

### GEOSTUDIO Tutorials Results and Procedure Comparison

#### Angel Francisco Martinez

Application Engineer MIDAS IT Estados Unidos



Geo-Technical analysis System New eXperience







- 1. SIGMAW: Stress Bulbs beneath tank
- 2. SEEPW: Seepage through embankment
- 3. SLOPEW: Strength reduction comparison





## Stress Changes in the Ground Beneath a Round Tank

#### (SIGMA W Comparison)

#### Angel Francisco Martinez

Application Engineer MIDAS IT Estados Unidos









### 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 twodimensional 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**

#### **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.

GTS NX

# Symmetry Axis



### **Start Axisymetric File**

#### **Open New**

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

| () |                       |   |
|----|-----------------------|---|
|    | New (Ctrl+N)          | C |
|    | Create a new document |   |

| Analysis Setting        | ×                        |
|-------------------------|--------------------------|
| Project Title           | Engineer                 |
| Desc.                   |                          |
| Model Type              | Gravity Direction        |
| ⊖ 3D                    | Y                        |
| 0.20                    | ⊖z                       |
| 2 • Axisymmetric        |                          |
| Unit System             |                          |
| kN ~ m                  | ~ sec ~ 3                |
| Initial Parameters      |                          |
| Gravity Acceleration(g) | 9.806 m/sec <sup>2</sup> |
| Unit Weight of Water    | 9.80665 kN/m³            |
| Initial Temperature     | 0 [T]                    |
| Plane Strain Thickness  | 1 m                      |
|                         | OK Cancel                |

### **Materials and Properties**

#### **Define Materials and Properties**

Add/Modify Material

Material

ID

Model Type

Define material properties of the 2 layers

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



GTS NX

Upper soil: E = 3000 kPa Poisson's ratio v = 0.45Lower soil: E = 4000 kPaPoisson's ratio v = 0.45



#### Create... No Name Type Isotropic-Elastic 1 upper soil Isotropic 2 lower soil Isotropic-Elastic 0-11-1-1-1 2 lower soil Color Name Name upper soil Color odel Type $\checkmark$ Elastic Elastic General Time Dependent General Time Dependent Elastic Modulus (E) 4000 kN/m<sup>2</sup> Elastic Modulus (E) 3000 kN/m<sup>2</sup> Inc. of Elastic Modulus 0 kN/m³ Inc. of Elastic Modulus 0 kN/m<sup>3</sup> Inc. of Elastic Modulus Ref. Height 0 m Inc. of Elastic Modulus Ref. Height 0 m 0.45 Poisson's Ratio (Nu) 0.45 Poisson's Ratio (Nu) Unit Weight (Gamma) 20 kN/m<sup>3</sup> Unit Weight (Gamma) 20 kN/m³

Х

Create/Modify 2D Property



#### **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<sup>st</sup> Location as 0,0 press Apply
- 5. Set 2<sup>nd</sup> location as 38, 25



| 2 hber   | O Automa                                | atic Setting                            | 1 m      |
|----------|---|---|----------|
|          | OUser D                                 | efined                                  | 40 🚔     |
| Location |   |   |          |
|          | ○ 1Q                                    | ○ 2Q ○                                  | 3Q () 4Q |
| 👳 🥖      | O                                       | Cance                                   | Apply    |
| <br>     |   | <u></u>                                 |          |
|          |   |   |          |
|          |   |   |          |
|          |   |   |          |
|          |   |   |          |
|          |   |   |          |
|          |   |   |          |
|          | · • • • • • • • • • • • • • • • • • • • |   |          |
|          | ·····                                   | · • · · · · · · · · · · · · · · · · · · |          |
|          |   |   |          |
|          |   |   |          |
|          |   |   |          |
|          |   |   |          |
|          |   | · • · · · • · · · · · · · · · · · · · · |          |
|          | · • · · · · · · · · · · · · · · · · · · |   |          |
|          |   |   |          |
|          |   |   |          |
|          |   |   |          |
|          |   |   |          |
|          |   |   |          |
|          |   |   | Y        |
|          |   |   |          |
|          |   |   |          |
|          |   | · • · · · • · · · · · · · · · · · · · · |          |
|          |   |   |          |

+ tw tg

# Show/Hide Grid

田田

Define Grid

Size

2

GTS NX

Ha Define Grid

×

#### **Create Geometry Outline**

Draw additional lines to divide model into 4 segments

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







GTS NX



3

### **Mesh Size Control**

#### **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







### Mesh

#### 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



|  |        | 2D 2D->3D<br>Conversion of the second s |
|--|--------|--|
|  |        | 2D (F7)  |
| dvanced Option<br>Merge Nodes<br>Tolerance | 0.     | .0001  |
| Element Size Growth Rate -                 | Coarse | 1  |
| Min/Max Element Size                       | Large  | 1  |
| 2D Mesher<br>Delaunay Mesher               |        | ~  |
| Element Type<br>Quadrilateral              |        | ~  |
| Higher-Order Element                       |        |  |



### Boundary

#### GTS NX

#### **Boundary**

#### **Apply Boundary Conditions**

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







### Load

#### 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.





### **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



| d/Modify Analys                 | is Case   |   |   | × | Ratch Analysis        |
|---------------------------------|---|---|---|---|-----------------------|
| - Analysis Case Se              | tting   |   |   |   | S Out thay as         |
| Title                           | linear  |   |   |   | Perform Modeling      |
| Description                     |   |   | Analysis Control                                | P | 🔒 Results             |
| Solution Type                   | Linear Static   |   | <ul> <li>Output Control</li> </ul>              | P | Analysis              |
| Construction Sta                | age Set   |   | $\sim$  |   | GTS NX Solver         |
|                                 |   |   |   |   | Name Type Description |
| Analysis Case Mo                | All Sets  | << >> 3   | Active Sets                                     |   | linear Linear Static  |
| Mesh     Defa     lowe     lowe | ault Mesh Set<br>erleft<br>erright                                | Mesh     Default     lower le     lower le     lower le   | Mesh Set<br>sft<br>ght                          |   |                       |
| Boundar                         | er lett<br>er middle<br>er right<br>ry Condition<br>Indary, Set-1 | upper i<br>upper i<br>upper i<br>upper i<br>Se Boundary C | ent<br>niddle<br>ight<br>Condition<br>anv Set-1 |   | Check On/Off OK       |
| Elocation Location              | bad<br>d Set-1<br>Pair  | Static Load   | et-1  |   |                       |

#### **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





#### GTS NX

sec

#### **Post Processor**

#### **Check Displacements**

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





#### **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)







### **Compare Results to SIGMAW Tutorial**

**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.



Max:76.8338 0.4%+72.314 0.8%+67.794 0.7%+63.275 0.9%+58.755 0.9%+58.755 0.9%+54.236

GTS NX



#### MIDAS GTS NX

19

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





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<br>(m)    | Wn (%) | LL (%) | IP(%) | Peso unitario<br>total γt<br>(kN/m3) | Es (MPa)<br>Valor<br>conservador | е   | Presión de<br>preconsolidación<br>(kPa) | OCR | Сс   | Cr   | K (m/día) | c' (kPa) | ¢,(ō) |
| 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)







Plane cut of layers



Initial CS stage



Stage1: Excavation







21

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



MT 830123204-8





Stage1: Excavation



10 year consolidation

Conscient 

20 year consolidation



50 year consolidation

### Seepage Thru Earth Embankment

#### (SEEPW Comparison)

#### Angel Francisco Martinez

Application Engineer MIDAS IT Estados Unidos









### Introduction

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**

#### **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**

#### **Open New**

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



| Analysis Setting        | ×                        |
|-------------------------|--------------------------|
| Project Title           | Engineer                 |
| Desc.                   |                          |
| Model Type              | Gravity Direction        |
| () 3D                   | ● Y                      |
|                         | ⊖z                       |
| () Axisymmetric         |                          |
| Unit System             |                          |
|                         |                          |
| kN ~ m                  | <u> </u>                 |
| Initial Parameters      |                          |
| Gravity Acceleration(g) | 9.806 m/sec <sup>2</sup> |
| Unit Weight of Water    | 9.80665 kN/m³            |
| Initial Temperature     | 0 [T]                    |
| Plane Strain Thickness  | 1 m                      |
|                         |                          |
|                         | OK Cancel                |

### Materials

|   | 1 Geometry Mesh  | 🔊 core X10   |
|---|--|--|
| Define Materials  | Comp. Pro  | ы core X100  |
|   | Material Property  | 🔊 silt k function  |
| 1. Menu > Mesh > Material> Create> Isotropic                      | Function *   | 🔊 water content  |
| 2. Define General Fill material as shown for the General tab.     | Prop./CSys./Func   |  |
| 2. In the Demonstrate setting Uncertained and                     |  |  |
| 3. In the Porous tab, active Unsaturated                          | Maine Suction Kx (m/sec)<br>0.1 1.00E-05   | Matric Suction (kPa) VWC (m <sup>3</sup> /m <sup>2</sup>       |
| Property>ADD>Individual>User Defined and copy paste               | 0.14384499 9.96E-06  | 0.16237767 0.449683  |
| corresponding excel shoots Press Ok                               | 0.20691381 9.91E-06<br>0.29763514 9.82E-06   | 0.28366509 0.449748  |
| corresponding excersivels. Fress OK                               | 0.42813324 9.69E-06  | 0.6951928 0.448796   |
| 4. Repeat for core 10X less and core 100X less materials          | 0.61564821 9.50E-06  | 1.1288379 0.44738  |
|   | 1 274275 8.79E-06  | 2.9763514 0.437809   |
|   | 1.8329807 8.18E.06   | 4.8329302 0.424379<br>7.8475997 0.398415                       |
| No Name Type Create 🔻   | 2.6366509 7.30E.06<br>3.7926902 6.10E.06   | 12.74275 0.353926  |
| 1 granular toe Isotropic-Elastic Isotropic                        | × 5.4555948 4.58E-06   | 20.691381 0.29121<br>33.598183 0.222567                        |
| 2 general fill Isotropic-Elastic Orthotrop                        | 7 8475997 2 92E 06   | 54 555948 0 163773   |
| 3 core X10 less Isotropic-Elastic 2D Equival                      | 11.266579 1.472-06   | 88.586679 0.121111<br>143.84499 0.092185                       |
| 4 Core 100 X less Tsoropic-Elastic                                | 23.357215 1.58E-07   | 233.57215 0.072563   |
| General Porous Time Dependent                                     | 33.598183 3.63E-08<br>49.326302 7.28E.09   | 379.26902 0.058798<br>615.84821 0.048704                       |
|   | 69.51928 1.36E-09  | 1,000 0.040948   |
| aterial X Unit Veget (Saturated ) 44 Mum                          | 100 2.44E-10   |  |
| ID 2 Name general fill Color Initial Void Ratio (eo) 0.1          | Add/Molify Unsaturated Fairthon  | ×  |
| Unsaturated Property silk function                                | Parkter Here     Sole Pactor 2 mobile Deph Op     elis function     E     Oth even ine write | ten Wate Canterion Option<br>IV ans kan under ⊡t ans kan under |
| Model Type Elastic V Structure Drainage Parameters                | Permediative Function Colo   | and an all and an all and an                                   |
| General Porous Time Dependent Drained                             | Punction Type: Liber Sethind:  |  |
| Elastic Modulus (E) 50000 kN/m <sup>2</sup>                       | Postsare<br>(004/m <sup>2</sup> ) K.Rata <sup>Pr</sup>                                       |  |
| Tor of Elactic Modulus 0, 17625087                                | 2 0.1438 0.9000 4 Hot  |  |
| Inc. of Edde Products 0 KV/m <sup>3</sup>                         | 3 0.2053 0.5050 Here<br>4 0.2975 0.0000 Here   |  |
| Inc. of Elastic Modulus Ref. Height 0 m Permeability Coefficients | 5 0.4281 0.0000<br>8 0.418 0.0000  |  |
| Poisson's Ratio (Nu) 0.3 kx ky kz                                 |  |  |
| Unit Weight (Gamma) 20 kV/m <sup>3</sup> 1 1 1 m/sec              | Paraton Type 1366r dather  |  |
| Initial Stress Parameters   | Pressure Wolfer = 1  |  |
| Ko Anisotropy   | 2 3.1024 3.4400 1 1.14   |  |
| Name Type Add   | 4 84281 84494 5.2<br>  |  |
| silk k function Individual ModiFy                                 | 2 0.6852 0.4468<br>E 1.1286 0.4474 = **  |  |
| 100 X Indvidual Delete  | Relies Dati  | CR CANAR 27  |

#### **Properties**

#### **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.



#### Add/Modify Property

| No | Name         | Туре | Sub-Type     | Create |
|----|--------------|------|--------------|--------|
| 1  | general fill | 2D   | Plane Strain | 1D.    |
| 2  | toe          | 2D   | Plane Strain | 2D.    |
| 3  | 10X          | 2D   | Plane Strain | 20     |
| 4  | 100X         | 2D   | Plane Strain | 30.    |

#### Create/Modify 2D Property

Х

Ŧ

.

2



#### GTS NX

×

1. 2.

3.

4.

5.

#### **Create Geometry Outline** 日本 19 19 一世 日 2 Define the WORK GRID as a drawing plane with lines H Show/Hide Grid Ha Define Grid spaced 1m apart to create rectangles Define Grid Menu > SHOW GRID Menu > Define Grid > Width 1m, User Define 55 m Size Menu>Geometry > PolyLine 1 m Width Select the ABS x,y method, set 1<sup>st</sup> Location as 0,0 Automatic Setting Number press Apply OUser Defined 2 55 🚔 Set End location as 23, 10 Location PolyLine $\times$ () 1Q $\bigcirc 2Q$ O 3Q O 4Q 2D 3D B Apply Cancel 23, 10 Location Point & C 5 REL dx, dy $\sim$ Method 💰 🐟 Polyline Make Face Link Start and End Point Line $\times$ ~ ... Geometry Set Geometry Set-1 2D 3D ОК Apply Cancel Back Input Start Location Location 0,0 Method ABS x, y $\sim$

#### **Create Geometry Outline**

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

PolyLine



#### **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

|    | 2D 3      | 3D      |        |        |        |
|----|-----------|---------|--------|--------|--------|
|    | Input St  | art Lo  | cation |        |        |
| 2) | Location  |         |        |        | 23, 10 |
|    | Method    | AB      | Sx,y   |        | ~      |
|    | 2D 3      | ID.     |        |        |        |
| 3  | Input St  | art Loc | ation  |        |        |
|    | Location  |         |        |        | 27, 10 |
|    | Method    | AB      | Six, y |        | $\sim$ |
|    | Rectangle |         |        |        | ×      |
| 4  | 2D        |         |        |        |        |
|    | – Meth    | od      | 1      | $\Box$ |        |
|    | Input O   | ne Corr | her    |        |        |
|    | Location  | 1       |        |        | 40, 0  |
|    | Method    | ABS     | S x, y |        | $\sim$ |

| 20 3L     | )                       |          |
|-----------|-------------------------|----------|
| Input End | Location                |          |
| Location  |                         | -2, -10  |
| Method    | REL dx, dy              | ~        |
| ine       |                         | $\times$ |
| 2D 3D     |                         |          |
| Input End | Location                |          |
| Location  | -2                      | 2, -10   |
| Method    | REL dx, dy              | $\sim$   |
| Rectangle |                         | ×        |
| 2D        |                         |          |
| - Method  |                         |          |
| Input Dia | gonally Opposite Corner |          |
| Location  | 10,                     | -1       |
| Method    | REL dx, dy              | $\sim$   |



Point & Cu

Rectangle

/ Line





|    | Geo  | metr   | у     | Me | sh    | St   |
|----|------|--------|-------|----|-------|------|
| 0~ | 800  | (a)    | r     | ×  | 5     | E    |
| ĵ  | 00   | Ø      |       | ê  | 10    | E    |
|    | Poin | it & ( | Curve | 1  |       |      |
| e  | 10   | -      | Ċ     | X  | Inter | sect |

| ntersect     |                     | ×     |
|--------------|---------------------|-------|
| 3D           |                     |       |
|              | Selected 7 Curve(s) |       |
| Geometry Set | Geometry Set-1      | ~     |
| 🐺 🕒 📫        | OK Cancel           | Apply |
|              | 5                   |       |

### Mesh

#### GTS NX

#### 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





### **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 Om
- 4. Select right Edge, apply Review boundary



1



### **Analysis Case**

toe base

#### 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







### **Analysis Case**



- 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



#### GTS NX Solver

 $\land$ Description Name Type • Case 1 Homogen Seepage(Stea 2 • Case 2 Dam with t Seepage(Stea 2 Case 3 Core with Seepage(Stea Case 4 Core with Seepage(Stea  $\mathbf{\nabla}$ Check On/Off ОК Cancel

 $\times$ 









### **Compare Results to SEEPW Tutorial**

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

GS E&C



Stage2: Banking Height 160m



Stage5: Banking Height 210m + Hydrostatic pressure

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



Stage 2: Total Displacements



Stage 4: Vertical deflection



Stage 5: Horizontal Displacements



#### Stage 5: Hydrostatic pressure

GS E&C

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



Stage 5: Total Head





S GS E&C

+9.756

+6.081

+4.921

+4.238

+3.723

+3.264

+2.859

+2.498

+2.175

+1.628

+1.397

+1.194

+1.016

+0.858 +0.714

+0.586

f0.477

f0.371

f0.236

#### Stage 5: Hydraulic Gradient



# Strength Reduction Stability Comparison (SLOPEW Comparison)

#### Angel Francisco Martinez

Application Engineer MIDAS IT Estados Unidos







### 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 stressstrain 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**

#### **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**

#### **Open New**

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



| Analysis Setting        | ×                        |
|-------------------------|--------------------------|
| Project Title           | Engineer                 |
| Desc.                   |                          |
| Model Type              | Gravity Direction        |
| () 3D                   | €Y                       |
|                         | ⊖z                       |
| () Axisymmetric         |                          |
| Unit System             |                          |
|                         |                          |
| kN ~ m                  | ✓ sec ✓ 3                |
| Initial Parameters      |                          |
| Gravity Acceleration(g) | 9.806 m/sec <sup>2</sup> |
| Unit Weight of Water    | 9.80665 kN/m³            |
| Initial Temperature     | [1] 0                    |
| Plane Strain Thickness  | 1 m                      |
|                         | OK Cancel                |

### **Materials**



#### **Define Materials**

1. Menu > Mesh > Material> Create> Isotropic>Mohr Coulomb

Material

ID

6

General Porous

Cohesion (C) Inc. of Cohesion

Inc. of Cohesion Ref. Height

Frictional Angle (Phi)

Model Type

Name

Mohr-Coulomb

- 2. Define 6 materials as shown in the General tab
- 3. Define Non linear properties as shown in table

| Add/Modif                  | y Material   |  |  | ×  |
|----------------------------|--|--|--|--|
| No                         | Name   | Туре   |  | Create 💌   |
| 1<br>2<br>3<br>4<br>5<br>6 | in situ<br>srf 1.2<br>srf 1.3<br>srf 1.4<br>srf 1.5<br>srf 1.6 | Isotropic-Mohr-C<br>Isotropic-Mohr-C<br>Isotropic-Mohr-C<br>Isotropic-Mohr-C<br>Isotropic-Mohr-C<br>Isotropic-Mohr-C | oulomb<br>oulomb<br>oulomb<br>oulomb<br>oulomb | Isotropic<br>Orthotrop<br>2D Equiva<br>Interface a<br>Import |
| ID ID                      | 1 Name   | in situ  | Cold   | r 🔽  |
| Model T                    | ype Mohr-Co  | oulomb   | ~  | Structure  |
| Gener                      | Porous Non   | -Linear Time Depe  | endent   |  |
| Elas                       | tic Modulus (E)  |  | 10000  | kN/m²  |
| Inc.                       | of Elastic Modul   | JS   | 0  | kN/m³  |
| Inc.                       | of Elastic Modul   | us Ref. Height   | 0  | ] m  |
| Pois                       | son's Ratio (Nu)   |  | 0.3334   | ]  |
| Unit                       | Weight (Gamma  | )  | 20   | kN/m³  |
|                            |  |  |  |  |

|                    |       | Name           | Cohesion | Friction<br>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             |    |  |  |  |  |  |
|                    | 3     | SRF 1.6        | 3.2      | 18.78             |    |  |  |  |  |  |
| srf 1.6<br>mb      | Color | Struct         |          |                   |    |  |  |  |  |  |
| ear Time Dependent | t     |                |          |                   |    |  |  |  |  |  |
|                    | 3.2   | kN/m²<br>kN/m³ |          |                   |    |  |  |  |  |  |
| ight               | 0     | m<br>[deg]     |          |                   | 49 |  |  |  |  |  |

Mesh

Comp. Prc ↔ CSys

Function \*

Prop./CSvs./Func

Geometry

Material Property

#### **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.



GTS NX

#### Add/Modify Property

| No | Name    | Туре | Sub-Type     | Create 🔻 |
|----|---------|------|--------------|----------|
| 1  | in situ | 2D   | Plane Strain | Modify   |
| 2  | srf 1.2 | 2D   | Plane Strain | Hourym   |
| 3  | srf 1.3 | 2D   | Plane Strain | Сору     |
| 4  | srf 1.4 | 2D   | Plane Strain |          |
| 5  | srf 1.5 | 2D   | Plane Strain | Delete   |
| 6  | srf 1.6 | 2D   | Plane Strain | Import   |

Х

| eate/Modify 2 | O Property  |   | ×  |
|---------------|---|---|--|
| Plot Only(    | 2D) G   | auging Shell  | Axisymmetric   |
| Shell         | Plane Stress  | Plane Strain  | Geogrid(2D)  |
| ID 7          | Name in sit   | u   | Color  |
| Material      | 3   | 1: in situ  | ~ 🙋  |
| -Material C   | Sys   |   |  |
| (             | CSys  | Global Rectangula   | ar ~   |
| (             | Angle   |   | 0 [dea]  |
|               | Plot Only(<br>Shell<br>ID 7<br>Material<br>Material C | Plot Only(2D) G<br>Shell Plane Stress<br>ID 7 Name in sit<br>Material 3<br>Material CSys<br>© CSys<br>O Angle | Plot Only(2D) Gauging Shell<br>Shell Plane Stress Plane Strain<br>ID 7 Name in situl<br>Material 3 1: in situ<br>Material CSys<br>© CSys Global Rectangula |

#### **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 1<sup>st</sup> Location as 0,0 press Apply
- 5. Set End location as 0, 20

| SS.                              | S.  |   | Pol   | Line   |   |   |   |
|----------------------------------|---|---|---|--|---|---|---|
|                                  | 3   |   | 2   | D 3D   |   |   |   |
| Poinc                            | ne  | 5   |   | Input Nex<br>Location  | t Location (RB to   | Stop)   | 0, 20   |
| 2D 3D                            |   | ,   |   | Method   | REL dx, dy  |   | ~   |
| Input Star<br>Location<br>Method | t Location<br>ABS x, y  | 0,0   | Ge  | Make F<br>ometry Sei   | ace Link Sta<br>t Geometry Set  | -1<br>Cancel  | Appl  |
|                                  | C Point<br>Point<br>C Polyli<br>ne<br>2D 3D<br>Input Star<br>Location<br>Method | Point & C   Point & C | Point & C   Point & C | Point & C<br>Point & C<br>Point & C<br>Point & C<br>S<br>Point & C<br>S<br>Point & C<br>S<br>Point & C<br>S<br>S<br>Point & C<br>S<br>S<br>S<br>S<br>S<br>S<br>S<br>S<br>S<br>S<br>S<br>S<br>S | PolyLine<br>Point & C<br>Point & C<br>Point & C<br>Point & C<br>Point & C<br>Point & C<br>PolyLine<br>1<br>PolyLine<br>1<br>PolyLine<br>1<br>PolyLine<br>1<br>Dual<br>PolyLine<br>1<br>Dual<br>PolyLine<br>S<br>PolyLine<br>S<br>PolyLine<br>S<br>PolyLine<br>Method<br>Make F<br>Geometry Se<br>Method<br>ABS x, y | Point & C   Point & C | Point & C   Point & C |

| 開中<br>弾<br>Show | Hide Grid |         | ×                   | a # g<br>⊞ <sub>g</sub> Defi | ne Grid X    |
|-----------------|-----------|---------|---------------------|------------------------------|--------------|
|                 | Size      |         |                     |                              |              |
|                 | Width     |         |                     |                              | 1 m          |
|                 | Number    | O Autor | matic Set<br>Define | ting<br>255                  | ; <b>_</b>   |
|                 | Location  |         |                     |                              |              |
| $\times$        |           | () 1Q   | () 2Q               | ○ 3Q                         | <b>○ 4</b> Q |
|                 | 👳 🥔       |         | ОК                  | Cancel                       | Apply        |
| 0, 20<br>~ .    |           |         |                     |                              |              |

(1)

| Polyline points   | 20  |
|---|---|
| <ol> <li>Input next location (15,0)</li> <li>Input next location (20, -10)</li> <li>Input next location (16, 0)</li> <li>Input next location (0, -10)</li> <li>Input next location (-51, 0)</li> <li>Press OK</li> </ol>  | 2D     3D       Input Next Location (RB to Stop)       Location       Method       REL dx, dy |
| PolyLine × 2D 3D Input Next Location (RB to Stop)   |   |
| Image: Decision of the second seco | 2D 3D<br>Input Next Location (RB to Stop)   |
| 2D 3D<br>Input Next Location (RB to Stop)<br>Location 20, -10<br>Method REL dx, dy  | Location  |
| PolyLine  | Geometry Set Geometry Set-1  Back OK Cancel   |
| 3 Input Next Location (RB to Stop)<br>Location 16, 0<br>Method REL dx dy  |   |
| THE UNITY OF  |   |

PolyLine

GTS NX

52

 $\times$ 

0, -10

 $\sim$ 

 $\times$ 

-51, 0

 $\sim$ 

~ ...

Apply

### Mesh

#### 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



| Advanced Option                                  | × |
|--|---|
| Merge Nodes                                      |   |
| Tolerance 0.0001                                 | ] |
| Element Size Growth Rate                         |   |
| Fine Coarse                                      |   |
| Min/Max Element Size                             |   |
| Small Large                                      |   |
| 2D Mesher  |   |
| Delaunay Mesher 🗸 🗸                              |   |
| Element Type<br>Tri+Quad ~                       |   |
| Higher-Order Element                             |   |
| Pattern Mesh<br>Register Each Mesh Independently |   |
| OK Cancel  |   |
| ∃… 🗖 🌐 Mesh                                      |   |
| - 🗍 🌐 Default Mesh Set                           | 1 |
| ground   | 2 |
|  | 5 |
| 1.4  | 6 |
|  | 7 |
|  | 8 |
| □ 🗿 1.6  | 9 |
|  |   |



GTS NX

### Boundary



### **Construction Stages**

#### **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
- Create Stage 2 and activate sets as shown. Press Save > New 4.



GTS NX

Stage Set

Auto Set

Seepage/Consolidat

Construction Stage

📾 Simulate Stage 🕍 Volum

Static/Slope Analysis

0

Stage

Wizard

Define

Contact

Contact

### **Construction Stages**

#### **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 St | tage Set Name   | Construction Stage | e Set-1                                | ~                |  |   |  |                  |   |   |
|-----------------|---|--------------------|--|------------------|--|---|--|------------------|---|---|
| Stage ID        | 3: 1.2  | ~ <b>_</b>         | Move to Previous                       | Move to Next     |  | New   | Inser  |                  |   |   |
| Stage Name      | 1.2   |                    |  |                  |  |   | Analysis   |                  |   |   |
| Stage Type      | Stress  | ~                  |  |                  |  |   | Output   |                  |   |   |
| Set Data        | .2<br>.3<br>.4<br>.5<br>.6<br>lefault Mesh Set<br>round<br>dary Condition<br>.2 pground | Activated Data     | a<br>2<br>dary Condition<br>Load<br>ct | Deactivated Data | a<br>und<br>ny Condition<br>oad                  |   | ndition<br>ine Water Le<br>0 m No<br>ine Water Le<br>Input Wate<br>LDF |                  |   |   |
| 1               |   |                    |  | 2                | Stage ID<br>Stage Name<br>Stage Type<br>Set Data | 4: 1.3<br>1.3<br>Stress<br>h<br>1.2<br>1.3<br>1.4<br>1.5<br>1.6<br>Default Mesh Set<br>ground<br>ndary Condition<br>1.2 pground | Activated Dat  | Move to Previous | Move to Next  Deactivated Data  Deactivated Data  Mesh  1.2  Boundary Condition Static Load Contact | New     Insert       Analysis (       Output C       Initial Condition       Define Water Lev       0     m       Define Water Lev       Input Water       LDF       Clear Displayment       Slope Stability(SRM) |

### **Construction Stages**

#### **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**

#### 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



| 1 | General 🥵 Setting | Analysis |
|---|-------------------|----------|
|   | Analysis Case     |          |
|   | 💰 🕏 Batch A       | nalysis  |
|   | Modeling          |          |
| 3 | Perform 🔂 Results |          |
|   | Analysis          |          |

|   | Name | Type             | Description |
|---|------|------------------|-------------|
| ~ | . C8 | Operatuation St. |             |
|   |      |                  |             |
|   |      |                  |             |
|   |      |                  |             |
|   |      |                  |             |
|   |      |                  |             |
|   |      |                  |             |
|   |      |                  |             |

#### GTS NX





s, 1.3-5RM, INCR=8 (FO5=1.1945), [UNIT] kN, m





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







### **Compare Results to SLOPEW Tutorial**



### **Compare Results to SLOPEW Tutorial**

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





GTS NX

+97.287 +92.470 +87.653 +82.836 +78.019 +73.202 +68.385 +63,568 +58.751 +53.934 +49.117 +44.300 +39.483 +34.666 +29.850 +25.033 +20.216 +15.399 +10.582 +5.765

+0.948

### 3D Case Study: Slope Stability (Colombia)





# Construction stage excavation of slope for housing hydroelectric dam machinery



### 3D Case Study: Slope Stability (Colombia)





### 3D Case Study: Slope Stability (Colombia)





### 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**



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

GTS NX

#### 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