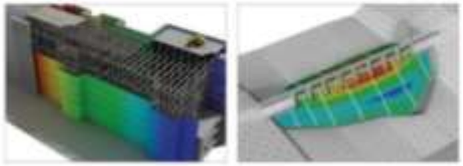
# GEOSTUDIO Tutorials Results and Procedure Comparison

GTS NX
Geo-Technical analysis System New eXperience




Angel Francisco Martinez

Application Engineer
MIDAS IT
Estados Unidos


Integrated Solver Optimized for the next generation 64-bit platform

**Finite Element Solutions for Geotechnical Engineering**

MIDAS

# Contents

# Stress Changes in the Ground Beneath a Round Tank

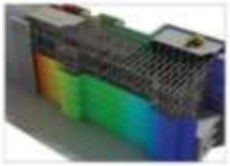
## (SIGMA W Comparison)

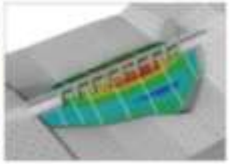
Angel Francisco Martinez

Application Engineer
MIDAS IT
Estados Unidos

GTS NX

Geo-Technical analysis System New eXperience





Integrated Solver Optimized for the next generation 64-bit platform

**Finite Element Solutions for Geotechnical Engineering**

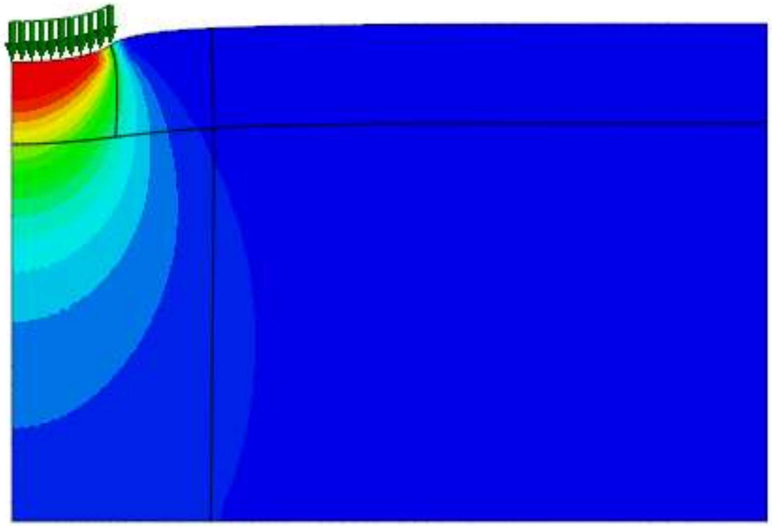
MIDAS

# Introduction



The primary purpose of this tutorial is to demonstrate how to get started using MIDAS GTS NX, to introduce the steps to run analysis, and to illustrate how to get results. This problem show how to obtain stress changes in the ground beneath a round tank filled with a fluid. A direct comparison will be done against Intro SIGMA/w Tutorial under same conditions.

When studying a problem that is symmetric about a vertical axis, the three-dimensional effects can be analyzed with a two-dimensional finite element mesh. Axisymmetric modeling in GTS NX for this case considers the stresses in the X-Y plane as well as the circumferential stresses.



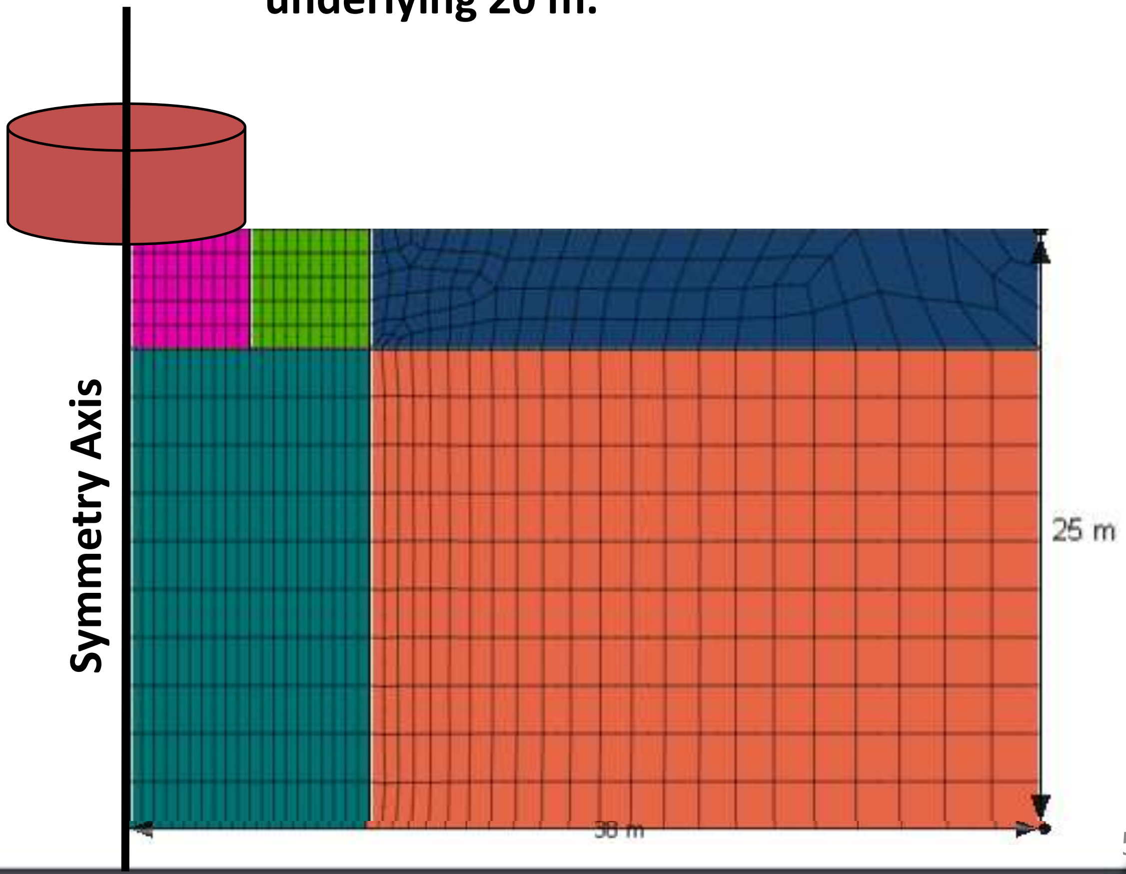
# Initial model dimensions

GTS NX

## Model Conditions

- Axisymmetric
- 5 meter radius tank
- Ground is 38 m wide, 25 m deep
- 2 layers
    1. Upper soil: E = 3000 kPa and Poisson's ratio v = 0.45
    2. Lower soil: E = 4000 kPa and Poisson's ratio v = 0.45
- 40 Kpa pressure load

**Figures show the corresponding problem setup. The radius of the tank is 5 m and the applied pressure is 40 kPa. The soil region is 38 m wide and 25 m high. The upper 5 m of soil has different properties than the underlying 20 m.**

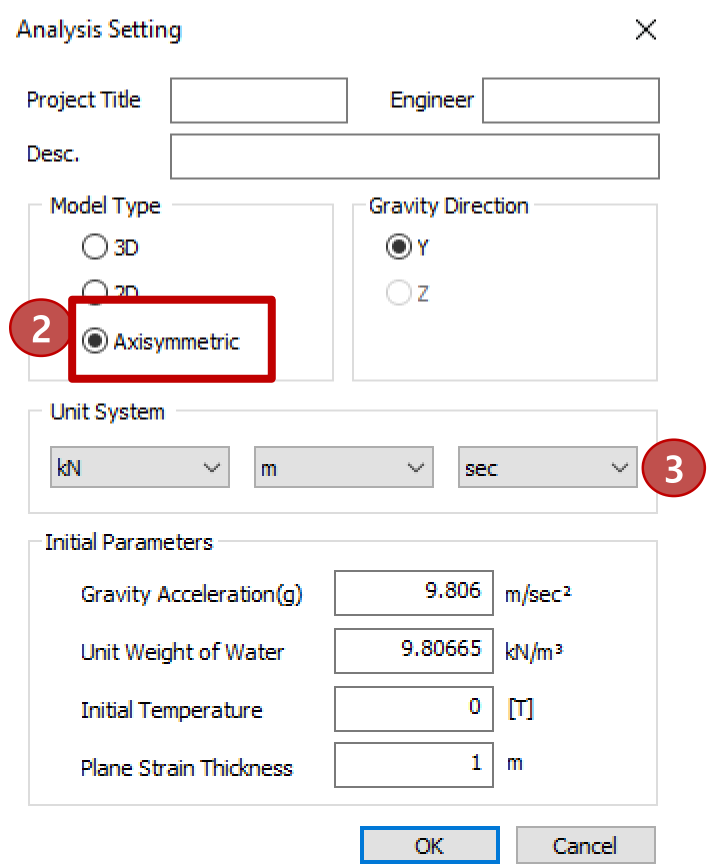


# Start Axisymetric File

## Open New

1. File > New
2. Axisymmetric
3. Units kN, m, sec


1

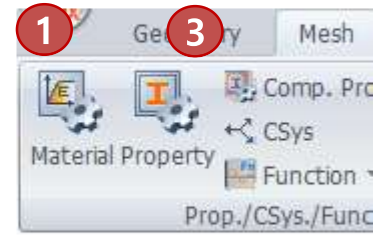


# Materials and Properties

## Define Materials and Properties

**Define material properties of the 2 layers**

1. Menu > Mesh > Material> Create> Isotropic > Elastic
2. Define upper and lower layers as shown
3. Menu > Mesh > Properties > Create > 2D > Axis symmetric
4. Create 2 Axis symmetric properties for the 2 layers

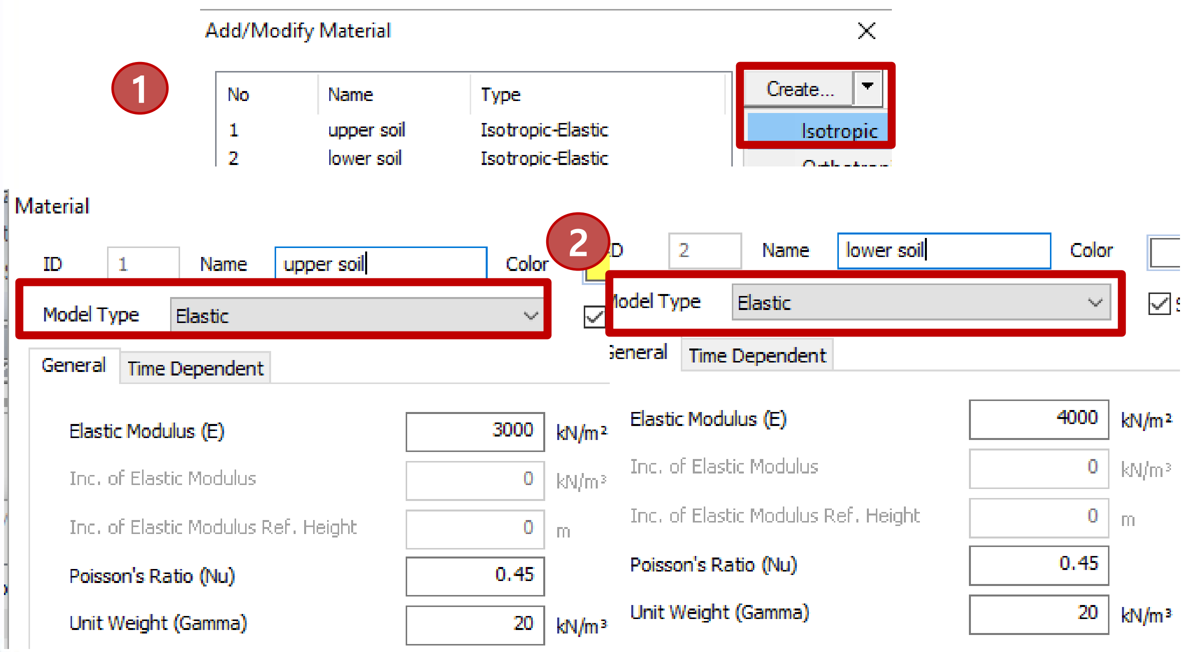


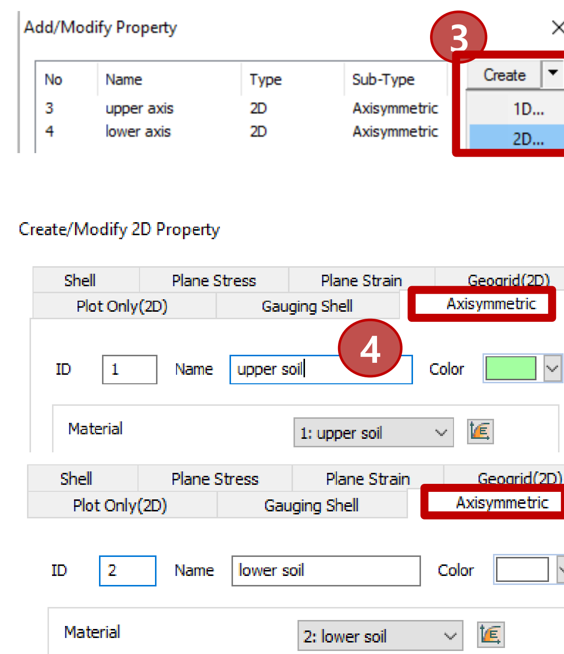

**Upper soil: E = 3000 kPa**

**Poisson's ratio v = 0.45**

**Lower soil: E = 4000 kPa**

**Poisson's ratio v = 0.45**



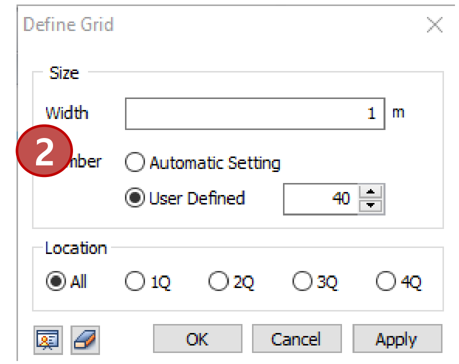



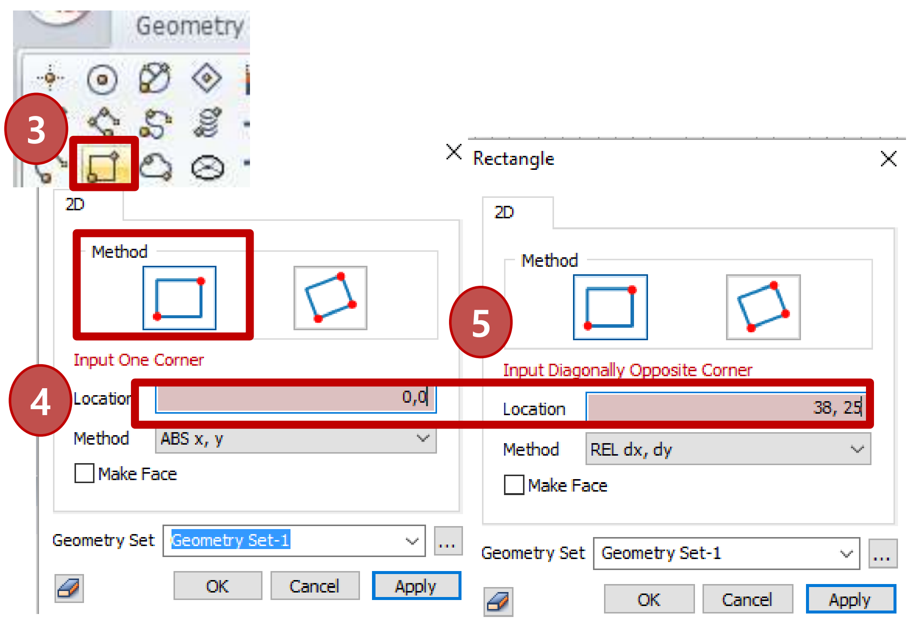
## Create Geometry Outline

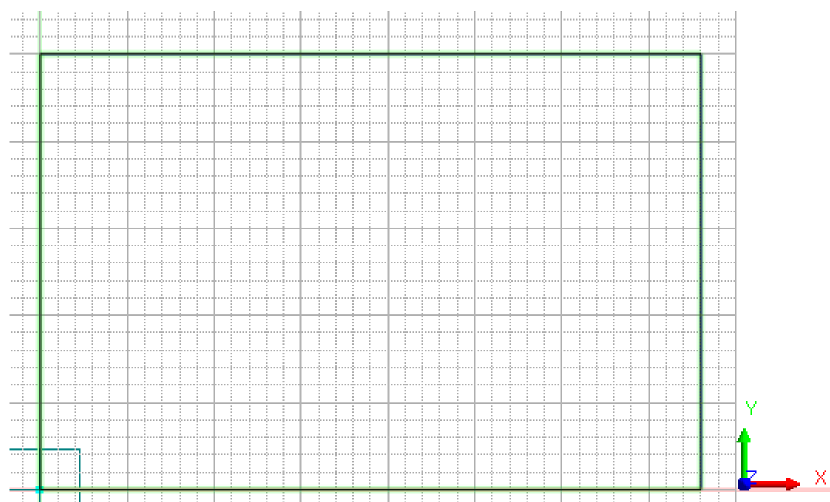
Define the WORK GRID as a drawing plane with lines spaced 1m apart to create rectangles

1. Menu > SHOW GRID
2. Menu > Define Grid > Width 1m, User Define 40 m
3. Menu>Geometry > Rectangle
4. Select the 2 corner method, set 1$^{st}$ Location as 0,0 press Apply
5. Set 2$^{nd}$ location as 38, 25





# Geometry

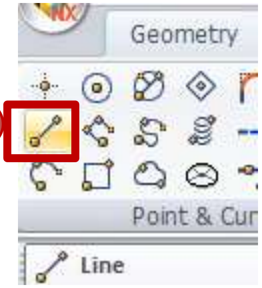
GTS NX

## Create Geometry Outline

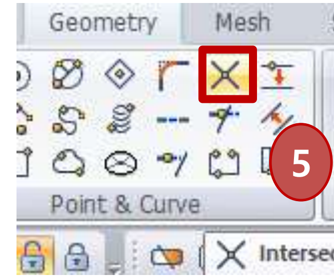
**Draw additional lines to divide model into 4 segments**

1. **Menu>Geometry > Line**
2. **Draw a horizontal line from (0,20) to (38,20)**
3. **Draw a Vertical Line from (5,25) to (5,20)**
4. **Draw a Vertical Line from (10,25) to (10,0)**
5. **Geometry > Intersect > Select All > Apply**

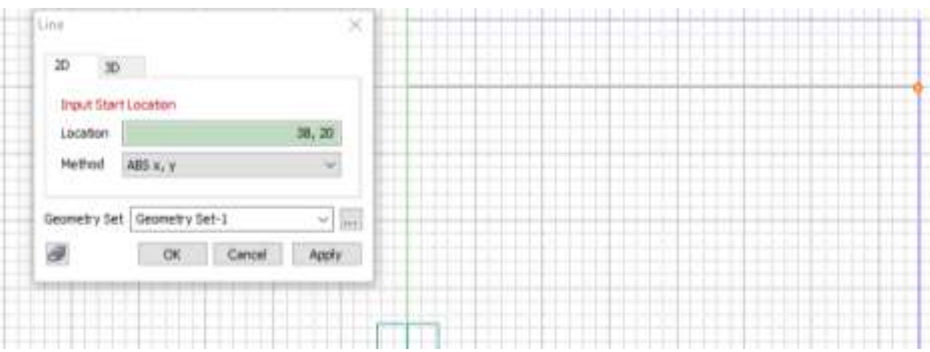
1



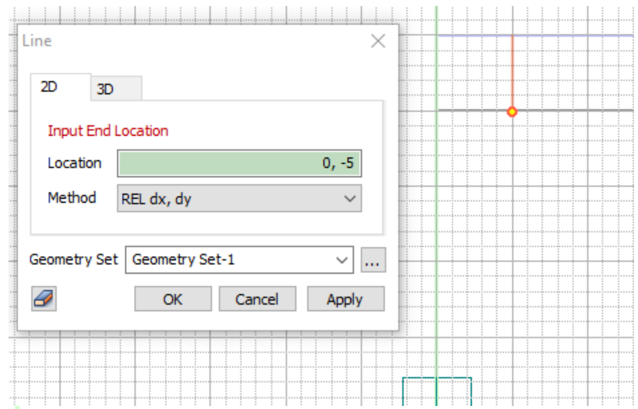

5



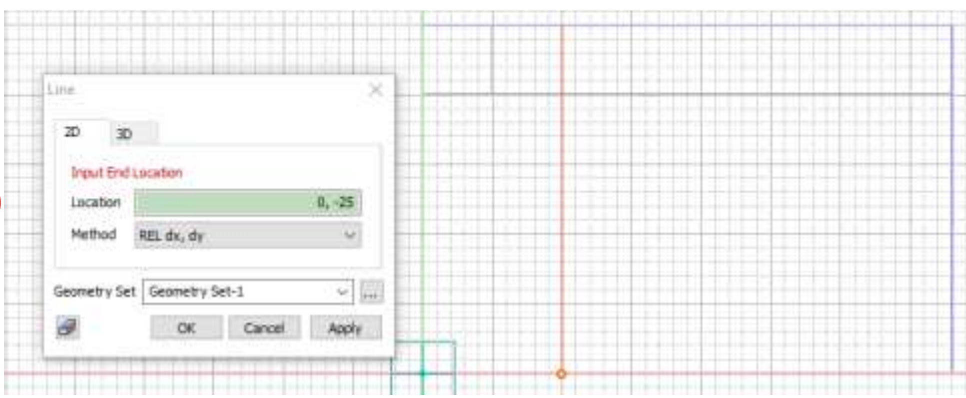

2



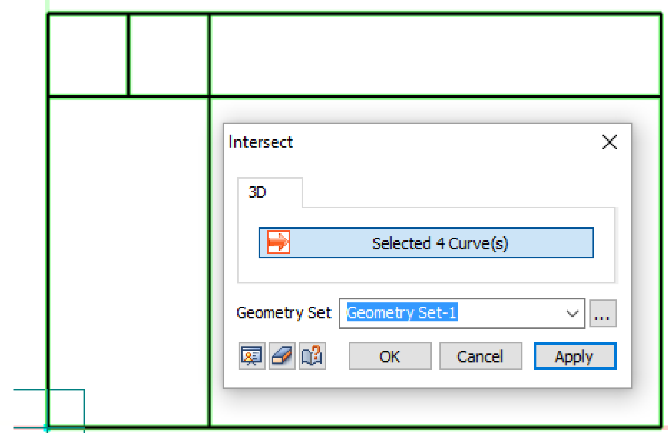

3




4




5



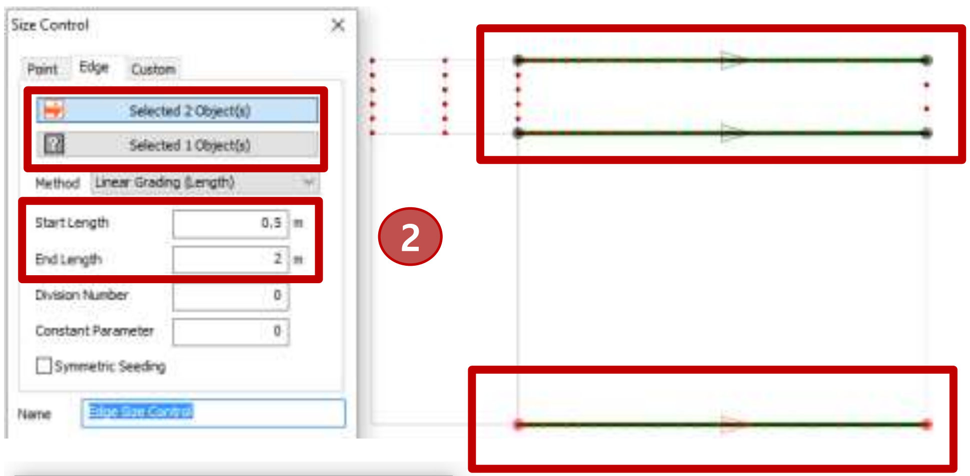
# Mesh Size Control

GTS NX

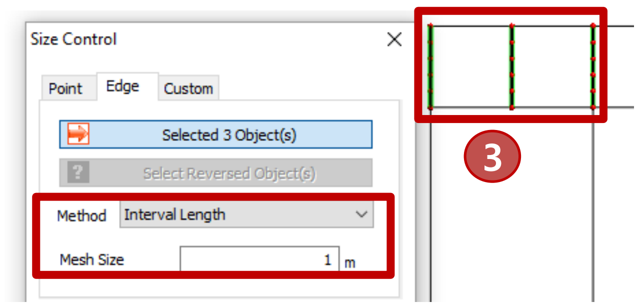
## Define Mesh Conditions

1. Menu> Mesh > Size Control
2. Define Linear Grading 0.5m -> 2m from left to right
3. Define Interval Length of 1m for the upper left corners
4. Define Interval Length of 2m for upper right corner
5. Define Integral length 0.5m for left horizontal lines

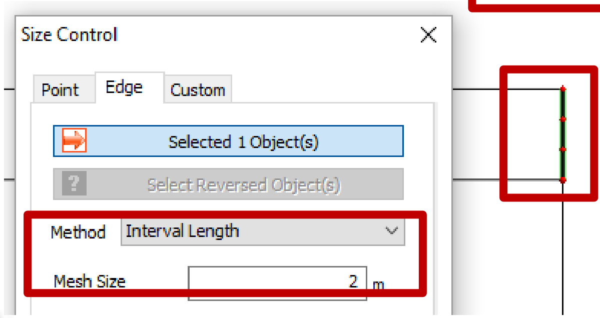

1



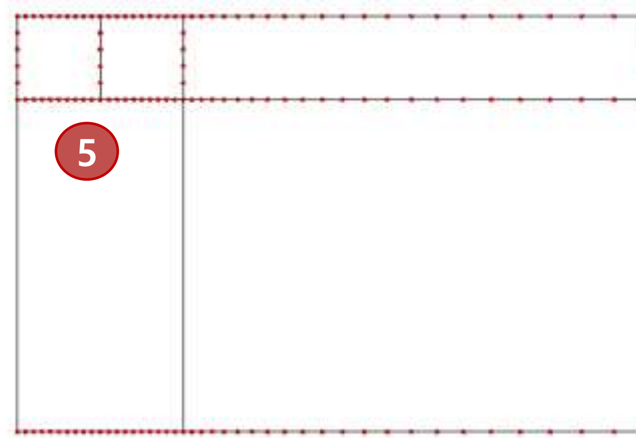

2




3




4



5

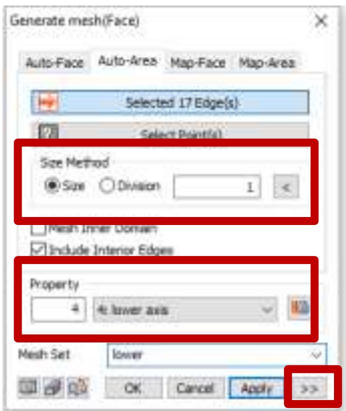
# Mesh

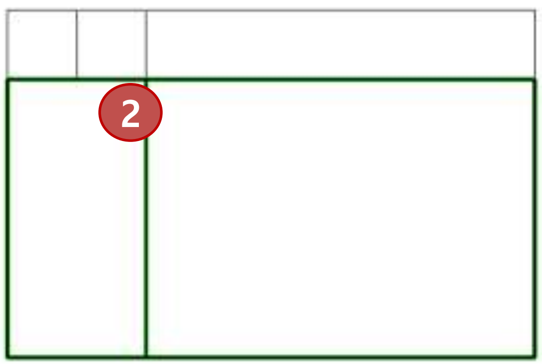
GTS NX

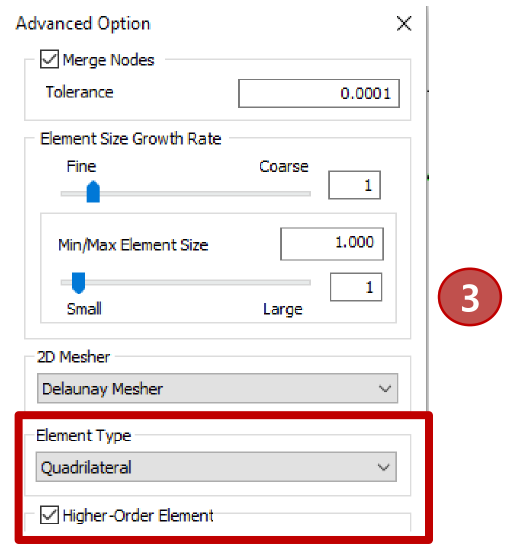
## Mesh

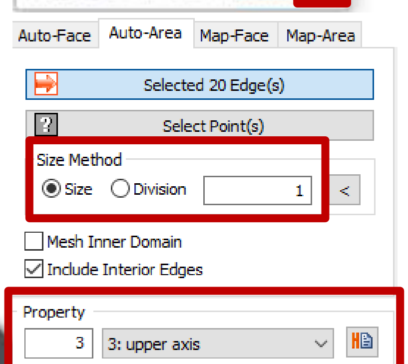
**Mesh Upper and Lower Domains**

1. Menu> Mesh > Generate > 2D > Auto Area
2. Mesh lower 2 areas with 1m and lower axis property
3. Click>> and select quadrilateral and higher order elements. Click OK > Apply
4. Mesh upper 3 areas with 1m and upper axis property

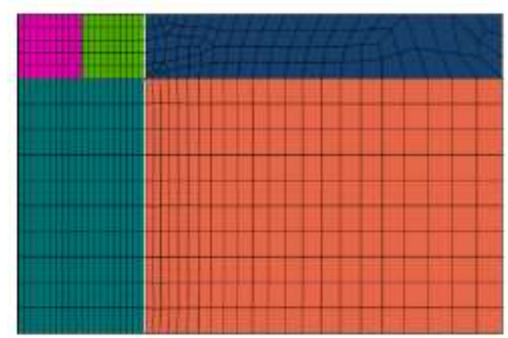







# Boundary

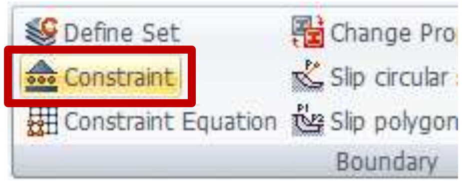


## Boundary

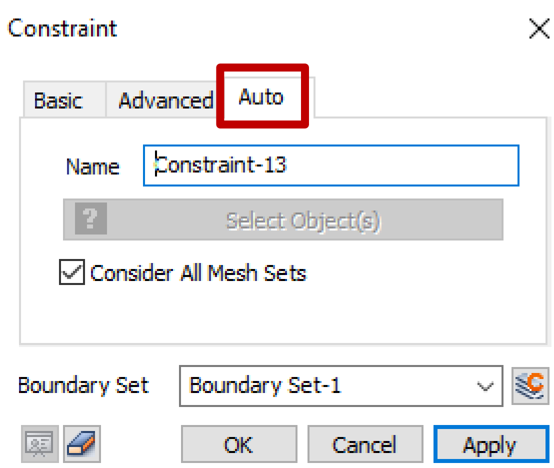
**Apply Boundary Conditions**

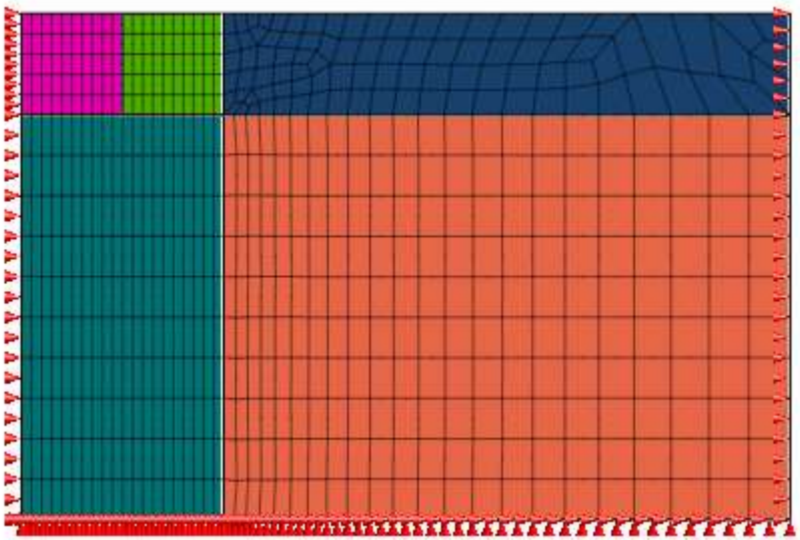
1. **Menu> Static/Slope Analysis>Boundary>Constraint**
2. **Select Auto>Apply**

1




2



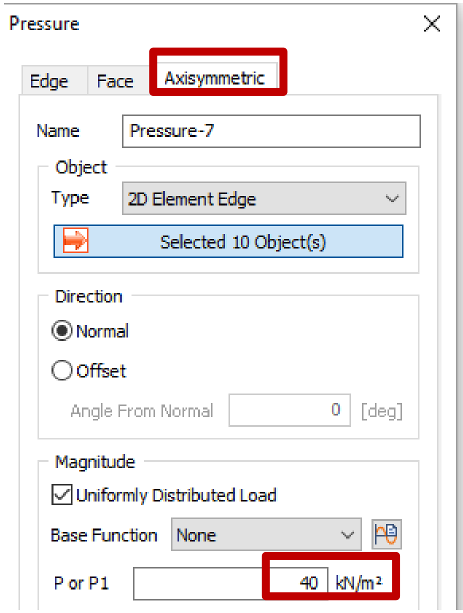
## Load

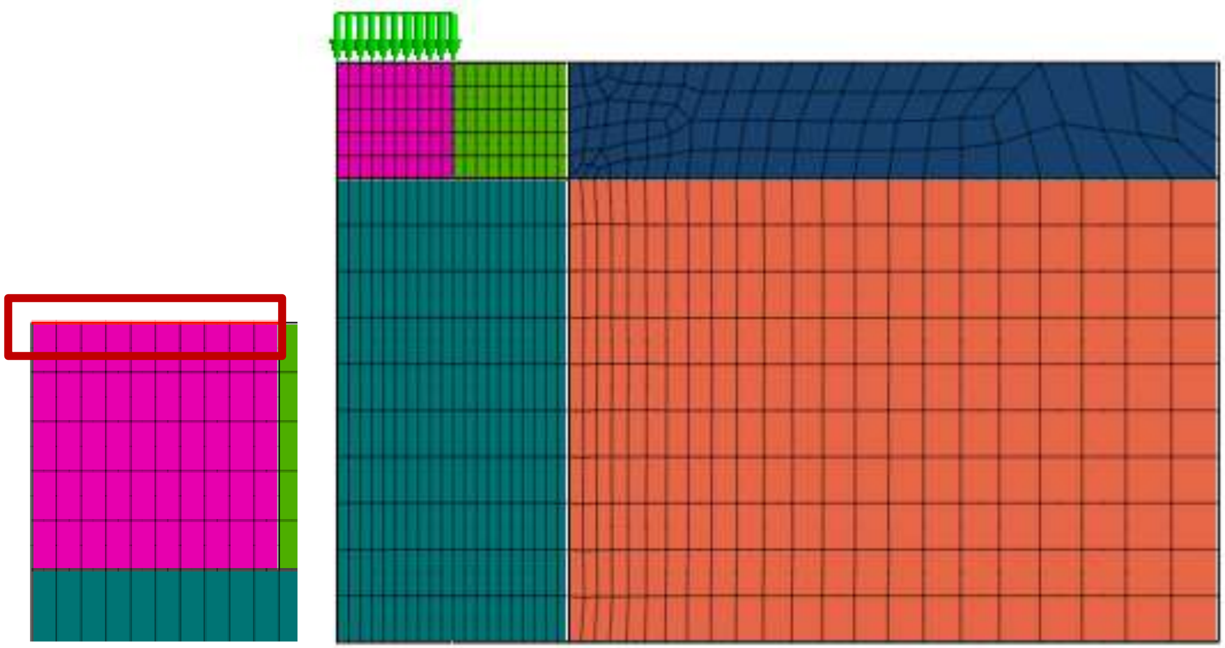
**Apply Pressure Load**

1. Menu> Static/Slope Analysis>Load>Pressure
2. Select Axisymmetric and apply 40kN/m^2 load to upper left nodes as shown.

1


2



# Analysis Case

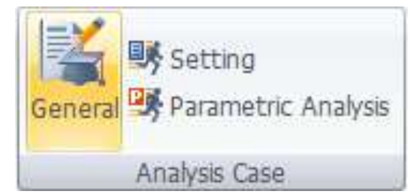
GTS NX

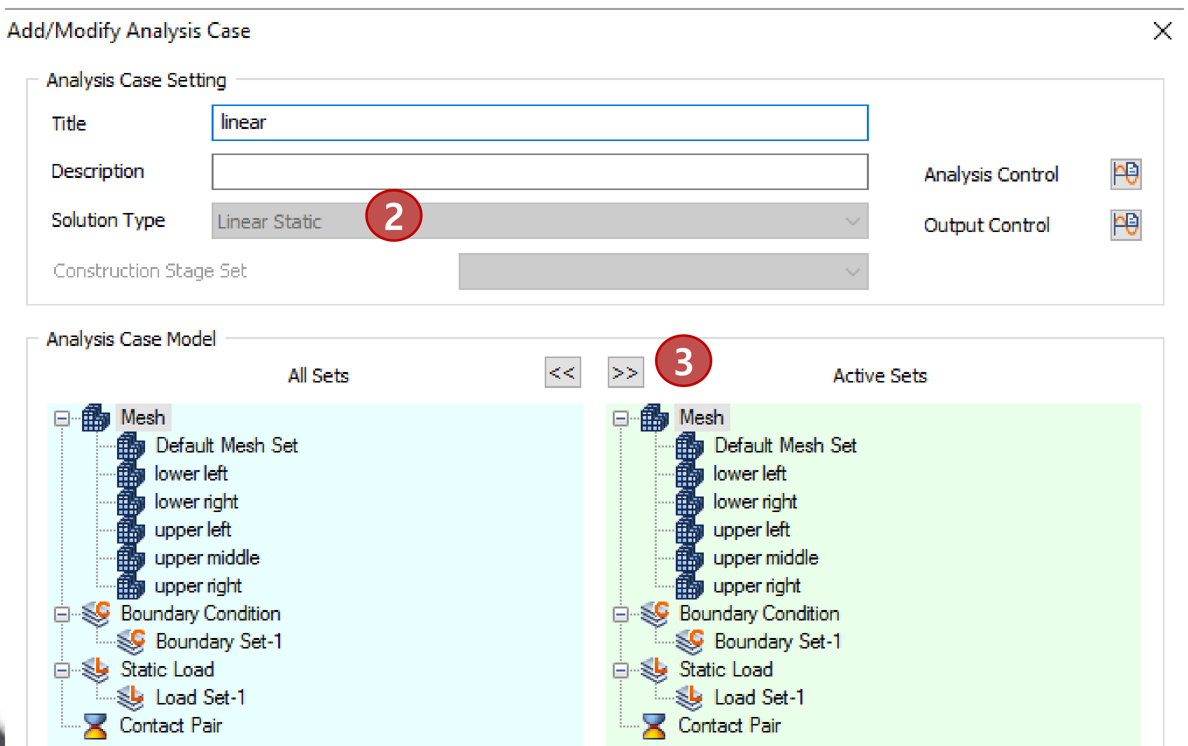
## Analysis Case

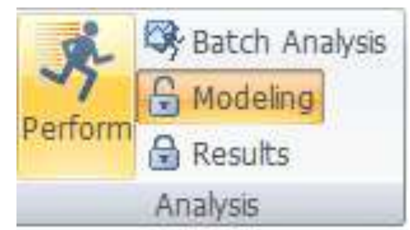
**Create Static Analysis Case**

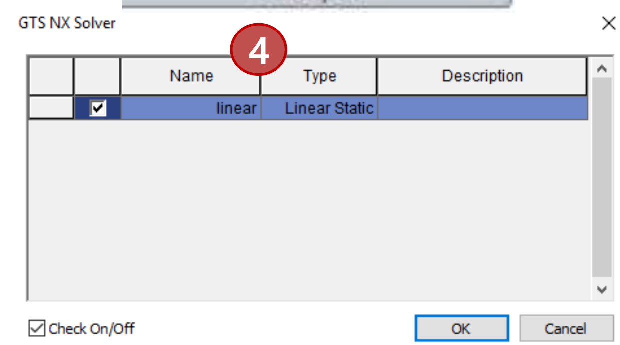
1. **Menu> Analysis>Analysis Case >General**
2. **Select Linear Static**
3. **Activate All**
4. **Run Analysis > Press OK**

1



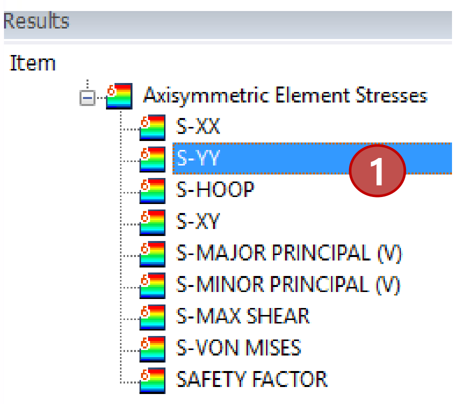
# Results

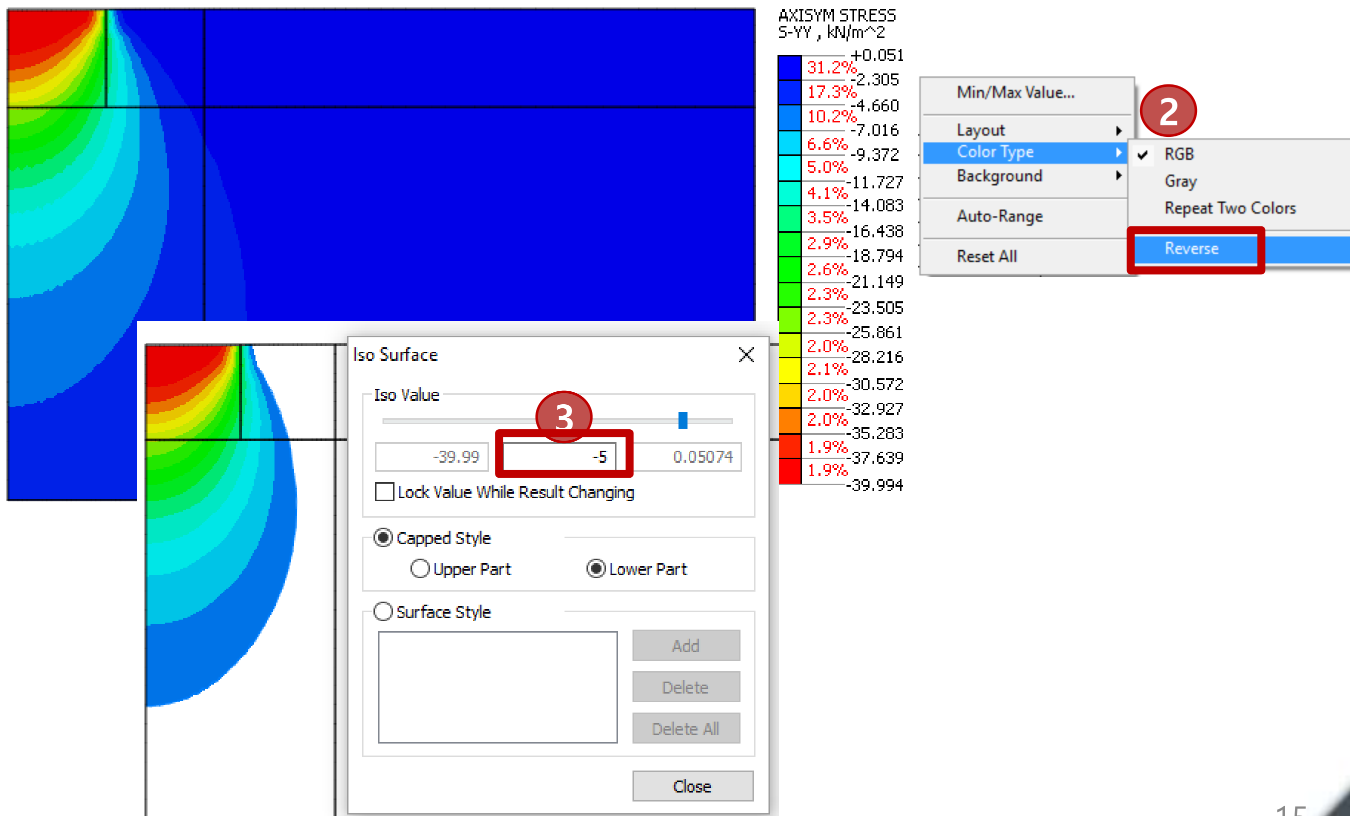
GTS NX

## Post Processor

**Check Vertical Stresses SYY**

1. **Works Tree > Results > Stresses > S-YY**
2. **Right Click Legend > Color Type > Reverse**
3. **Activate Iso surface > Lower Part > Set to -5**

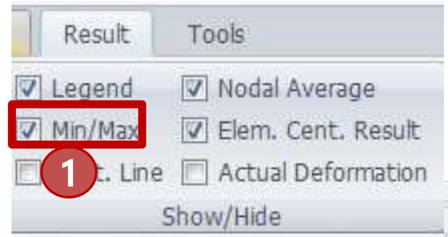
# Results

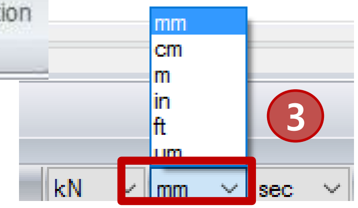
GTS NX

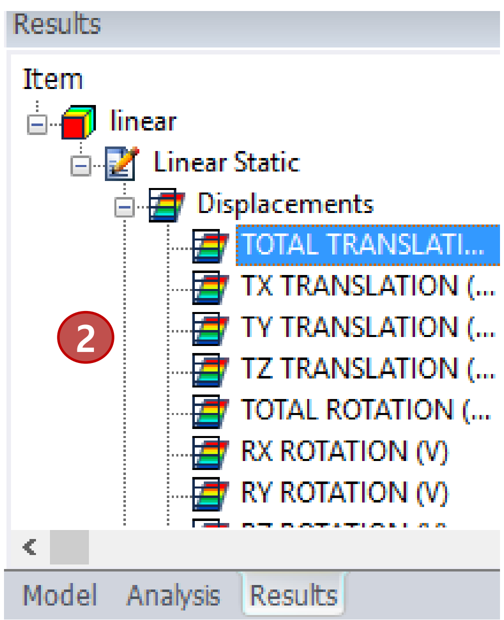
## Post Processor

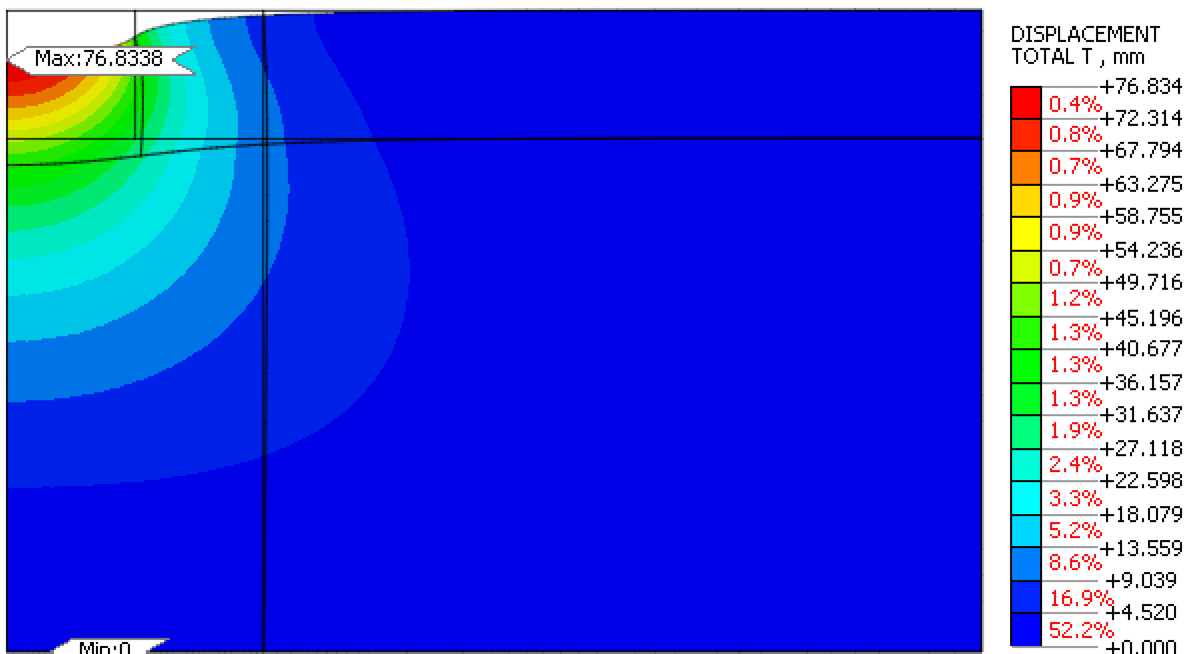
**Check Displacements**

1. **Menu> Results>Show>Min/Max**
2. **Works Tree > Results > Displacements > Total**
3. **Change units to mm**



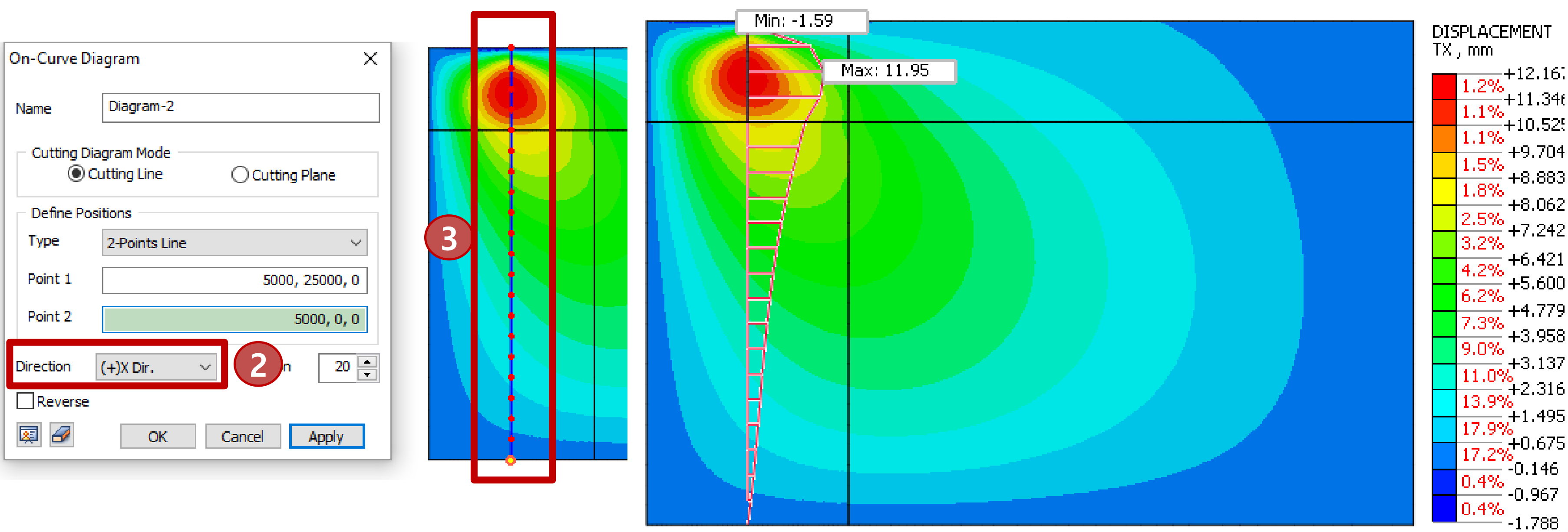
# Results

GTS NX

## Post Processor

**Check Horizontal Displacements**

1. Menu> Results>Advanced>Cutting Diagram
2. Set to X direction
3. Draw vertical diagram by selecting 2 points (top and bottom as shown)

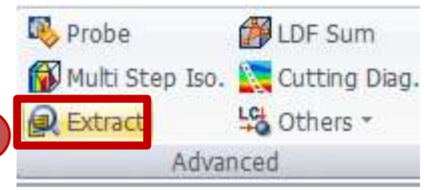
# Results

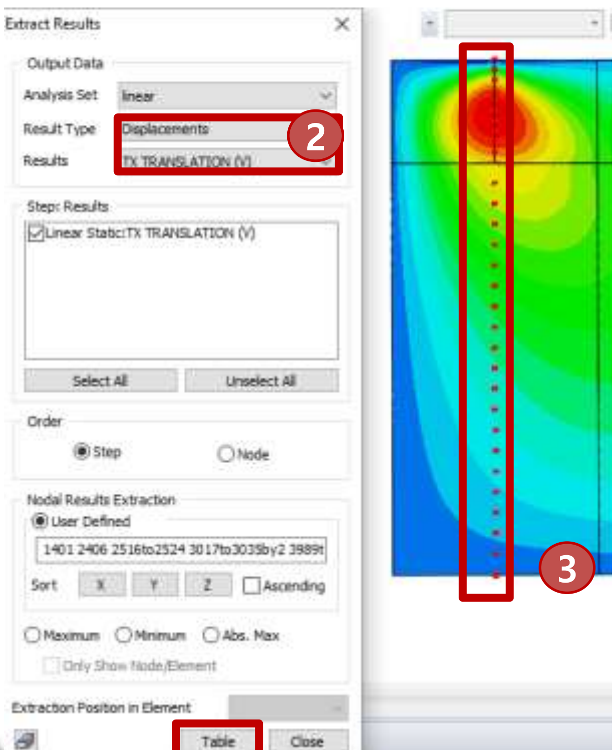
GTS NX

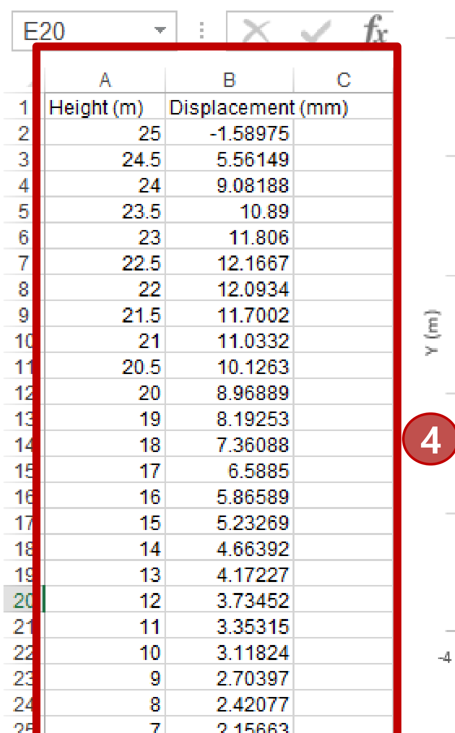
## Post Processor

**Graph Displacements**

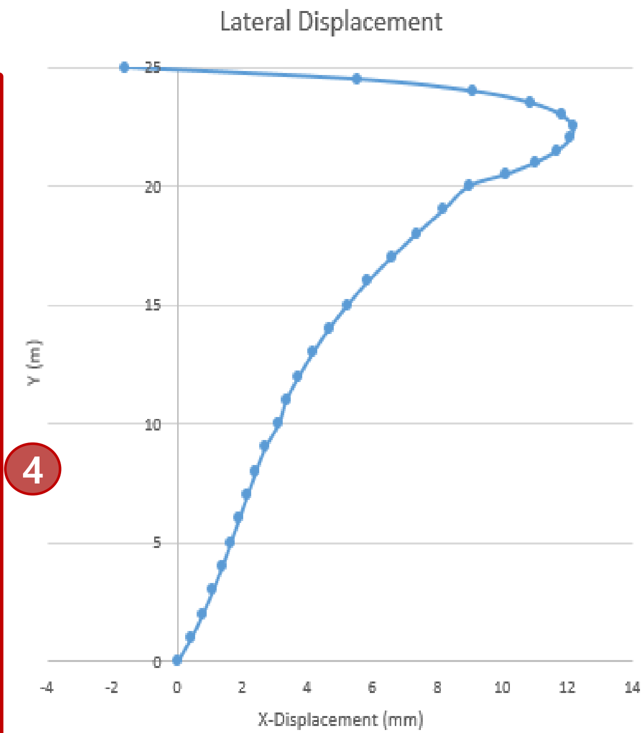
1. Menu> Results>Advanced>Extract
2. Set Type > Displacements / Results > TX Translation
3. Select nodes as shown > Click Table
4. Select depth and displacement columns, right click export to excel to graph.



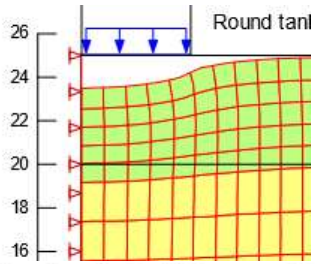

| | A | B | C |
|---|---|---|---|
| 1 | Height (m) | Displacement (mm) | |
| 2 | 25 | -1.58975 | |
| 3 | 24.5 | 5.56149 | |
| 4 | 24 | 9.08188 | |
| 5 | 23.5 | 10.89 | |
| 6 | 23 | 11.806 | |
| 7 | 22.5 | 12.1667 | |
| 8 | 22 | 12.0934 | |
| 9 | 21.5 | 11.7002 | |
| 10 | 21 | 11.0332 | |
| 11 | 20.5 | 10.1263 | |
| 12 | 20 | 8.96889 | |
| 13 | 19 | 8.19253 | |
| 14 | 18 | 7.36088 | |
| 15 | 17 | 6.5885 | |
| 16 | 16 | 5.86589 | |
| 17 | 15 | 5.23269 | |
| 18 | 14 | 4.66392 | |
| 19 | 13 | 4.17227 | |
| 20 | 12 | 3.73452 | |
| 21 | 11 | 3.35315 | |
| 22 | 10 | 3.11824 | |
| 23 | 9 | 2.70397 | |
| 24 | 8 | 2.42077 | |
| 25 | 7 | 2.15663 | |

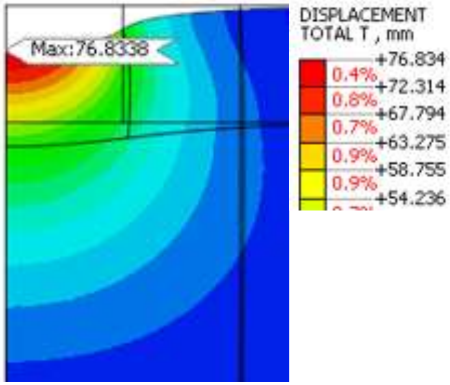


# Compare Results to SIGMAW Tutorial

GTS NX

**Displacement and Stresses match exactly with results from SIGMA W tank tutorial**




**10 Vertical settlement**

Figure 6 shows the vertical settlements as a deformed mesh at a 20X exaggeration. The maximum settlement is about 0.076 m (76 mm) under the center of the tank.

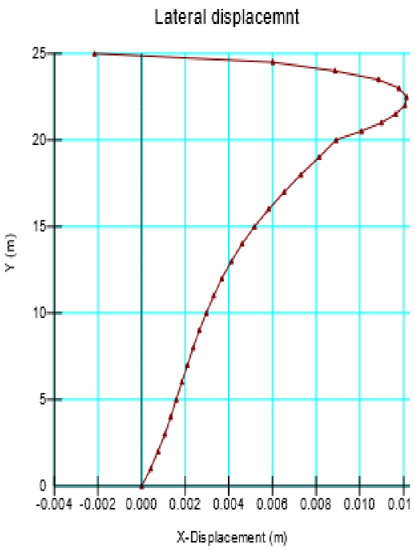



**Figure 7 Lateral displacement profile**

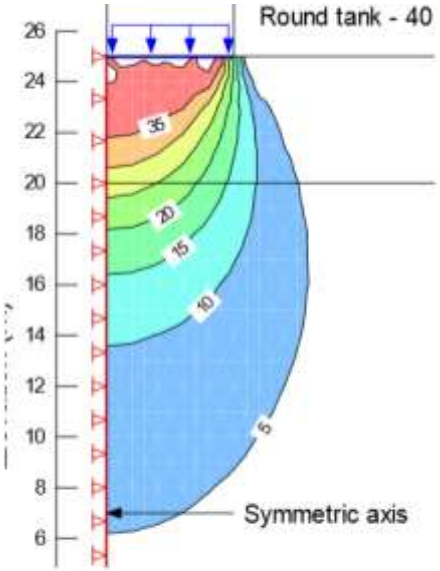
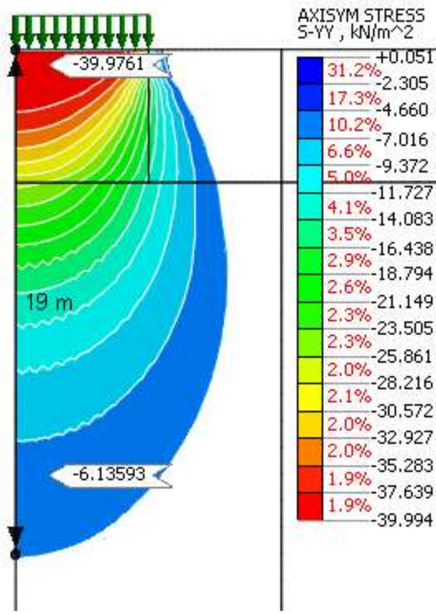
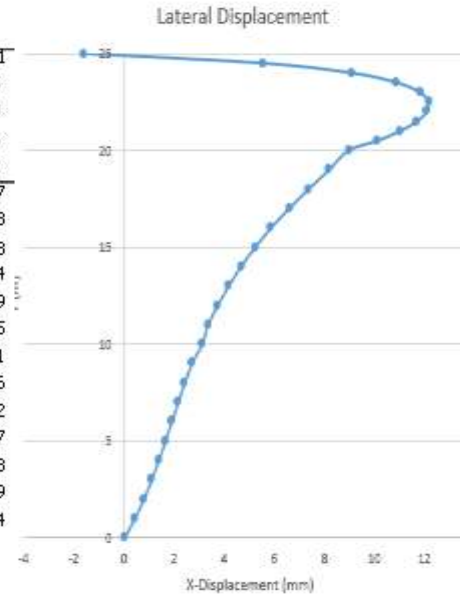



**Figure 5 Vertical stress change contours**

*SIGMA/W Example File: SIGMA Tutorial.docx (pdf) (gsz)*



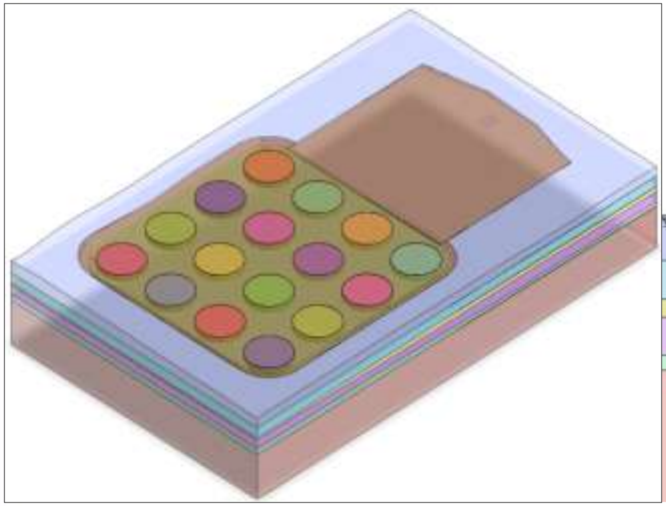




*MIDAS GTS NX*

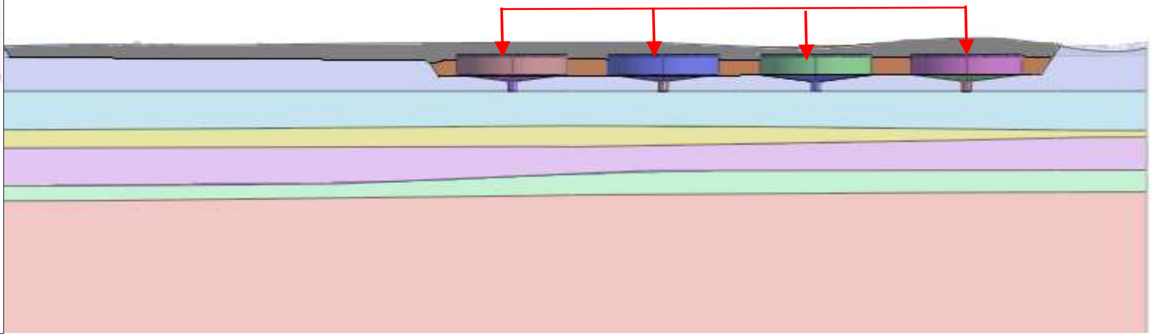
# 3D Case Study: Tanks in Clay (Colombia)

GT

**Long term consolidation analysis of water treatment tanks in soft clay**



3D isometric view of tanks



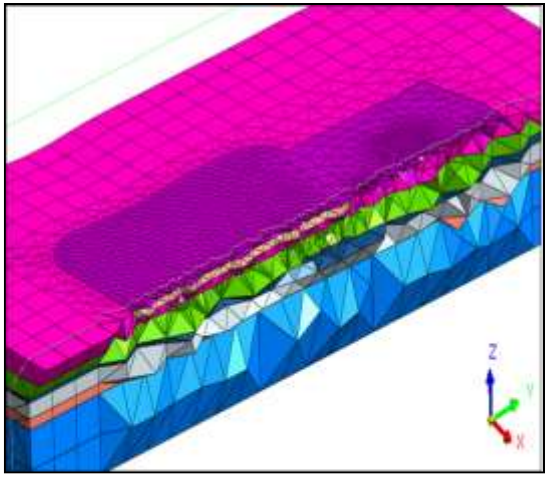
2D side view of tanks in clay

| PERFIL MODELACIÓN | | | | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| Material | Espesor (m) | Wn (%) | LL (%) | IP(%) | Peso unitario total γt (kN/m3) | Es (MPa) Valor conservador | e | Presión de preconsolidación (kPa) | OCR | Cc | Cr | K (m/día) | c' (kPa) | ϕ`(º) |
| 1 | 12.5 | 67 | 101 | 69 | 16.7 | 21 | 1.6 | 322 | 3.6 | 1.08 | 0.17 | 5.6E-03 | 21 | 23 |
| 2 | 11.7 | 56 | 80 | 51 | 17.4 | 30 | 1.3 | 427 | 2.1 | 0.63 | 0.10 | 2.7E-02 | 37 | 22 |
| 3 | 5.5 | 89 | 162 | 81 | 15.0 | 26 | 2.3 | 453 | 1.7 | 1.55 | 0.25 | 1.7E-03 | 46 | 24 |
| 4 | 11.5 | 41 | 66 | 39 | 17.5 | 39 | 1.2 | 571 | 1.8 | 0.45 | 0.07 | 1.0E-03 | 72 | 21 |
| 5 | 4.5 | 33 | 59 | 30 | 17.2 | 77 | 1.0 | 644 | 1.7 | 0.50 | 0.10 | 7.8E-06 | - | - |
| 6 | 46 ? | - | - | - | 19.7 | 125 | - | - | 1.6 | - | - | 7.8E-06 | - | - |

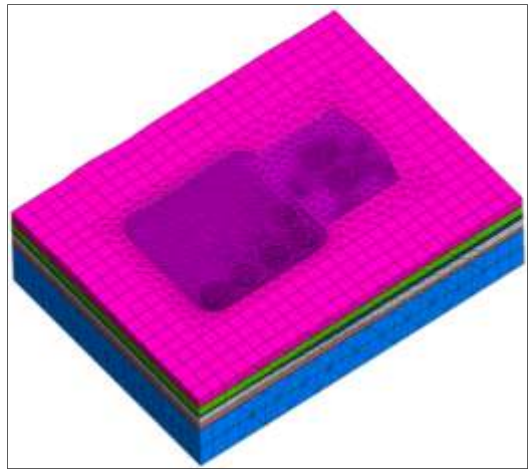
Material table of clay layers

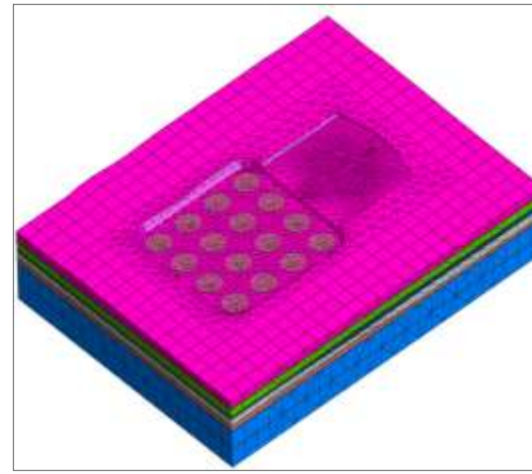
# 3D Case Study: Tanks in Clay (Colombia)

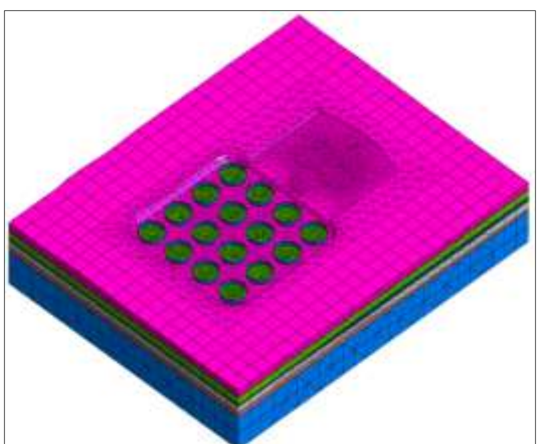
GT

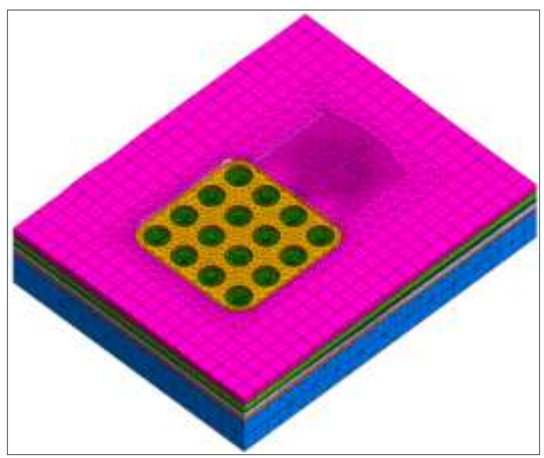


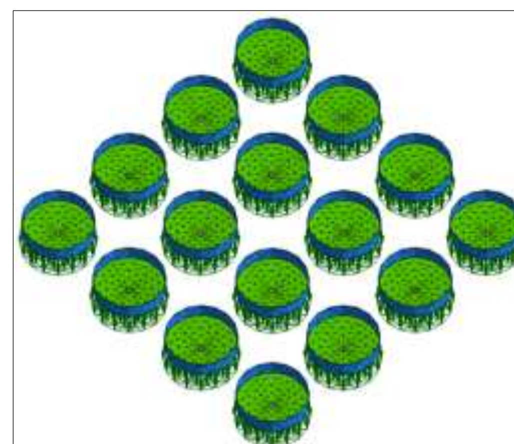
Plane cut of layers


Initial CS stage

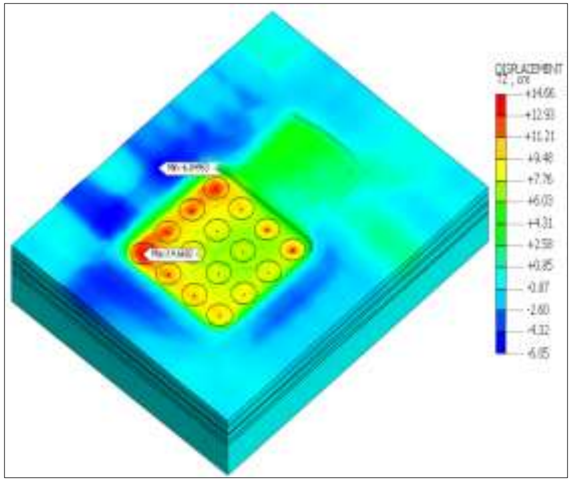

Stage1: Excavation


Stage2: Installation

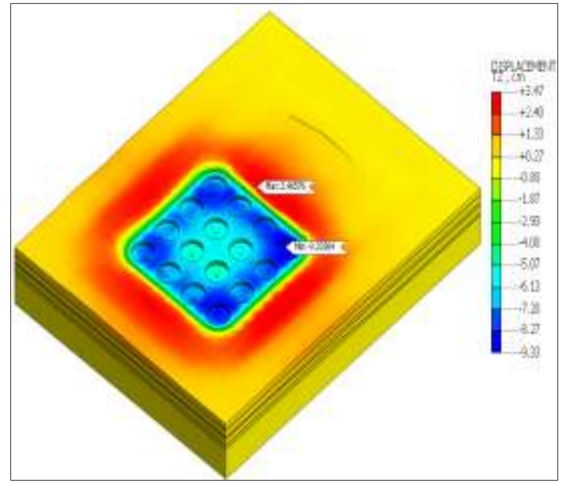

Stage3: Reinforcement


Stage4: Loads

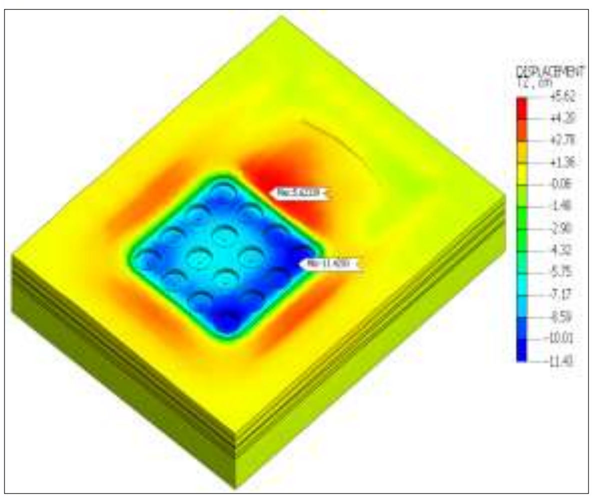
# 3D Case Study: Tanks in Clay (Colombia)



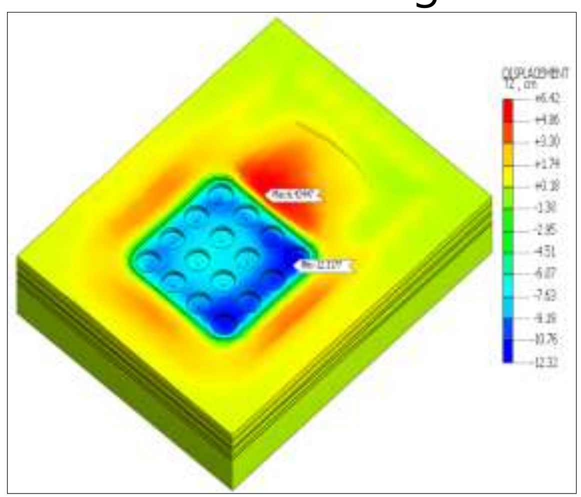
Stage1: Excavation



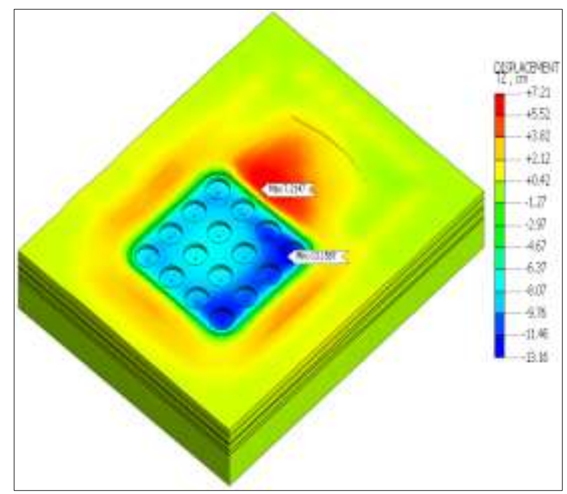
Stage4: Loads



10 year consolidation



20 year consolidation



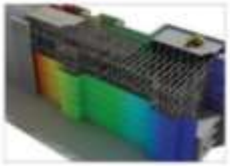
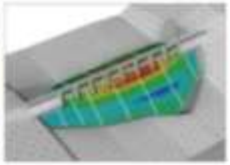
50 year consolidation

# Seepage Thru Earth Embankment

## (SEEPW  Comparison)

GTS NX

Geo-Technical analysis System New eXperience

Angel Francisco Martinez

Application Engineer
MIDAS IT
Estados Unidos

Integrated Solver Optimized for the next generation 64-bit platform

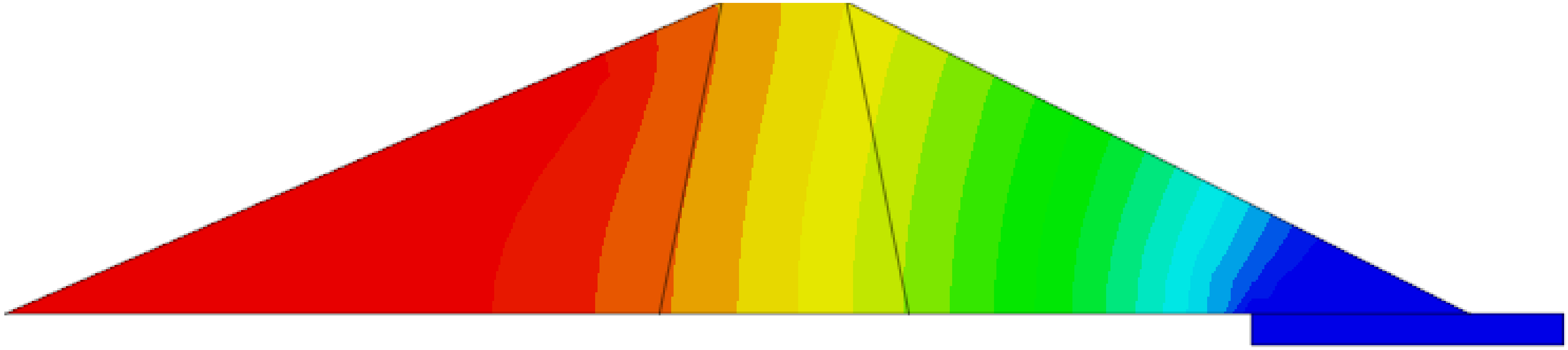
**Finite Element Solutions for Geotechnical Engineering**

MIDAS

# Introduction

GTS NX

**The objective of the tutorial is look at the 2D flow through an embankment dam.  This seepage analysis is a common example and consequently most users will have a good idea as to what the solution should look like. It shows how easy it is to find the downstream seepage outflow when the dam has a homogenous material.  The tutorial will also look at cases were a toe under-drain is included and when the central core's hydraulic conductivities are lowered.**



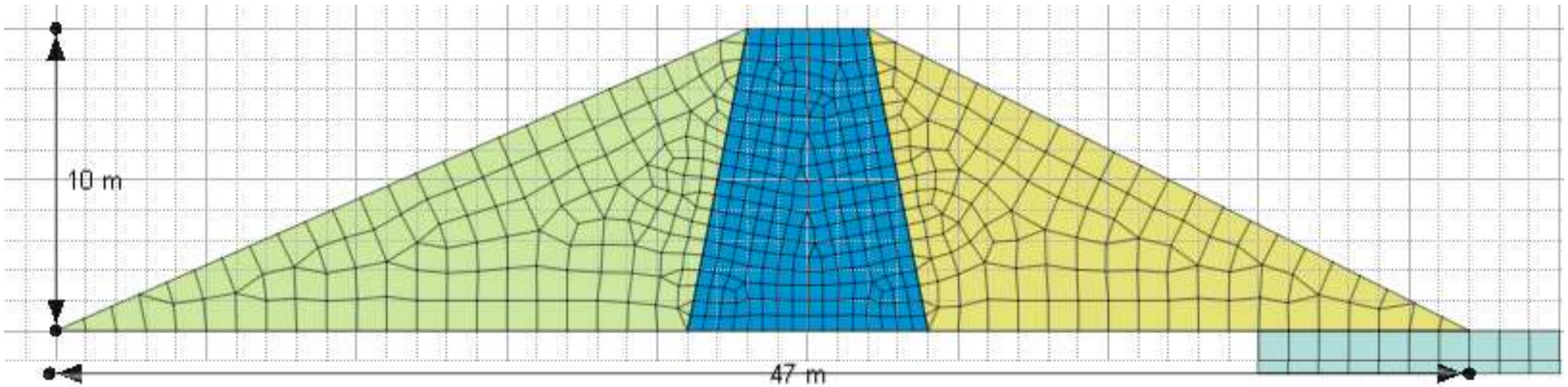
# Initial model dimensions

GTS NX

## Model Conditions

- 2D Steady State Seepage
- 10m tall X 47m wide Dam
- 3 layers
    1. Core
    2. Shell
    3. Toe Drain
- 8 m water level on right side of dam

**Figures show the corresponding problem setup. The dam consists of a center core (4m wide at top, 8m wide at base), 2 shells and a toe drain on the right bottom corner. Water level on left side is 8m.**

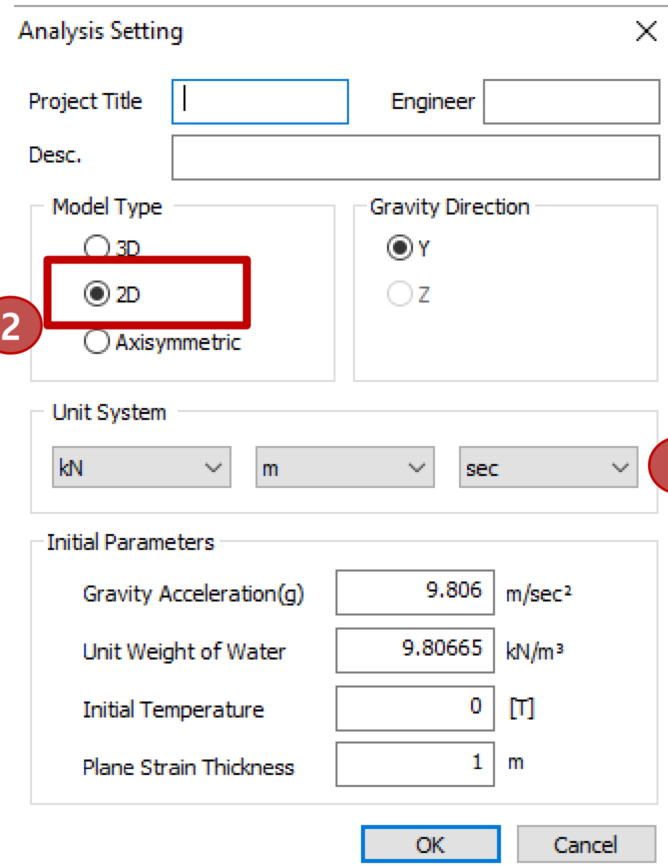


# Start Axisymetric File

GTS NX

## Open New

1. File > New
2. 2D
3. Units kN, m, sec

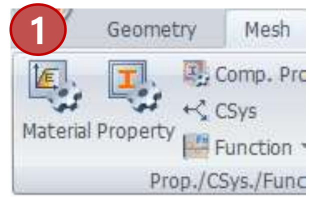
# Materials

GTS NX

## Define Materials

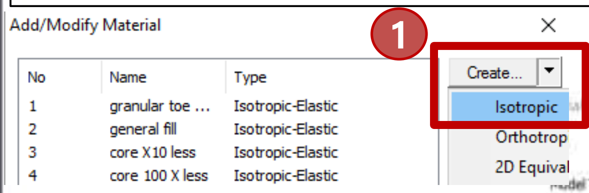
1. Menu > Mesh > Material> Create> Isotropic
2. Define General Fill material as shown for the General tab.
3. In the Porous tab, active Unsaturated Property>ADD>Individual>User Defined and copy paste corresponding excel sheets. Press Ok
4. Repeat for core 10X less and core 100X less materials

1

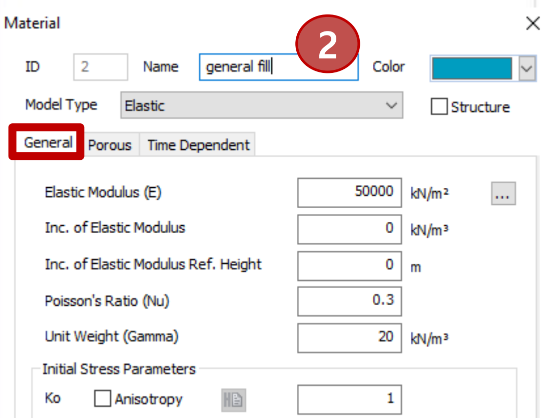



- core X10
- core X100
- silt k function
- water content

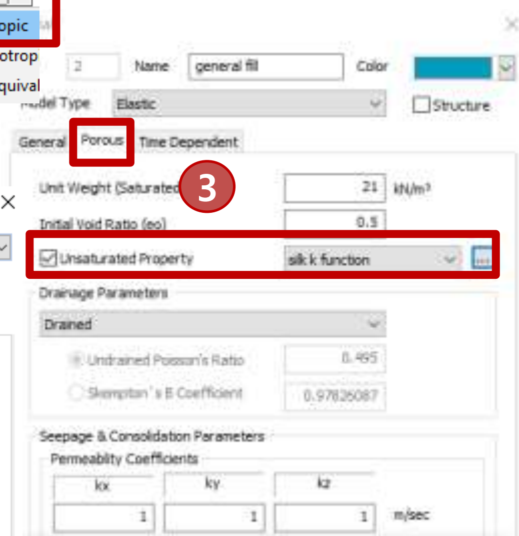
1




2




3





3

| Matric Suction | Kx (m/sec) |
| --- | --- |
| 0.1 | 1.00E-05 |
| 0.14384499 | 9.96E-06 |
| 0.20691381 | 9.91E-06 |
| 0.29763514 | 9.82E-06 |
| 0.42813324 | 9.69E-06 |
| 0.61584821 | 9.50E-06 |
| 0.88586679 | 9.21E-06 |
| 1.274275 | 8.79E-06 |
| 1.8329807 | 8.18E-06 |
| 2.6366509 | 7.30E-06 |
| 3.7926902 | 6.10E-06 |
| 5.4555948 | 4.58E-06 |
| 7.8475997 | 2.92E-06 |
| 11.288379 | 1.47E-06 |
| 16.237767 | 5.55E-07 |
| 23.357215 | 1.58E-07 |
| 33.598183 | 3.63E-08 |
| 48.329302 | 7.28E-09 |
| 69.51928 | 1.36E-09 |
| 100 | 2.44E-10 |

| Matric Suction (kPa) | VWC ($m^3/m^3$) |
| --- | --- |
| 0.1 | 0.449946 |
| 0.16237767 | 0.449883 |
| 0.26366509 | 0.449746 |
| 0.42813324 | 0.449447 |
| 0.6951928 | 0.448796 |
| 1.1288379 | 0.44738 |
| 1.8329807 | 0.444321 |
| 2.9763514 | 0.437809 |
| 4.8329302 | 0.424379 |
| 7.8475997 | 0.398415 |
| 12.74275 | 0.353926 |
| 20.691381 | 0.29121 |
| 33.598183 | 0.222567 |
| 54.555948 | 0.163773 |
| 88.586679 | 0.121111 |
| 143.84499 | 0.092186 |
| 233.57215 | 0.072563 |
| 379.26902 | 0.058798 |
| 615.84821 | 0.048704 |
| 1,000 | 0.040948 |

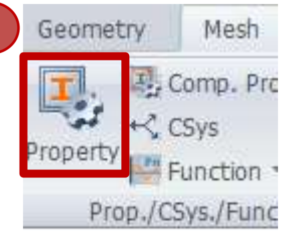
# Properties

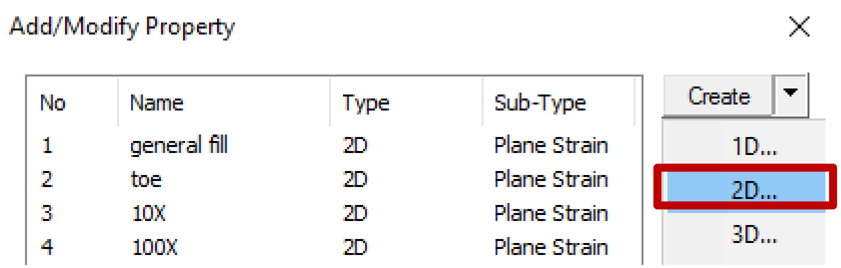
GTS NX

## Define Materials and Properties

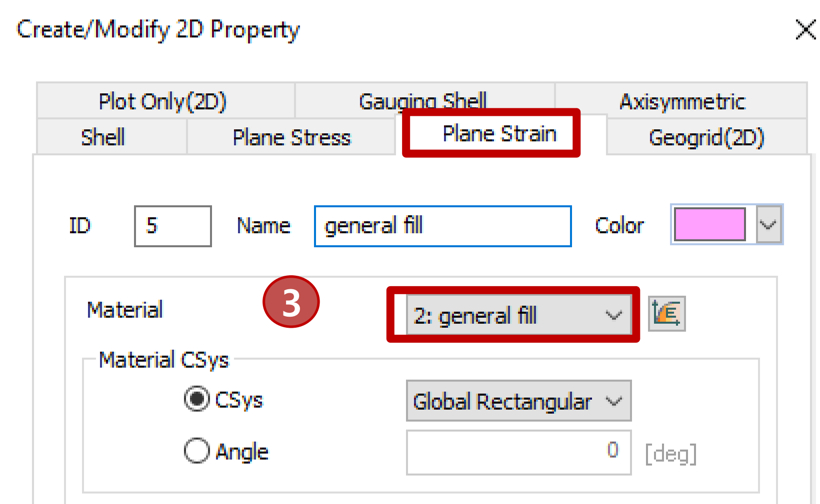
1. Menu > Mesh > Properties > Create > 2D > Plain Strain
2. Create 4 2D Plain Strain properties for the layers
3. Match the name with the material property.

1




2

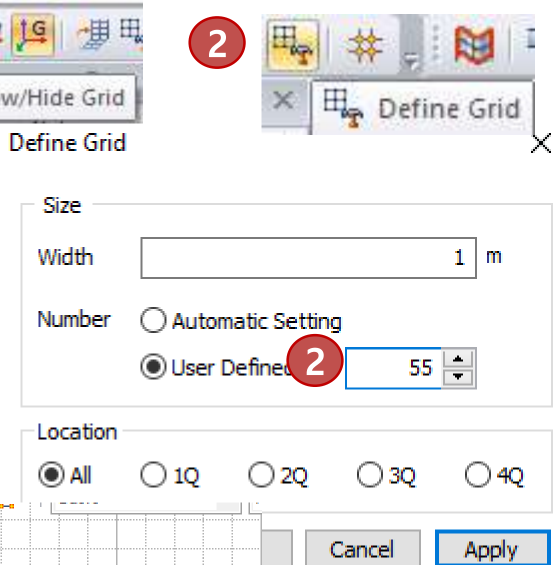


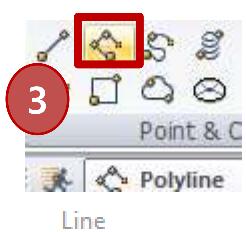

3

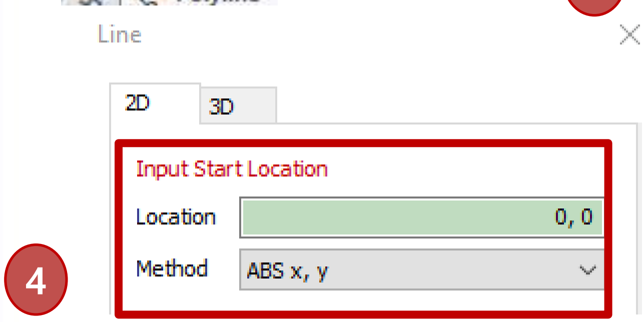
## Create Geometry Outline

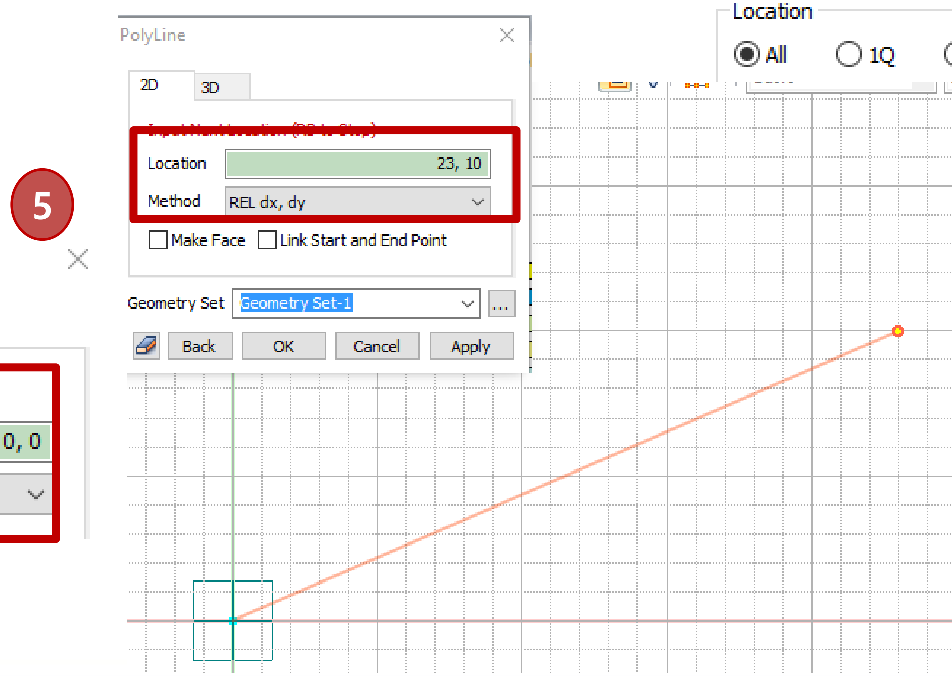
**Define the WORK GRID as a drawing plane with lines spaced 1m apart to create rectangles**

1. **Menu > SHOW GRID**
2. **Menu > Define Grid > Width 1m, User Define 55 m**
3. **Menu>Geometry > PolyLine**
4. **Select the ABS x,y method, set 1st Location as 0,0 press Apply**
5. **Set End location as 23, 10**

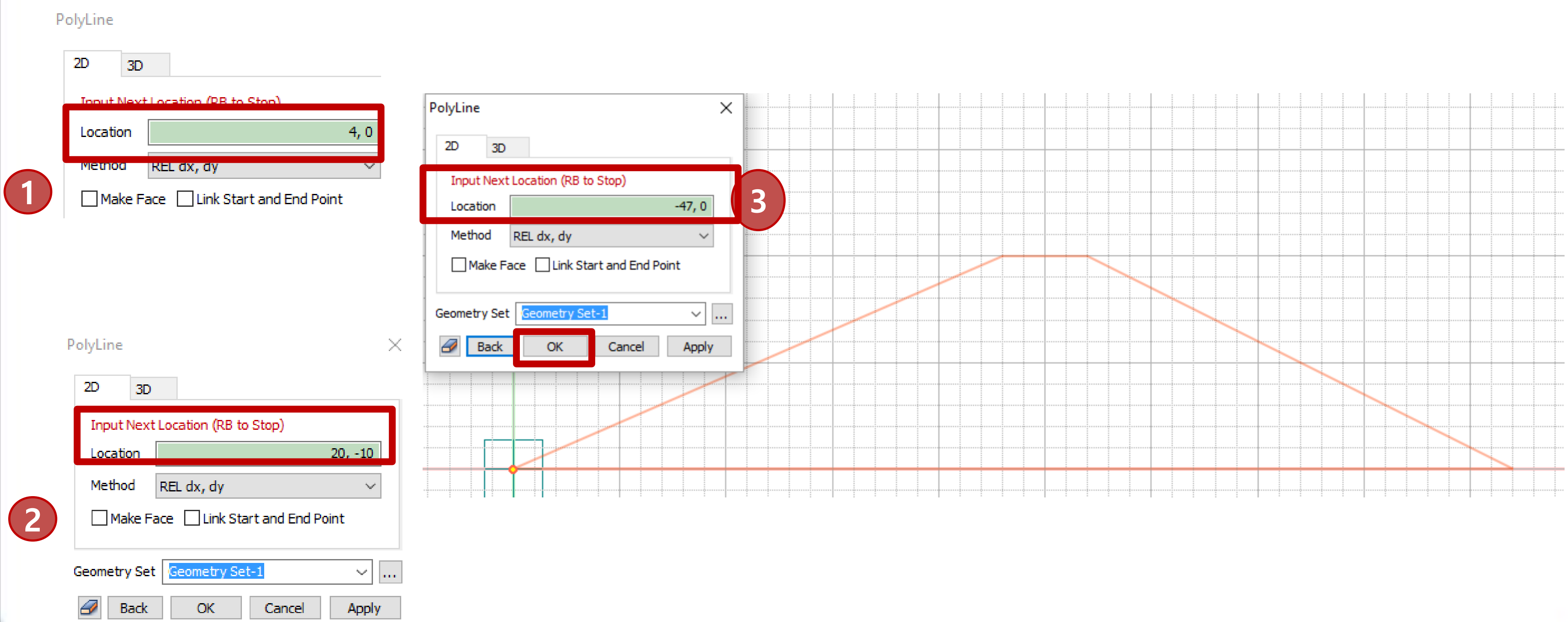



# Geometry

## Create Geometry Outline

1. Input next location (4,0)
2. Input next location (20, -10)
3. Input next location (-47,0)
4. Press OK

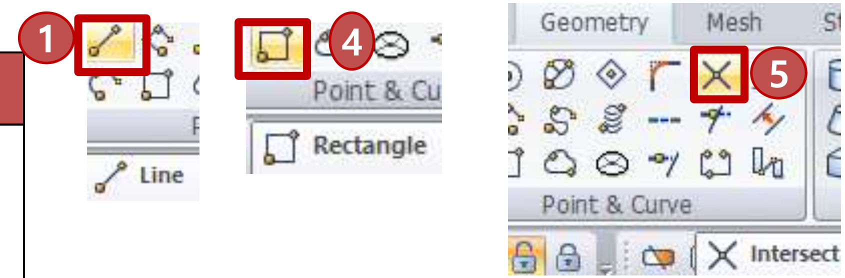


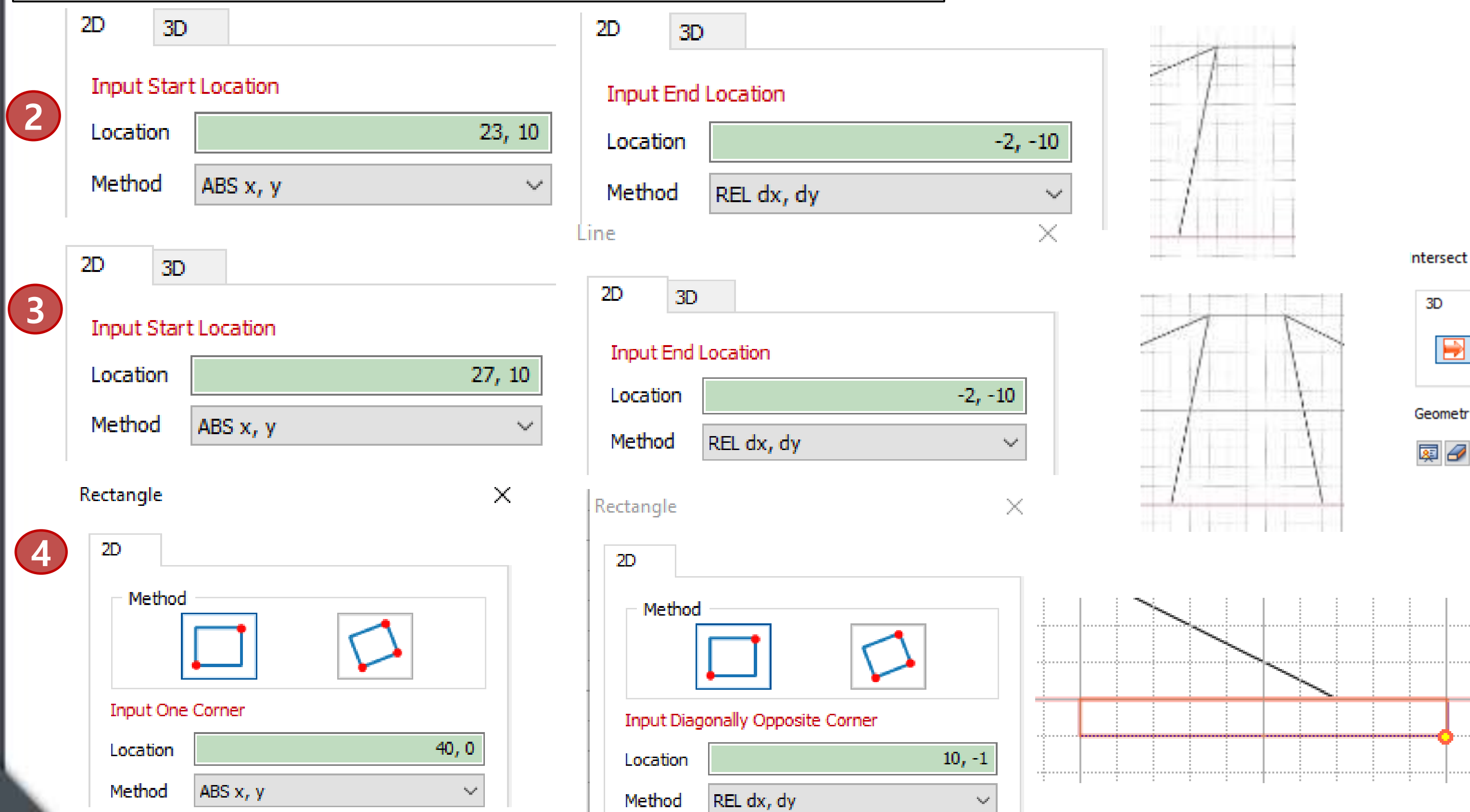
# Geometry

GTS NX

## Create Geometry Outline

1. Draw a Vertical Line Start (23,10) End (-2,10)
2. Draw a Vertical Line Start (27,10) End (-2,10)
3. Draw a Rectangle Start Corner (40,0) End Corner(10,-1)
4. Geometry > Intersect > Select All > Apply



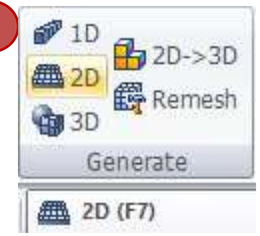
# Mesh



## Mesh

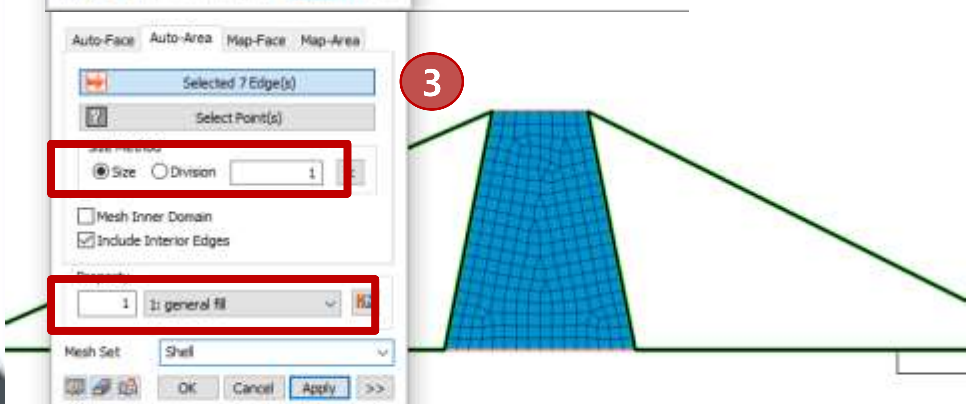
1. Menu> Mesh > Generate > 2D > Auto Area
2. Mesh the Core with 0.5m size and General Fill Property
3. Repeat for Core for 10X less and 100X less property
4. Mesh the 2 Shells with 1m size and General Fill
5. Mesh Toe Drain with 1m size and Toe Property

1

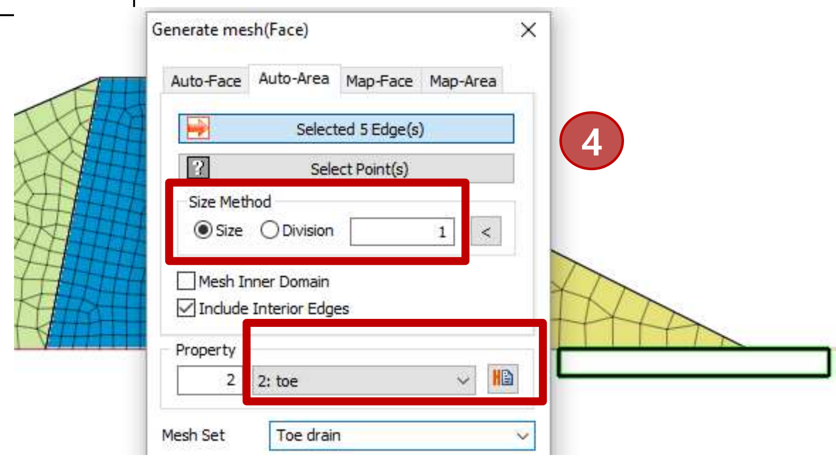




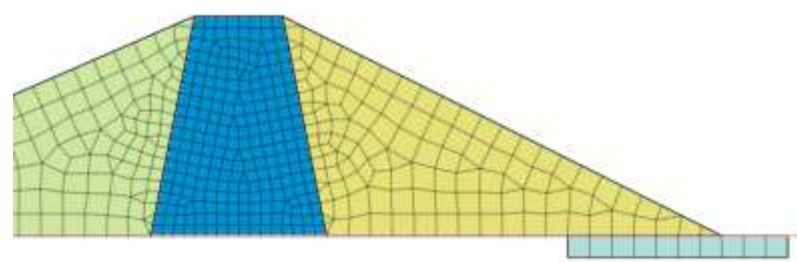
2




3




4



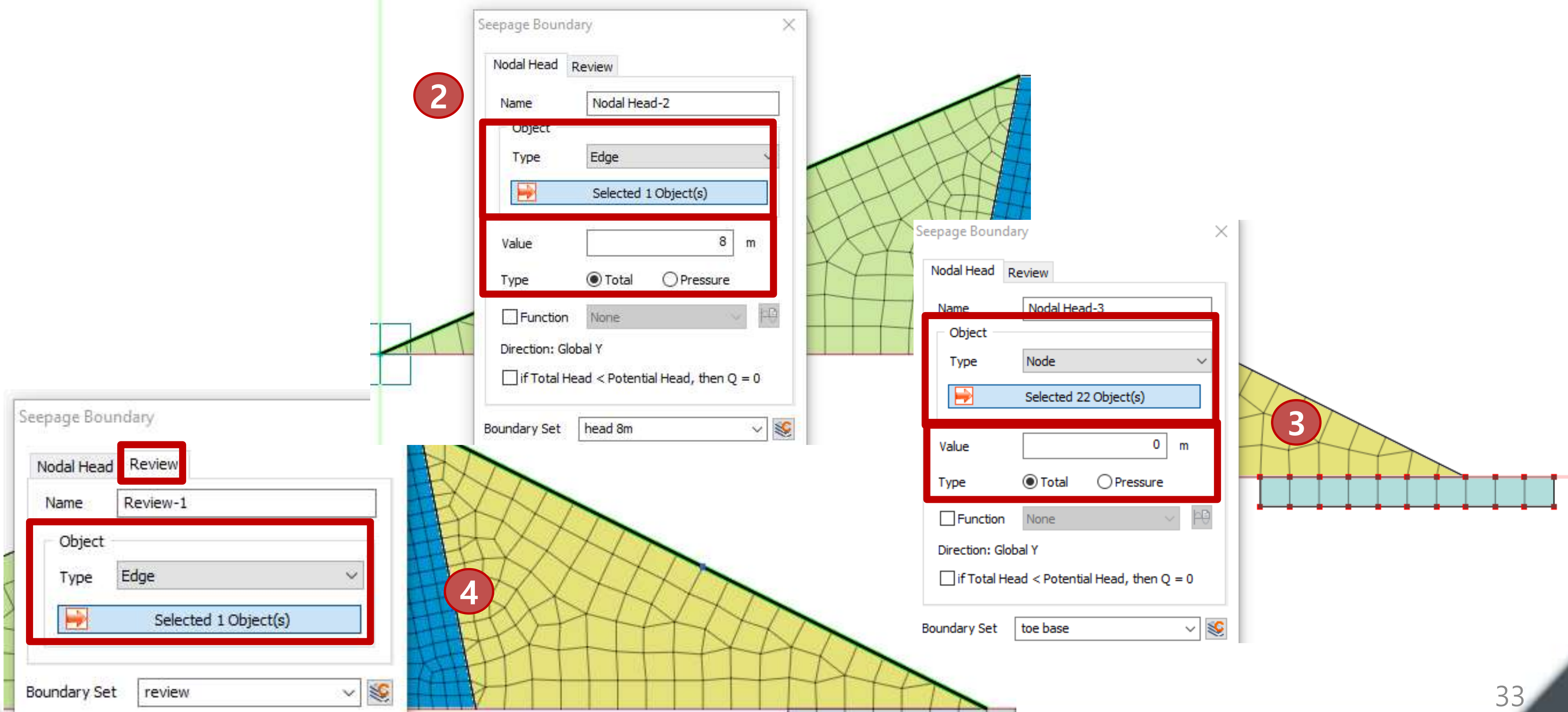
# Seepage Boundary

GTS NX

## Boundary

**Apply Seepage Boundary Conditions**

1. **Menu> Seepage/Consolidation Analysis>Nodal Head**
2. **Select the left Edge, apply Total head 8m as shown**
3. **Select Toe Drain Nodes, apply Total head of 0m**
4. **Select right Edge, apply Review boundary**

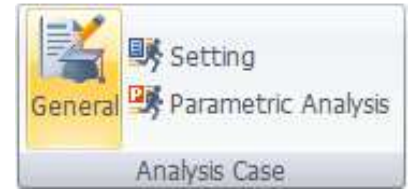
# Analysis Case

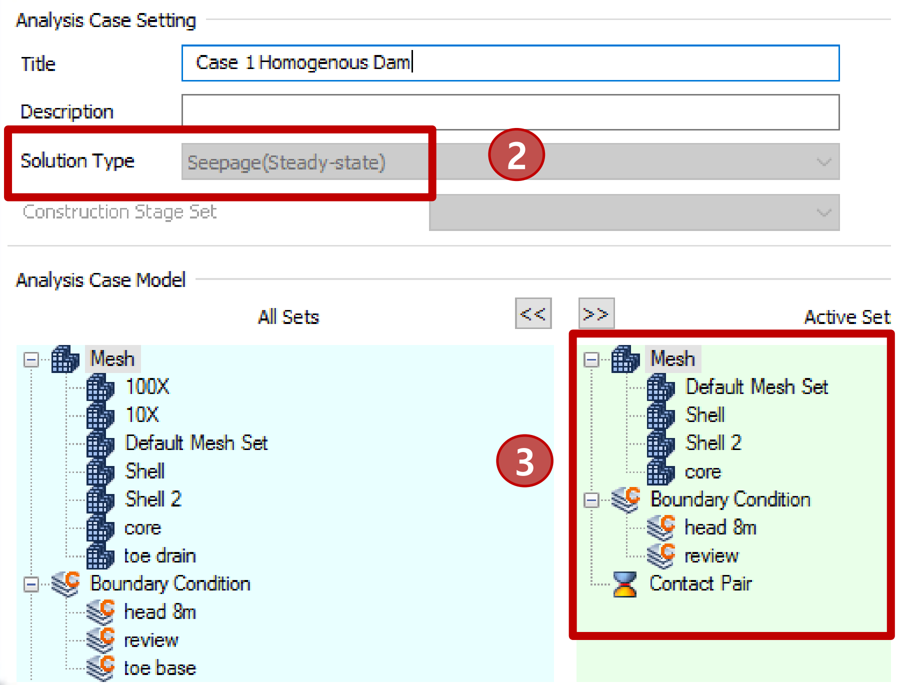
GTS NX

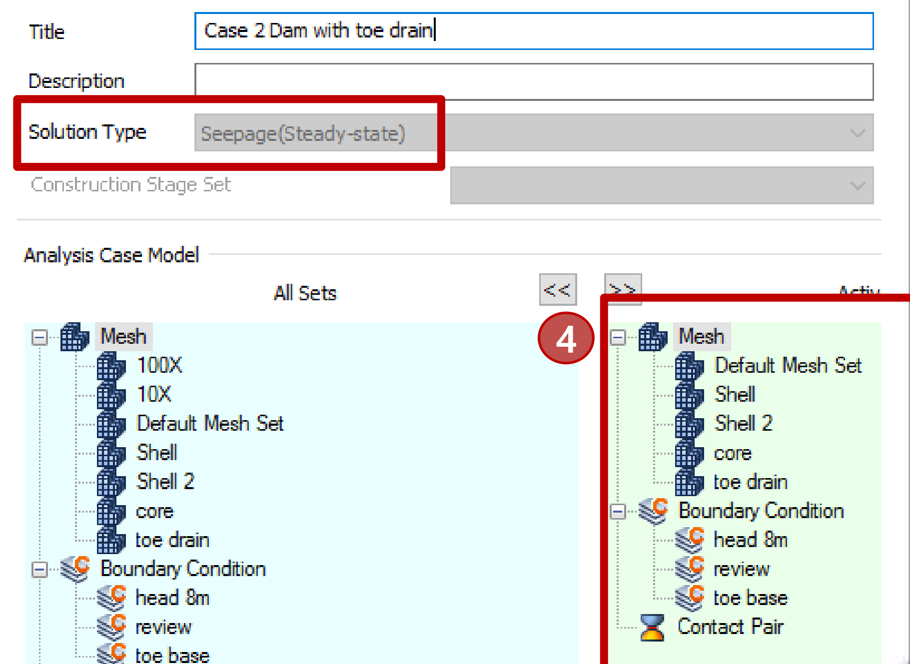
## Create 4 Seepage Steady State Analysis Cases

1. Menu> Analysis>Analysis Case >General
2. Select Seepage Steady State Solution Type
3. Create Case 1 and activate sets as shown. Press Apply
4. Create Case 2 and activate sets as shown. Press Ok

1



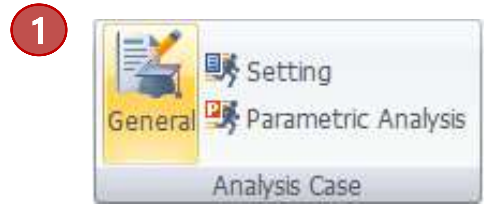



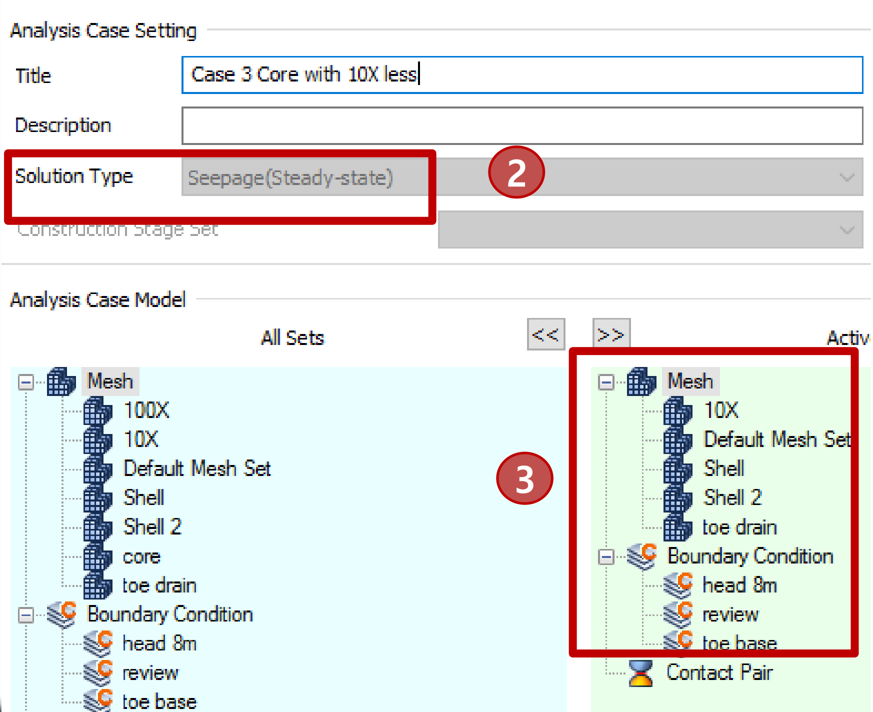
# Analysis Case

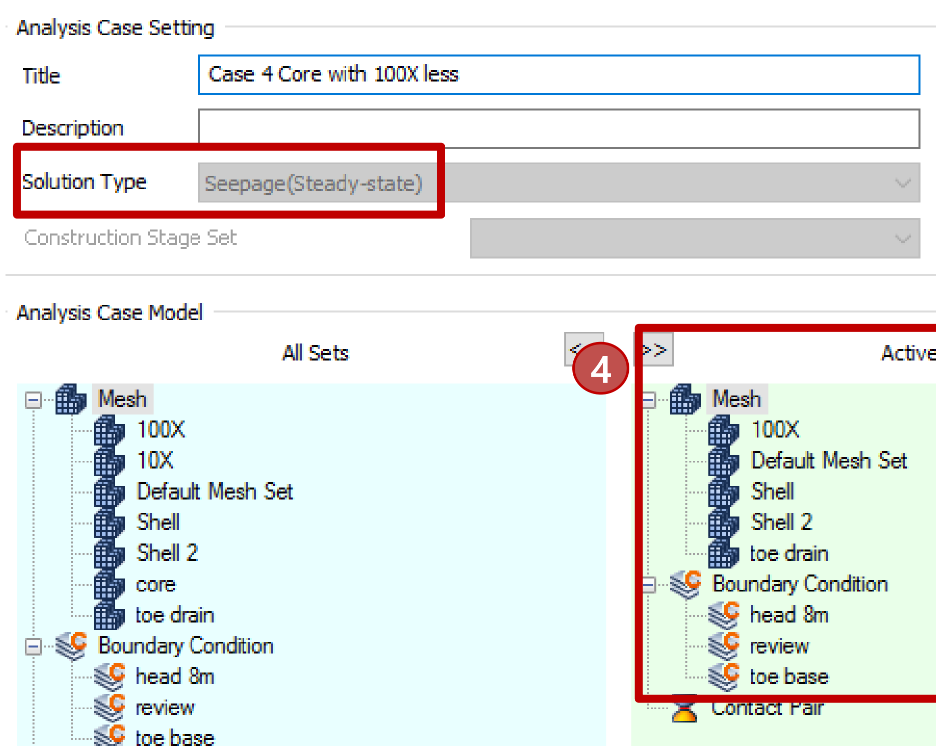
GTS NX

## Create 4 Seepage Steady State Analysis Cases

1. Menu> Analysis>Analysis Case >General
2. Select Seepage Steady State Solution Type
3. Create Case 3 and activate sets as shown. Press Apply
4. Create Case 4 and activate sets as shown. Press Ok



# Run Analysis

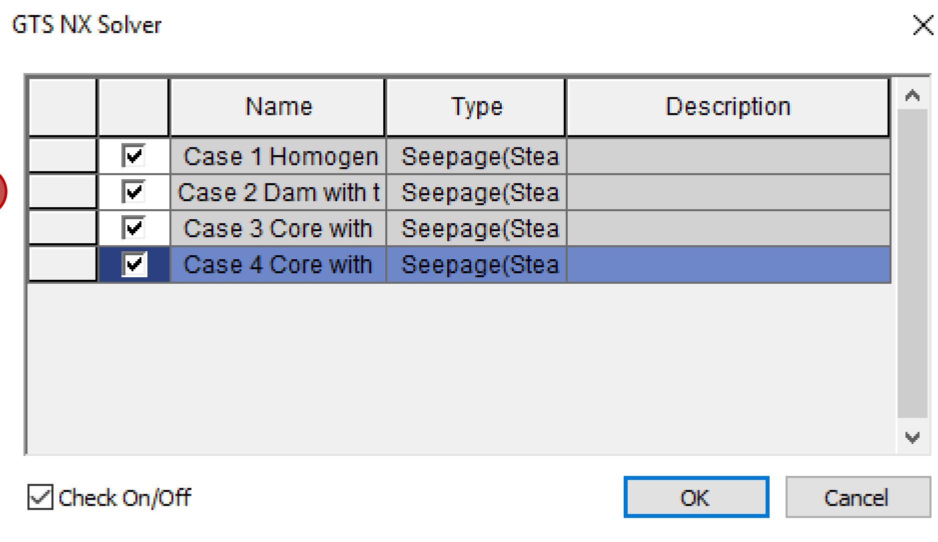
GTS NX

## Run all 4

1. Menu > Analysis > Perform
2. Select All
3. Press OK

1


2



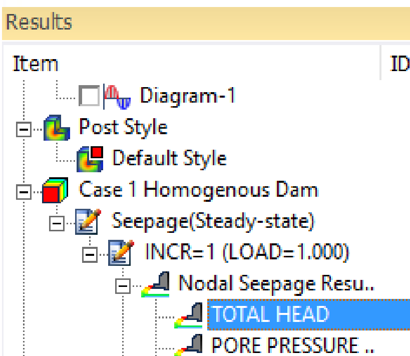
# Results

GTS NX

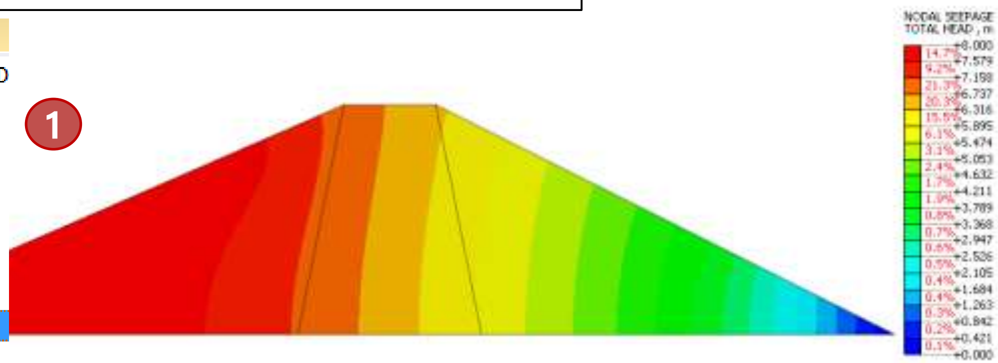
## Post Processor

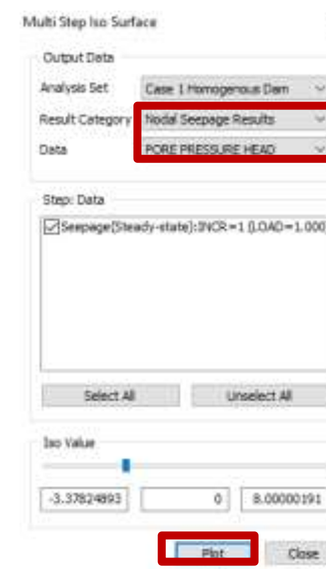
**Check the Total Head and water level for Case 1**

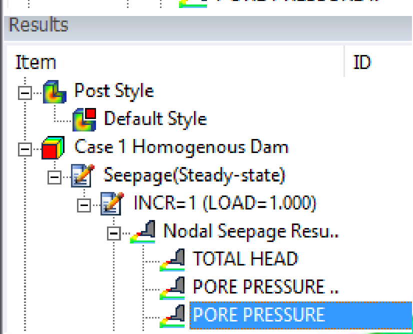
1. **Results > Nodal Seepage > Total Head**
2. **Results > Nodal Seepage > Pore Pressure**
3. **Results>Advanced>Multi Step Iso>Pore Pressure>0**

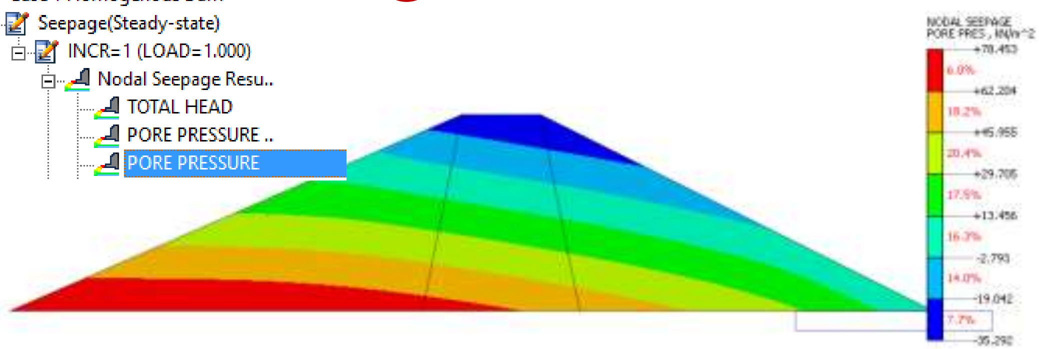

1




2




3

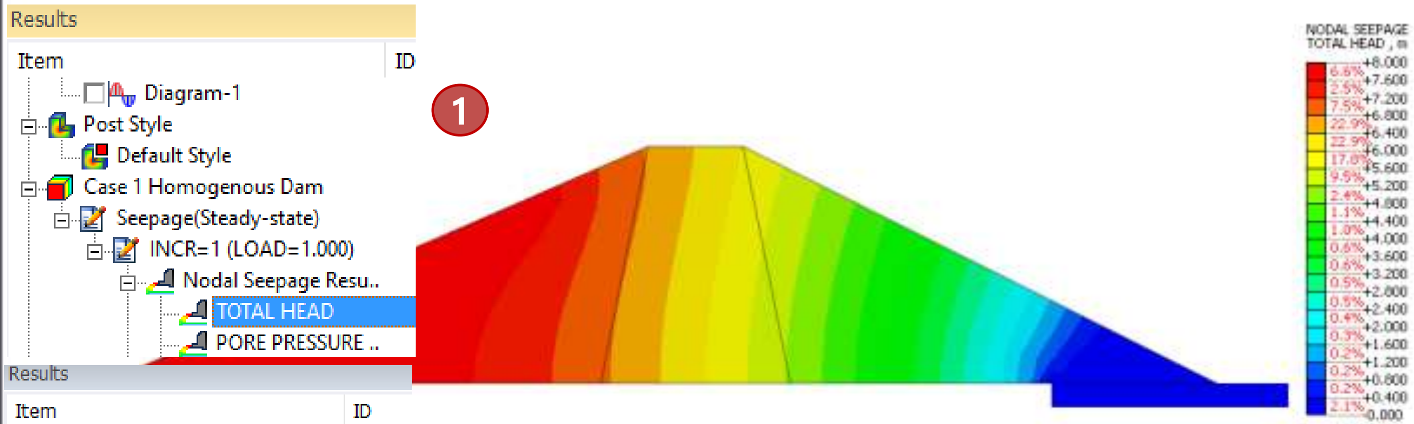
# Results

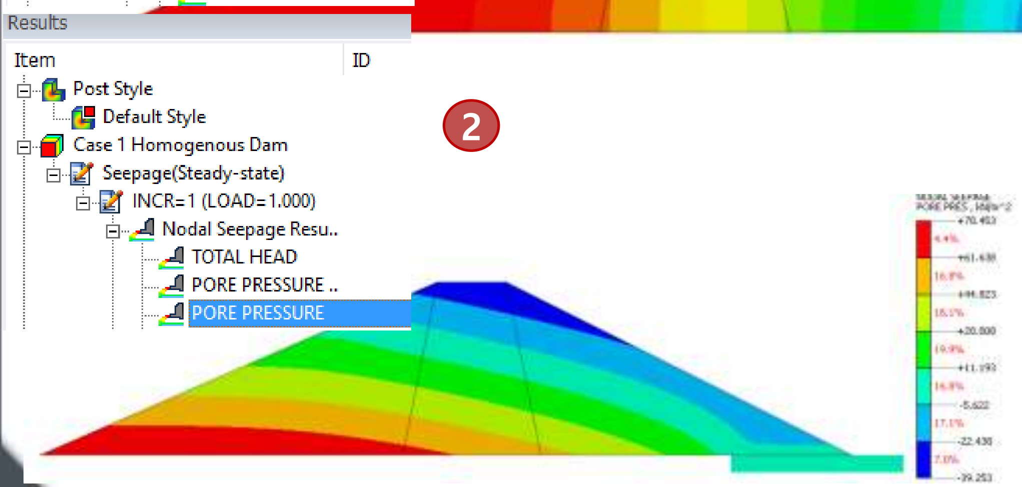
GTS NX

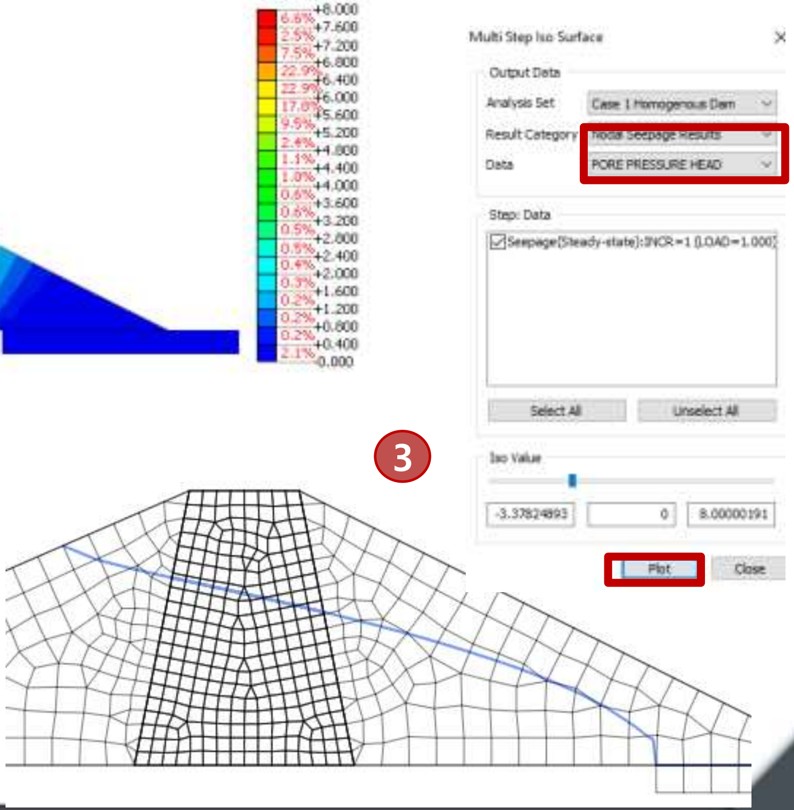
## Post Processor

**Check the Total Head and water level for Case 2**

1. **Results > Nodal Seepage > Total Head**
2. **Results > Nodal Seepage > Pore Pressure**
3. **Results>Advanced>Multi Step Iso>Pore Pressure>0**

# Results

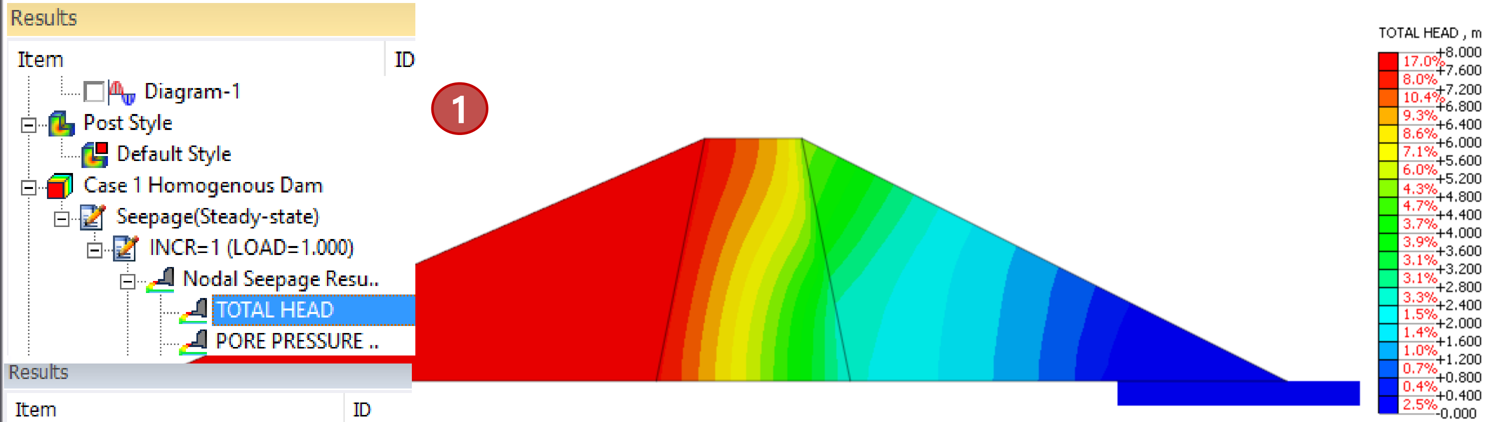
GTS NX

## Post Processor

**Check the Total Head and water level for Case 3**

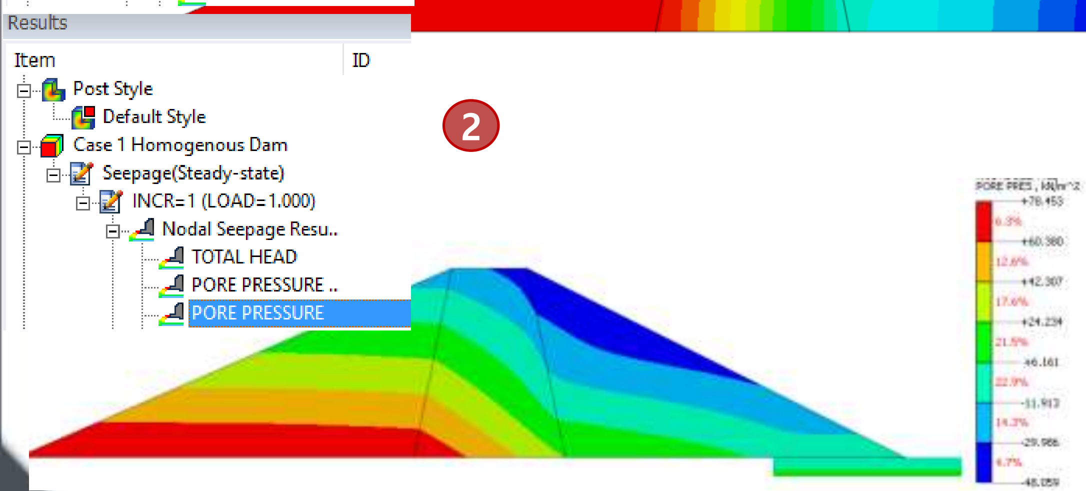
1. **Results > Nodal Seepage > Total Head**
2. **Results > Nodal Seepage > Pore Pressure**
3. **Results>Advanced>Multi Step Iso>Pore Pressure>0**


1

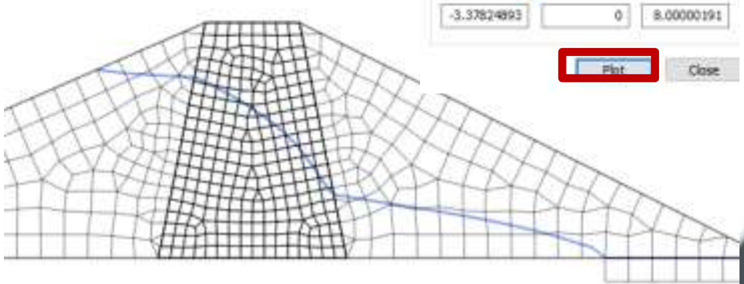




2




3



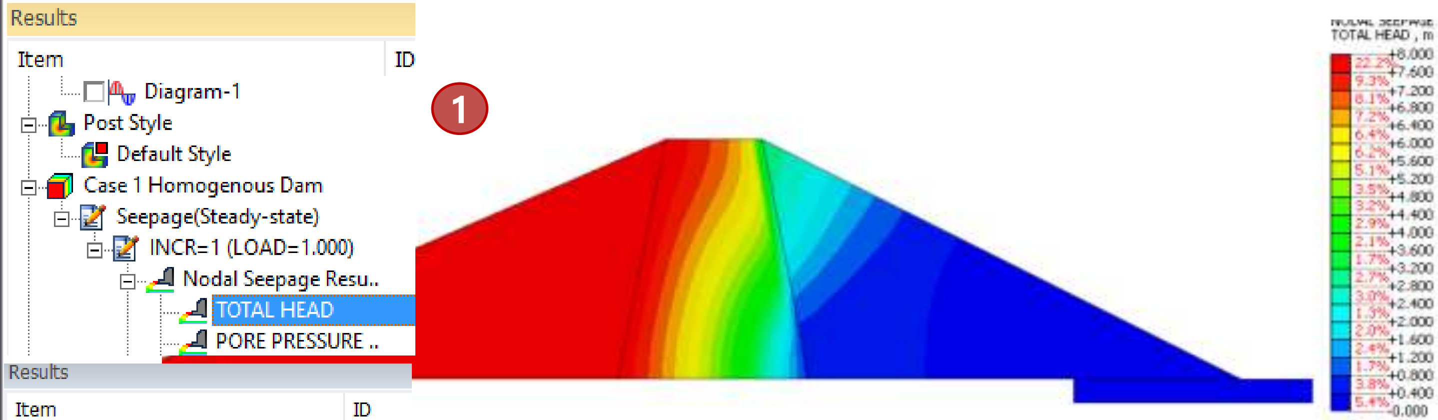
# Results

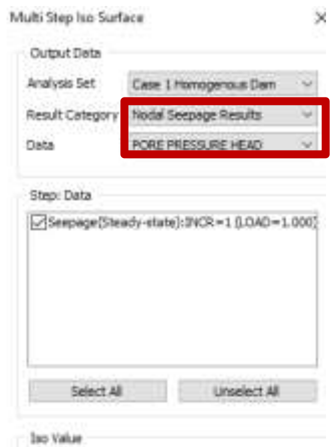
GTS NX

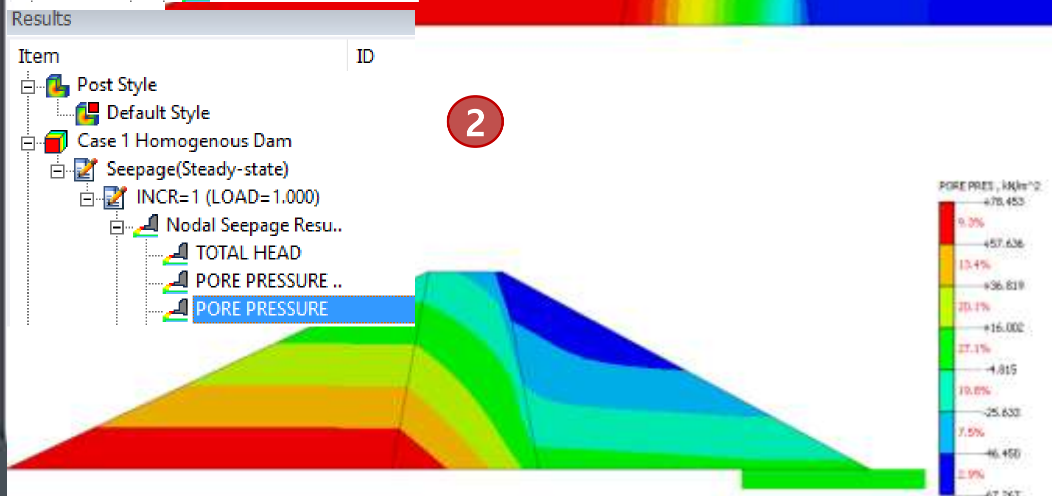
## Post Processor

**Check the Total Head and water level for Case 4**

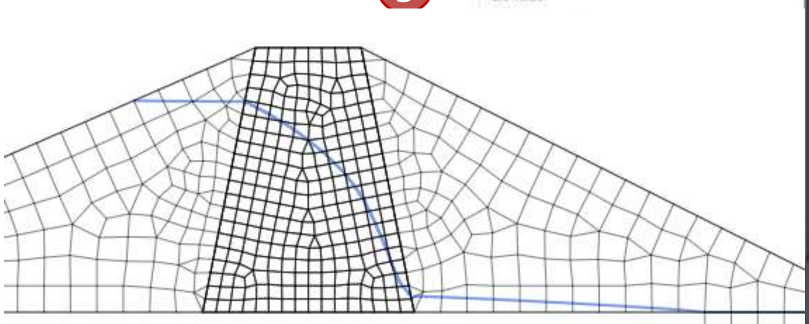
1. **Results > Nodal Seepage > Total Head**
2. **Results > Nodal Seepage > Pore Pressure**
3. **Results>Advanced>Multi Step Iso>Pore Pressure>0**

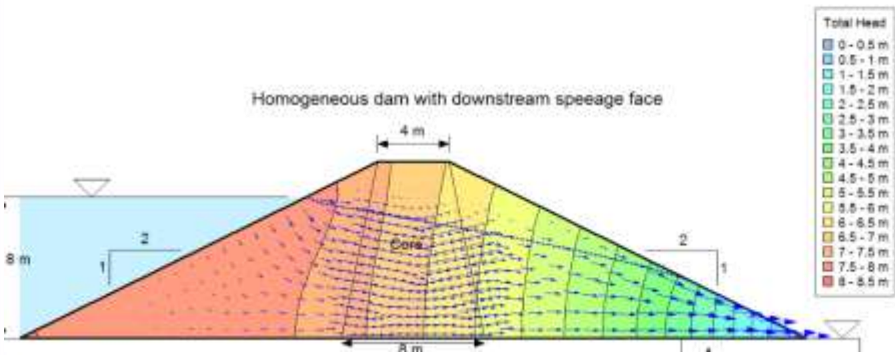

3

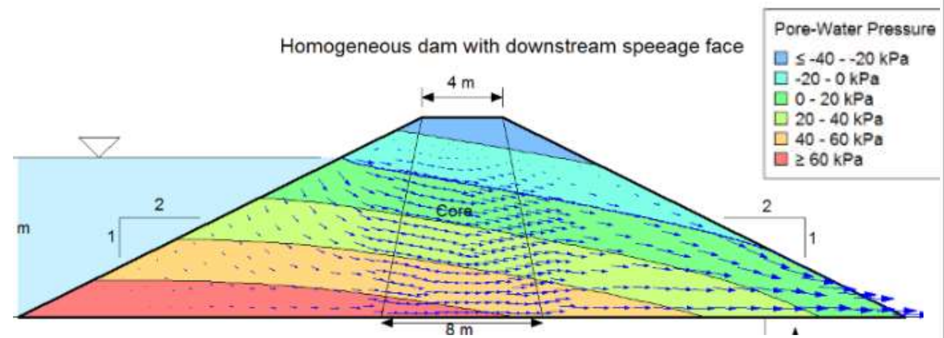


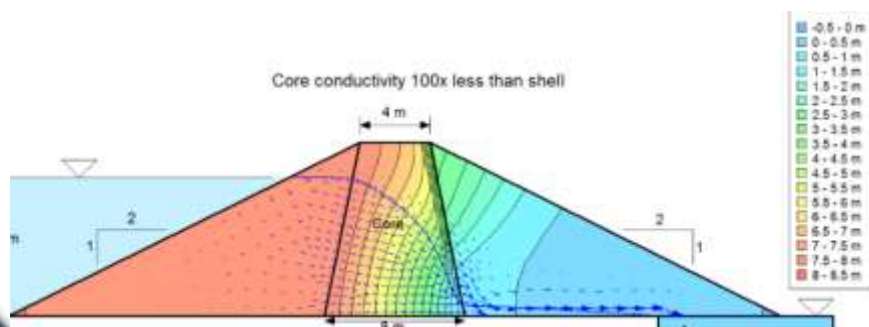
# Compare Results to SEEPW Tutorial

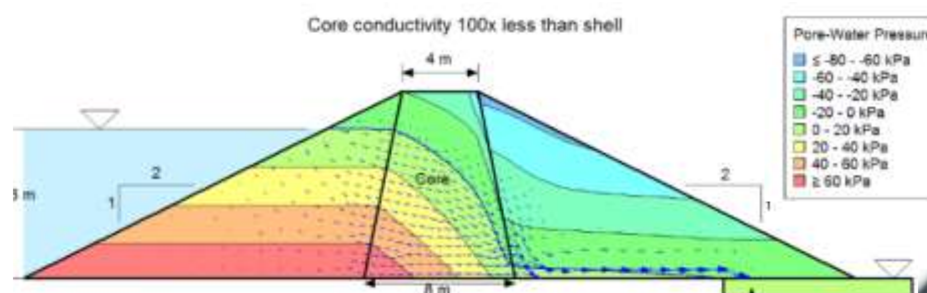
GTS NX

Total Head and Pore Pressure distribution match SEEPW tutorial. Shown here are Case 1 and Case 4 results.



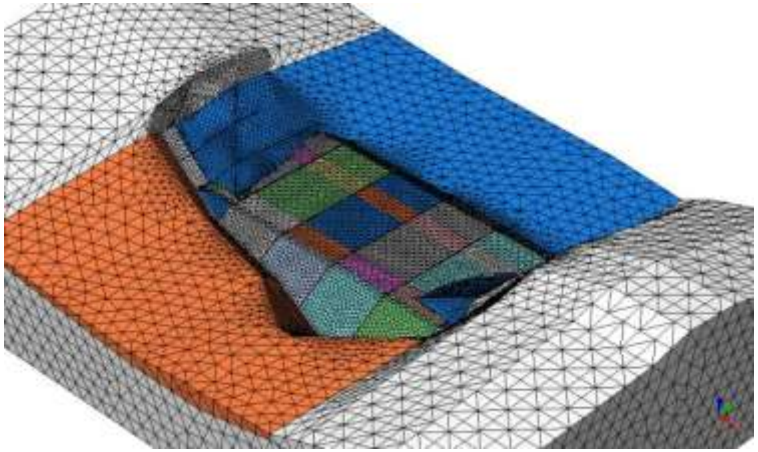



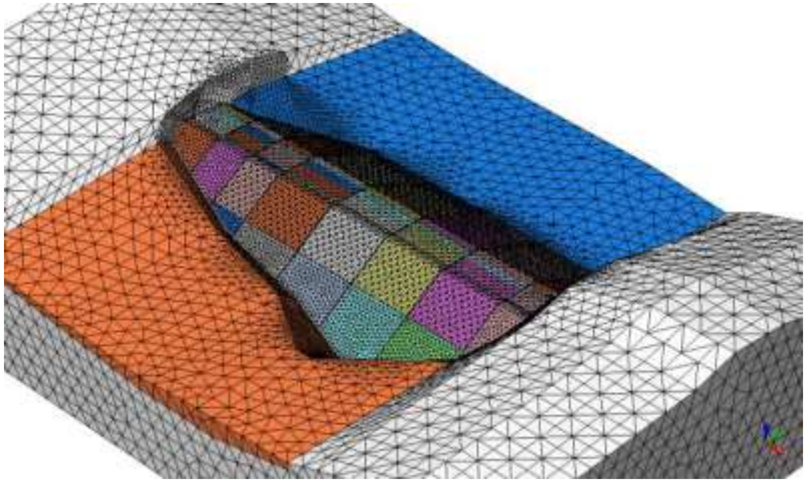
# 3D Case Study: CS of Dam(S. Korea)

Buhang Dam, South Korea

**Construction stage analysis of dam reflecting full water level hydrostatic pressure**

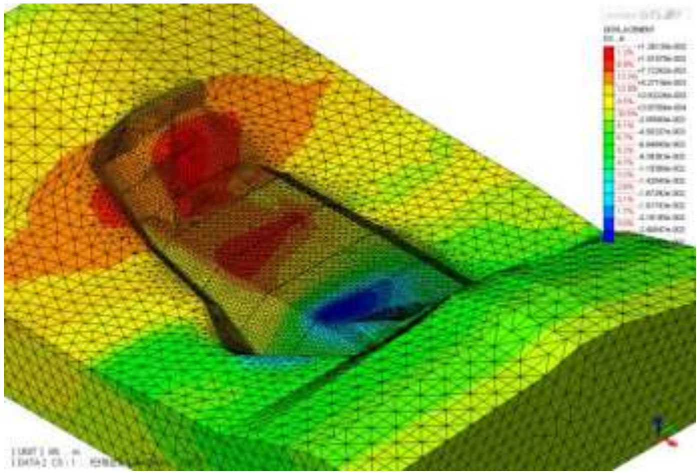


Stage2: Banking Height 160m

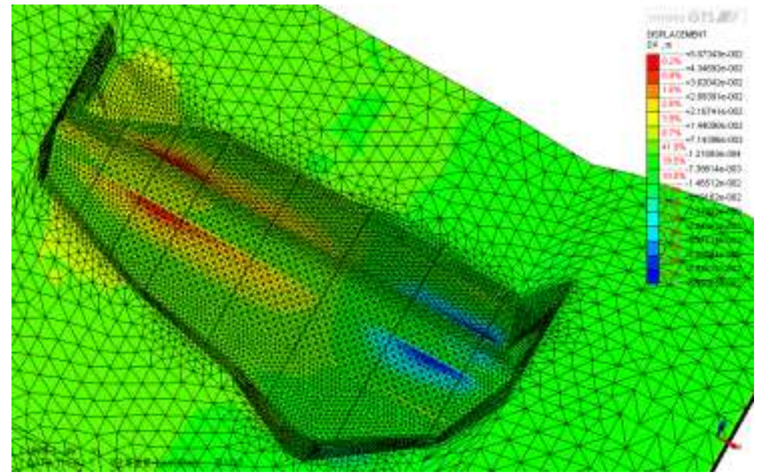


Stage5: Banking Height 210m + Hydrostatic pressure

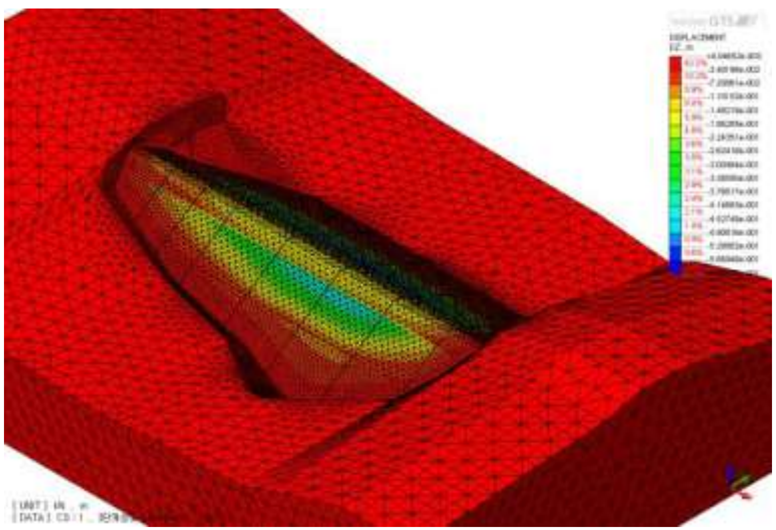
# 3D Case Study: CS of Dam(S. Korea)



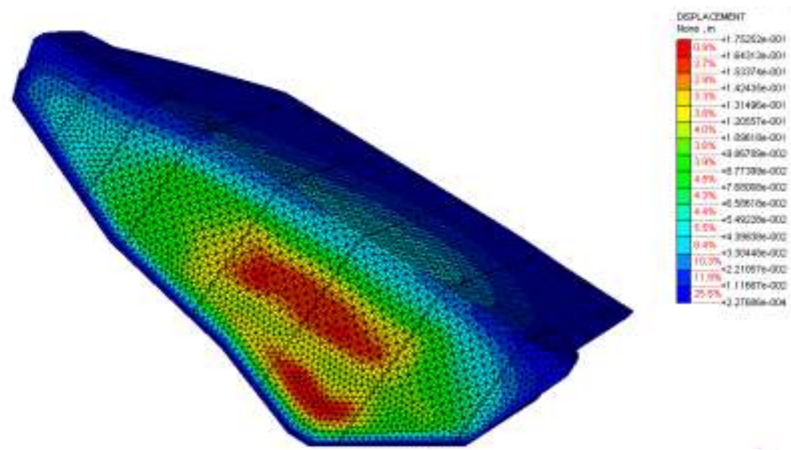
Stage 2: Total Displacements



Stage 5: Horizontal Displacements

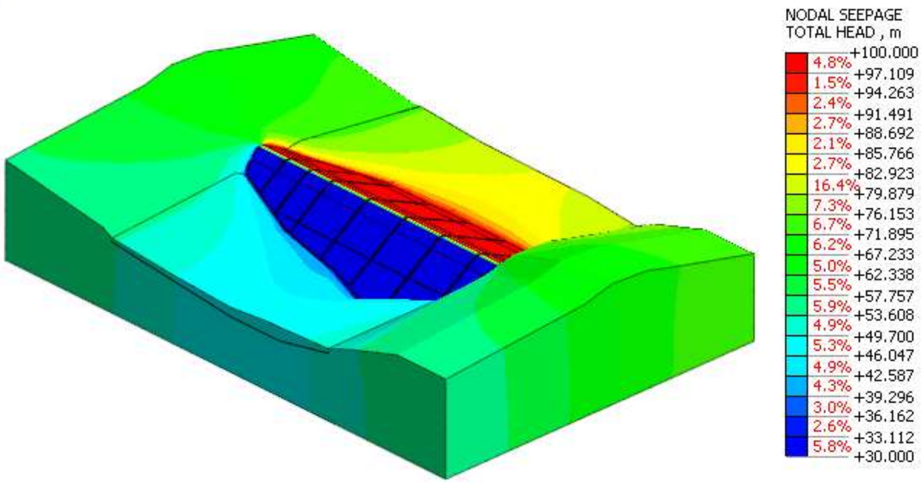


Stage 4: Vertical deflection

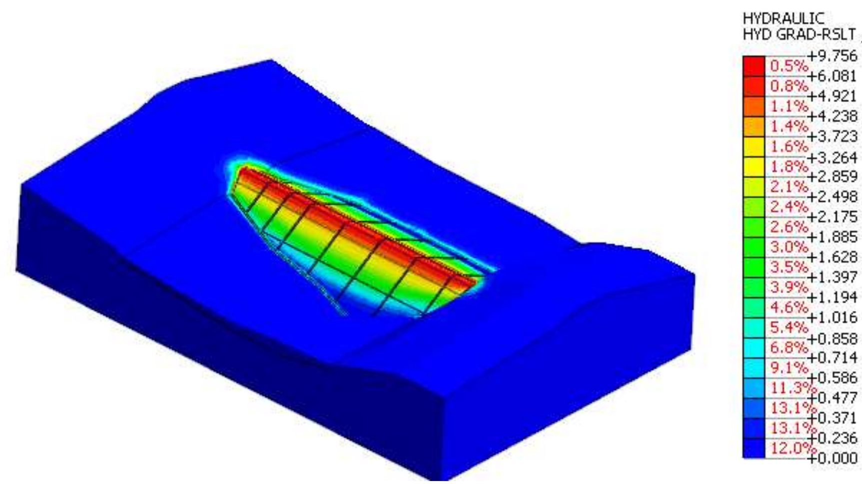


Stage 5: Hydrostatic pressure

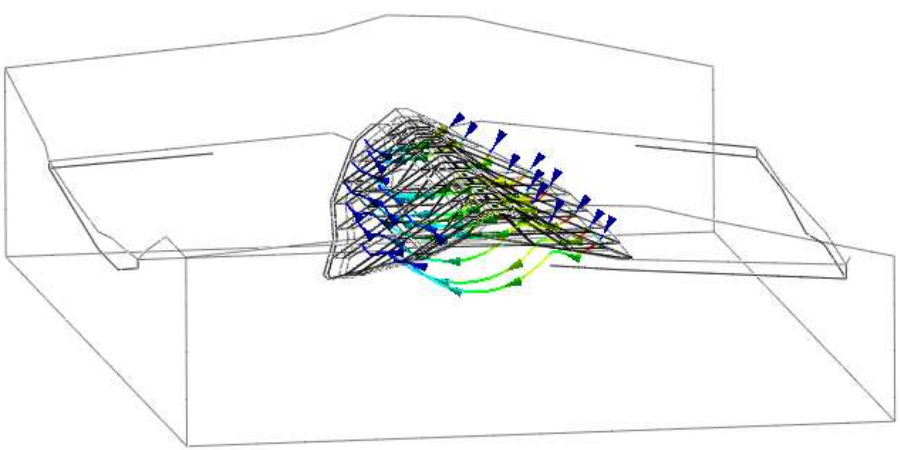
# 3D Case Study: CS of Dam(S. Korea)



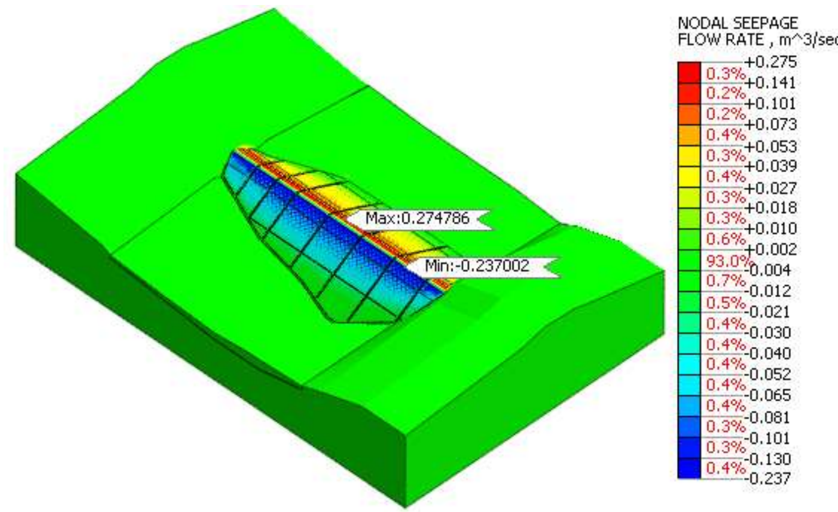

Stage 5: Total Head




Stage 5: Hydraulic Gradient



Stage 5: Flow Paths



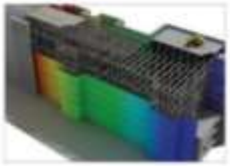

Stage 5: Flow Rate

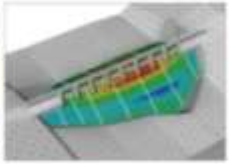
# Strength Reduction Stability Comparison

## (SLOPEW Comparison)

GTS NX

Geo-Technical analysis System New eXperience






Angel Francisco Martinez

Application Engineer
MIDAS IT
Estados Unidos


Integrated Solver Optimized for the next generation 64-bit platform

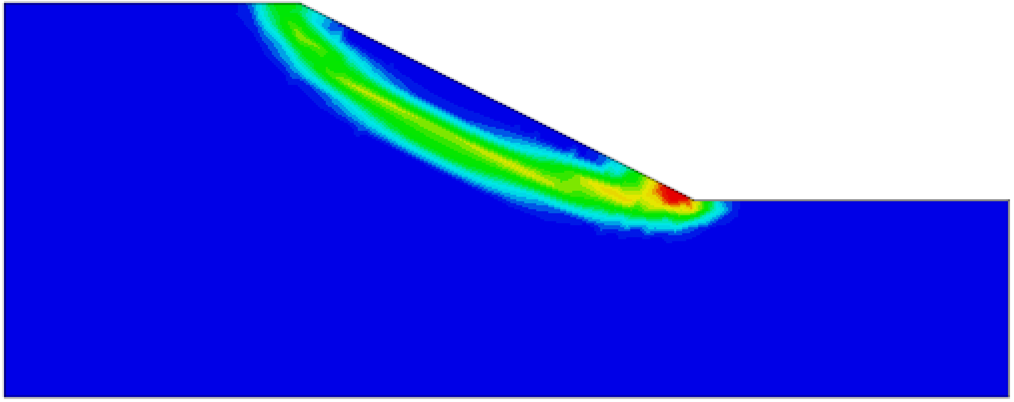
**Finite Element Solutions for Geotechnical Engineering**

MIDAS

# Introduction



Introduction Stability by strength reduction is a procedure where the factor of safety is obtained by weakening the soil in steps in an elastic-plastic finite element analysis until the slope “fails”. The factor of safety is deemed to be the factor by which the soil strength needs to be reduced to reach failure (Dawson et al., 1999; Griffiths and Lane, 1999). Numerically, the failure occurs when it is no longer possible to obtain a converged solution. The finite element equations for a stress-strain formulation are in essence equations of equilibrium. Not being able to obtain a converged solution therefore infers the system is beyond the point of limiting equilibrium. An alternative way to define “failure” is the point at which the deformations become excessive.
In the Strength Reduction method, the soil strength is artificially reduced, and so there is a need to redistribute the stresses. This can be done by the Stress Redistribution algorithm, and so this option can be indirectly used to do a Strength Reduction stability analysis.



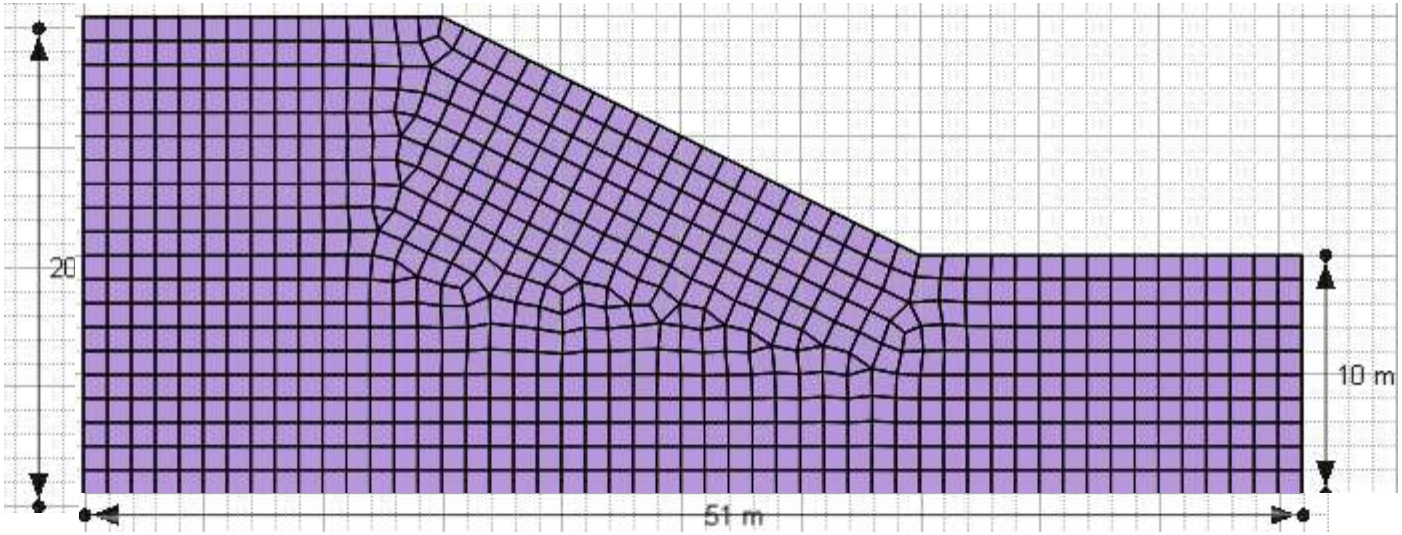
# Initial model dimensions

GTS NX

## Model Conditions

- 2D Strength Reduction Method Analysis
- 20m tall X 51m wide X 10 m Slope
- 1 layer
- 4 materials
    - 1.2
    - 1.3
    - 1.4
    - 1.5

Figures show the corresponding problem setup. It illustrates how a Strength Reduction Method analysis can be done with MIDAS GTS NX. In addition, the results are compared as the soil strength is reduced in stages. For each stage, the factor of safety is obtained for the new material. Each strength reduction analysis uses the previous analysis as its initial conditions. Lastly, the results are compared to a SLOPEW tutorial.



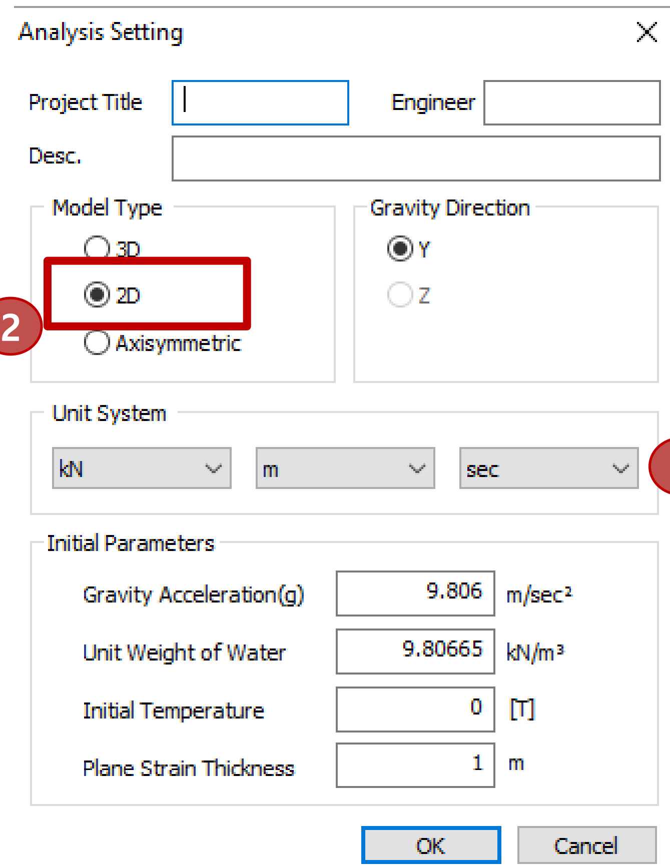
# Start Axisymetric File

GTS NX

## Open New

1. File > New
2. 2D
3. Units kN, m, sec


1




2

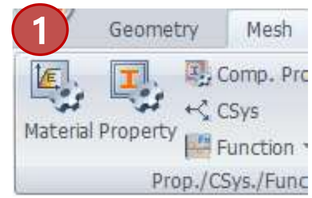
3

# Materials

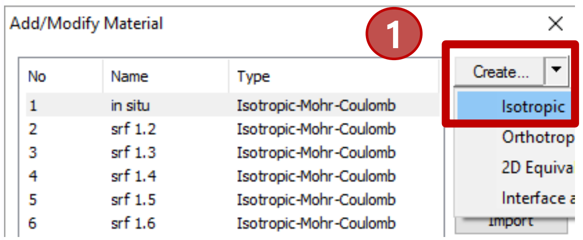
GTS NX

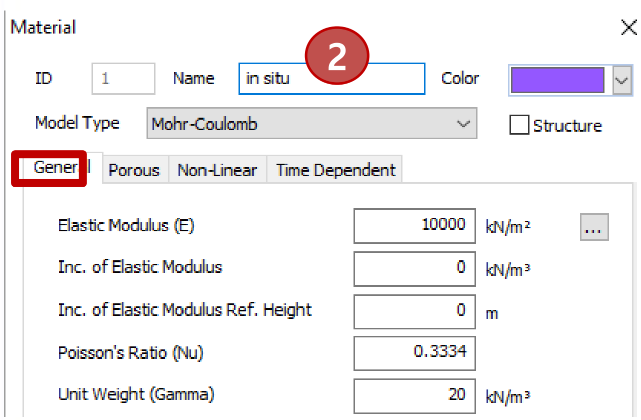
## Define Materials

1. Menu > Mesh > Material> Create> Isotropic>Mohr Coulomb
2. Define 6 materials as shown in the General tab
3. Define Non linear properties as shown in table

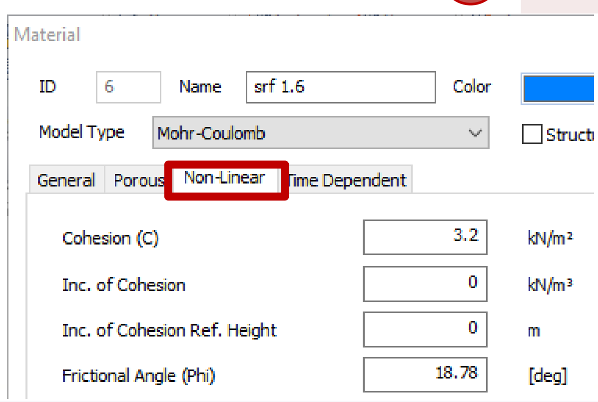



| Name | Cohesion | Friction Angle |
|---|---|---|
| In situ | 5 | 28 |
| SRF 1.2 | 4.17 | 23.9 |
| SRF 1.3 | 3.85 | 22.24 |
| SRF 1.4 | 3.57 | 20.8 |
| SRF 1.5 | 3.33 | 19.52 |
| SRF 1.6 | 3.2 | 18.78 |



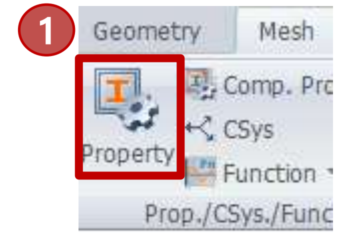

3

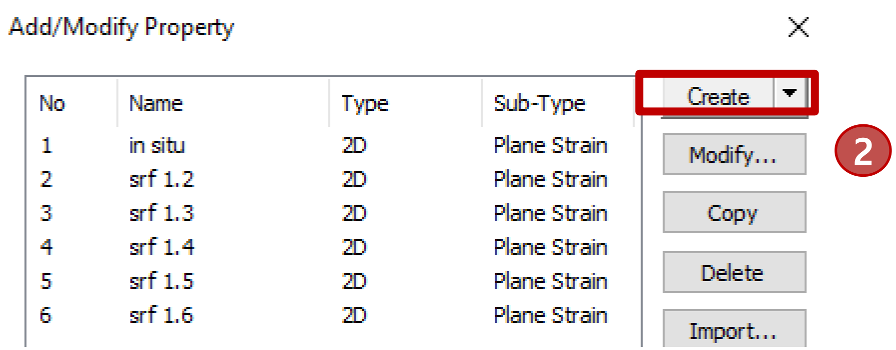


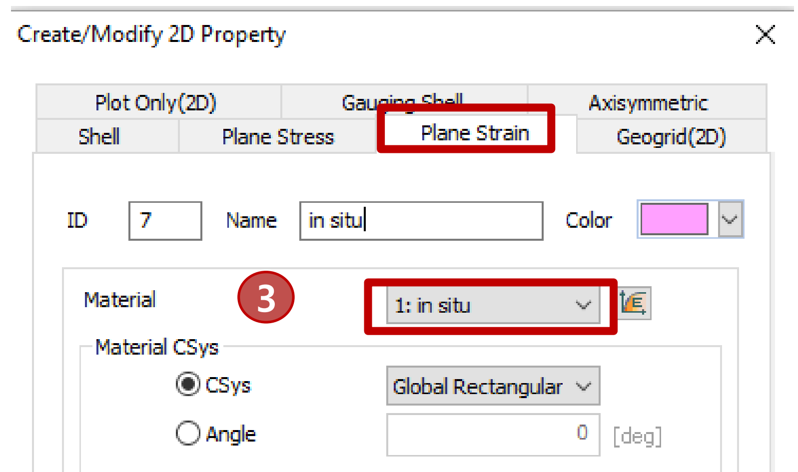
# Properties

## Define Materials and Properties

1. Menu > Mesh > Properties > Create > 2D > Plain Strain
2. Create 6 2D Plain Strain properties for the layers
3. Match the name with the material property.



# Geometry

GTS NX

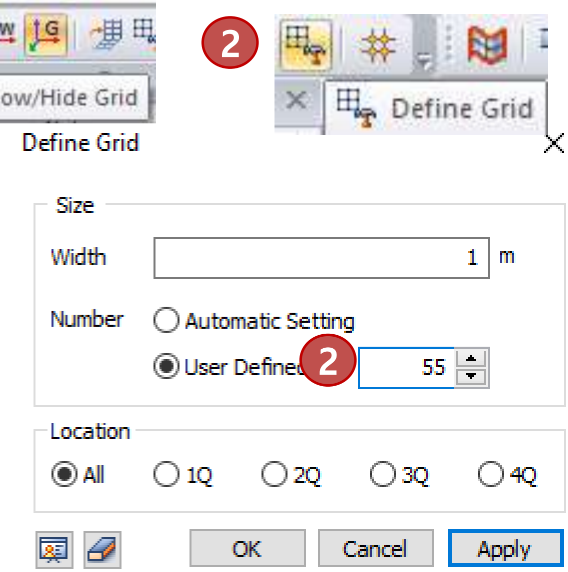
## Create Geometry Outline

**Define the WORK GRID as a drawing plane with lines spaced 1m apart to create rectangles**

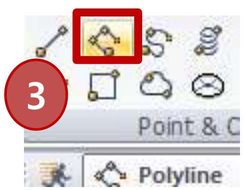
1. **Menu > SHOW GRID**
2. **Menu > Define Grid > Width 1m, User Define 55 m**
3. **Menu>Geometry > PolyLine**
4. **Select the ABS x,y method, set 1st Location as 0,0 press Apply**
5. **Set End location as 0, 20**

1

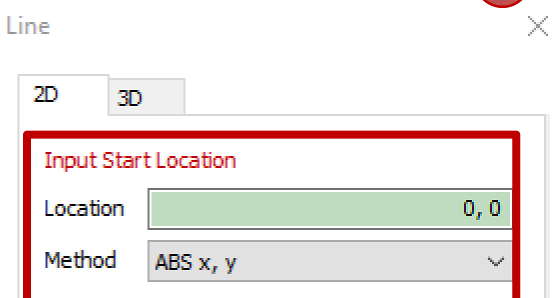

2



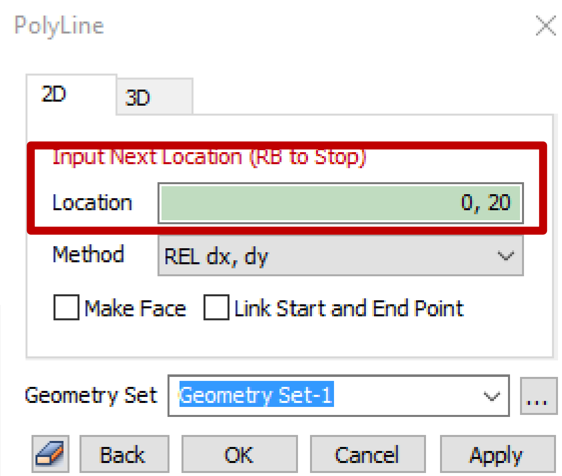

3

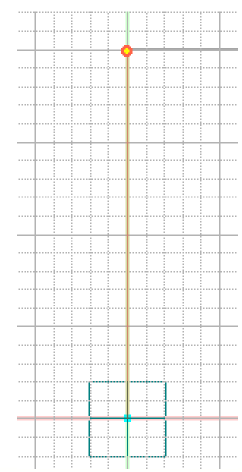



4




5



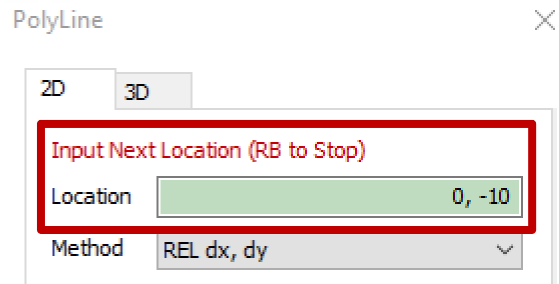



# Geometry

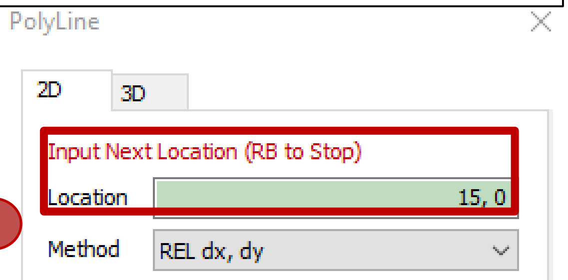
GTS NX

## Polyline points

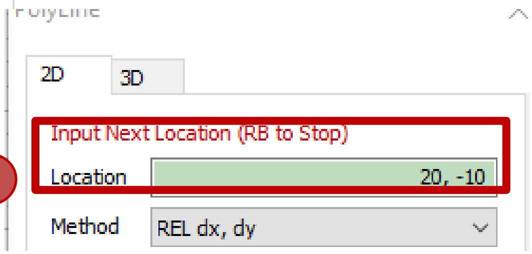
1. Input next location (15,0)
2. Input next location (20, -10)
3. Input next location (16, 0)
4. Input next location (0, -10)
5. Input next location (-51, 0)
6. Press OK

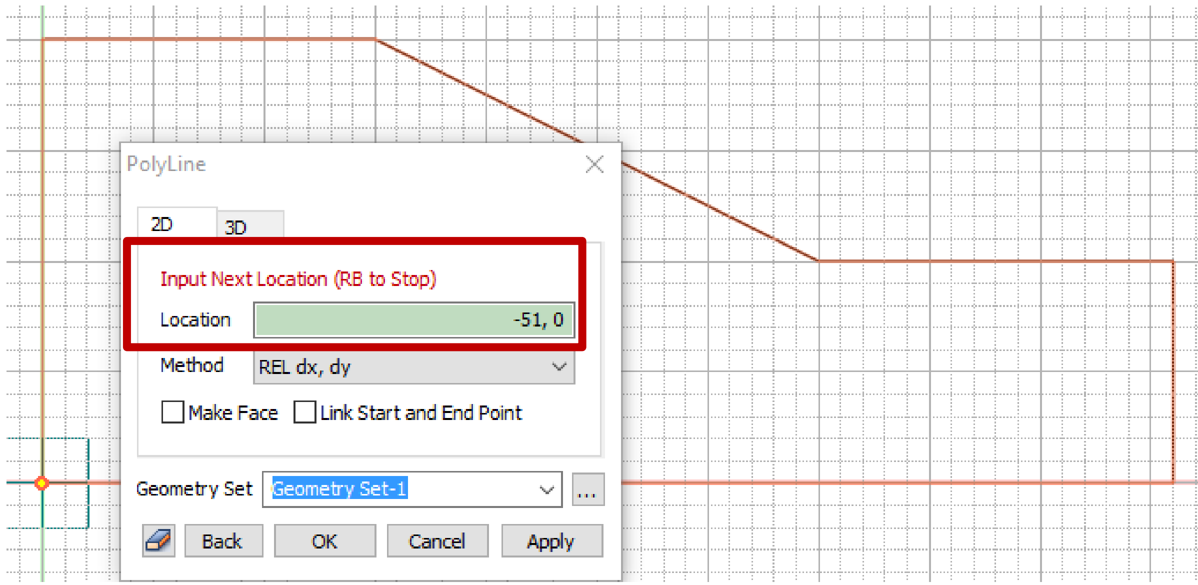



1

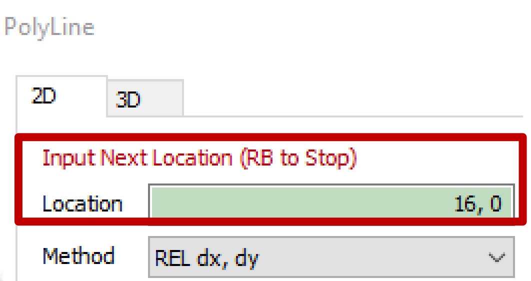



2




3

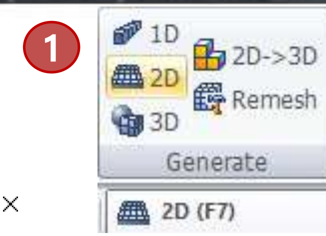


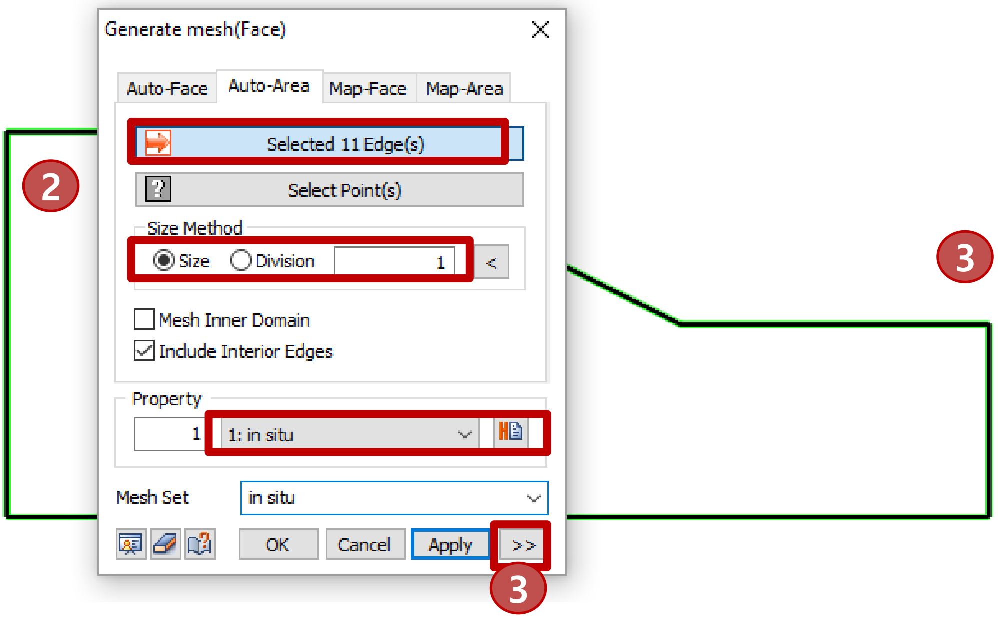
# Mesh

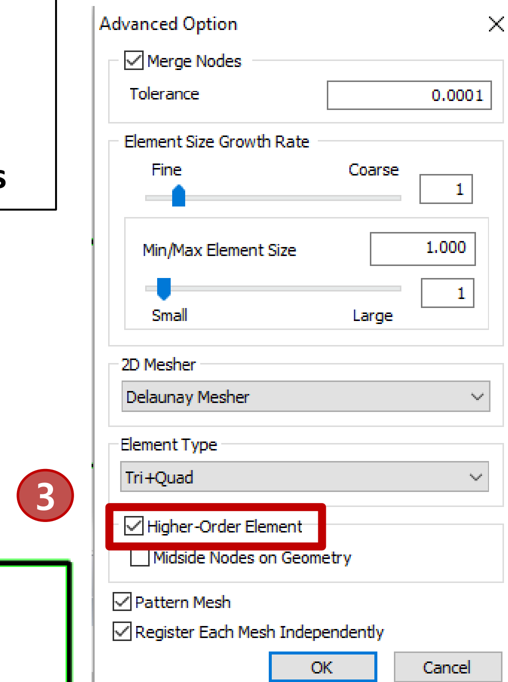
GTS NX

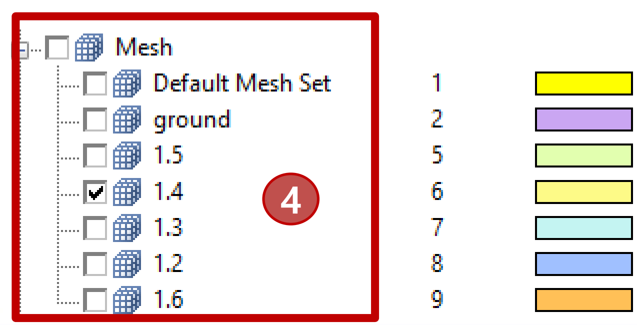
## Mesh

1. Menu> Mesh > Generate > 2D > Auto Area
2. Mesh with 1m size and in situ Property
3. Select >> Advance Option and activate Higher Order Element
4. Repeat meshing same area with the remaining materials



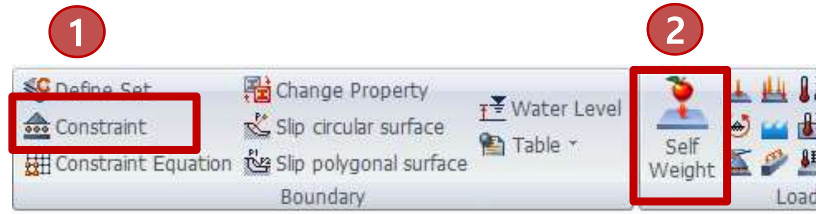
# Boundary

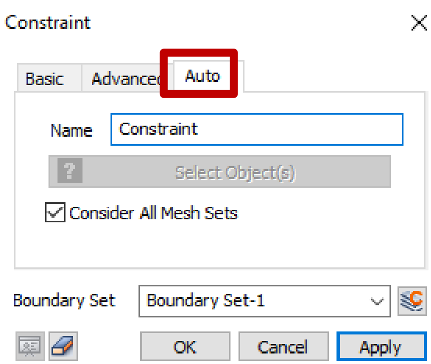
GTS NX

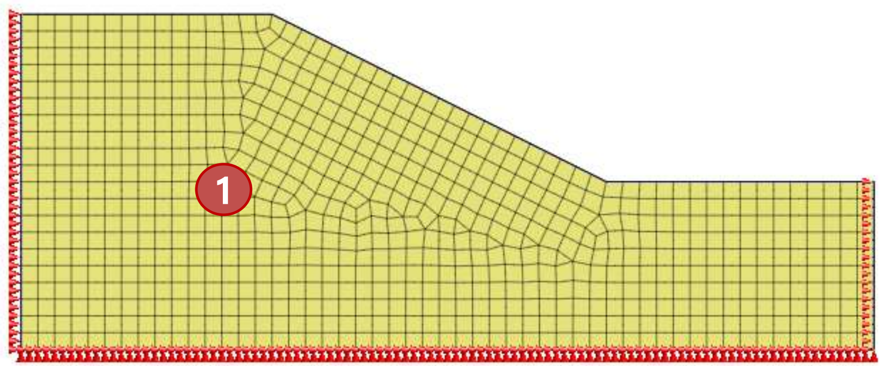
## Boundary

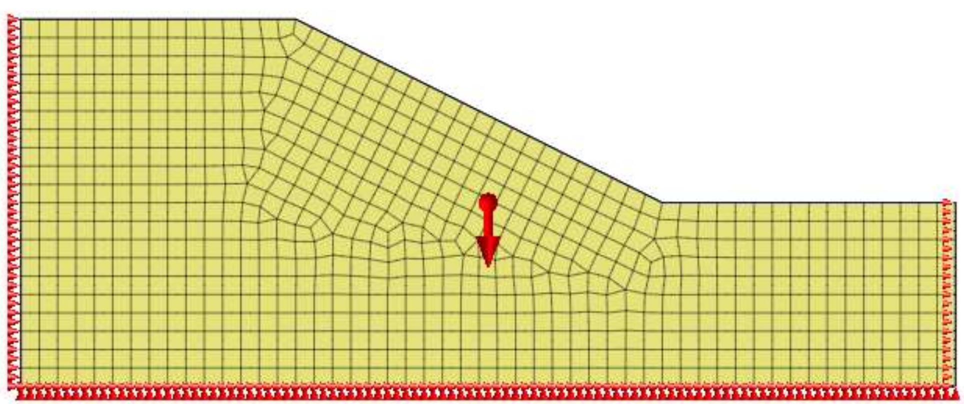
**Apply Seepage Boundary Conditions**

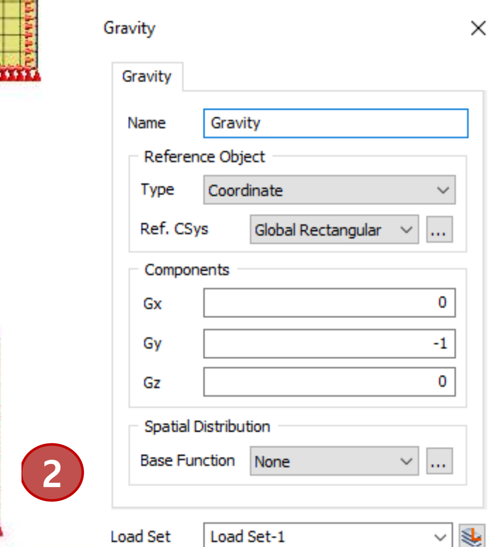
1. Menu> Static > Slope > Boundary > Constrain > Auto
2. Menu> Static > Slope > Load > Self Weight> Gy





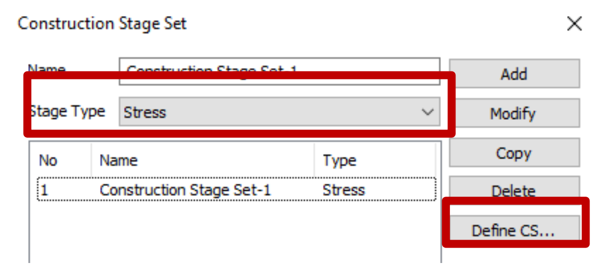
# Construction Stages

GTS NX

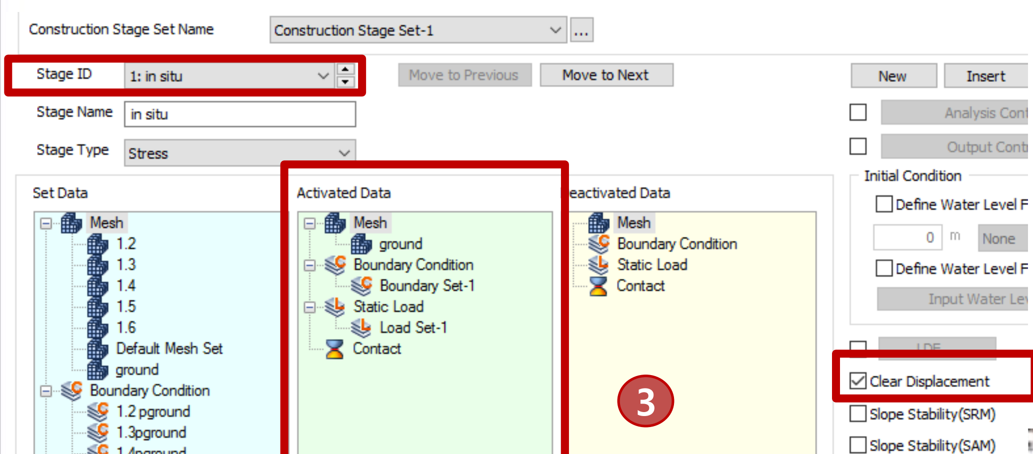
## Create Construction Stage Analysis

1. Menu> Static/Slope > Construction Stage> Stage Set
2. Select Stress Solution Type > Define CS
3. Create Stage 1 and activate sets as shown. Press Save > New
4. Create Stage 2 and activate sets as shown. Press Save > New

1

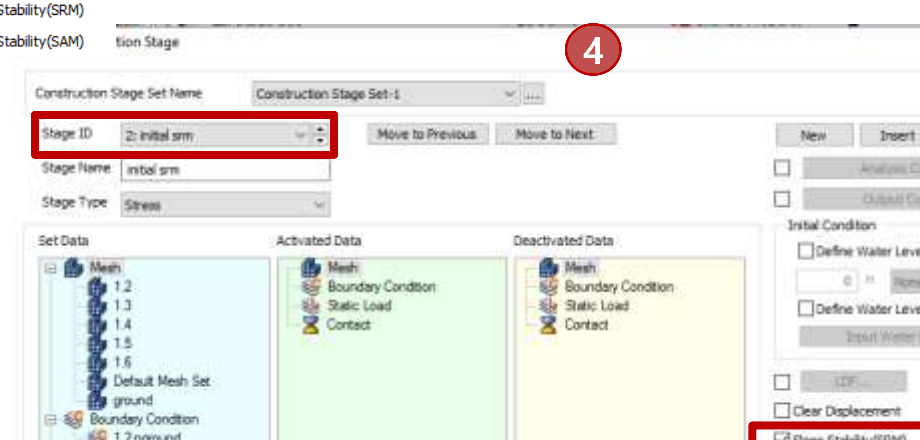




2




3

4

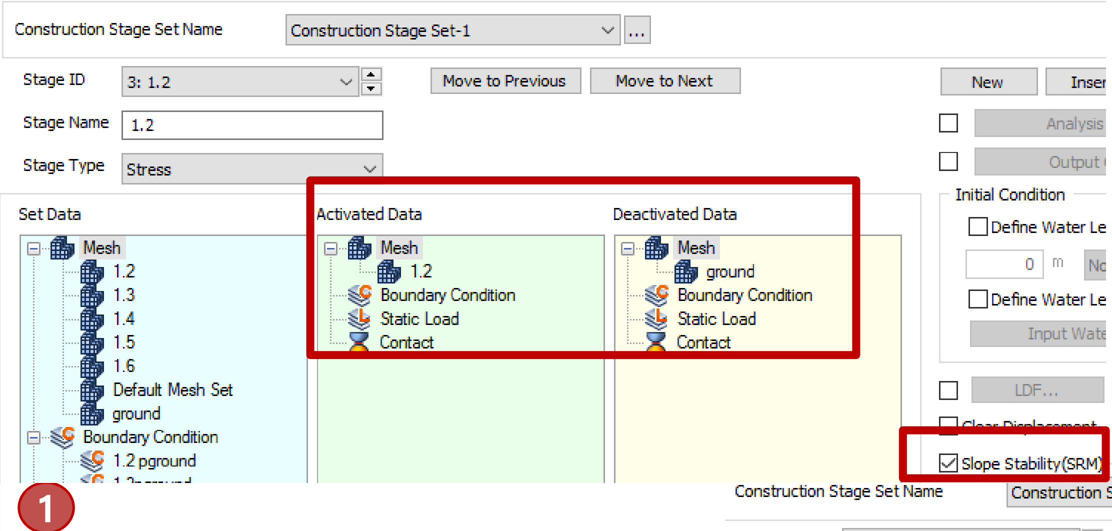


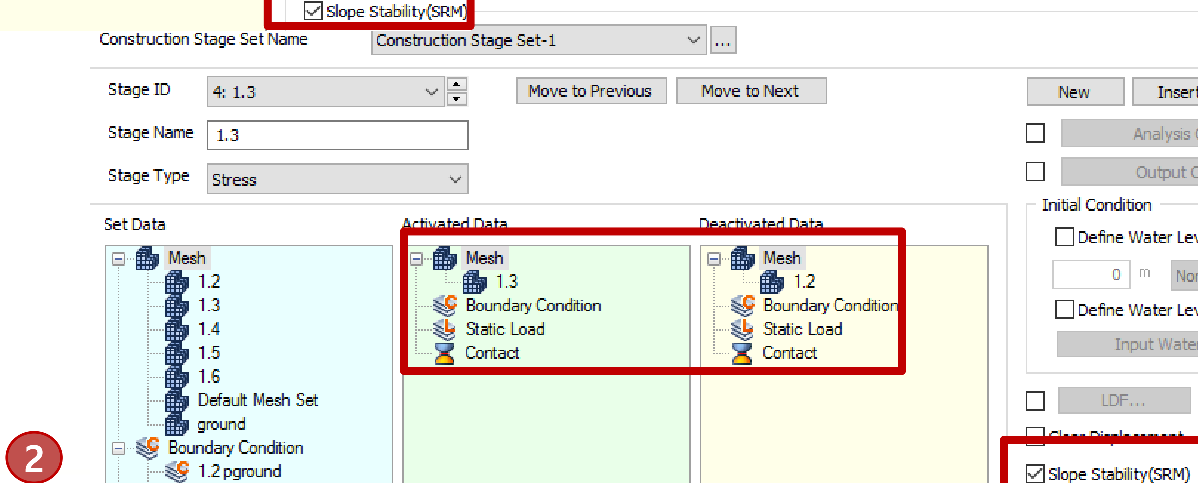
# Construction Stages

GTS NX

## Continue Stages

1. Create Stage 3 and activate sets as shown. Press Save > New
2. Create Stage 4 and activate sets as shown. Press Save > New



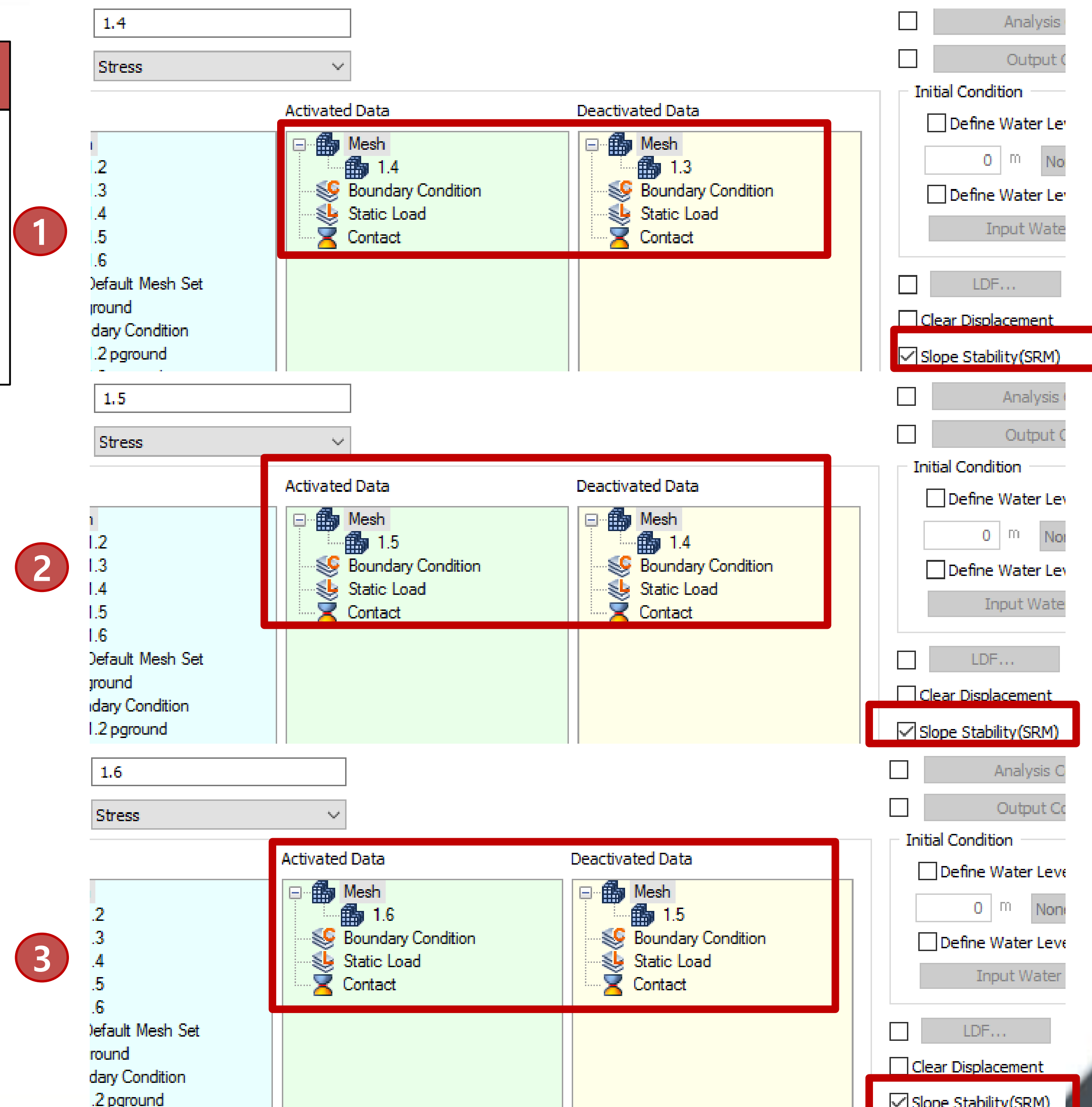



# Construction Stages

GTS NX

## Continue Stages

1. Create Stage 5 and activate sets as shown. Press Save > New
2. Create Stage 6 and activate sets as shown. Press Save > New
3. Create Stage 7 and activate sets as shown. Press Save > Close

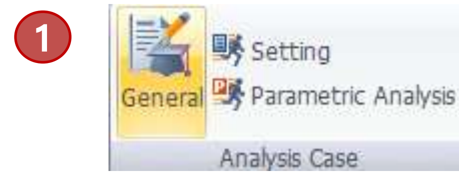


# Analysis Case

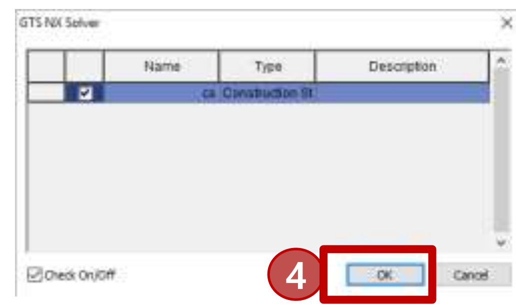
GTS NX

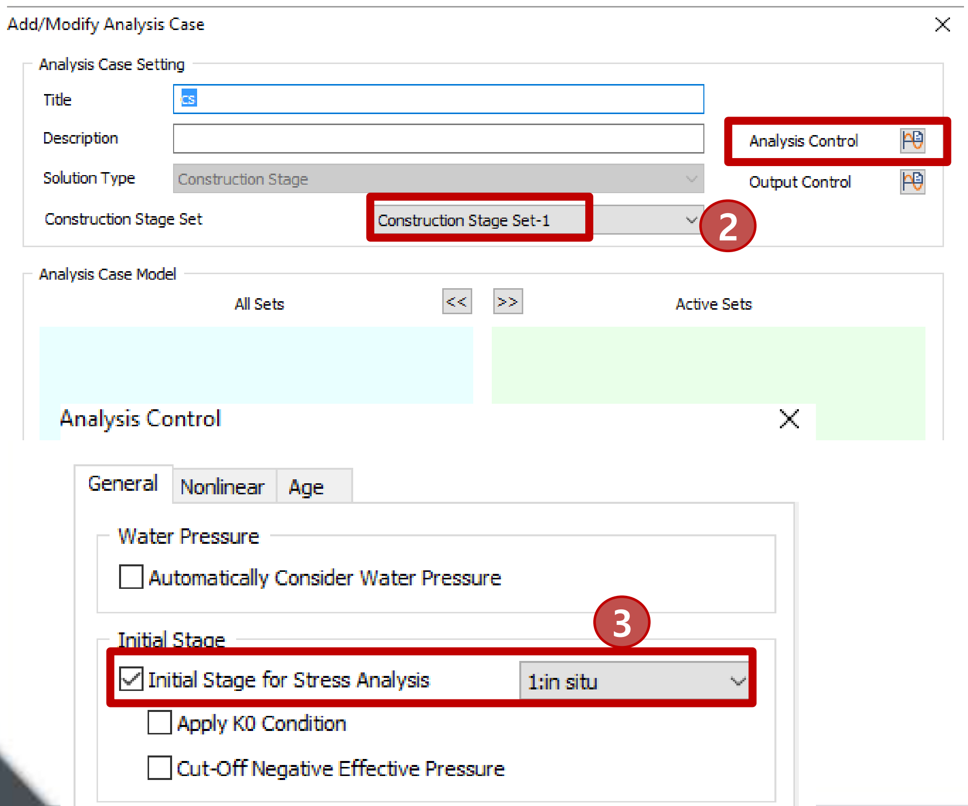
## Create and Run CS Case

1. Menu > Analysis > General
2. Create Construction Case
   Analysis Control > Initialize Stress > In situ (Stage 1)
3. Menu > Analysis > Perform
4. Press OK







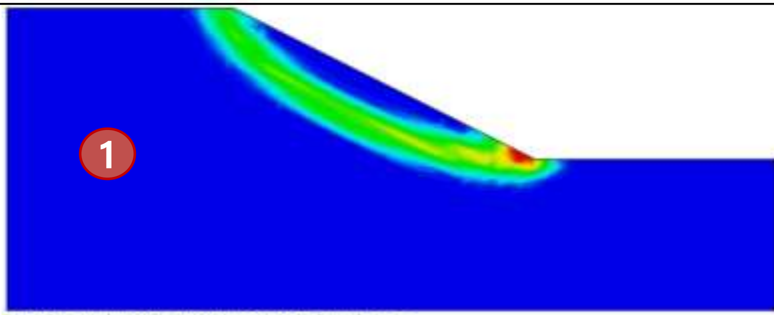



# Results

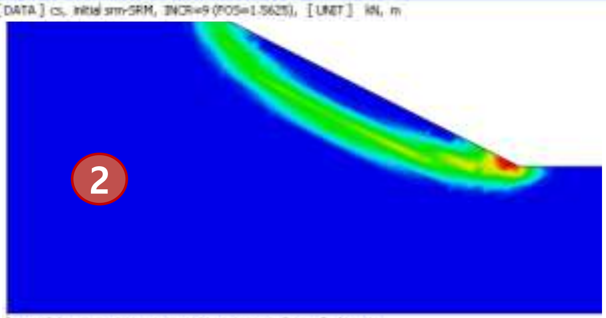
GTS NX

## Check the FOS for the SRM stages

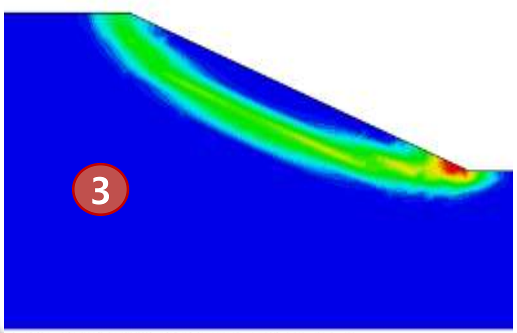
1. Results > initial srm > Plane Strain Strains > E-equivalent
2. Repeat for all other cases



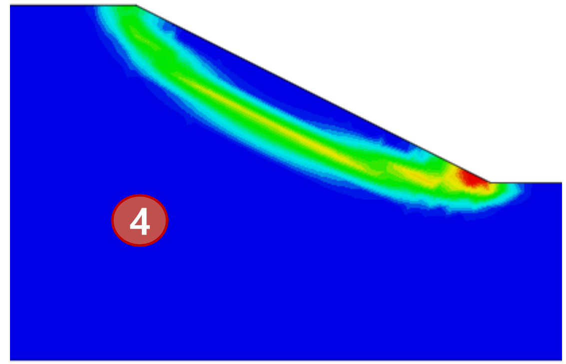

[DATA] cs, initial srm-SRM, INCR=9 (FOS=1.5625), [UNIT] kN, m



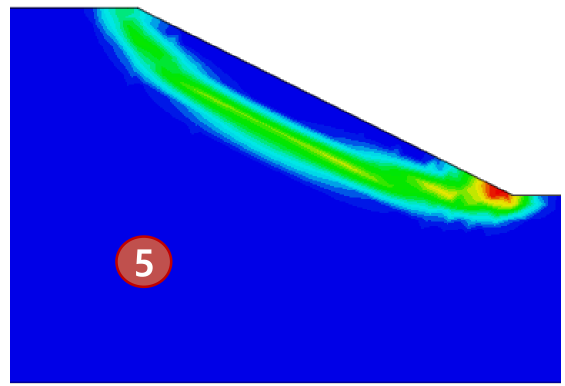

[DATA] cs, 1.2-SRM, INCR=10 (FOS=1.2906), [UNIT] kN, m



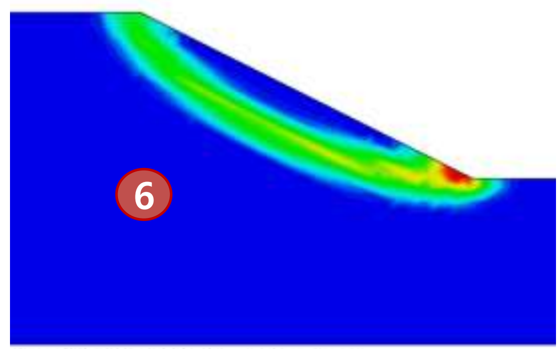

s, 1.3-SRM, INCR=8 (FOS=1.1945), [UNIT] kN, m




;, 1.4-SRM, INCR=8 (FOS=1.1063), [UNIT] kN, m




1.5-SRM, INCR=5 (FOS=1.0297), [UNIT] kN, m



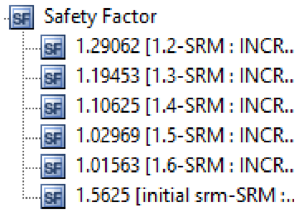

cs, 1.6-SRM, INCR=5 (FOS=1.0156), [UNIT] kN, m

| Case | FOS Result |
|---|---|
| Initial | 1.5625 |
| 1.2 SRM | 1.2906 |
| 1.3 SRM | 1.1945 |
| 1.4 SRM | 1.1062 |
| 1.5 SRM | 1.0297 |
| 1.6 SRM | 1.0156 |

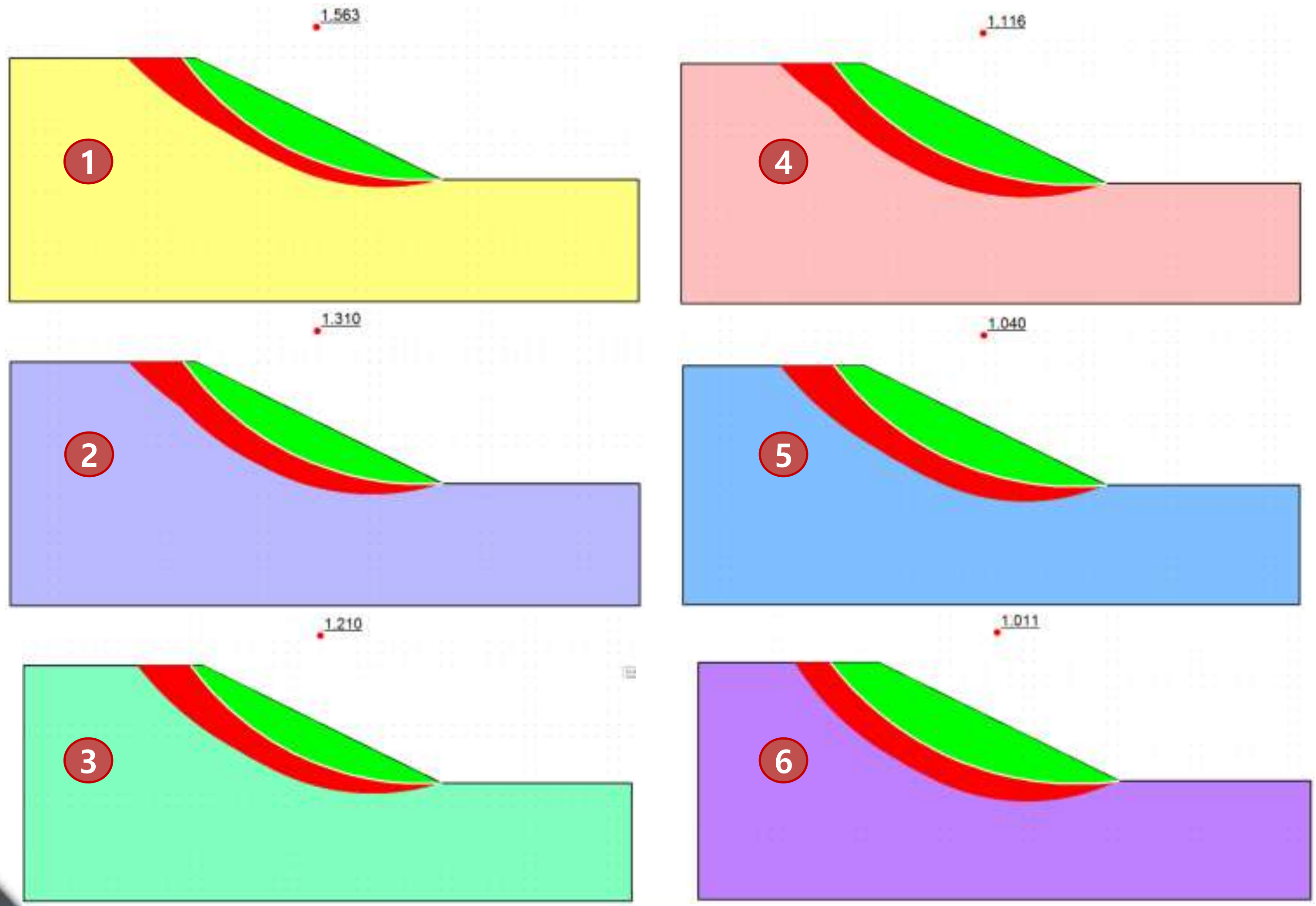
SF Safety Factor
- SF 1.29062 [1.2-SRM : INCR..
- SF 1.19453 [1.3-SRM : INCR..
- SF 1.10625 [1.4-SRM : INCR..
- SF 1.02969 [1.5-SRM : INCR..
- SF 1.01563 [1.6-SRM : INCR..
- SF 1.5625 [initial srm-SRM :..

# Compare Results to SLOPEW Tutorial

GTS NX



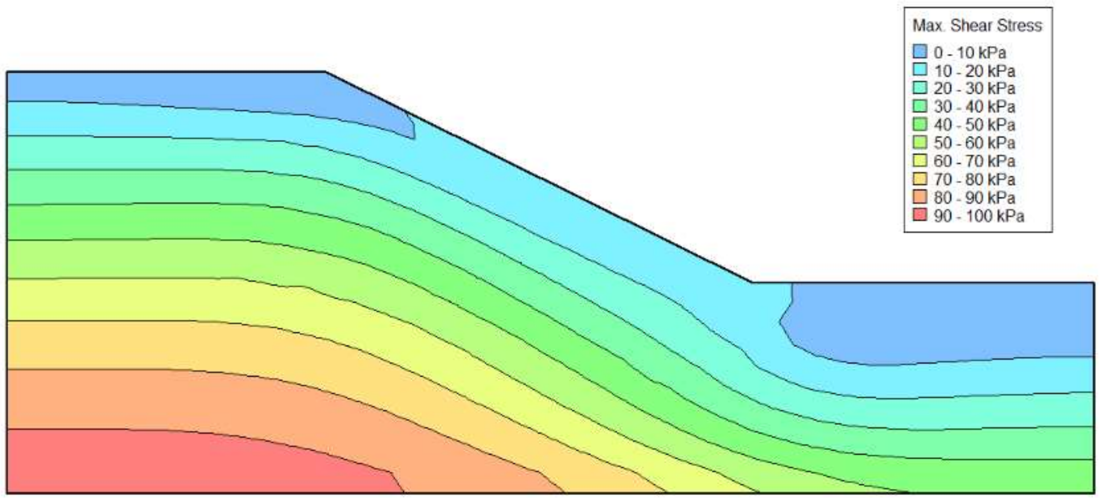

Results match SLOPEW tutorial. Shown here are FOS for all 6 cases.

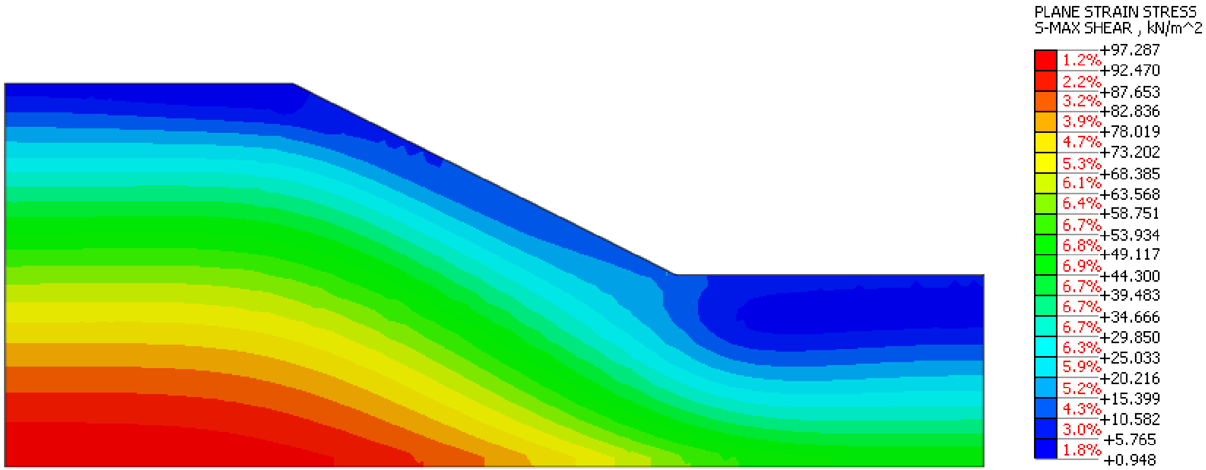


# Compare Results to SLOPEW Tutorial



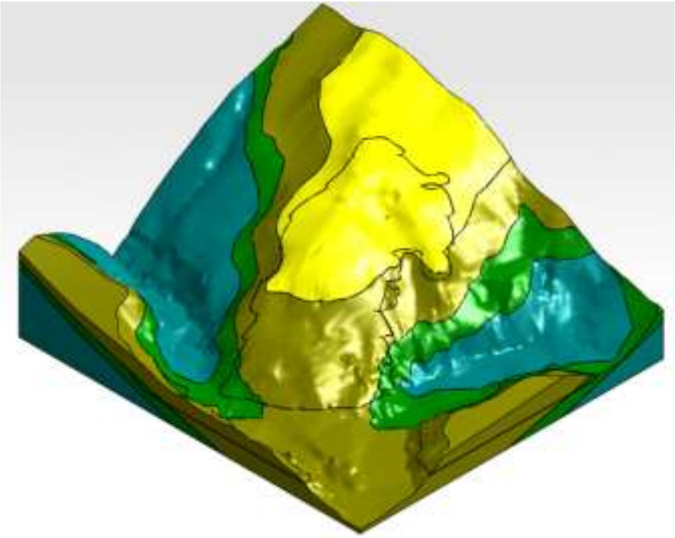
**Results match SLOPEW tutorial. Compare Shear Stresses for In Situ case**



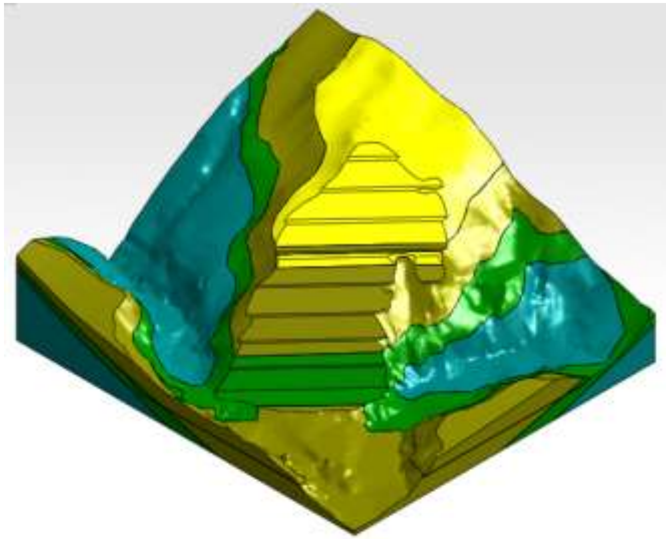



# 3D Case Study: Slope Stability (Colombia)

**Construction stage excavation of slope for housing hydroelectric dam machinery**

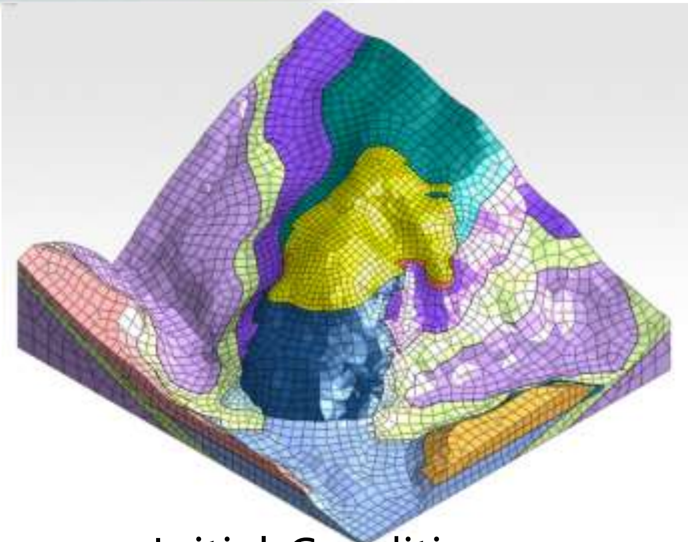


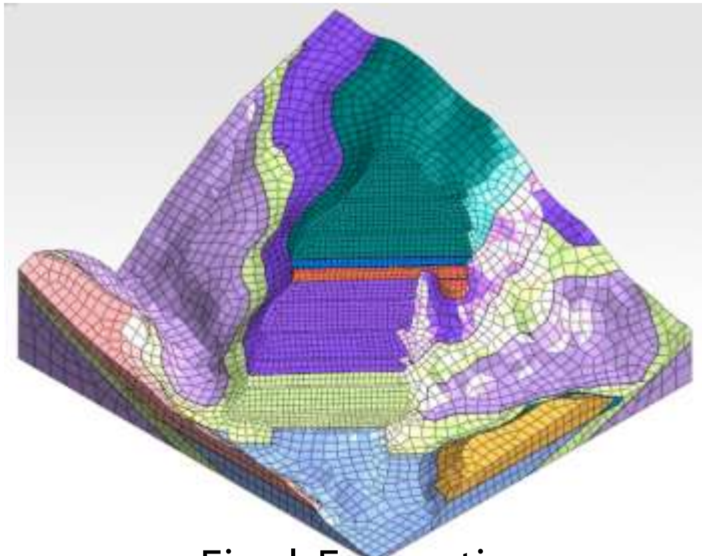
Initial Condition



Final Excavation

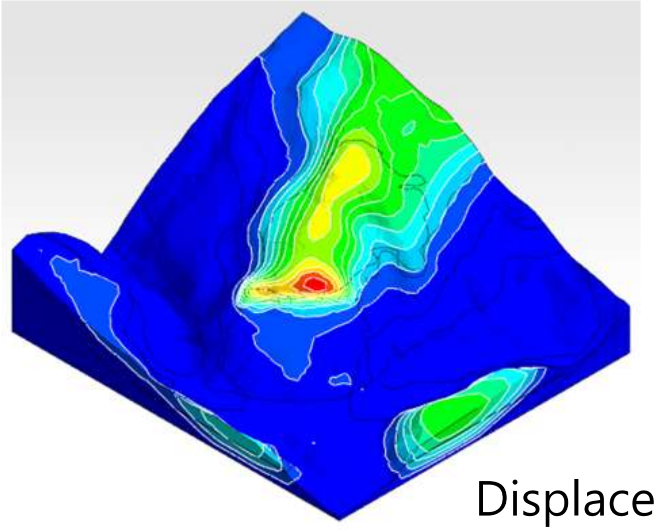
# 3D Case Study: Slope Stability (Colombia)

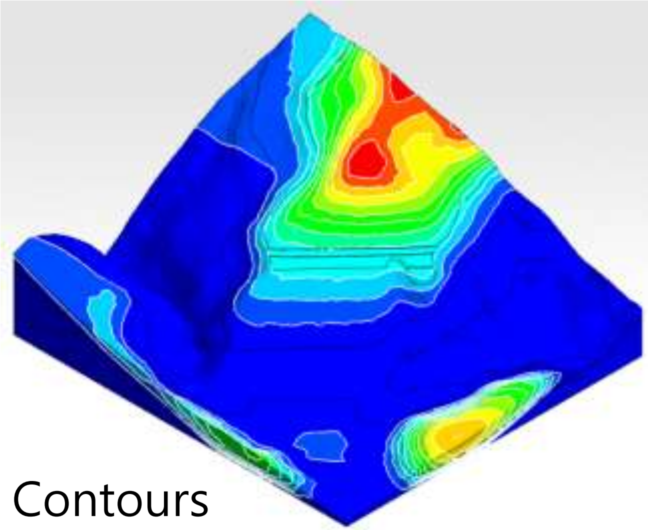




Initial Condition

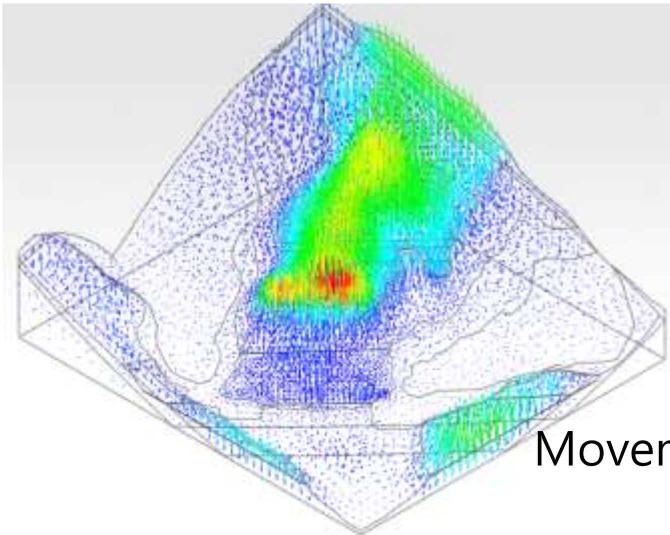
Final Excavation

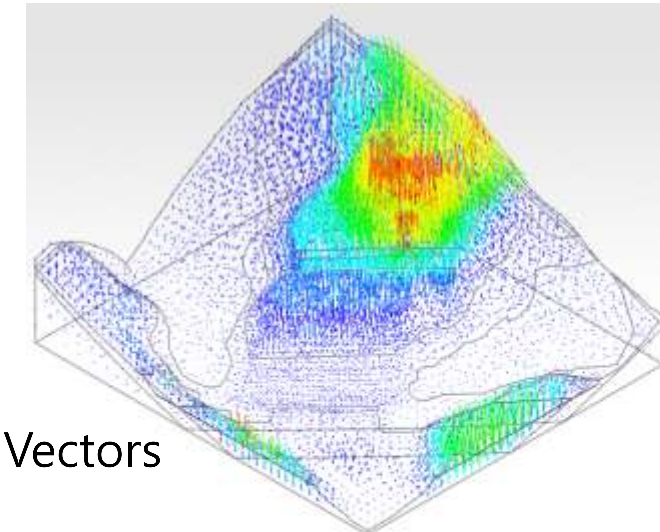




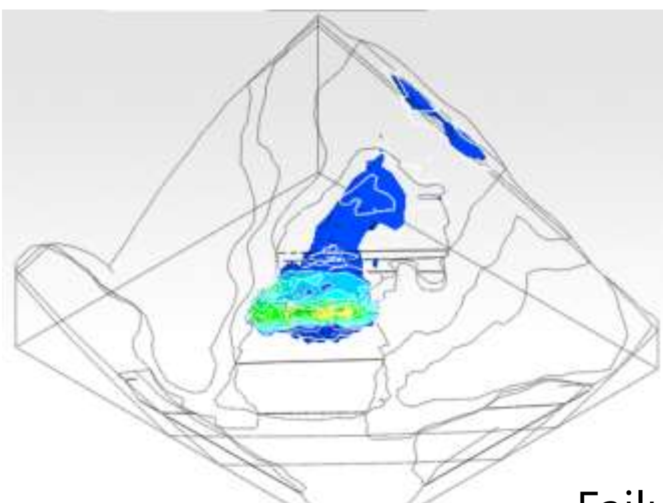
Displacement Contours

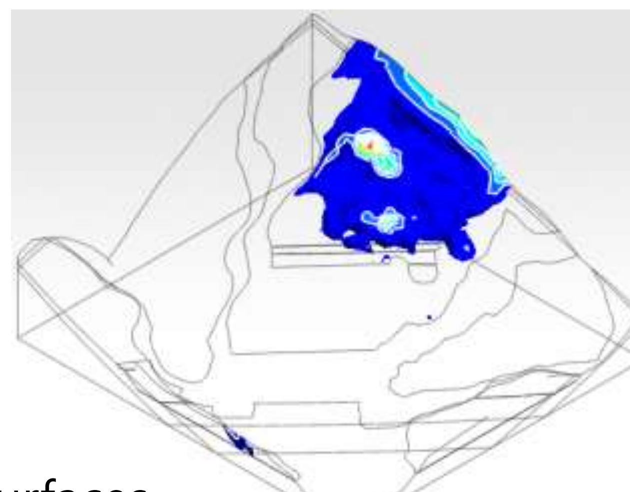
# 3D Case Study: Slope Stability (Colombia)





Movement Vectors





Initial Condition

Failure Iso Surfaces

Final Excavation

# Conclusions



- Comparison between GTS NX and Geostudio 2D Suite gives comparable results for the different kinds of analysis.
- GTS NX can additionally jump to 3D case from 2D starting file, but only practical for symmetrical cases.
- GTS NX has geometry modeling features and CAD compatibility that make it easy to create analysis cases for complex 3D problems.
- GTS NX allows coupled (stress – seepage – slope – seismic) analysis that would require multiple models in different platforms of Geostudio.

# Next Sesion

GTS NX

# MIDAS GTS NX TRANSITION FOR AMEC: 3D FINITE ELEMENT ANALYSIS

## AMEC SESSION: COMPARING 2-D AND 3-D FOR A RETAINING WALL

THIS IS AN EXCLUSIVE TWO-PART SERIES FOR AMEC ENGINEERS ONLY THAT WILL EXAMINE VARIOUS GEOTECHNICAL PROJECTS COMMON TO NORTH AMERICAN OFFICES. ATTENDEES WILL SEE A COMPARISON BETWEEN MIDAS GTS NX AND GEOSLOPE'S GEOSTUDIO SUITE FOR PUBLISHED TUTORIALS. THE SECOND AMEC SESSION TOPIC WILL BE: COMPARING 2-D AND 3-D ANALYSES FOR A RETAINING WALL.

**AMEC Session: Comparing 2-D and 3-D for a Retaining Wall**
**August 31st 2016 12:00 pm - 1:00 pm EDT**

List of Sessions | Register

### 3-D with midas GTS NX for Retaining Walls

Many Amec Foster Wheeler engineers are using GeoStudio Suite software from GeoSlope for 2D geotechnical analyses. Recent advances in computer modeling and client expectations have increased the need for state-of-the-art 3D capability. To meet this need, Amec Foster Wheeler has adopted midas GTS NX as the standard for 3D finite element analysis. The software will soon be available to all Amec Foster Wheeler geotechnical engineers. This session will show how midas GTS NX compares to software that Amec Foster Wheeler engineers currently use.

The session will show the engineers of AMEC the difference and steps of recreating a 2-D project into a 3-D project. This is a live project by AMEC in St Louis, MO for considering the retaining walls of a bridge rehabilitation and widening. There is consideration of seepage involved as well. GTS NX will be used to replicate this project in 3-D will accurate results and better graphical interface. AMEC engineers can see the steps to accomplish this project in midas GTS NX