Sequential Excavation for Mining in 2D and 3D

MIDASoft Inc.

Angel Francisco Martinez
Civil Engineer
Email: a.martinez@midasuser.com

Integrated Solver Optimized for the next generation 64-bit platform
Finite Element Solutions for Geotechnical Engineering
Contents

Part 1. Objectives
Part 2. Introduction
Part 3. 2D case
Part 4. 3D case
Part 5. Conclusion

Integrated Solver Optimized for the next generation 64-bit platform
Finite Element Solutions for Geotechnical Engineering
Objectives:

- Concepts of Modeling Excavation by stages
- Reinforcements
- Interface Elements
- Results Interpretation in 2D vs 3D
Contents

Part 1. Objectives

Part 2. Introduction

Part 3. 2D case

Part 4. 3D case

Part 5. Conclusion
MIDAS Products

midas Gen
Integrated Design System for Building and General Structures

midas Civil
Integrated Solution System for Bridge and Civil Structures

midas Design+
Structural Component Design & Detailing

midas FEA
Advanced Nonlinear and Detail Analysis System

midas GTS NX
2D / 3D Geotechnical and Tunnel analysis System

SoilWorks
Geotechnical Solutions for Practical Design

www.midasgtsnx.com
In sequential excavation, the stability analysis of a shoring wall, slope and the effects on the adjacent ground and structures are crucial.

Accurate ground material properties and geometry must be reflected. In addition, the analytical model needs to closely reflect the excavation sequence.

Stage Analysis by FEM
The analysis is carried out by the FEM and is applied to large scale, deep excavations in the design and construction of shoring wall and support systems. This method calculates displacements and stresses reflecting the elasto-plastic behavior of ground and interaction with the shoring wall and support structures as well as the deformations in the surrounding ground. Slope stability by strength reduction method is also necessary.
Wall-Interface-Ground Modeling

Interface elements account for normal and shear directional relative displacement and interface traction that simulate face/face and line/line behavior.

Interface function works in the following way. Right after the interface elements are generated, connected nodes are automatically detached at the spots of the interface. And, in between of the detached nodes, it creates a kind of elements which have specific rigidity in normal and tangent directions.

The objective is to closely reflect the behavior between the ground and wall surfaces by separating the behavior of the wall from the behavior of the ground. Appropriate stiffness of the interface elements needs to be specified.

Stiffness for interface elements is classified into the normal direction behavior (Kn) and the tangential direction behavior (Kt). If the nonlinear behavior of ‘Coulomb Friction’ needs to be defined for the interface elements, Cohesion and Internal Friction Angle must be specified.

Belytschco (1984) proposed the range of Kn values to be 2 to 1,000 times the Kt. This shows that the values of Kn and Kt are widely varied and are dependent on experiences.

Anchors

The anchor length can be classified as an ungrouted length or a grouted length. The ungrouted length is created with the ‘number of divisions’ as 1. The grouted length creates elements with a unit ‘length spacing’ as 1.

When using truss elements with different elements, the nodes must be shared. However, embedded truss elements do not require node sharing and are hence more convenient for modeling and analysis. Embedded truss elements are used in an embedded form inside a mother element, and the mother element can be a plane strain element or a solid element. The mother element is determined as the element that includes each embedded truss element node within itself, and the multi-point constraint is used to automatically constrain the nodal displacement of the embedded truss element to be the same as the internal displacement of the mother element.
Contents

Part 1. Objectives
Part 2. Introduction
Part 3. 2D case
Part 4. 3D case
Part 5. Conclusion

Integrated Solver Optimized for the next generation 64-bit platform
Finite Element Solutions for Geotechnical Engineering
Modeling

- Modeling in SoilWorks
  - Create Ground Materials
  - Create Structural Materials
  - Open CAD file
Modeling

- Modeling in SoilWorks
  - Generate Smart Surfaces
  - Drag and Drop materials
Structural Elements

- Assign Sheet Pile Wall Element
- Assign Anchors by Drag and Drop
- Generate Smart Mesh
- Create Interface 0.5; 1
- Change names of sets
Assign Loads and Boundary Conditions

- Apply Pretense Load to Anchors 220 kN
- Apply Self Weight
- Apply Boundary Conditions
Construction Stages

• Analysis Case
  - Construction Stage 1
Construction Stages

- Analysis Case
  - Construction Stages 2 and 3: Excavations of pipe and 1\textsuperscript{st} layer
Construction Stages

- Analysis Case
  - Construction Stages 4 and 5: Excavations 2 and 3
Analysis Case
Post-Processing
• Report Ready Results
  - Extract Results for construction stages
    - Displacements
    - Moments on Wall
    - Forces on Anchors
Post-Processing

- Report Ready Results
  - Extract Results for construction stages
    - Displacements
    - Moments on Wall
    - Forces on Anchors
Contents

Part 1. Objectives
Part 2. Introduction
Part 3. 2D case
**Part 4. 3D case**
Part 5. Conclusion
Modeling

- Modeling in midas GTS
  - Import 2D CAD Geometry
  - Extrude 2D geometry to make 3D solids (5m outer, 20m inner)
  - Import Materials (same as last model)
  - Copy anchors x direction 10@2m
  - Check duplicates
Mesh Generation

- Generate Mesh
  3D mesh of Ground layers
  Size 1 with Tetra
Mesh Generation

• Generate Mesh and Extract Elements
  - Extract 2D mesh for wall
  - Generate 1D mesh of Anchors
    - 1 division
Loads & Boundary Conditions

• Load Conditions
  - Self weight load
  - Pre Stress anchors 220 kN

• Boundary Conditions
  - Ground support conditions
  - Wall Interface with rigid links
Interface Wizard

When generating interface element using interface wizard, input the 2 parameters \((tv, R)\) as below so that the material properties will be automatically calculated according to the properties of the adjacent ground elements.

Apply [Virtual Thickness Factor \((tv)\)] and [Strength Reduction Factor\((R)\)] by using stiffness and nonlinear parameters of the adjacent elements.

\[
Kn = \frac{Eoed,i}{tv} \\
Kt = \frac{Gi}{tv} \\
Ci = R \times Csoil \\
Eoed,i = 2 \times Gi \times (1-vi)/(1-2 \times vi) \\
Gi = R \times Gsoil (Gsoil = E/(2(1+ vsoil)),
\]

\(tv\) = Virtual thickness factor (Generally use the value in the range of 0.01~0.1 if the stiffness is big, use smaller value.)
\(vi\) = Interface Possion Ratio = 0.45
\(R\) = Strength reduction factor
General strength reduction factor according to structural members and adjacent ground properties are listed as below.
Sandy soil/Steel material = \(R : 0.6\sim0.7\)
Clay/Steel material = \(R : 0.5\)
Sandy soil/Concrete = \(R : 1.0\sim0.8\)
Clay/Concrete = \(R : 1.0\sim0.7\)
Construction Stages

- Stage Definition
- Create 5 stages to match 2D model sequence by same Drag and drop method
- Create analysis case (initiate stress and activate K0)
Post-Processing

- Report Ready Results
  - Extract Results for stages
- Structural Results
| Engineer | China Pingmei Shenma Group  
<table>
<thead>
<tr>
<th></th>
<th>China University of Mining and Technology</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td><strong>Project Overview</strong></td>
</tr>
<tr>
<td><strong>Title</strong></td>
<td>Pingzhuang West Strip Mine, Inner Mongolia, China</td>
</tr>
<tr>
<td><strong>Project</strong></td>
<td>Steep / Side Slope Design for the Coal Mining</td>
</tr>
</tbody>
</table>
| **Scope**| Analysis results for design  
|          | - Y, Z Displacement with the changing of mining width  
|          | - Optimization of Mining Width under the various seepage conditions |
| **Overview**| Pingzhuang West Strip Mine, with the standard 374m mining depth, plan to produce 1,500,000 t coals per year, until 2024. After the steep/side slope stability analysis with mdias GTS, when the mining slope angle is over 40°, the years of the mining will be extended another 8-10 years, maximizing the economic efficiency. |
General Model

3D excavation part model for excavation stage

3D post-excavation analysis model
### midas GTS NX Modeling

<table>
<thead>
<tr>
<th>Y direction displacement with 20m mining width</th>
<th>Z direction displacement with 20m mining width</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Y direction displacement" /></td>
<td><img src="image2" alt="Z direction displacement" /></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Maximum displacement point for steep slope mining</th>
<th>Maximum stress point for steep slope mining</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image3" alt="Maximum displacement point" /></td>
<td><img src="image4" alt="Maximum stress point" /></td>
</tr>
</tbody>
</table>
Construction stage excavation with slope stability analysis by activate SRM box for each stage

1st Exca: FOS 2.225
2nd Exca: FOS 2.217
3rd Exca: FOS 2.209
Export CAD from GTS NX to Soilworks

Define LEM materials, Failure arc and stages for CS analysis
LEM for construction stage analysis can give FOS for each stage of excavation.

Initial: FOS 2.32

1st Exca: FOS 2.14
2nd Exca: FOS 2.04
3rd Exca: FOS 1.93
Contents

Part 1. Objectives
Part 2. Introduction
Part 3. 2D case
Part 4. 3D case
Part 5. Conclusion
Conclusion

• **Sequence of excavation and reinforcement is important for 3D deep excavation projects**

• **Earth Pressure Coefficient is important to consider when excavating in soft grounds**

• **Midas software allow for interface elements to better simulate soil structure interaction**

• **In 3D models it is possible to review in detail the stress distributions on cross-sections, which is not possible in 2D models**

• **Midas allows for import <-> export to Soilworks for results comparisons by different methods**
Questions

lamidas@midasuser.com

Integrated Solver Optimized for the next generation 64-bit platform

Finite Element Solutions for Geotechnical Engineering