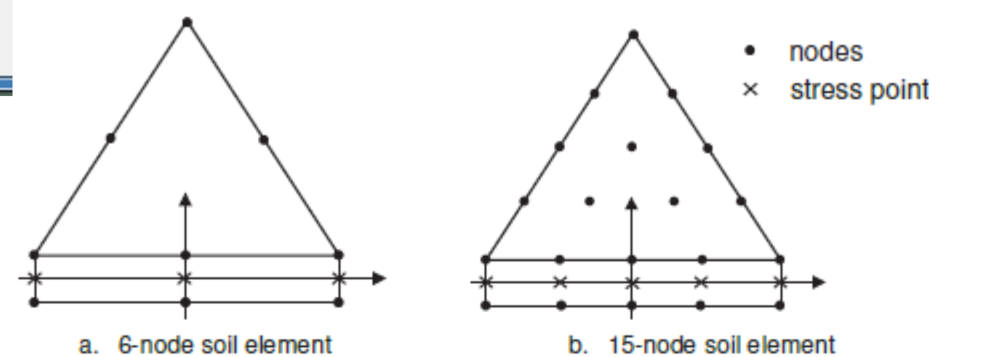
Figure 3.15 Flexible corner point with improved stress results

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**.

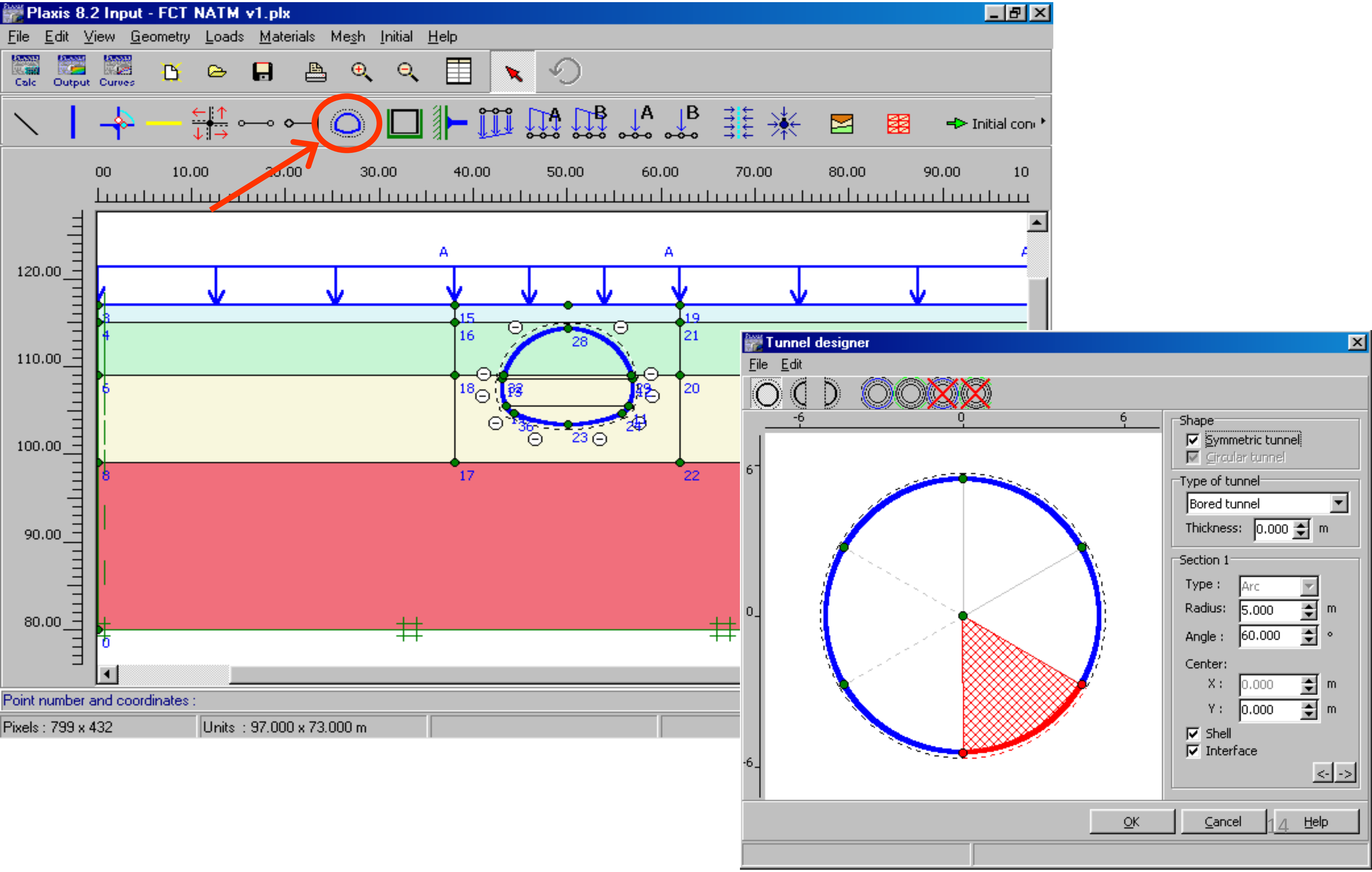
Interface Elements





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

Modelling tunnels

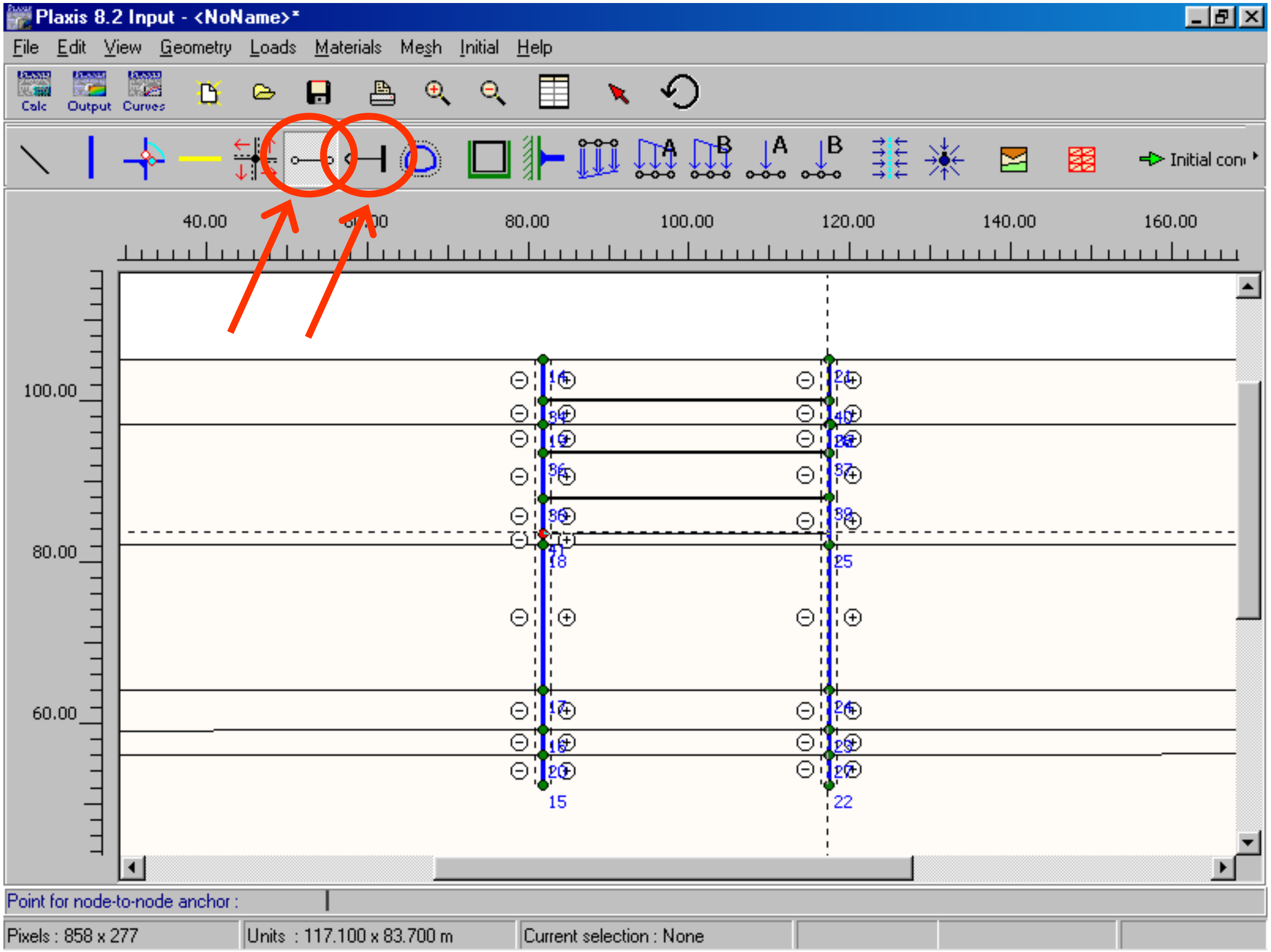


Anchors

- **Node to Node**: are sprints 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.

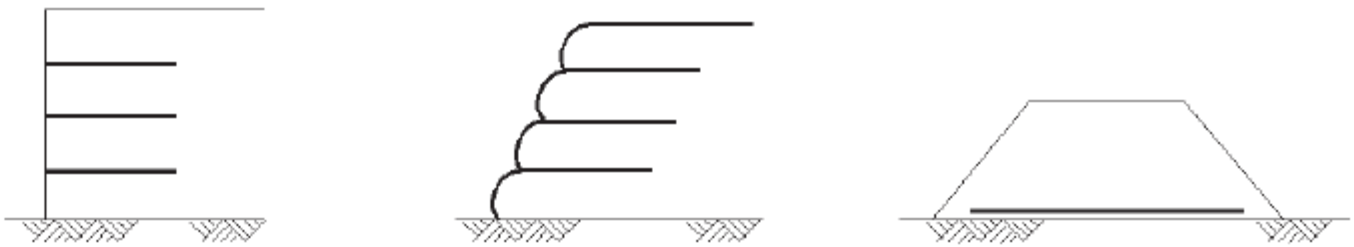
Modelling struts

- Use node to node anchors OR fixed end anchors

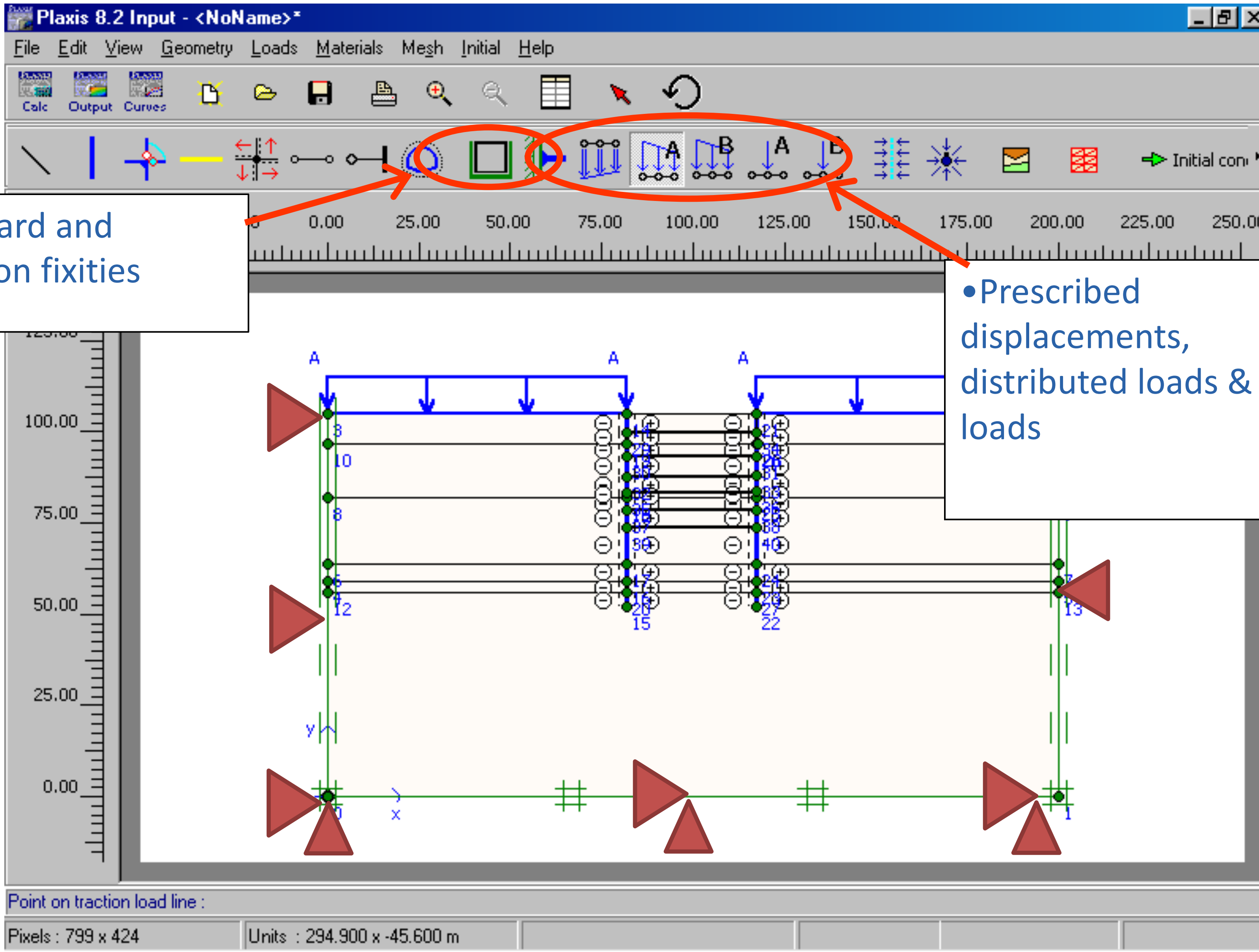


Geogrids

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



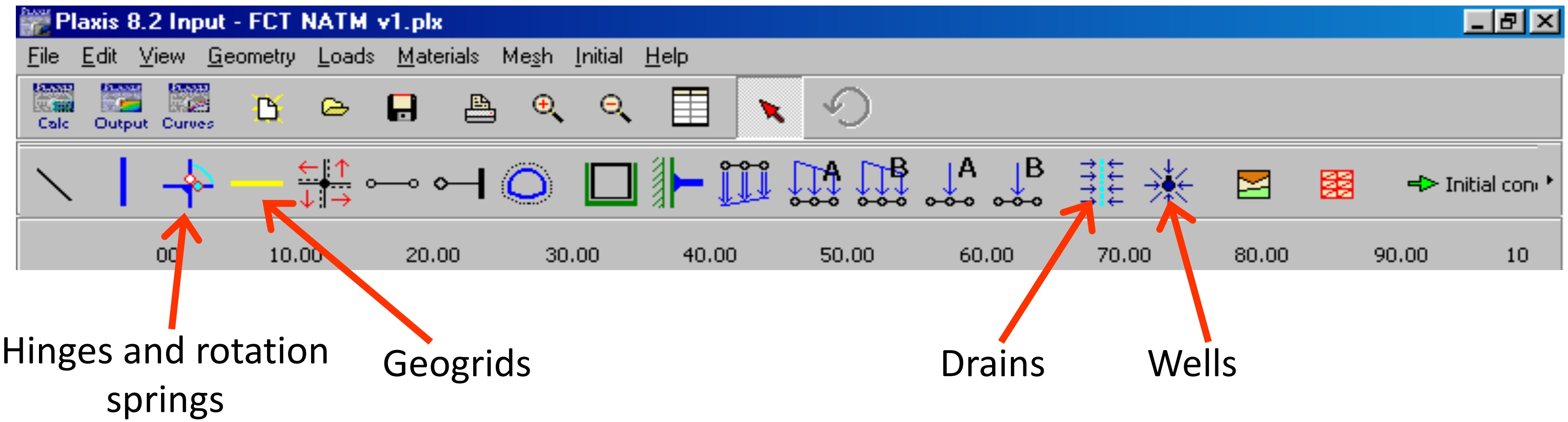
Defining load & boundary conditions



Standard and rotation fixities

- Prescribed displacements, distributed loads & point loads

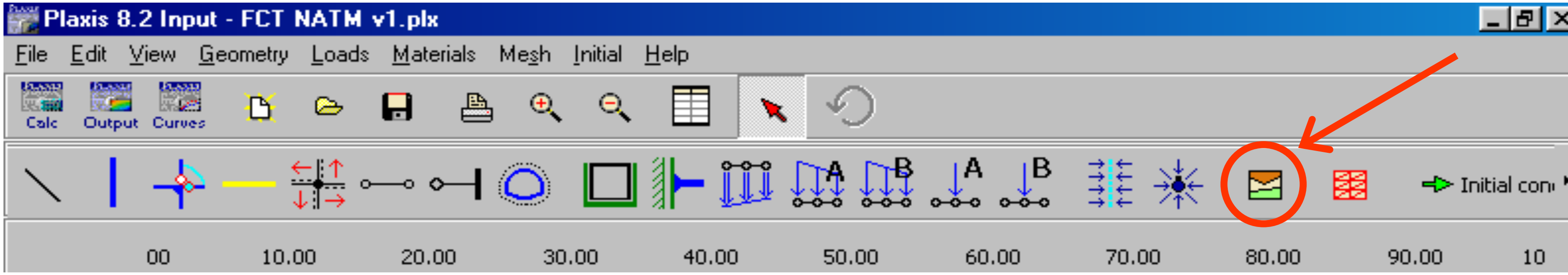
Other modelling capabilities

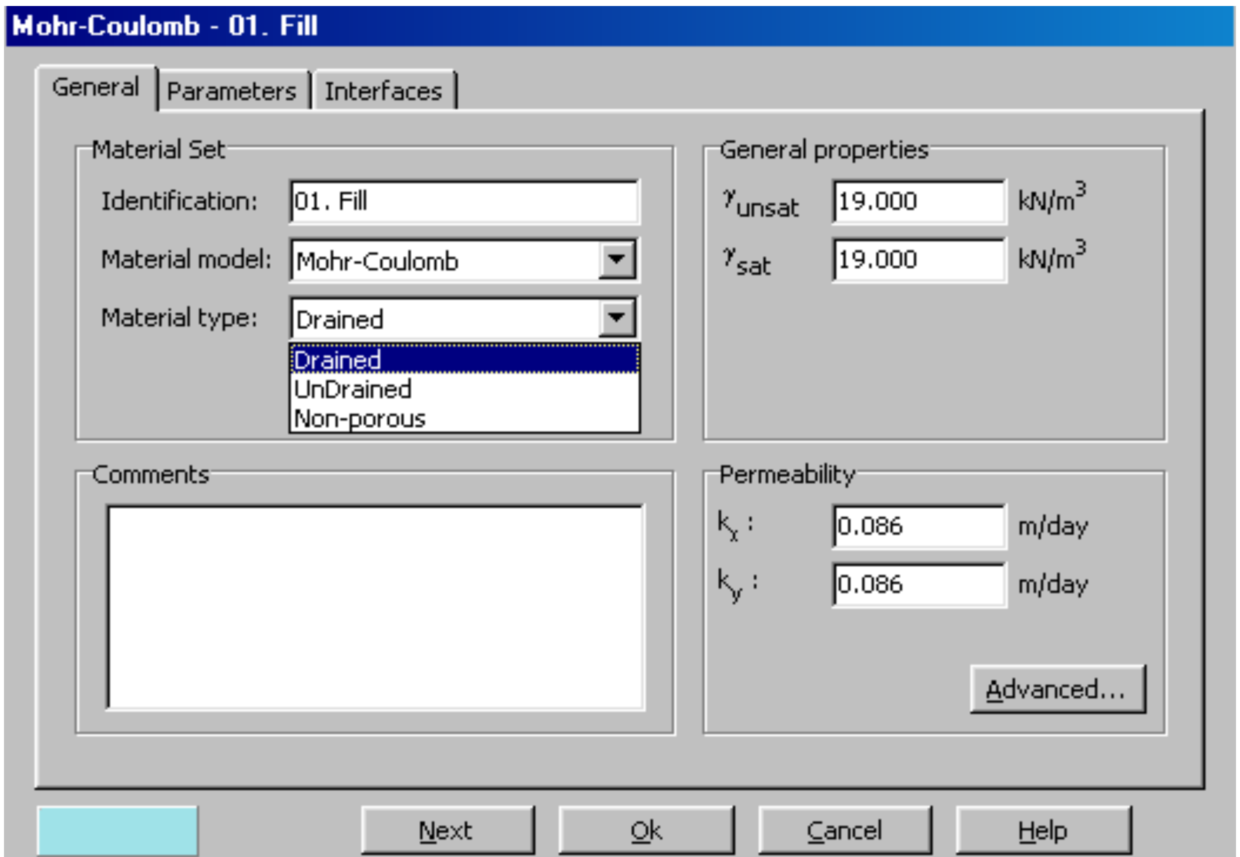


- Drains – lines that set pore pressures to zero

- 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, ν, c, φ and ψ

3. Soft-soil – a Cam Clay model for NC clays and peat

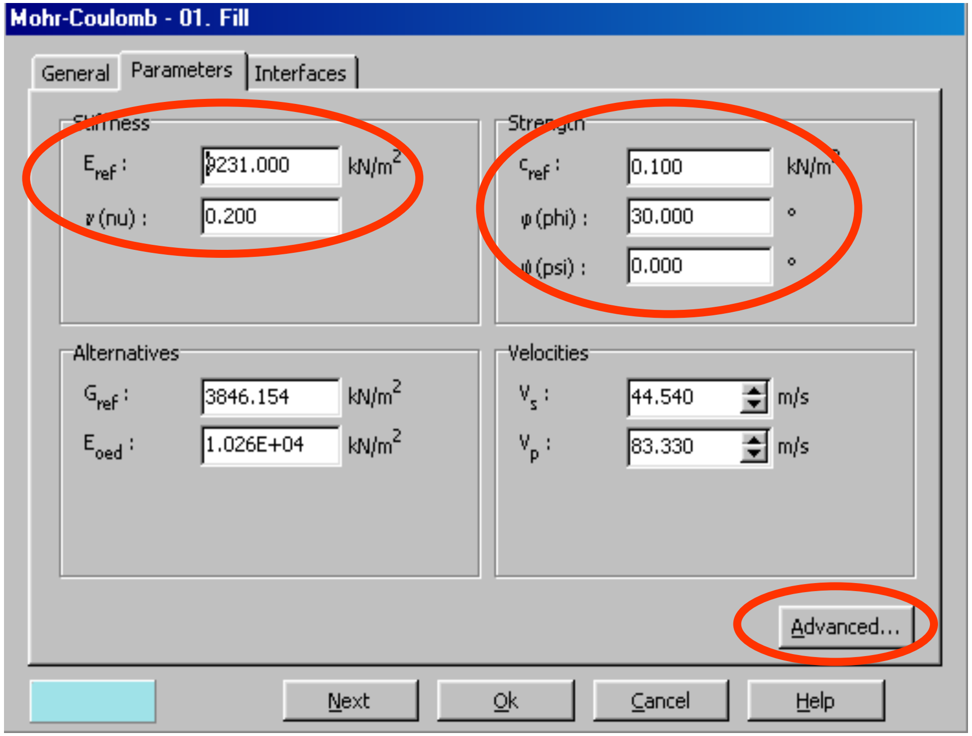
4. Hardening soil – an advanced elastoplastic hyperbolic model

5. Soft soil creep – similar to the soft-soil model, but can consider secondary compression

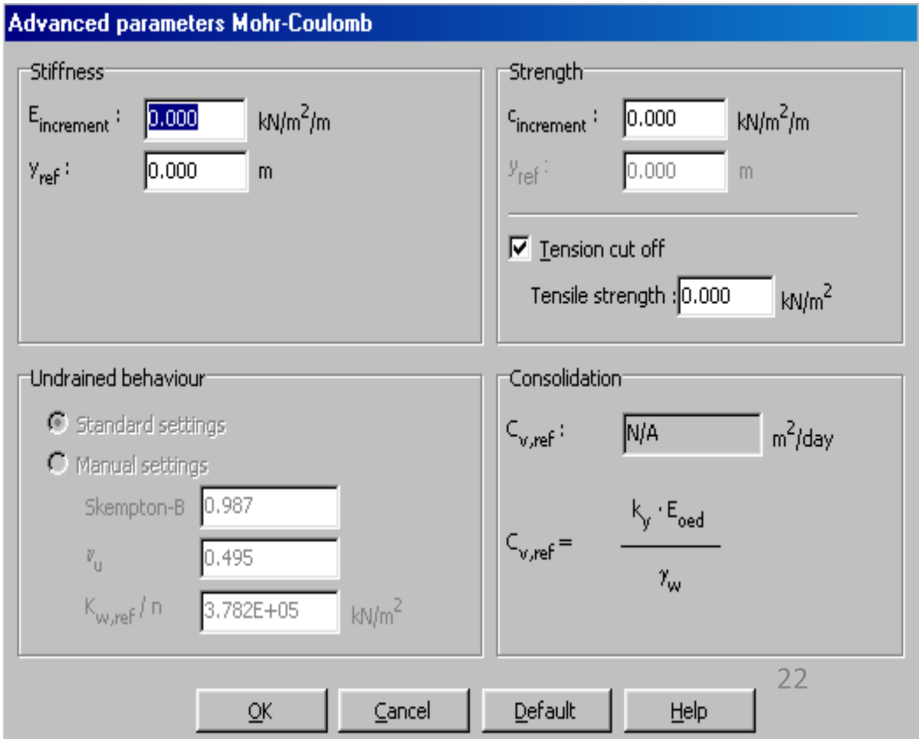
6. Jointed-rock – used for stratified or jointed rock.

7. User-defined– it is possible to use your own model.

The Mohr Coulomb model



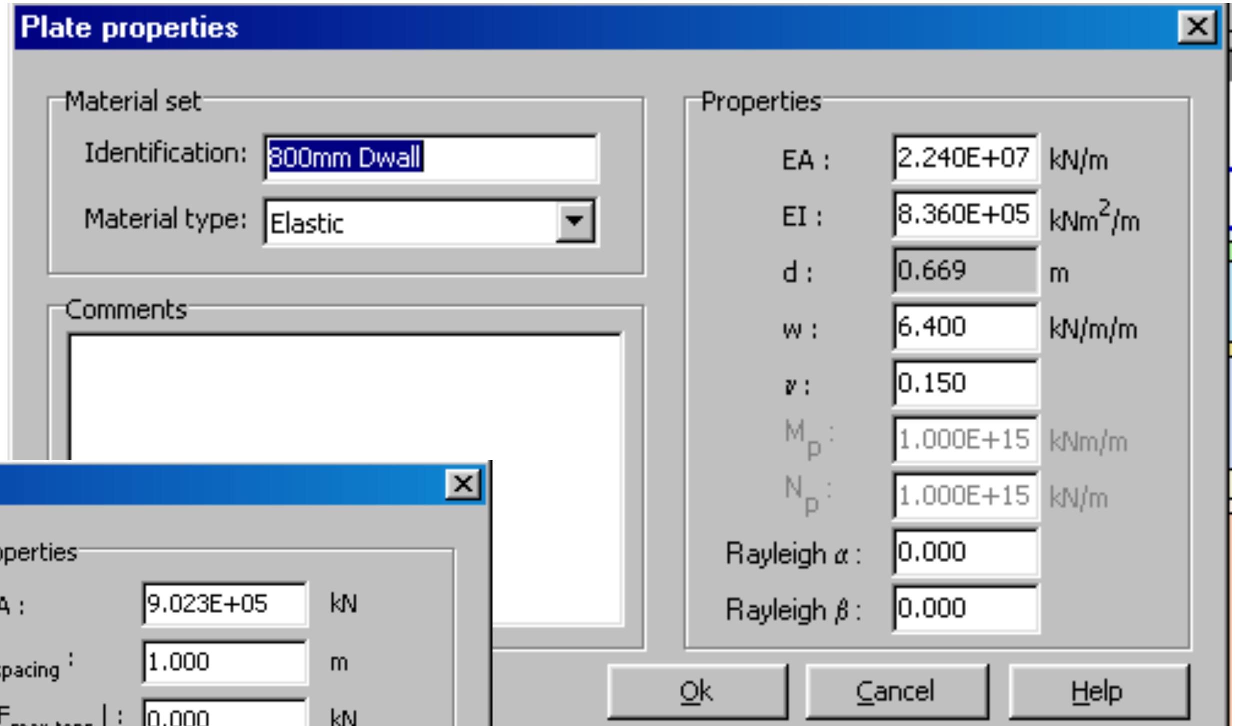
Stiffness & strength parameters

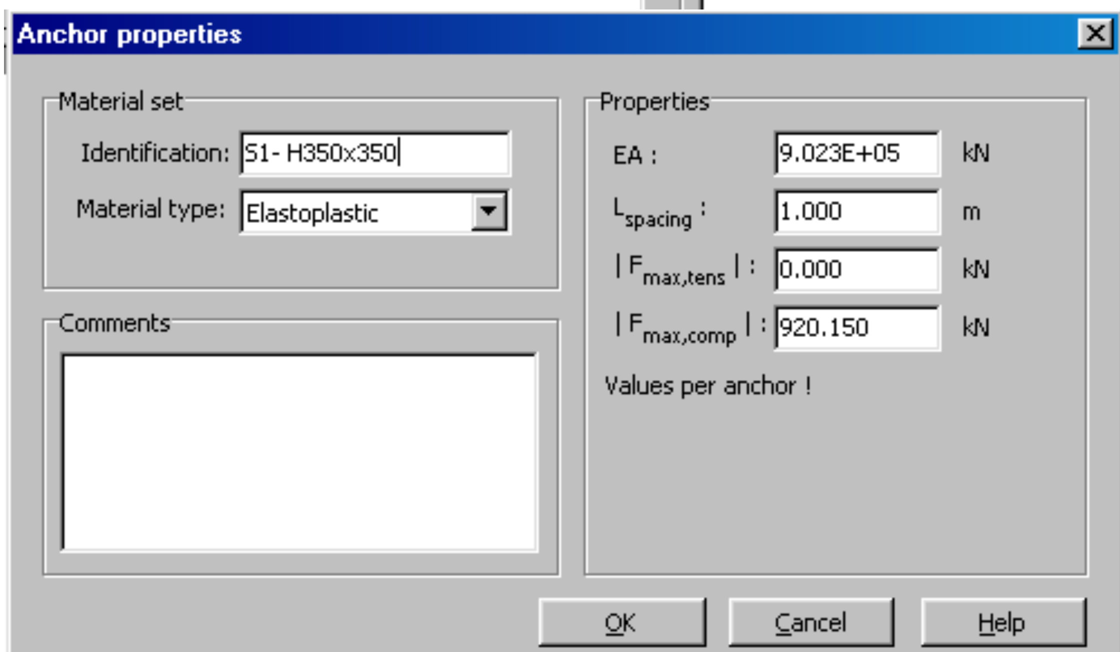


Variation of stiffness and strength with depth

Defining material properties – plates and anchors

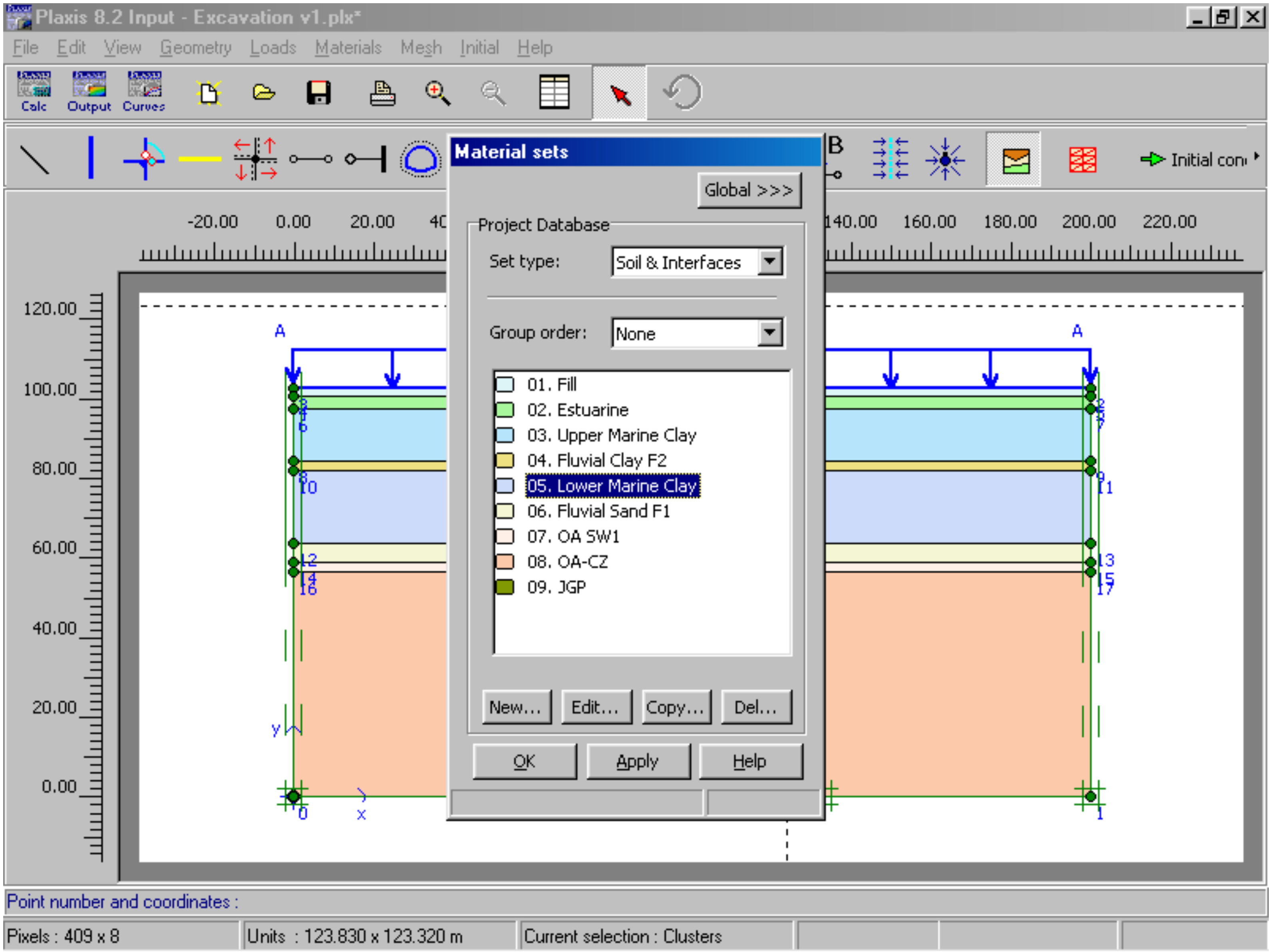
Plates





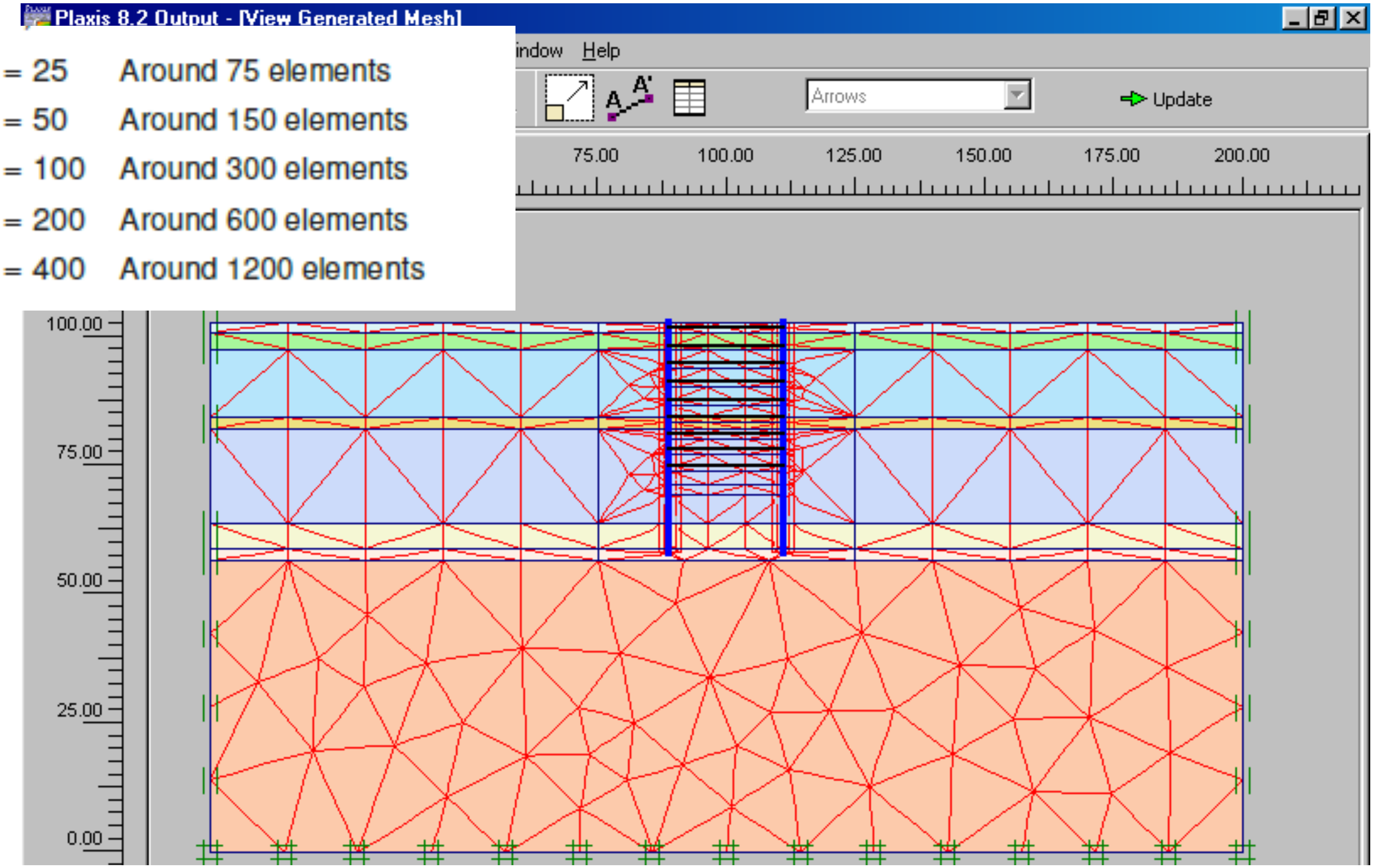
Anchors

Assigning materials



Generating the mesh

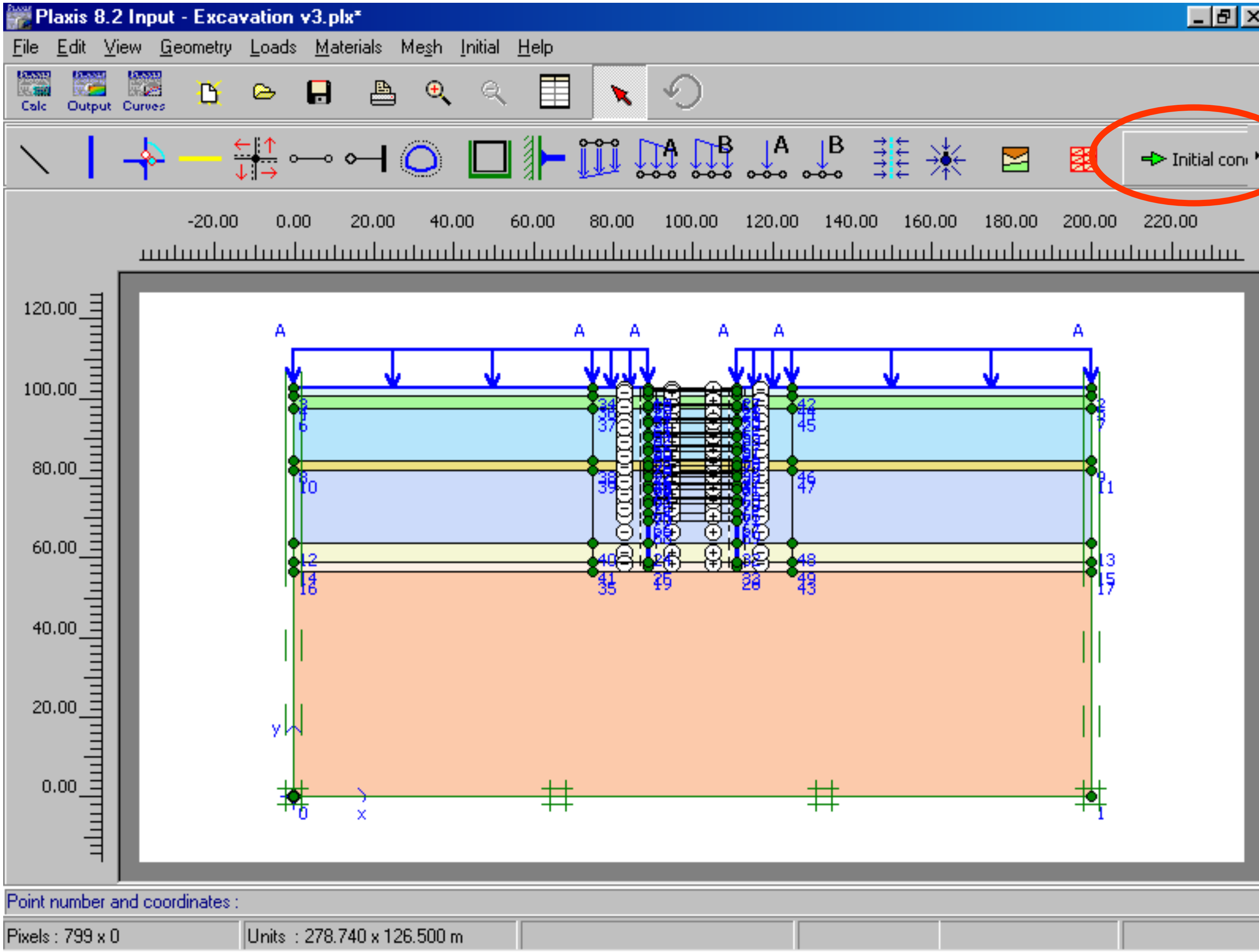
Specify global coarseness and local refinement: Element size: $l_e = \sqrt{\frac{(x_{max} - x_{min})(y_{max} - y_{min})}{n_c}}$



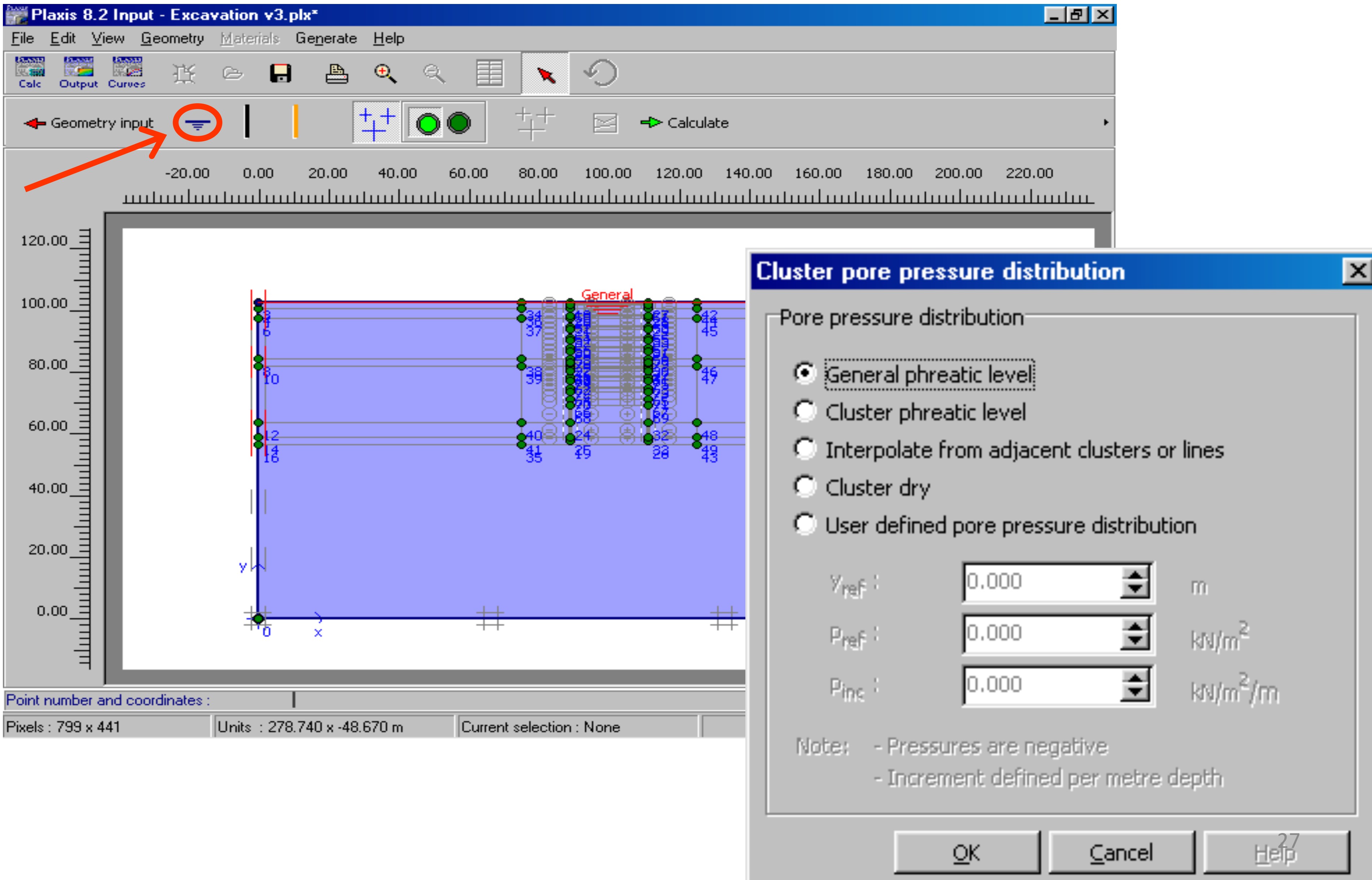
Very coarse:	$n_c = 25$	Around 75 elements
Coarse:	$n_c = 50$	Around 150 elements
Medium:	$n_c = 100$	Around 300 elements
Fine:	$n_c = 200$	Around 600 elements
Very fine:	$n_c = 400$	Around 1200 elements

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

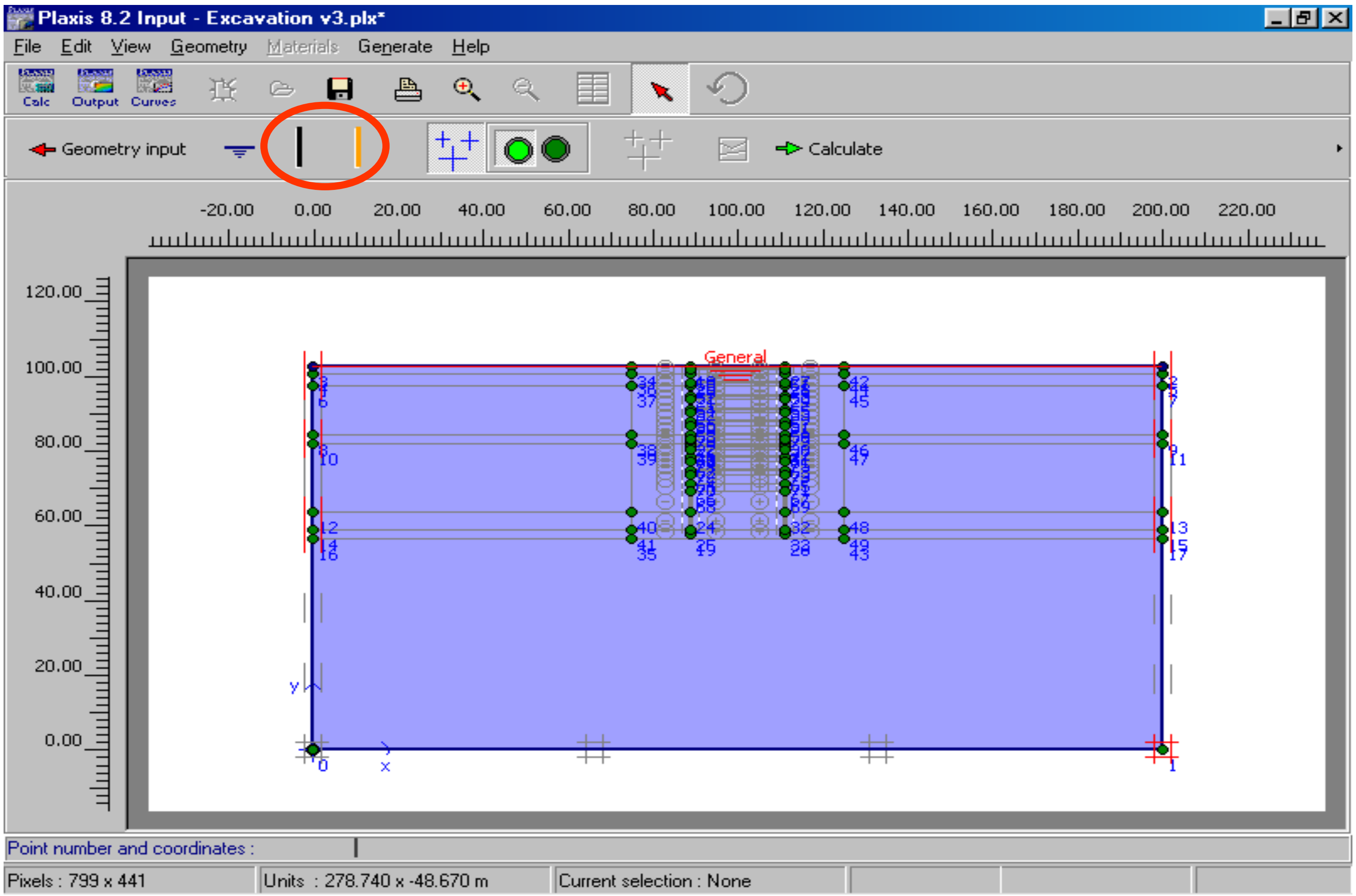


Defining initial pore pressures

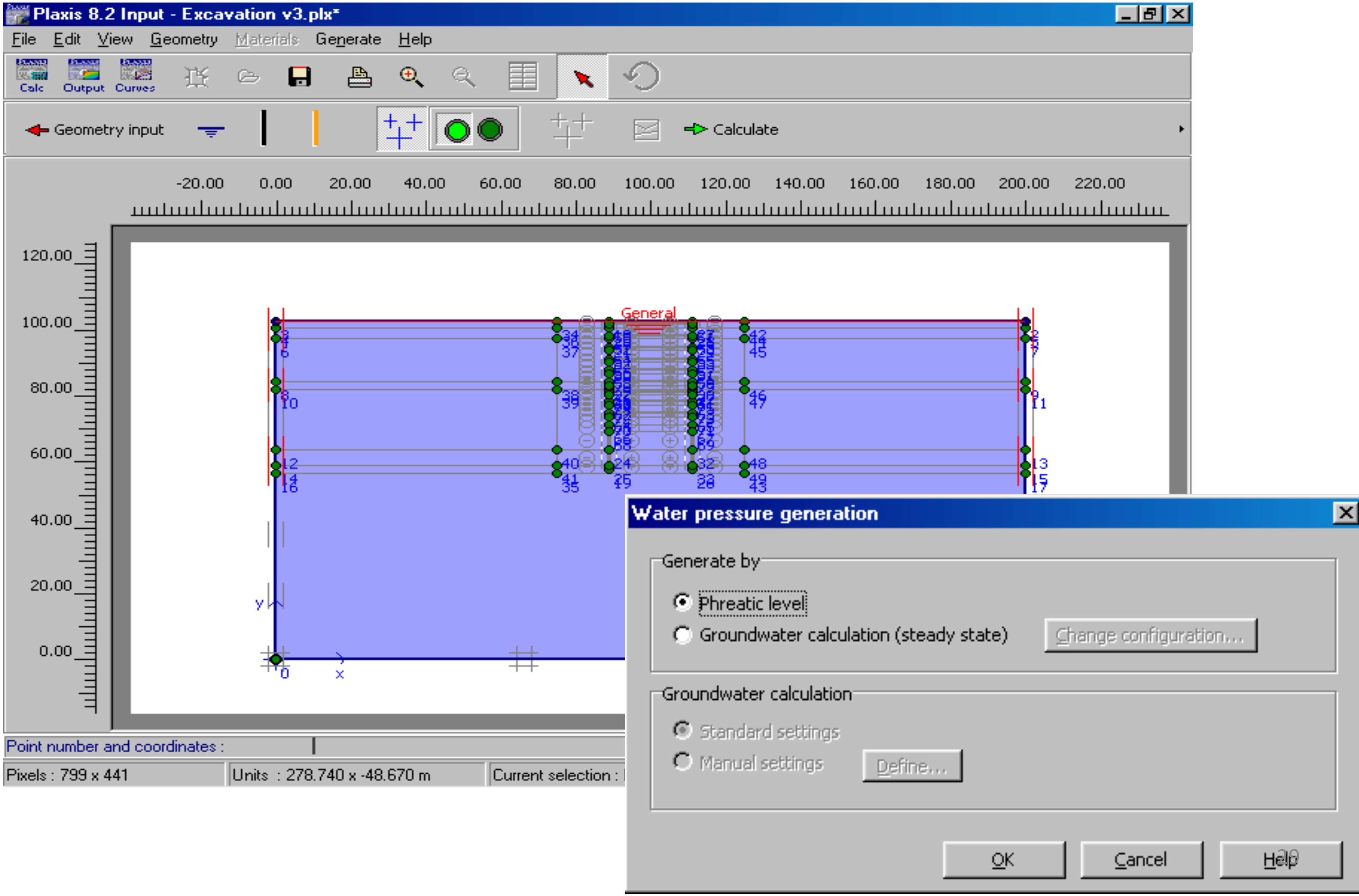


Defining initial pore pressures

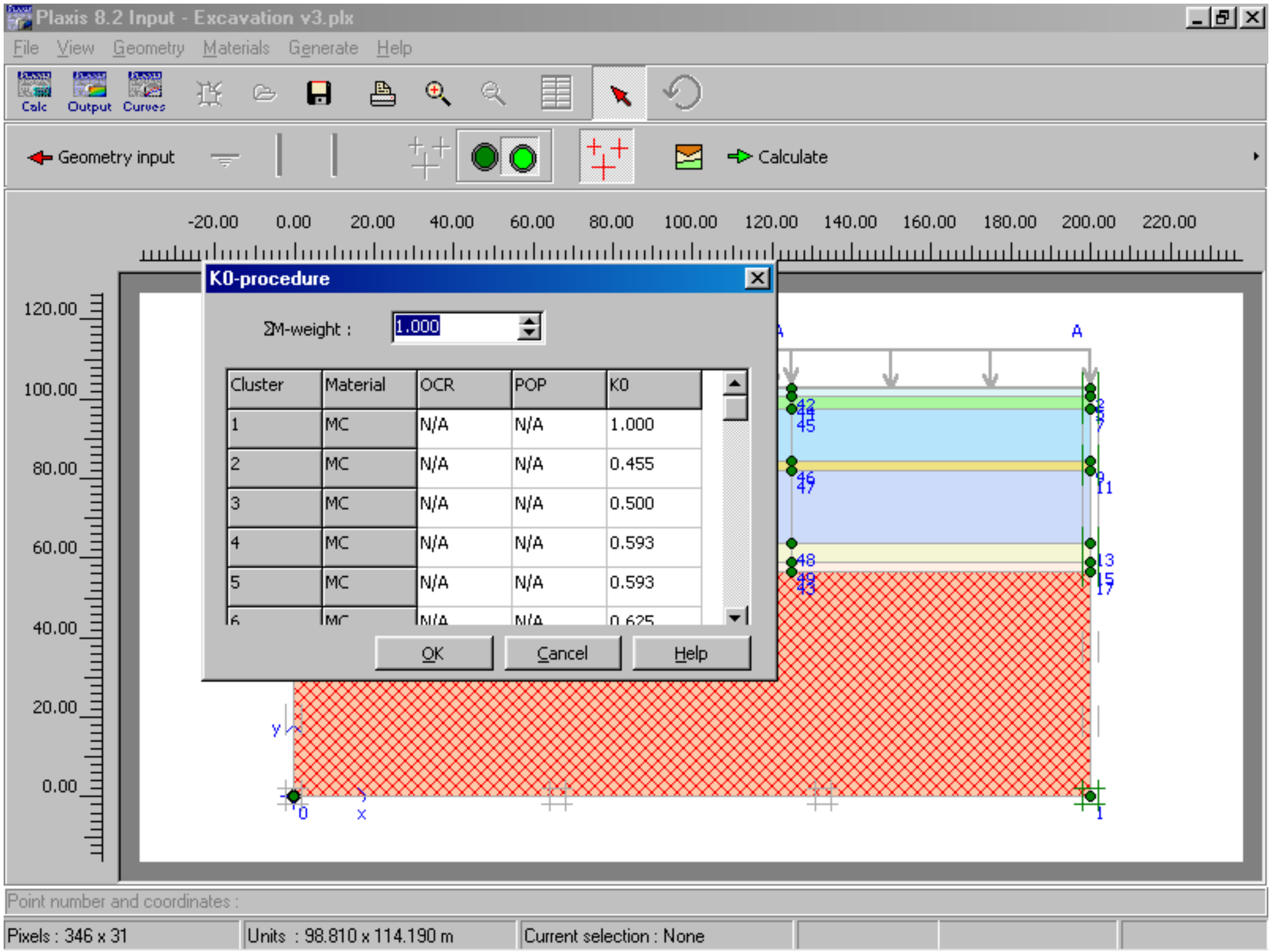
- Can specify closed flow and/or closed consolidation boundaries if necessary



Calculating pore pressures

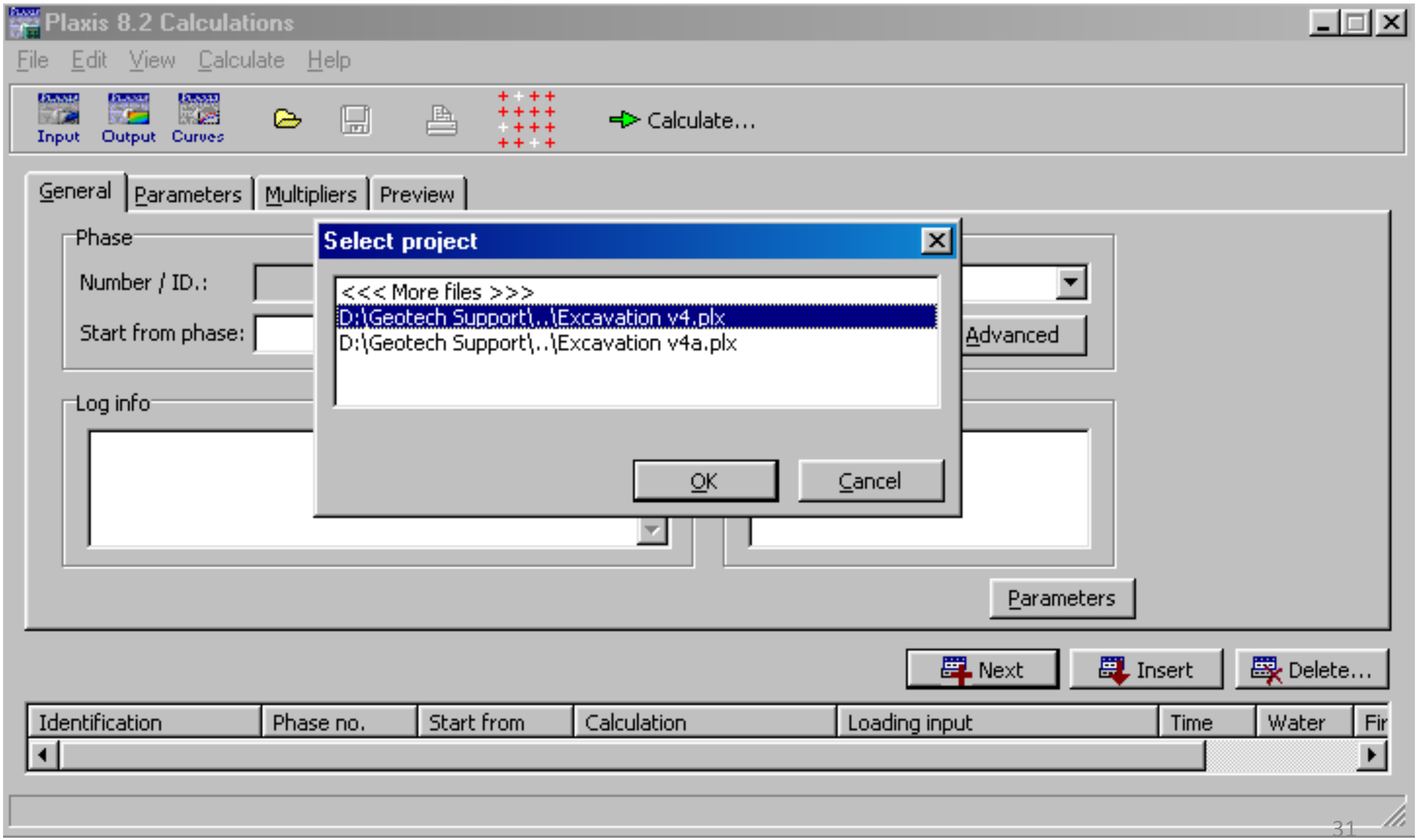


Generate initial soil stresses

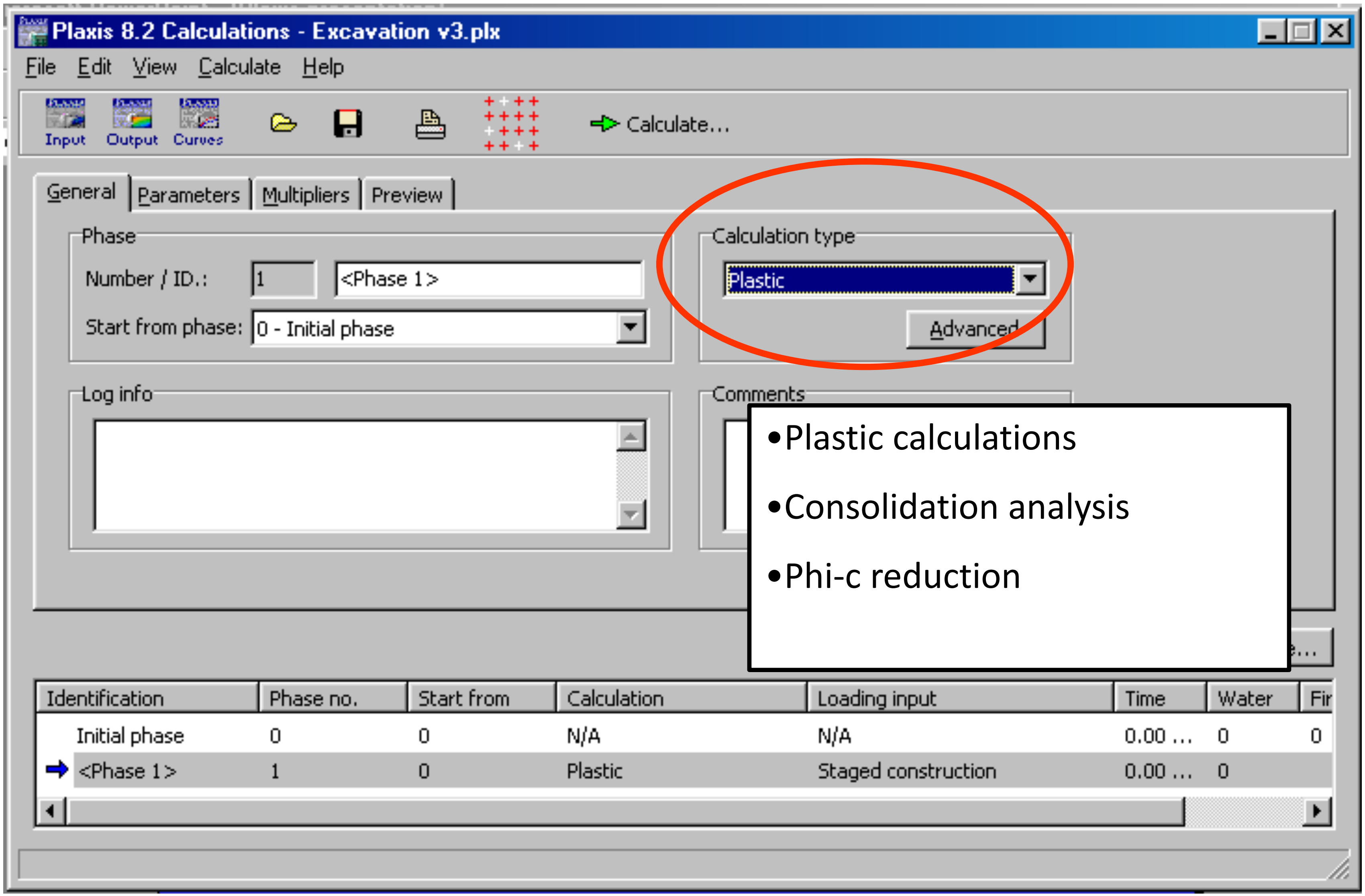


- Note: default K_0 value is 1 - sin ϕ in Plaxis
- Can also use gravity loading method

PLAXIS CALCULATIONS



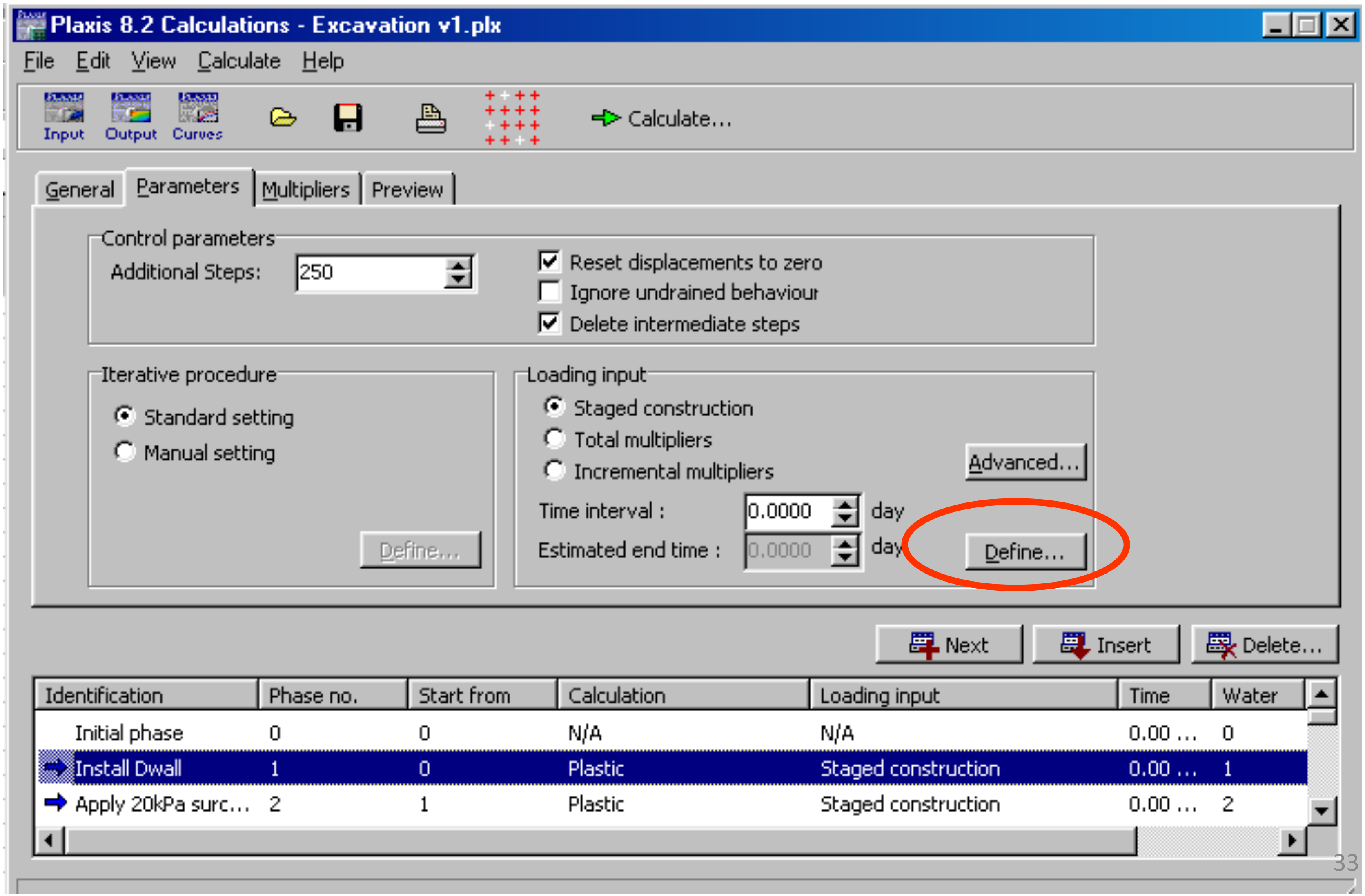
Type of Calculations



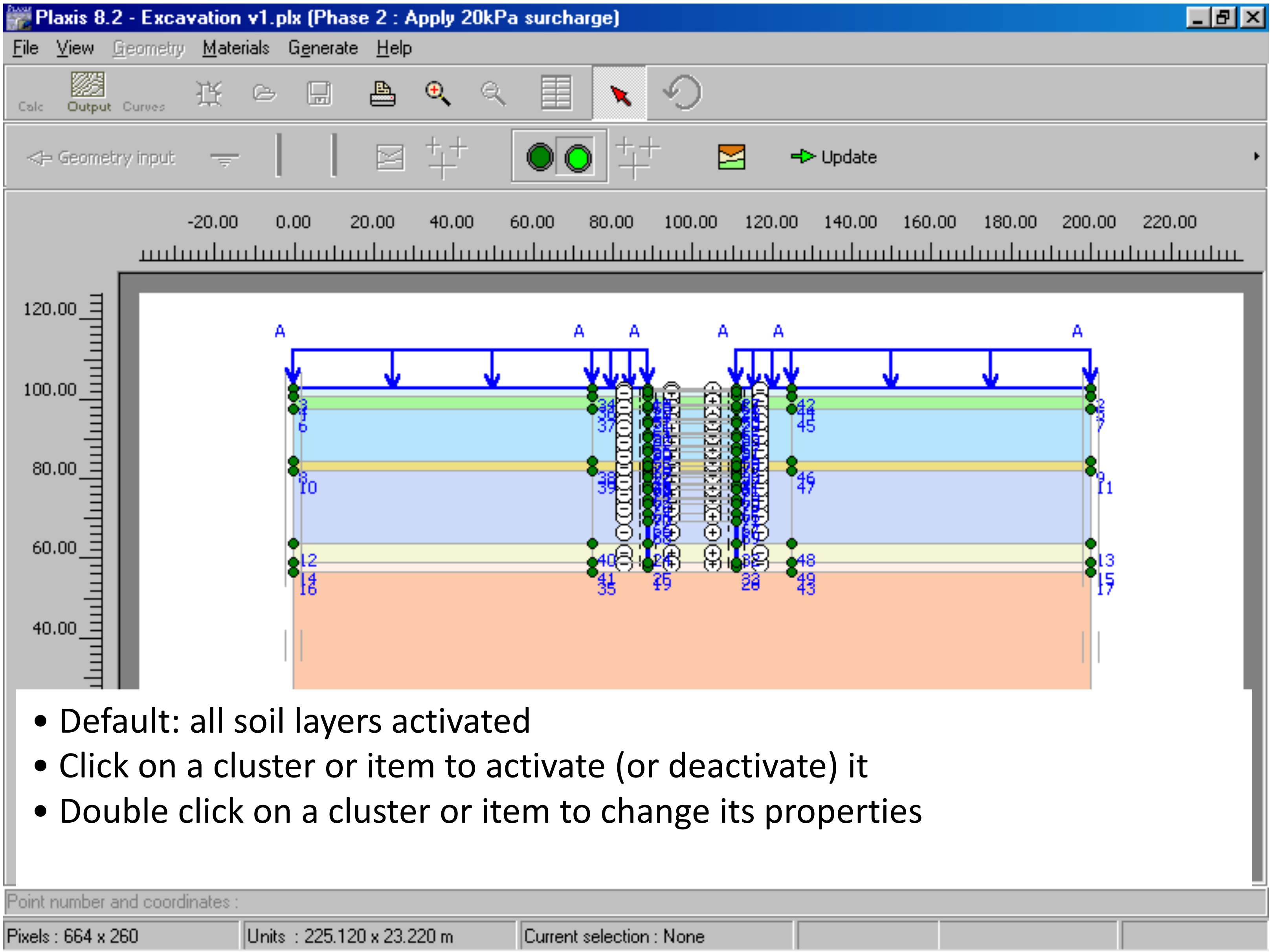
- Plastic calculations
- Consolidation analysis
- Phi-c reduction

Defining construction stages

• Define modelling phase, Insert next phase

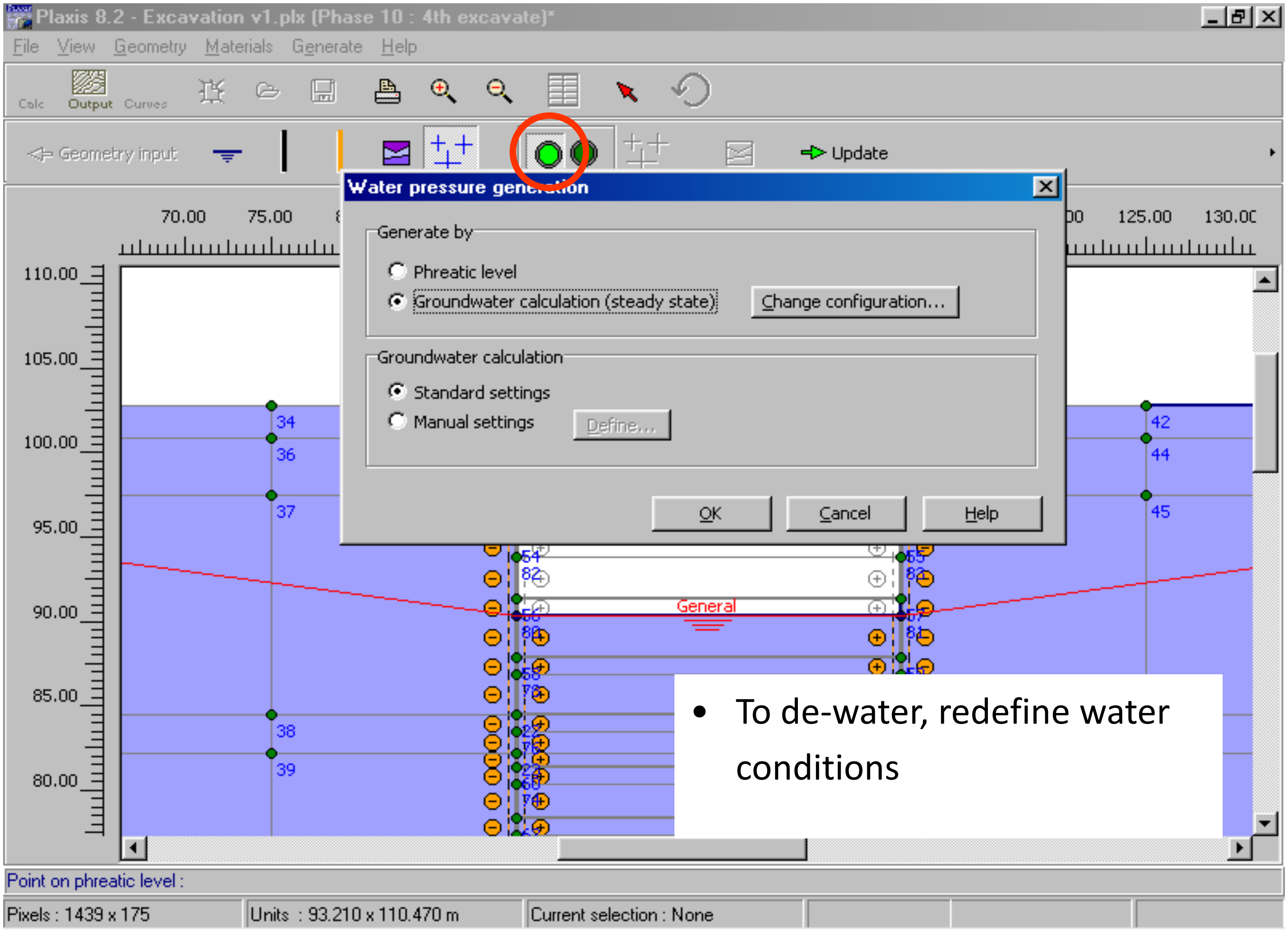


Activating elements for calculation



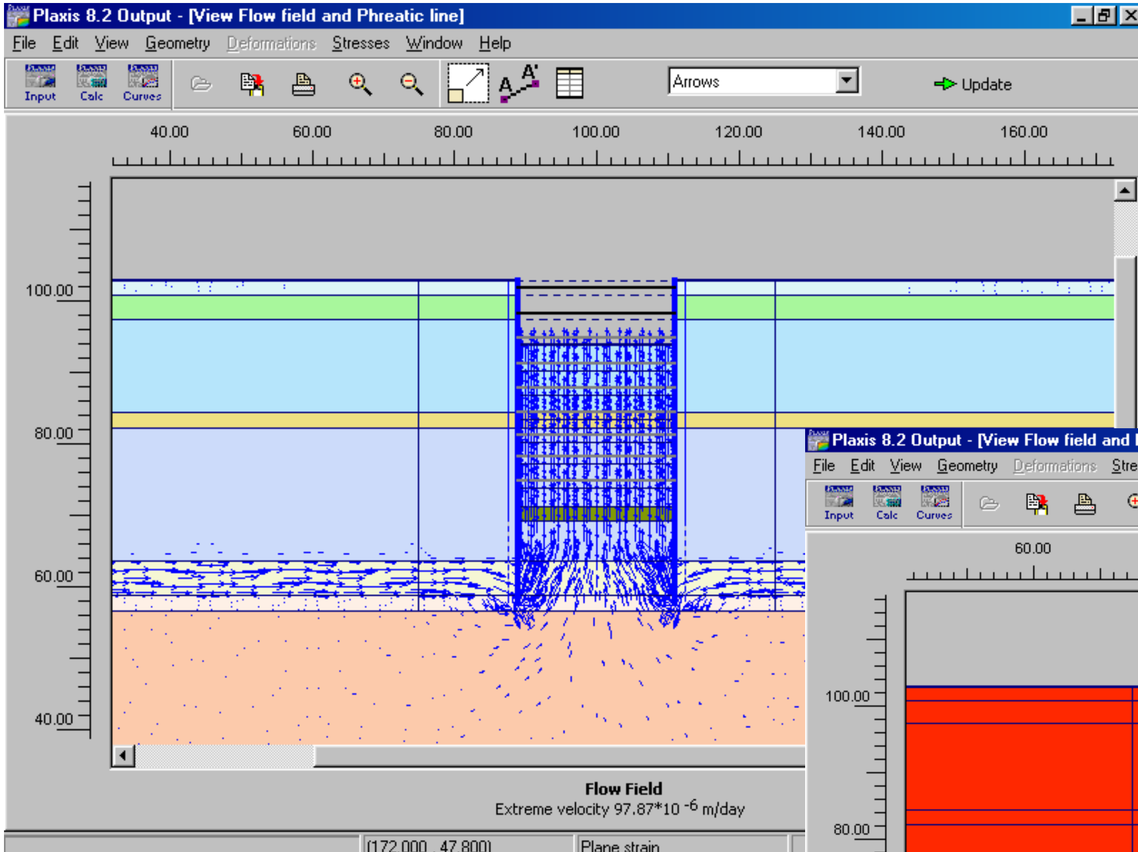
- Default: all soil layers activated
- Click on a cluster or item to activate (or deactivate) it
- Double click on a cluster or item to change its properties

Changing groundwater conditions

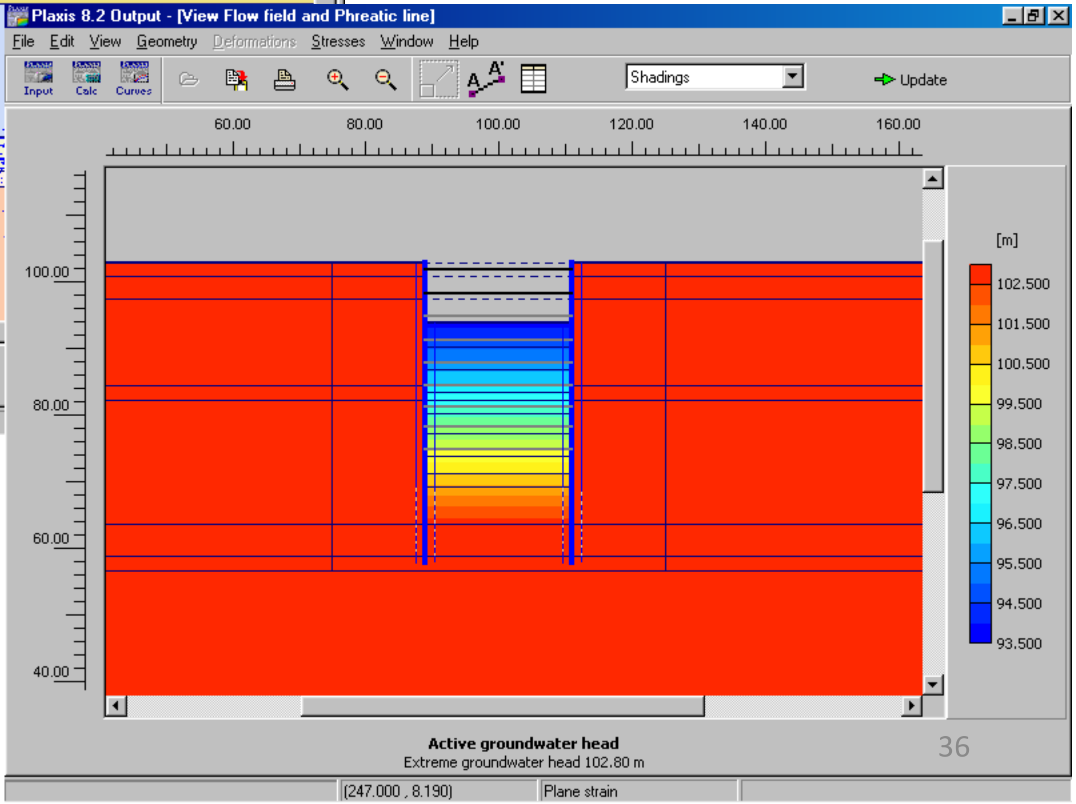


- To de-water, redefine water conditions

Groundwater calculations

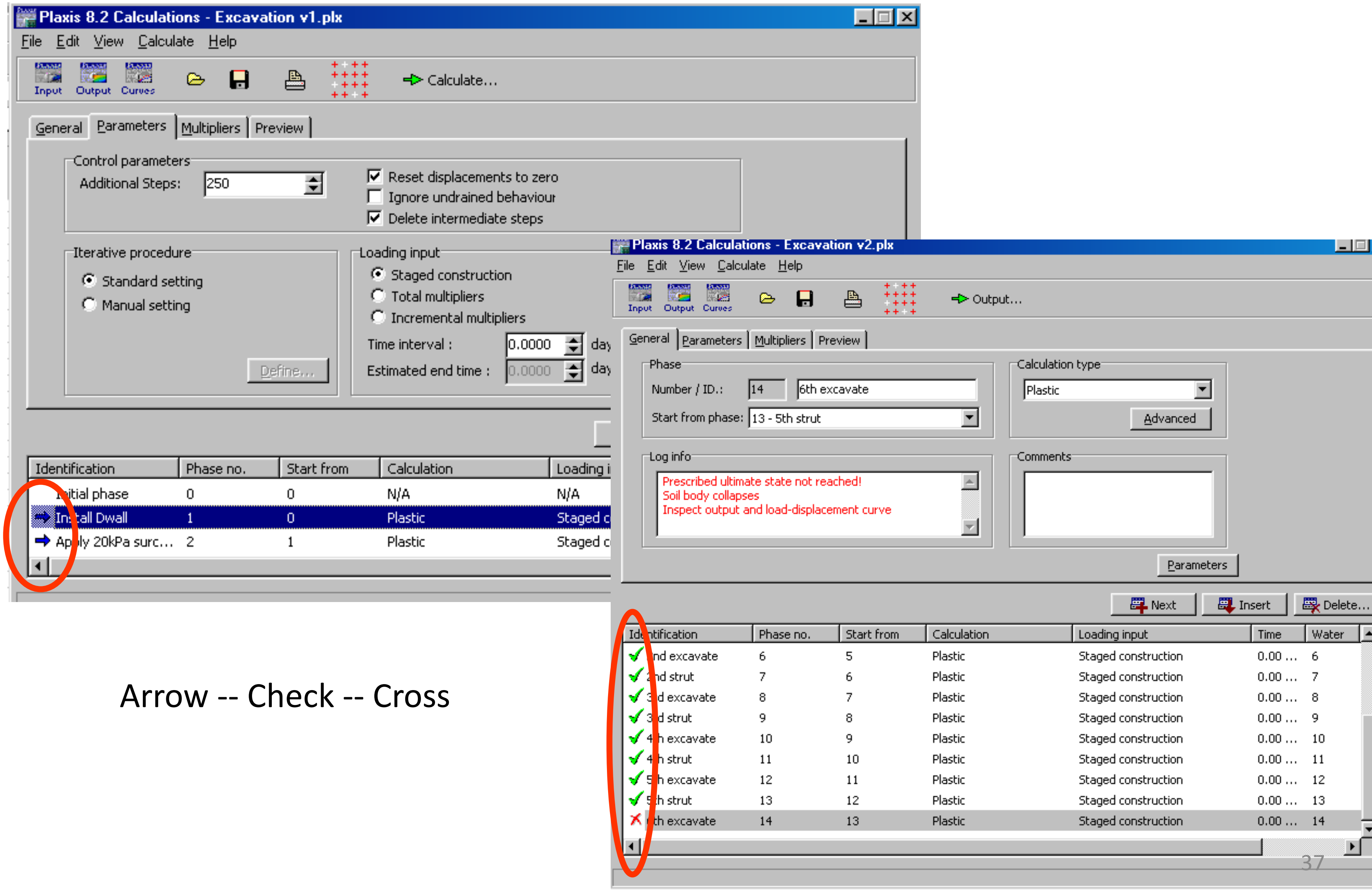


Flow field arrows



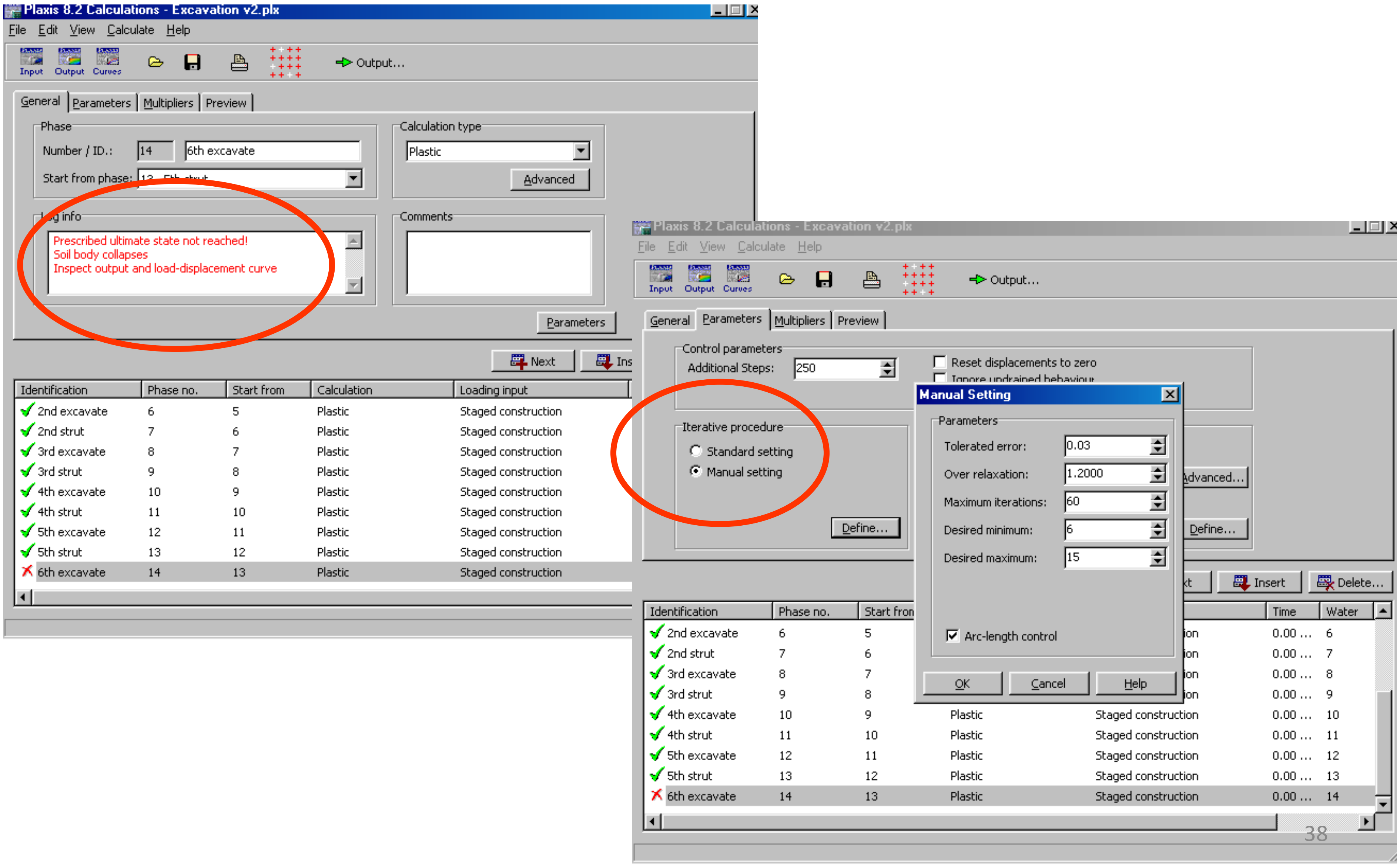
Active groundwater level contours

Selecting calculation phase

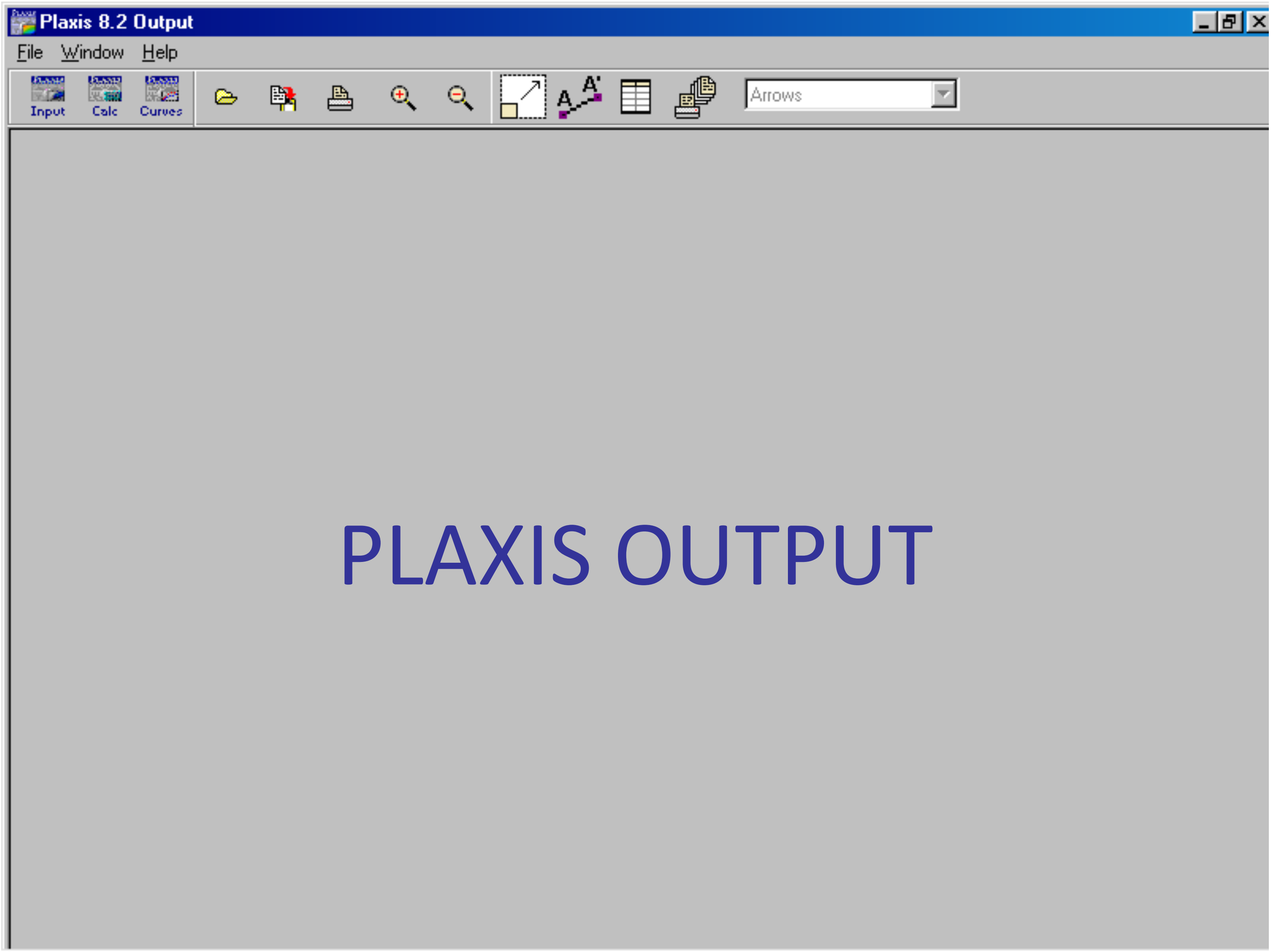


Arrow -- Check -- Cross

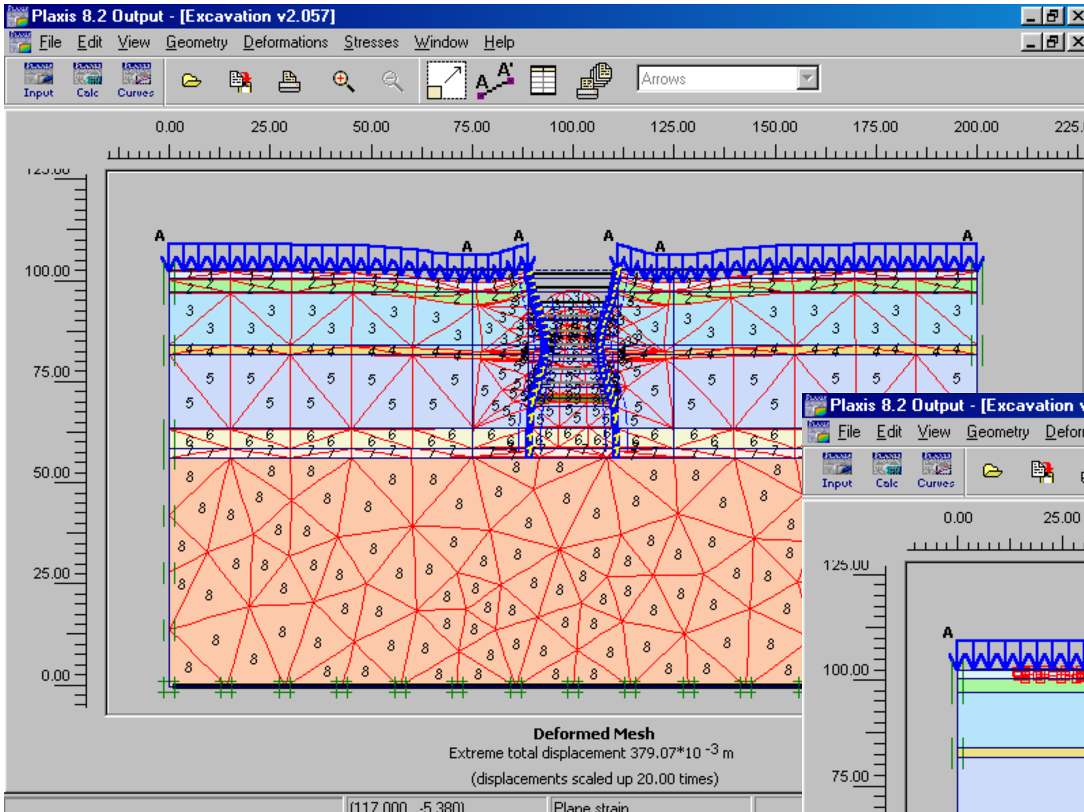
Achieving convergence



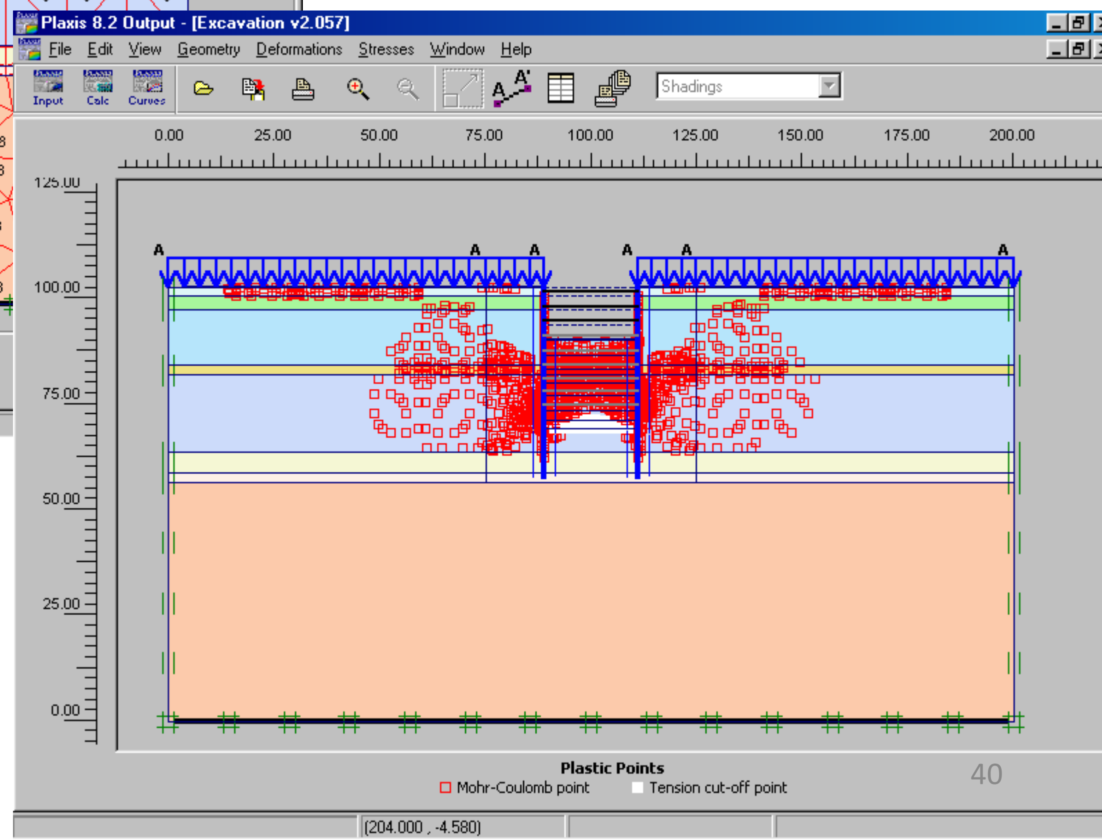
PLAXIS OUTPUT



Deformed mesh & plastic points



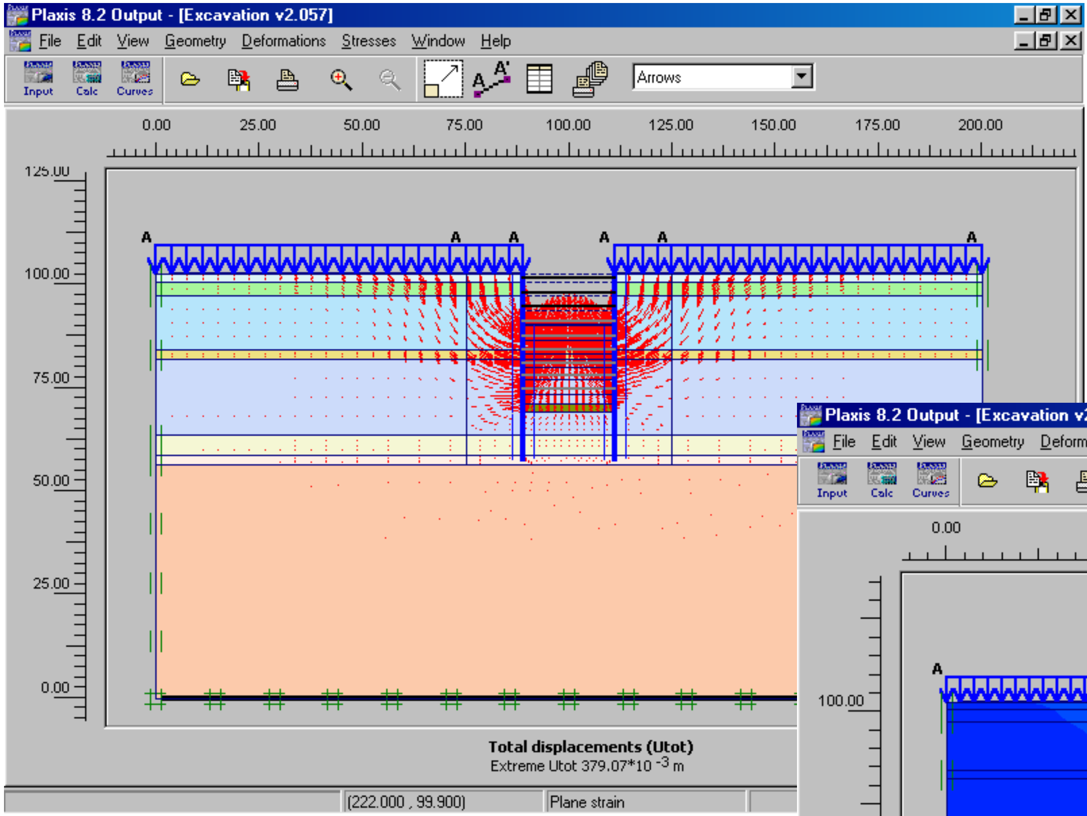
Deformed mesh



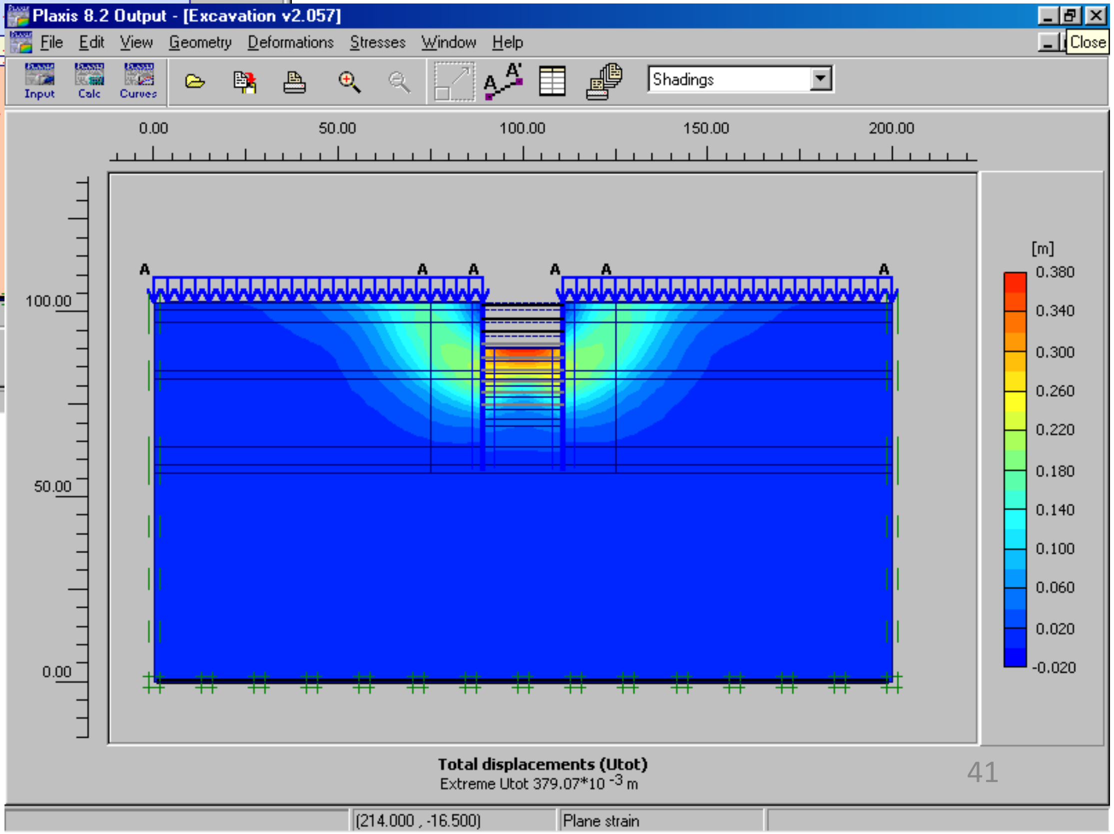
Red - Yielded Points

White - Tension cut-off

Soil Displacement Plots



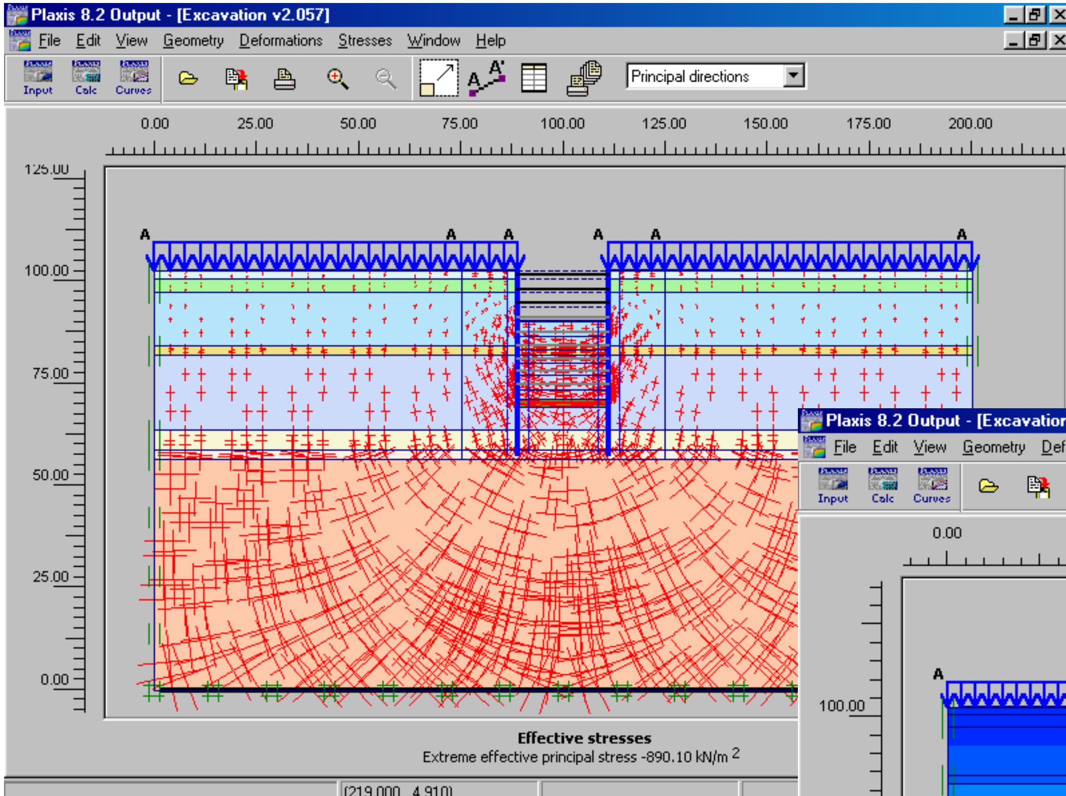
Arrows



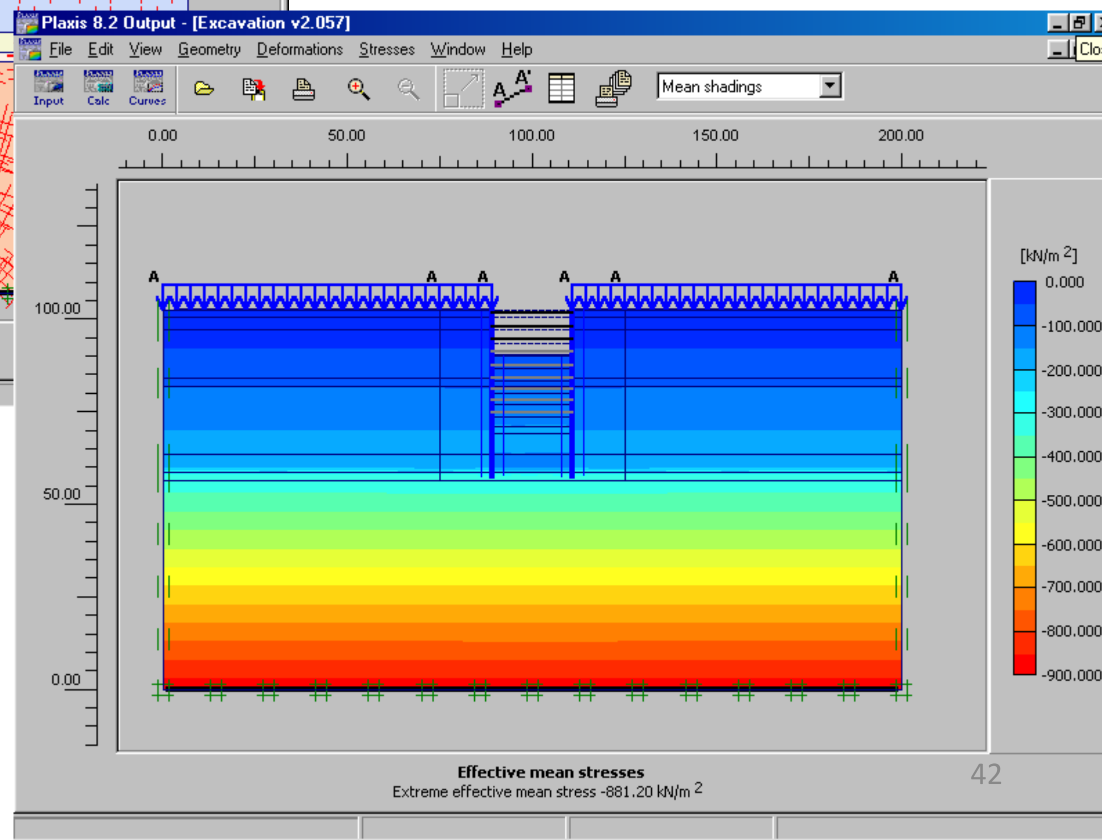
Contours

Soil Stress Plots

Principal directions

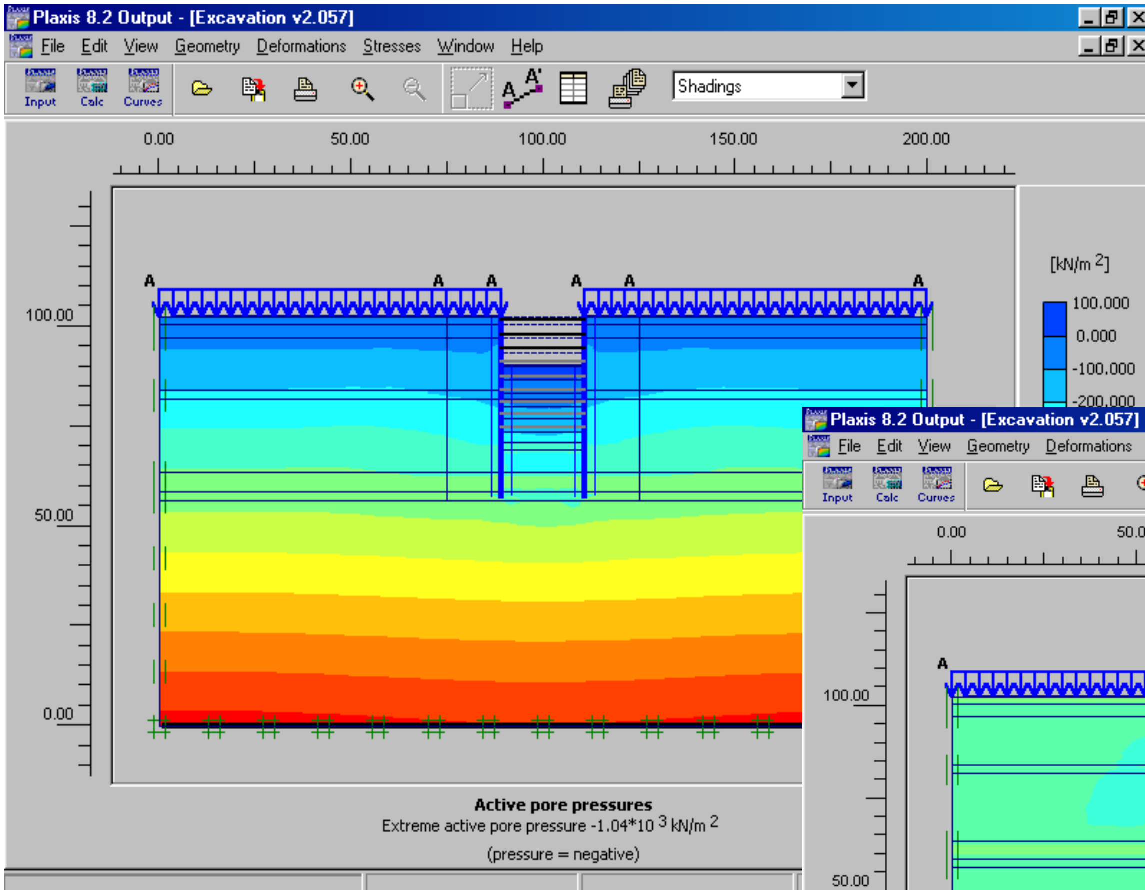


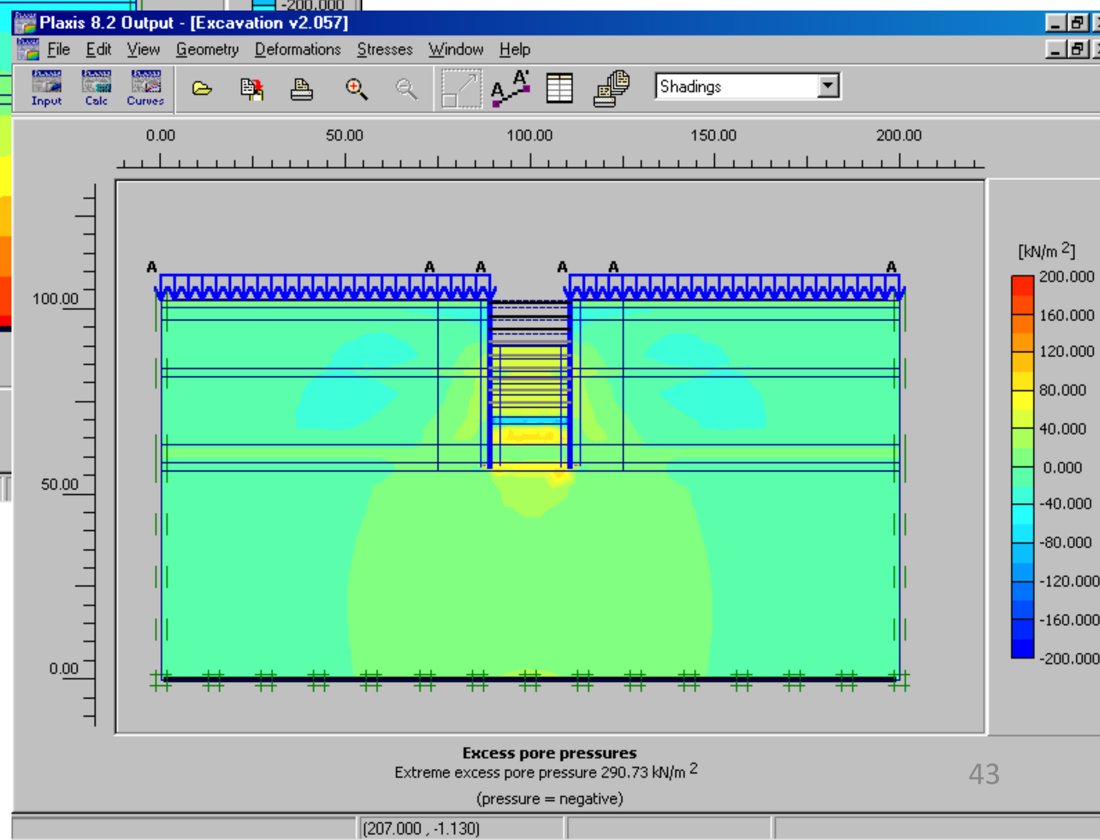
Contours



Pore Water Pressures

Active pore Pressures





Excess pore pressures

Strut loads

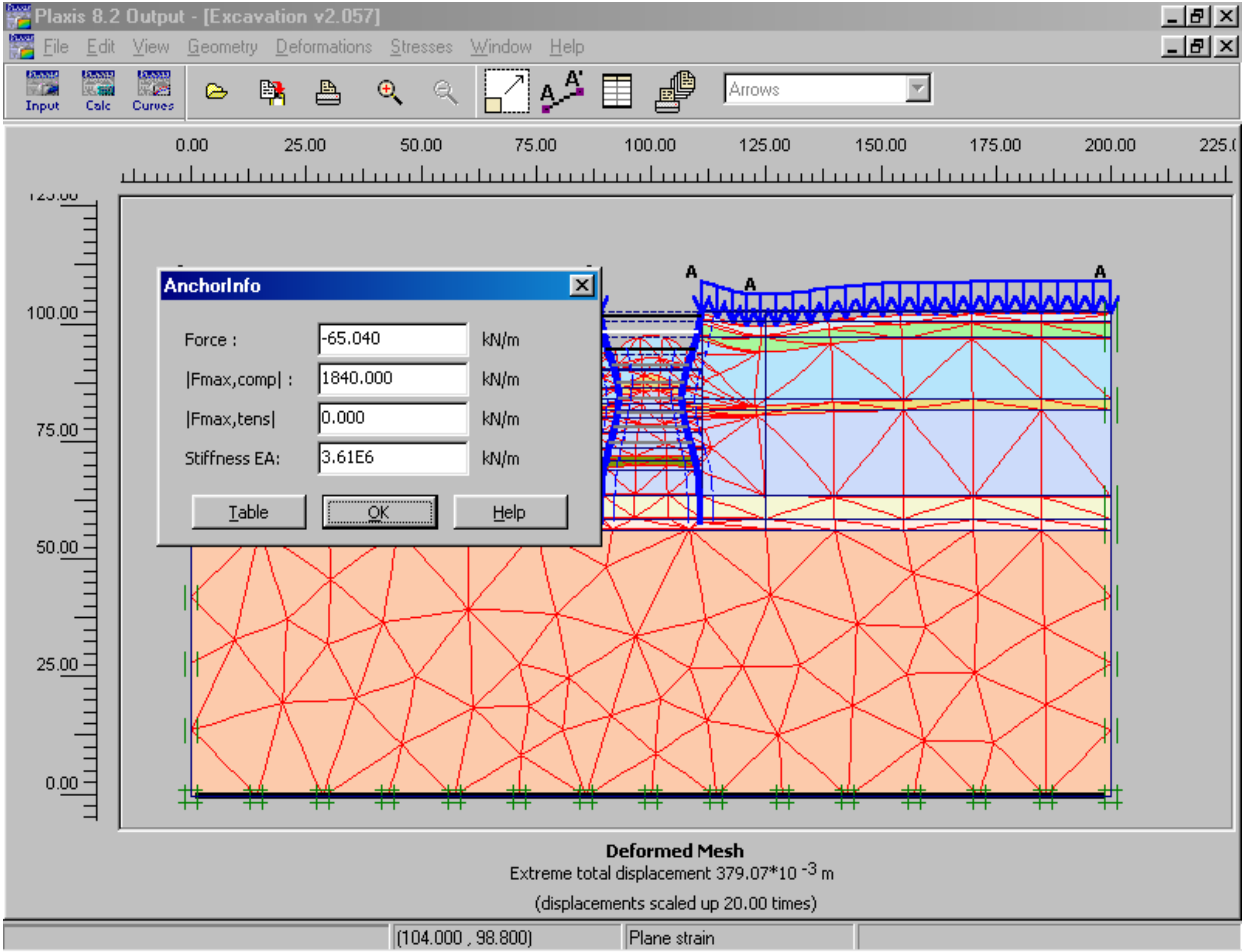

Plaxis 8.2 Output - [Excavation v2.057]
File Edit View Geometry Deformations Stresses Window Help
Input Calc Curves
Arrows
0.00 25.00 50.00 75.00 100.00 125.00 150.00 175.00 200.00
100.00 75.00 50.00 25.00 0.00
AnchorInfo
Force : -65.040 kN/m
|Fmax,comp| : 1840.000 kN/m
|Fmax,tens| 0.000 kN/m
Stiffness EA: 3.61E6 kN/m
Table OK Help
A A A
Deformed Mesh
Extreme total displacement 379.07*10 -3 m
(displacements scaled up 20.00 times)
(104.000 , 98.800)
Plane strain

Plate element plots

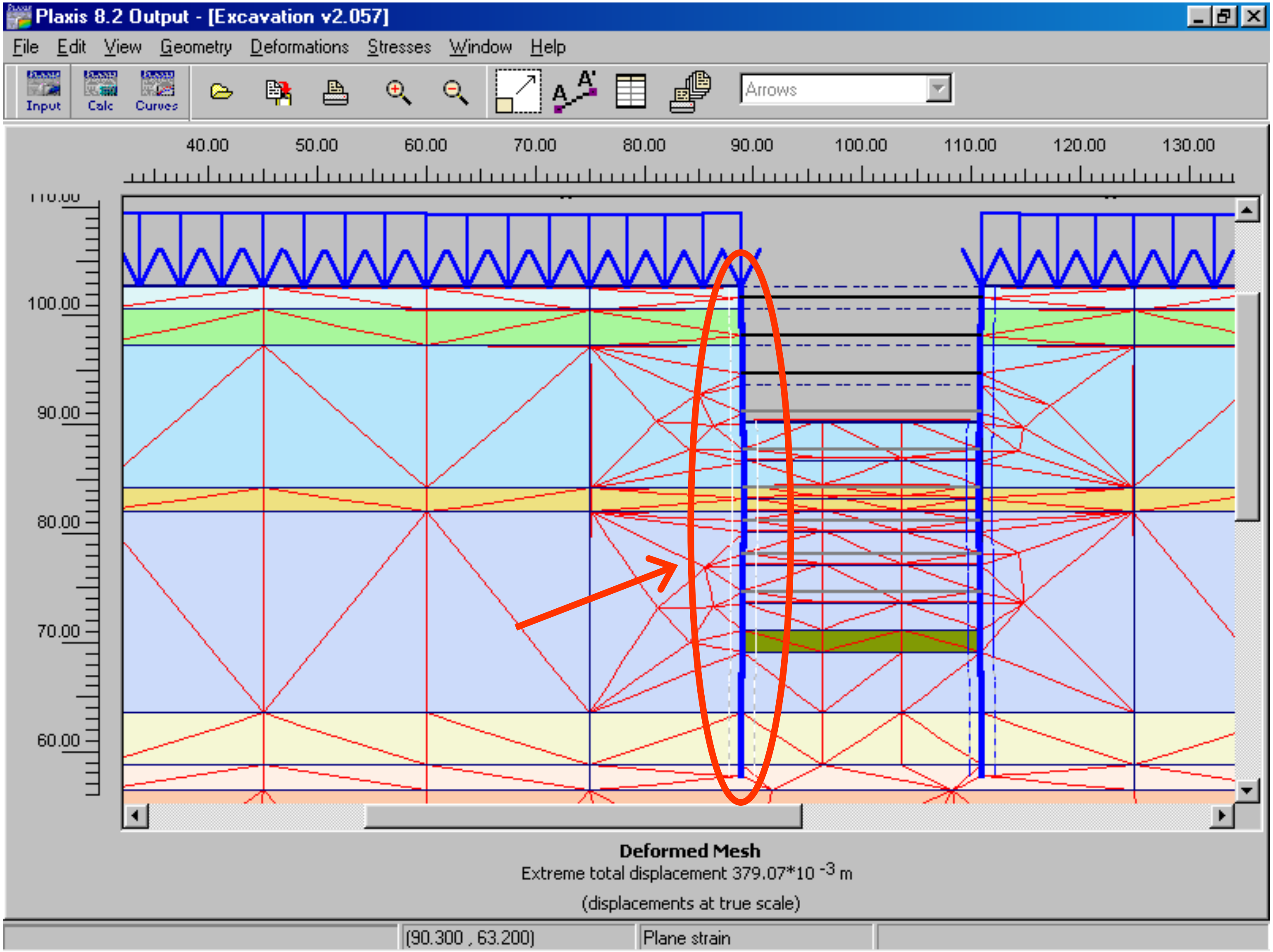
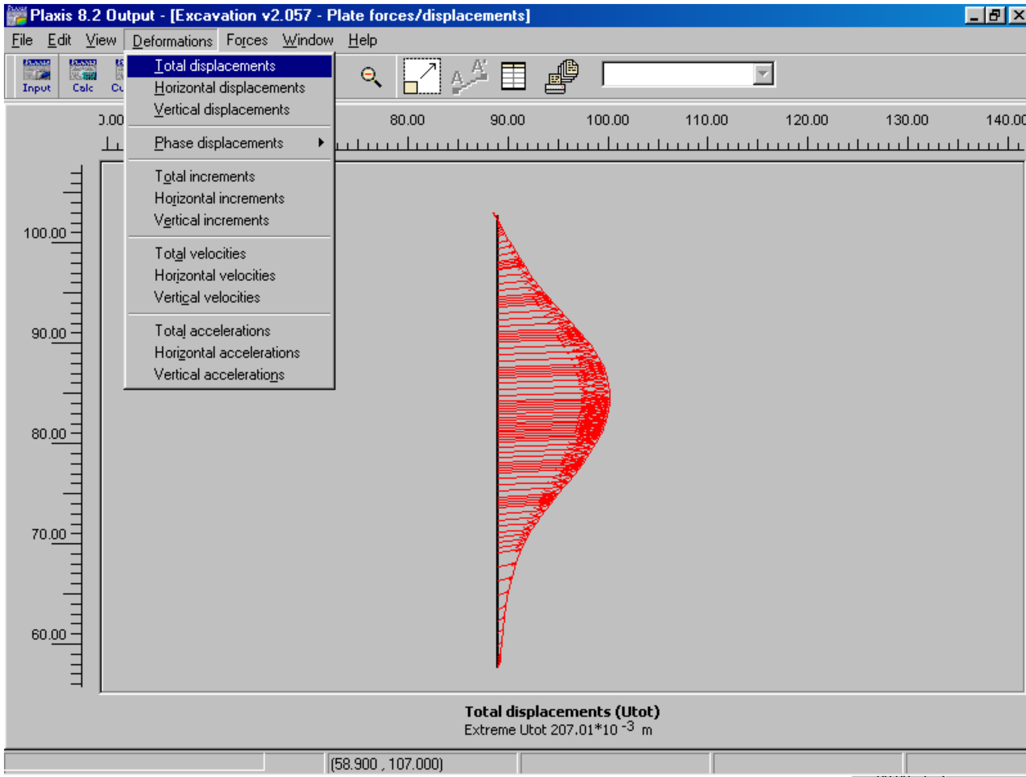
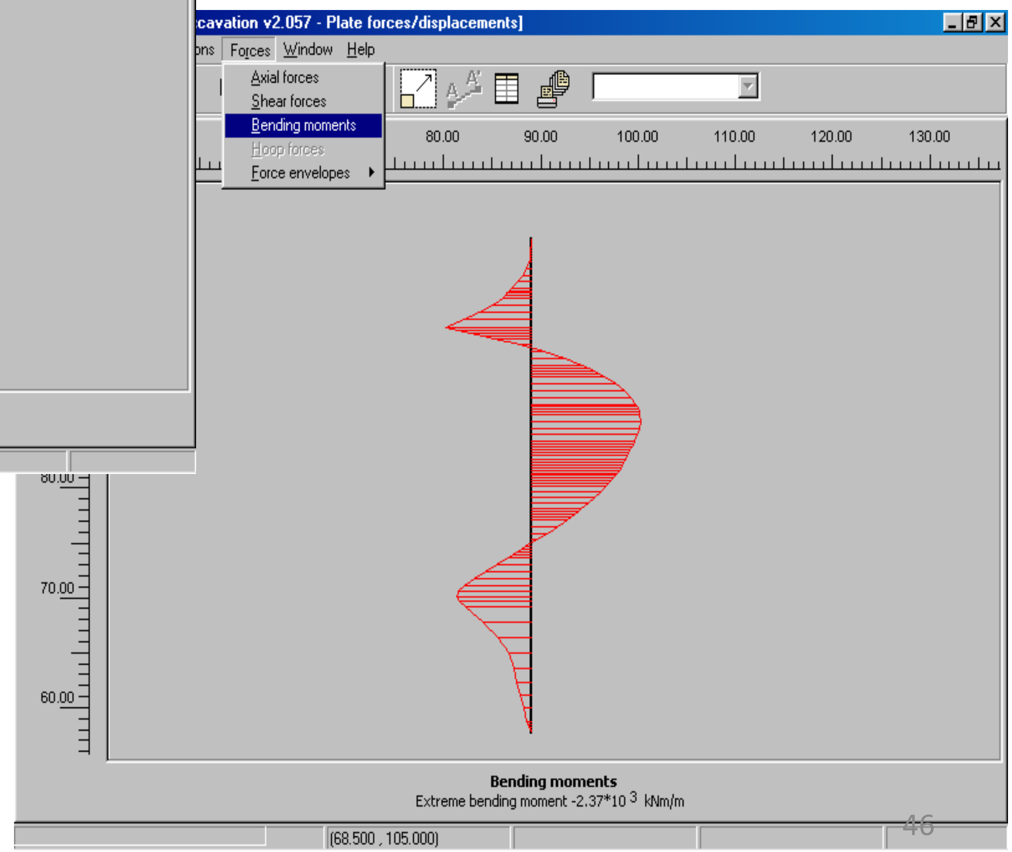


Plate element plots

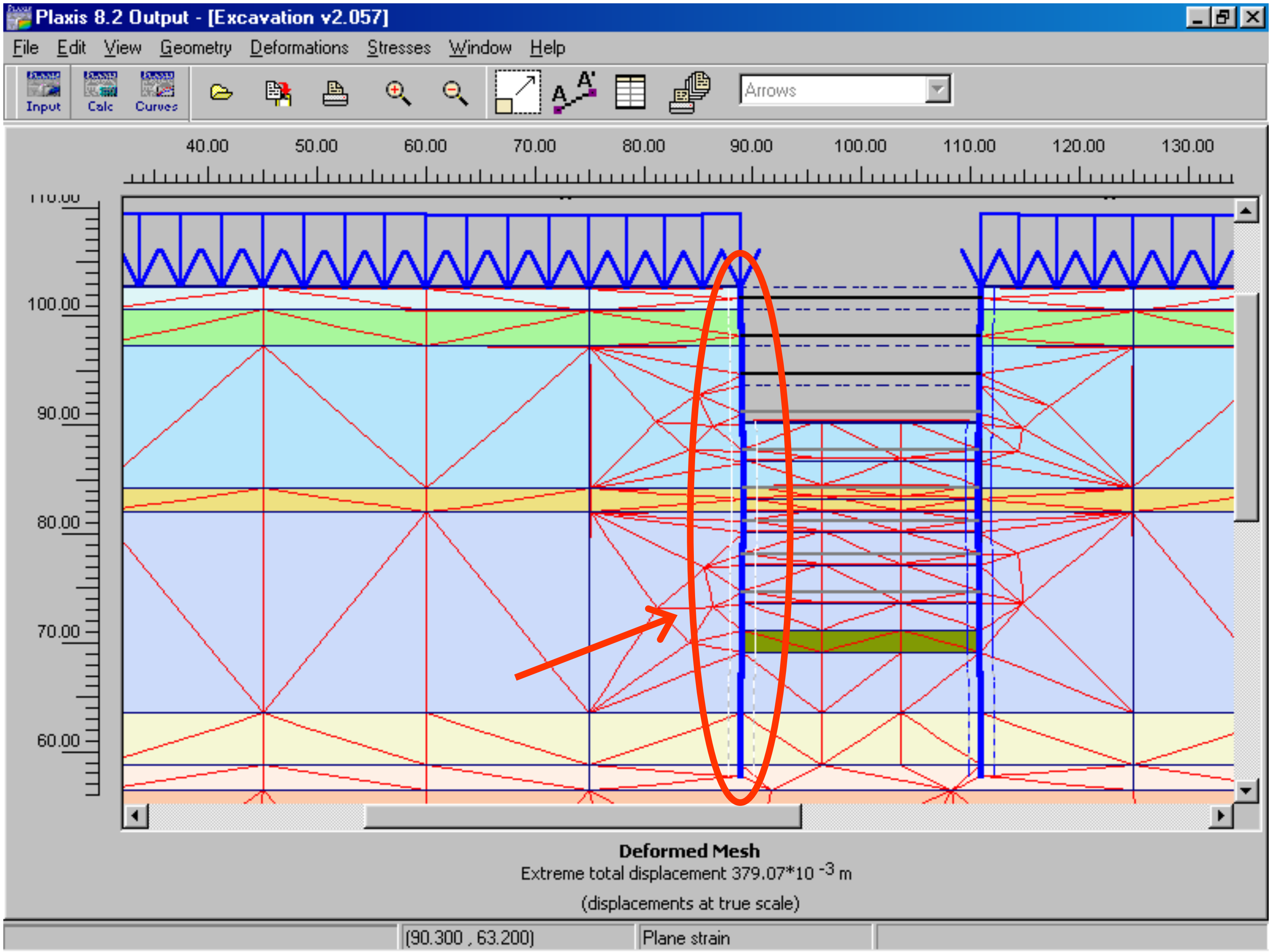


Total displacements

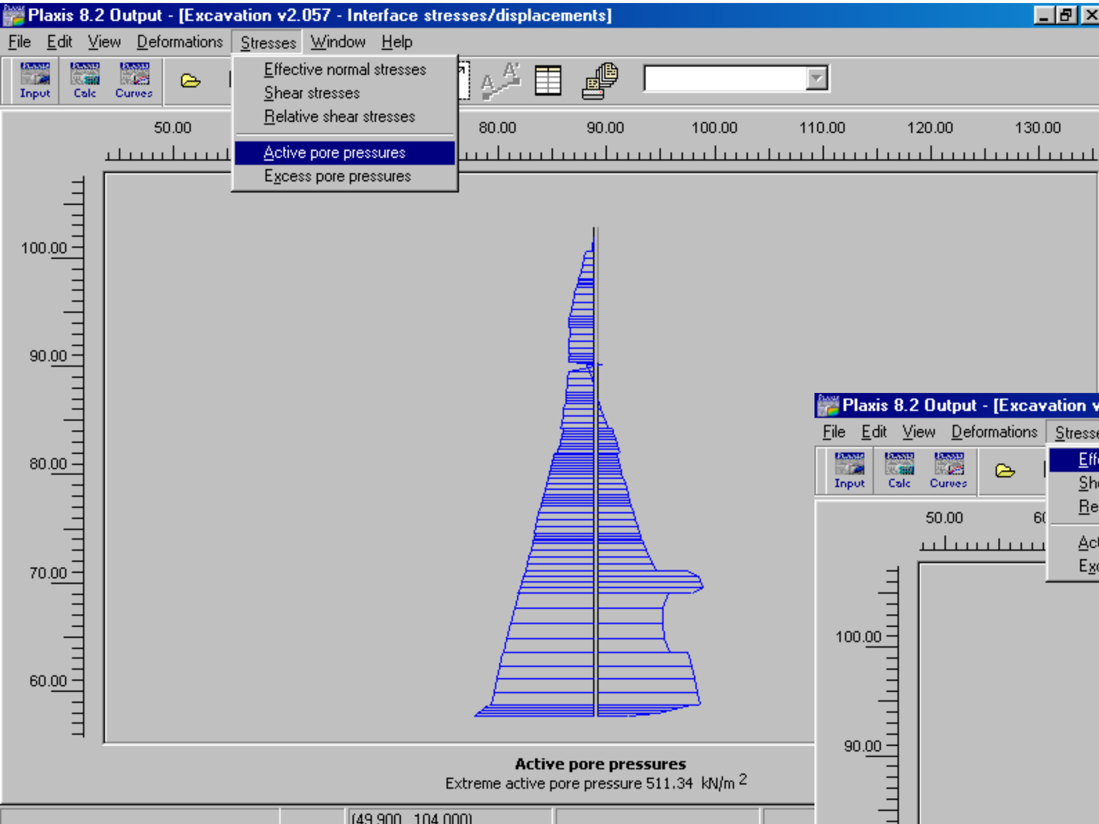


Bending moments

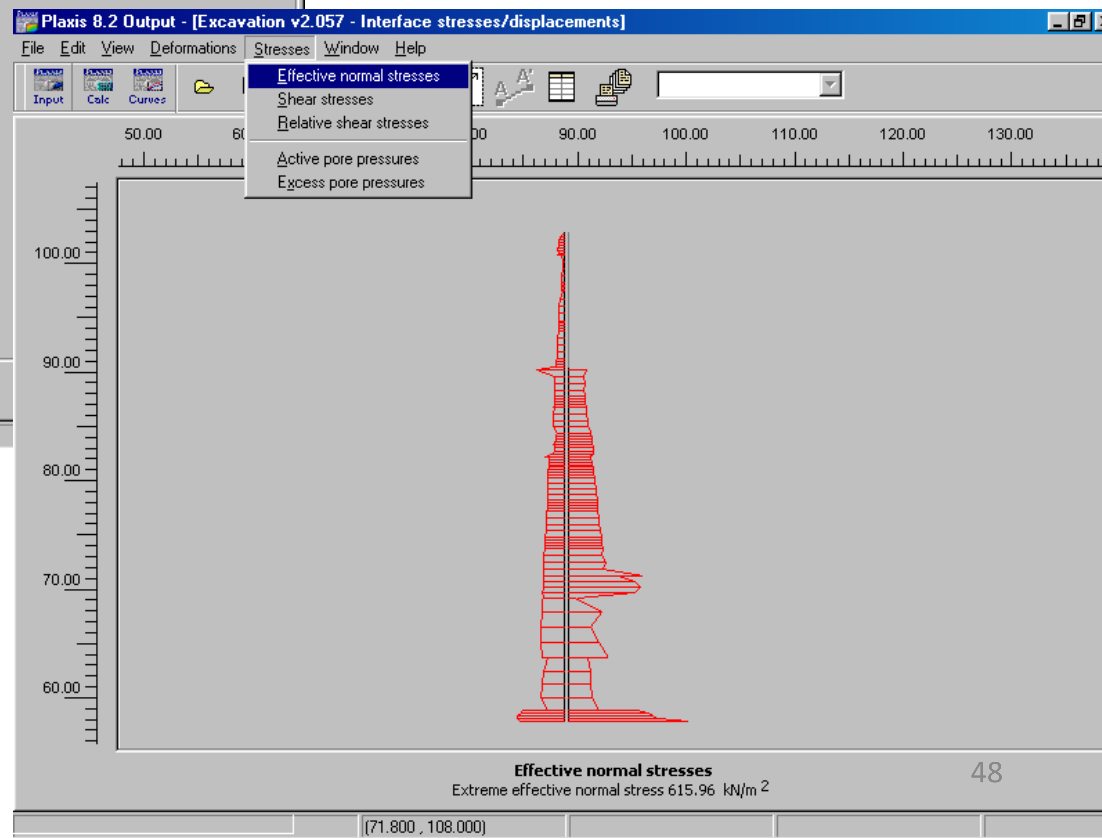
Interface element plots



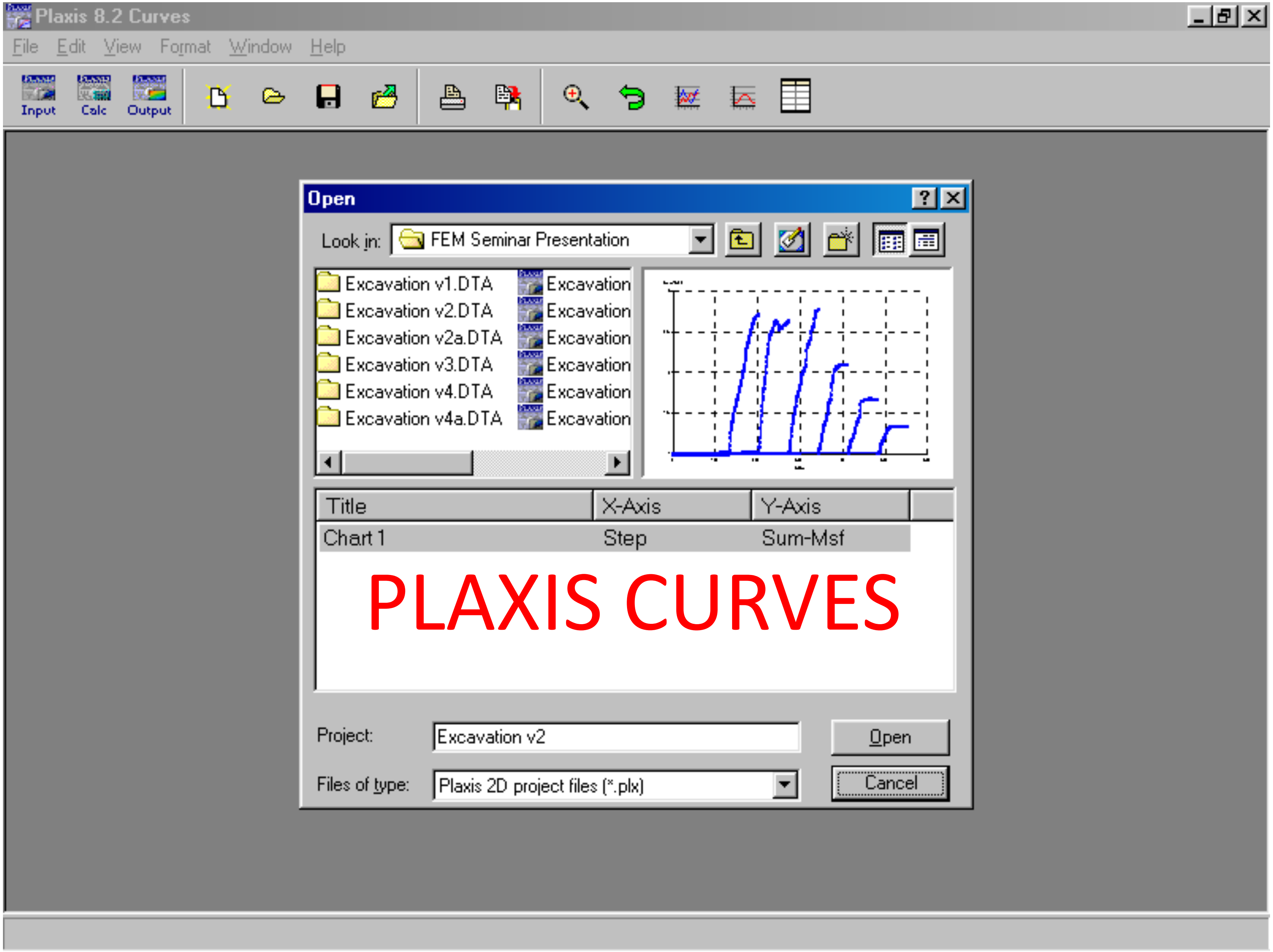
Interface element plots



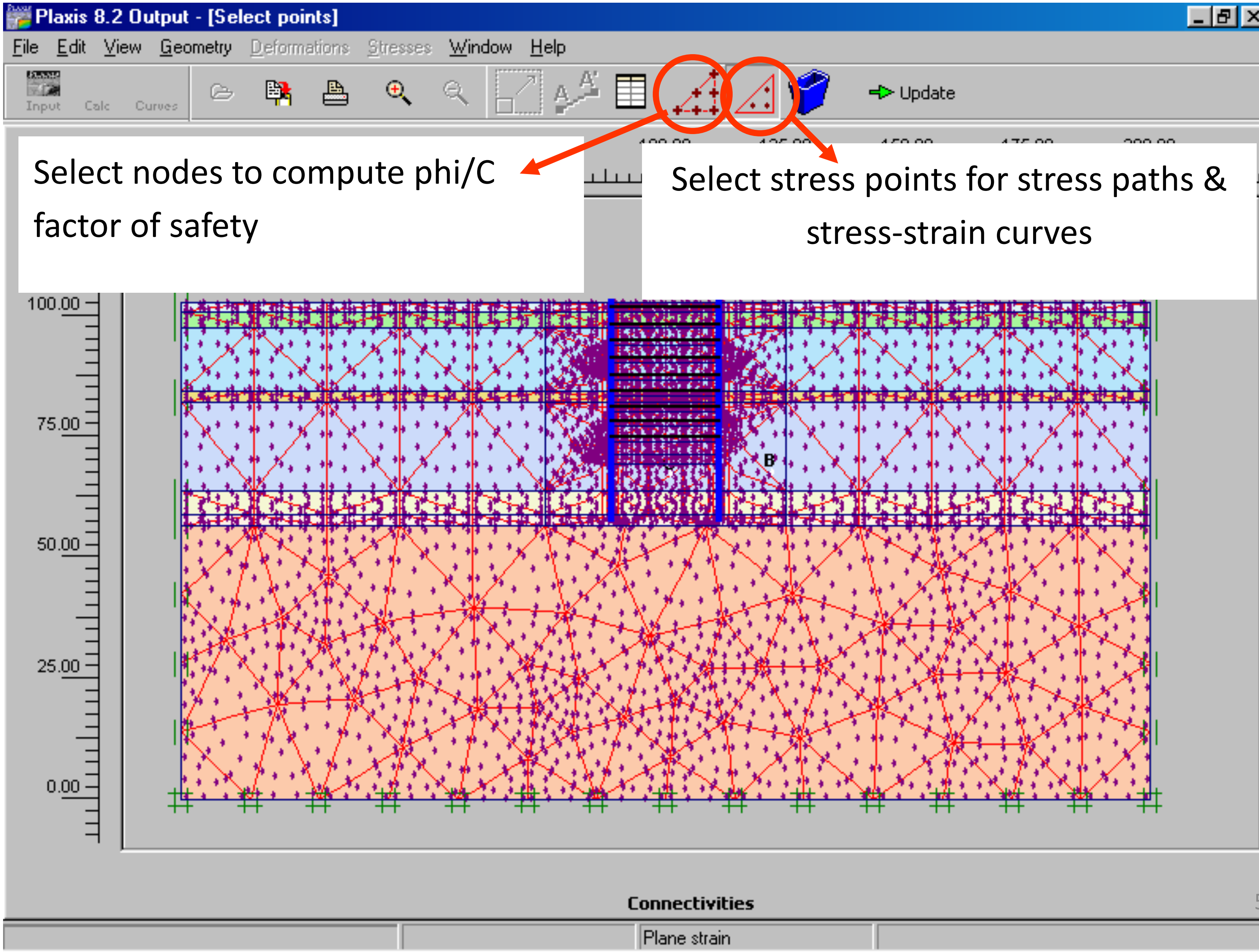
Active pore Pressures



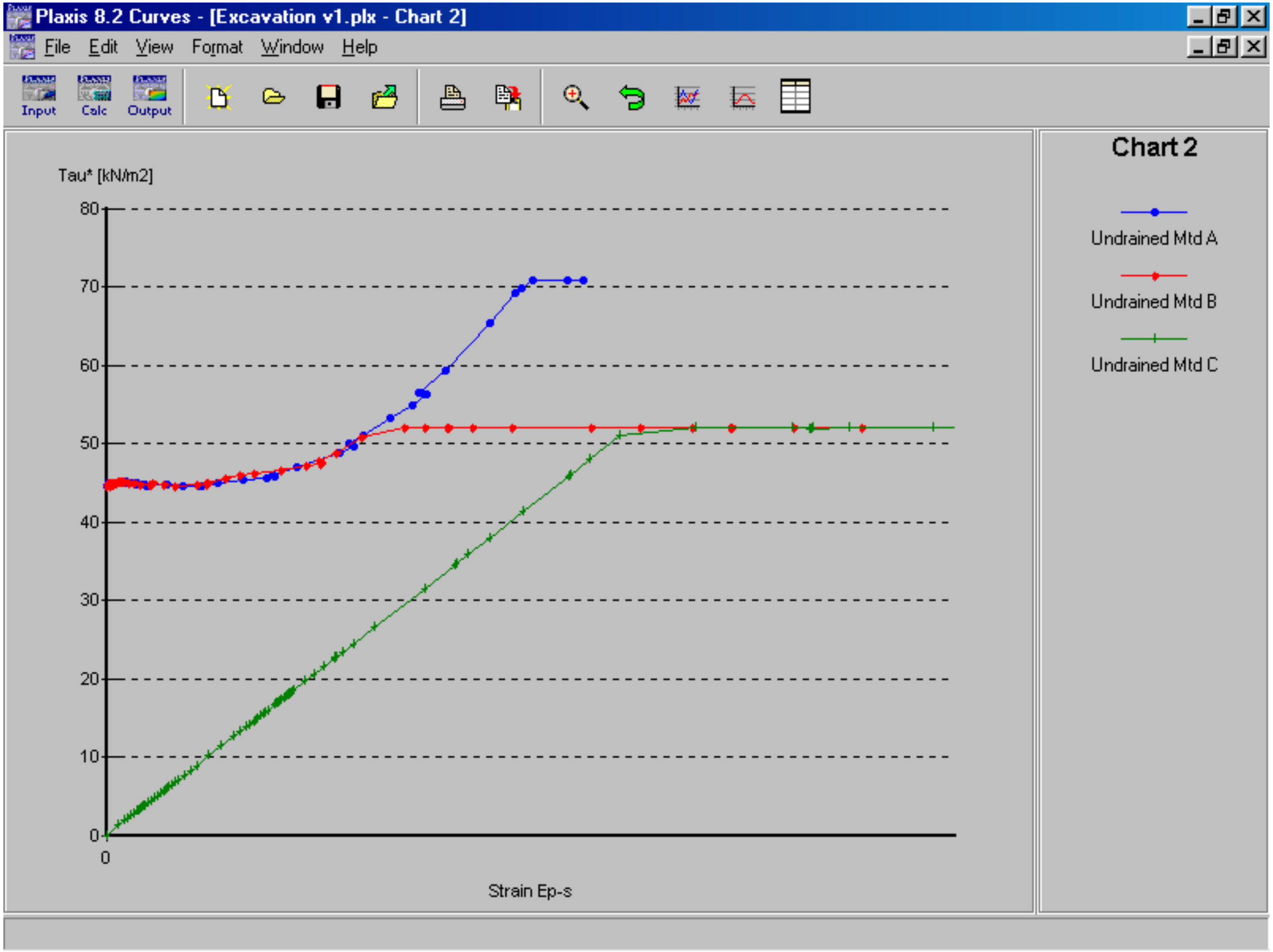
Excess pore pressures


Plaxis 8.2 Curves
File Edit View Format Window Help
Input Calc Output
Open
Look in: FEM Seminar Presentation
Excavation v1.DTA
Excavation v2.DTA
Excavation v2a.DTA
Excavation v3.DTA
Excavation v4.DTA
Excavation v4a.DTA
Excavation
Excavation
Excavation
Excavation
Excavation
Excavation
Title
X-Axis
Y-Axis
Chart 1
Step
Sum-Msf
PLAXIS CURVES
Project:
Excavation v2
Open
Files of type:
Plaxis 2D project files (*.plx)
Cancel

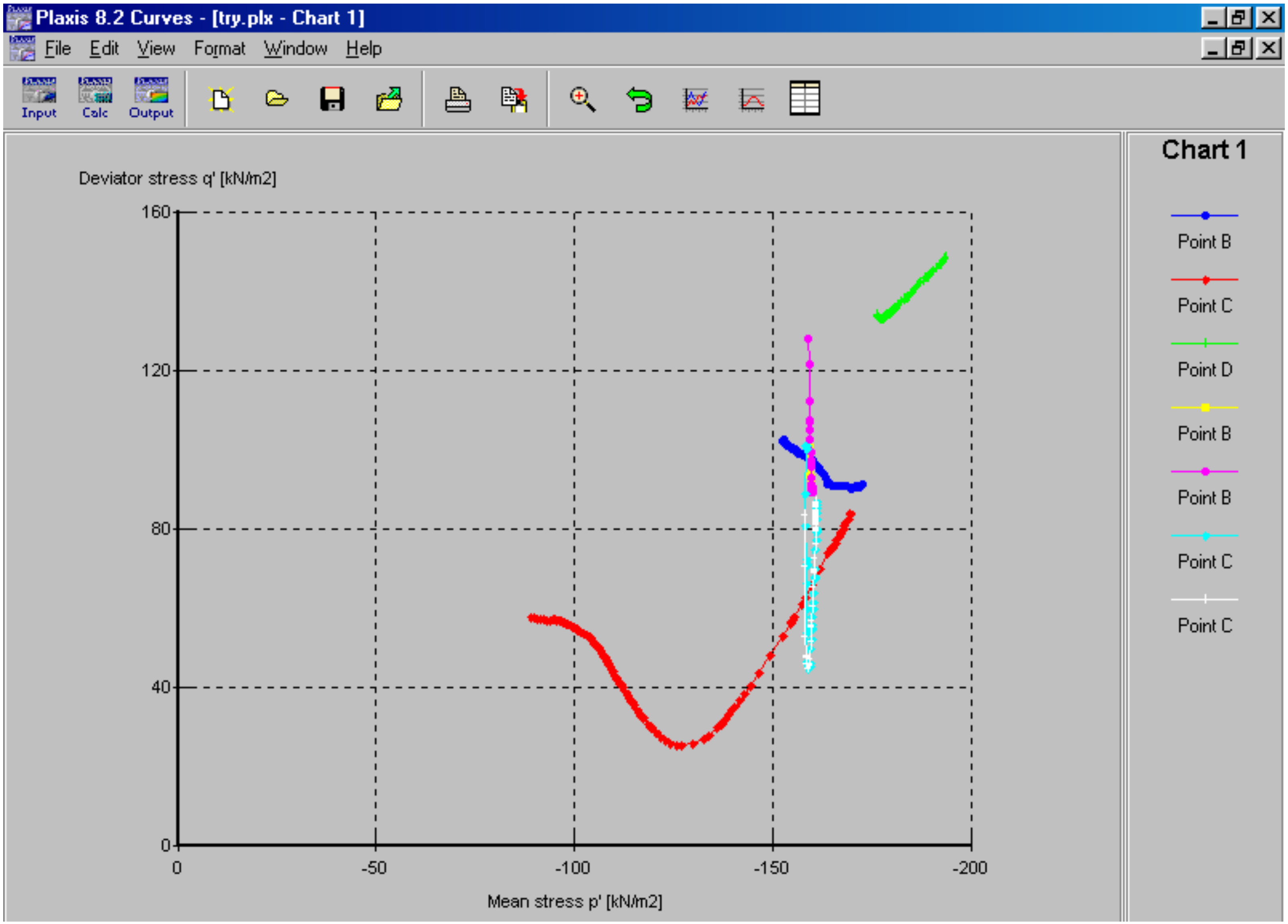
Selecting Nodes & Stress Points



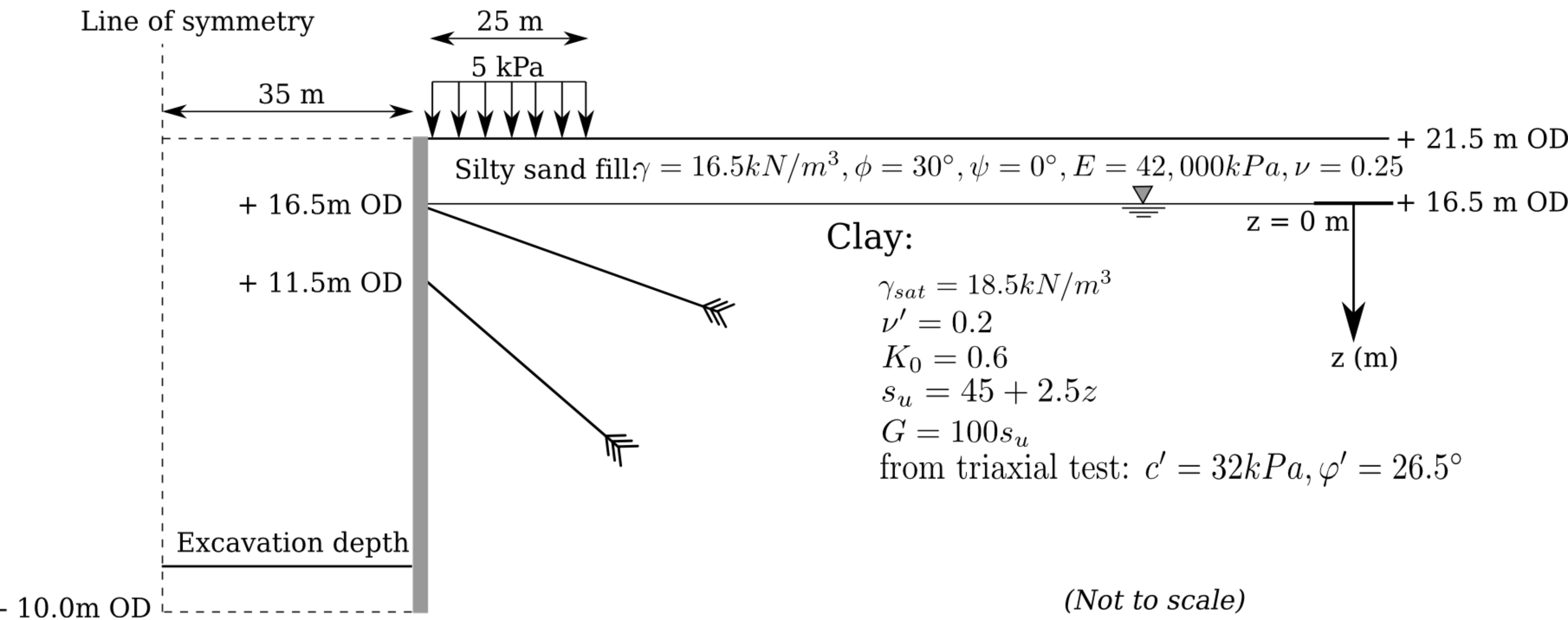
Plaxis Curves - Stress-Strain plots



Plaxis Curves - Stress Path plots



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



(Not to scale)

Total Stress Approach

- Stiffness is modelled using an undrained Young's modulus Eu and Poisson ratio.
- Pore pressures are not generated
- Undrained shear strength Su is an input parameter.
- Consolidation analysis has no effect and should not be performed.
- C_u (S_u) and undrained friction φ_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. Su 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, ν_u	c_u, φ_u=0	$K_{0,u}$
Effective Stress (triaxial parameters)	Undrained	E', ν'	c', φ'	K_0
Effective stress (strength profile)	Undrained	E', ν'	c', φ'	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

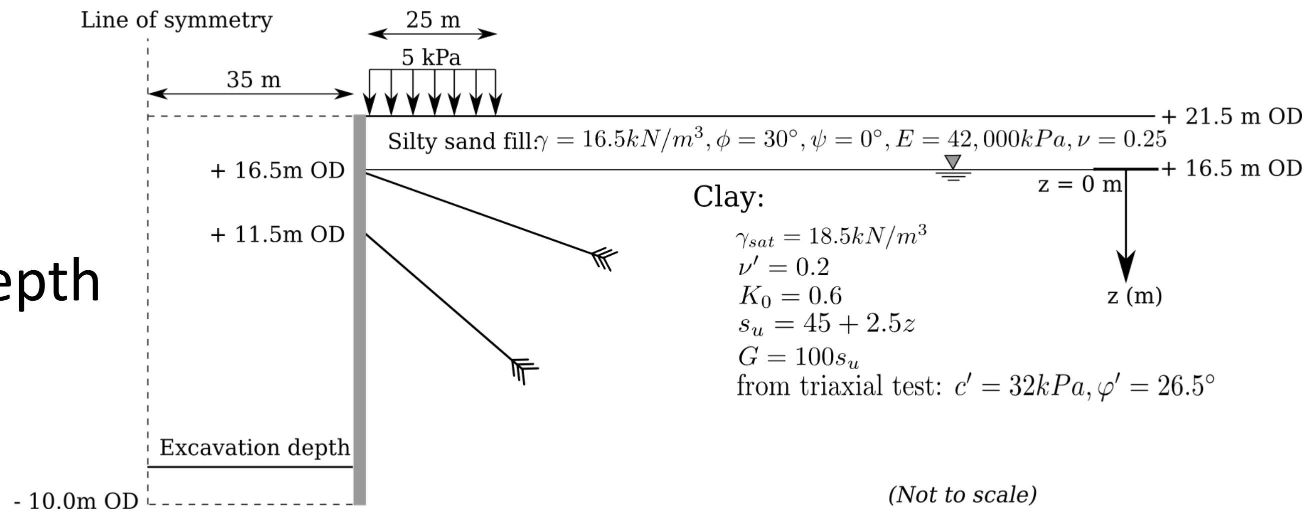
- Use Mohr's circles to determine s_u from c' and φ' and vice versa
- Remember to dewater the excavation

General:

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

Total Stress Approach

- *Use V = 0.495*
- K_0 Varies with depth
- φ'=0

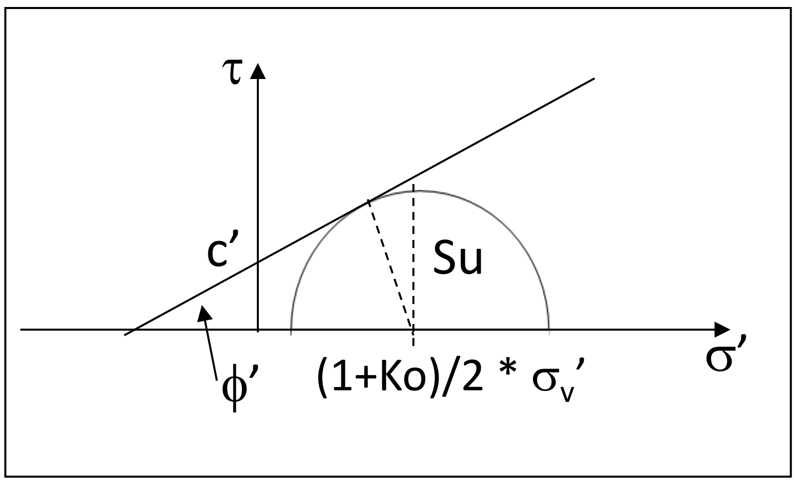


Depth	σ_v	u	σ_v'	$k_0\sigma_v'=\sigma_h'$	σ_h	K_0	K_0 Layer
0	82.5	0	82.5	49.5	49.5	0.6	
6.5	202.75	65	137.75	82.65	147.65	0.728	0.664
11.5	295.25	115	180.25	108.15	223.15	0.756	0.742
16.5	387.75	165	222.75	133.65	298.65	0.77	0.763

Undrained analysis using Effective stress method

Effective stress A

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



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

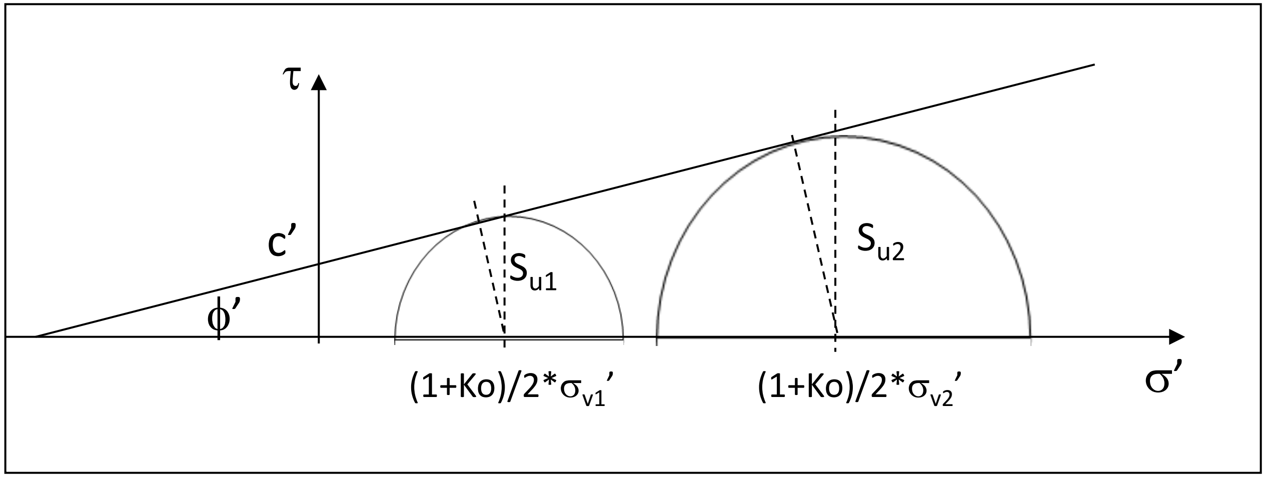
$S_u = c' \cos\phi + (1+K_o) \sin\phi' / 2 * \sigma_{vo}'$

Thus when z = 0, Su = c'

Undrained analysis using Effective stress method

Effective stress B

- Define strength c'-ϕ' in terms of the "equivalent" effective stress parameters, with zero dilation
- v'= 0.2
- E and Ko 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!