Chapter 7
3D Slope Stability

Workflow

: Geometry > Surface & Solid > Make Face	5
: Geometry > Surface & Solid > Box	5
: Geometry > Divide > Solid	6
: Mesh > Generate > 3D	7
: Static/Slope Analysis > Load > Self Weight	8
: Static/Slope Analysis > Boundary > Constraint	8
: Static/Slope Analysis > Water Level	8
: Analysis > Analysis Case > General	9
: Analysis > Analysis Case > Perform	10

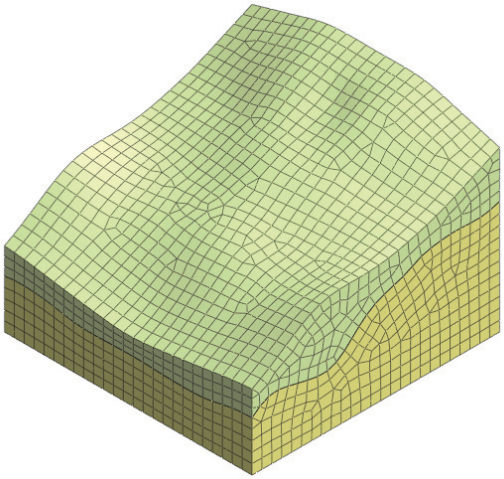
3D Slope Stability

Section 1
Overview

1.1 Learning Purpose

Recently it becomes possible for numerical slope stability analysis to simulate destroyed shapes similar to reality and to reflect the real site condition. However, there is still limitation of analyzing 3D slope behavior with 2D analysis, which only considers the cross section of a slope. Major difference between 2D analysis and 3D analysis is whether components which affect slope's behavior such as slip surface shape, ground material distribution, and slip surface rigidity can be reflected or not. Since 3D analysis can consider components which are neglected in 2D analysis, it is possible to get more reliable result. In other words, through 3D slope stability analysis slope slip range can be considered; location where slop activity is concentrated can be found; and according to this location, solution plan can be established.

▶Modeling overview



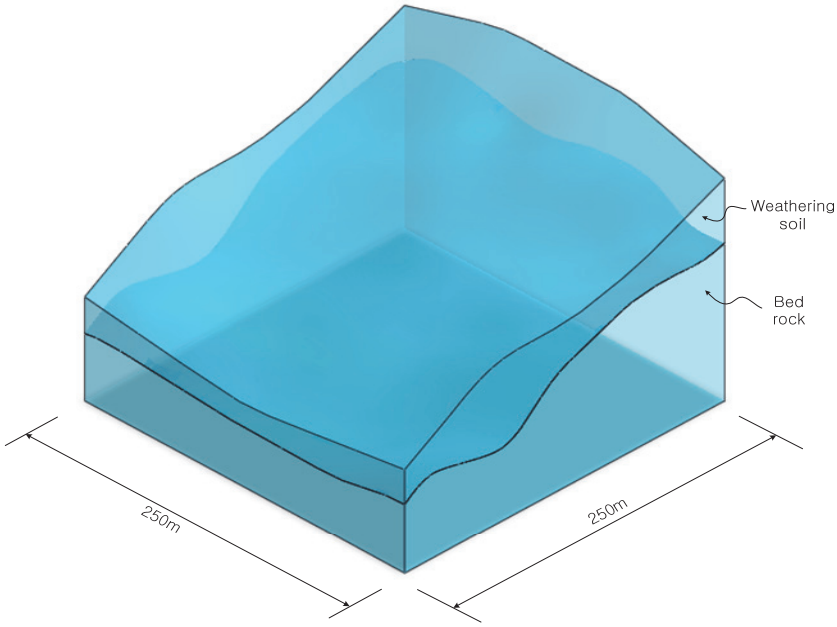
In this tutorial, the following main concepts will be explained:

- Generate surface and bedding planes using grid face
- Generate meshes
- Slope stability analysis
- Analysis result – Safety factor and maximum shearing strain
- Analysis result – Check results on specific cross section using 'Clipping plane'

1.2 Learning Purpose

This is 3D model which consists of weathering soil and bed rock. By reviewing slope stability of the model in dry / rainy season, we will locate the part which risks being destroyed.

▶Cross-section diagram



Section 2

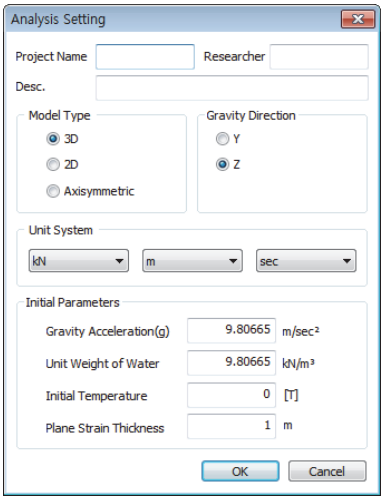
Analysis Setting

[Open the attached start file. (07_3DSlope_start)]

* : Analysis > Analysis Case > Setting

- Set model type, gravity direction, initial parameters and unit system for the analysis. The unit system can be changed whenever you want during the modeling process and after performing the analysis. The input parameters will be automatically converted by the right unit system.
- This tutorial is a 3D model with gravity in Z direction and using SI unit system (kN, m).

▶Analysis setting



Section 3
Define Material and Property

3.1 Definition of Ground and Structural Materials

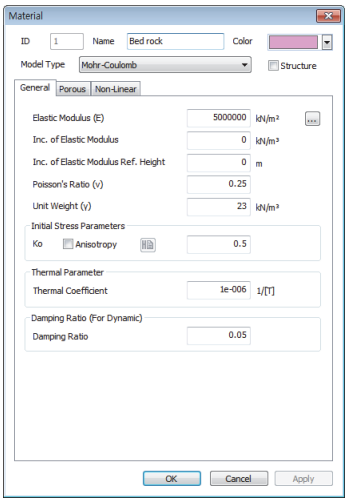
Define the soil model type as 'Mohr-Coulomb'. Ground materials are defined as below:

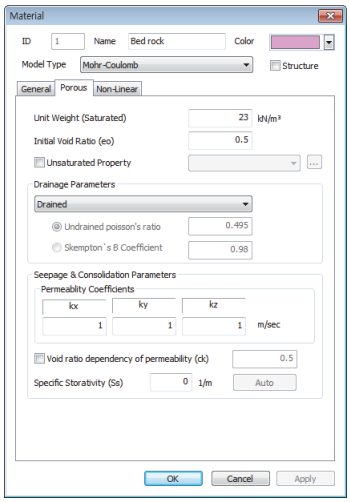
▶Table. Ground material

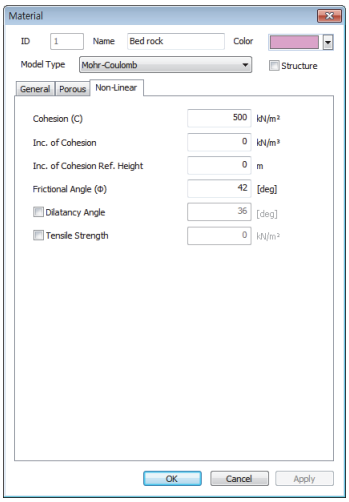
[Unit: kN, m]

Name	Bed rock	Weathering soil
Model Type	Mohr-Coulomb	Mohr-Coulomb
General		
Elastic Modulus(E)	5.0E+06	1.0E+05
Poisson's Ratio(v)	0.25	0.30
Unit Weight(r)	23	18
Ko	1.0	0.5
Porous		
Unit Weight(Saturated)	23	18
Initial Void Ratio (e0)	0.5	0.5
Drainage Parameters	Drained	Drained
Non-Linear		
Cohesion(C)	500	100
Frictional Angle	42	19

▶Define ground material -General
▶▶Define ground material - Porous
▶▶▶Define ground material – Non-Linear







3.2 Definition of Properties

Properties represent physical attributes of the meshes and will be assigned to mesh sets during mesh generation. Define the ground properties by assigning the material be used in each ground layer.

Name	Bed rock	Weathering soil
Type	3D	3D
Material	Bed rock	Weathering soil

Section 4
Modeling

The main purpose of this tutorial is generating 3D bedding planes from grid face, slope stability analysis (SRM), and result analyzing. You can start the tutorial by opening the start file in which basic materials and properties have already been predefined.

4.1 Modeling Geometry

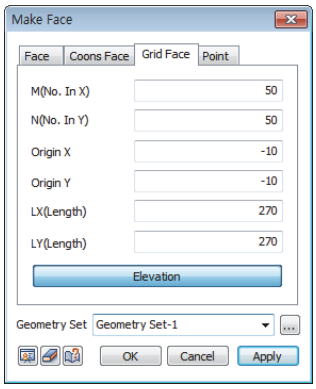
* : Geometry > Surface & Solid > Make Face

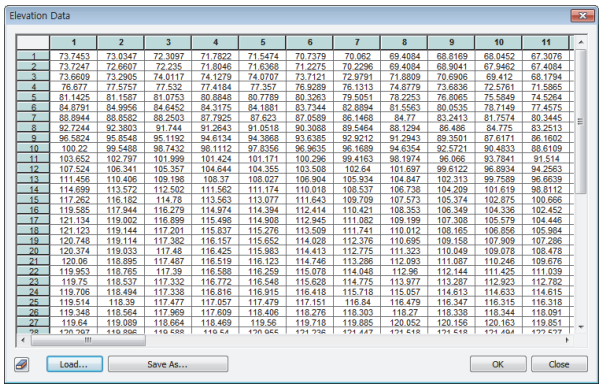
Generate a face with the coordinate data which contains strata and ground geometry information. To dive solid with bedding plane, make a face larger than target solid.

- Under [Grid Face] tab, enter '50', '50' for 'M (No. In X)' and 'N (No. In Y)'
- Input '-10','-10' at origin X, Y.
- Input '270','270' at LX, LY.
- Press [Elevation], and select [Load...] to import geometry data of '3d_slope_terrain.txt'.
- Click [OK], Press [Apply] button.
- In the same way, generated strata by importing '3d_slope_strata.txt' .

▶Make face

▶▶Grid face information





* : Geometry > Surface & Solid > Box

Generate box solid which represents the ground.

- Enter Origin point (0, 0, 0).
- Input '250' , '250, '250' to length, width, height.
- Name geometry set as 'ground'. Click [OK].

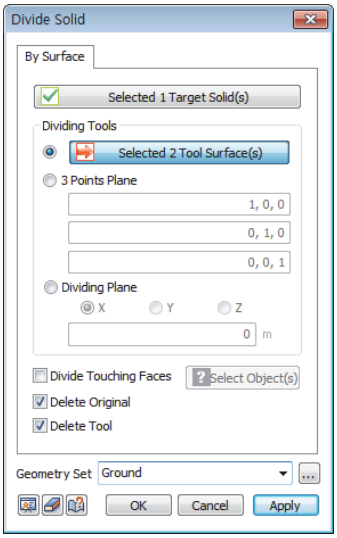
* : Geometry > Divide > Solid

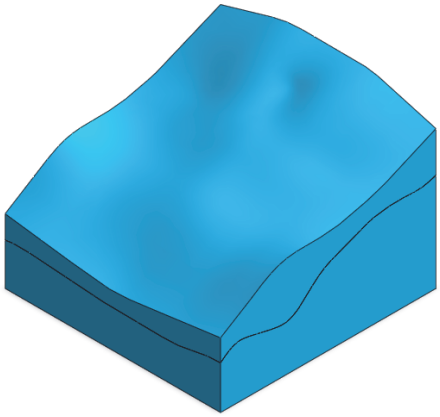
Divide ground solid with surface and bedding plane made with grid face.

- Select ground solid as target object.
- Change selection filter to 'Face(A)'.
- For tool surface, select the ground surface and bedding plane generated in the former step.
- Select geometry set to 'Ground'.
- Click [OK].
- Press [Delete] key to remove the useless solid above the ground surface

▶ Divide solid

▶▶ Divided solid





Tip

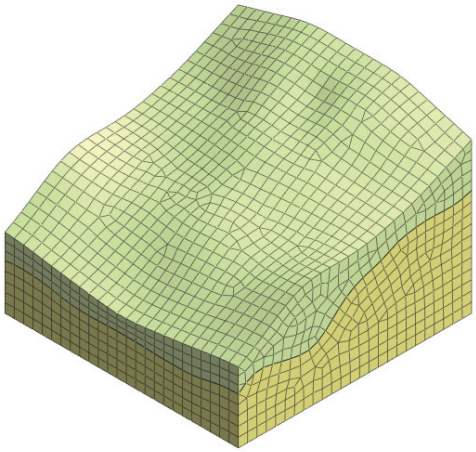
Tool surface must be bigger than or equal to the target solid area. If it is not, solid can be transferred to 'Shell' or 'Compound'.

4.2 Generate Mesh

* : Mesh > Generate > 3D

- Select upper solid in the [Auto-Solid] tab.
- Input size to '10'.
- Select 'Hybrid Mesher'.
- Check [Match Adjacent Faces].
- Choose '2: Weathering soil' as property. Name the mesh set as 'Weathering soil'.
- Press >> button. In the Advanced option, check [Higher-Order Element] and Click [OK].
- Generate weathering soil meshes by clicking [Apply].
- With the same method, generate 'Bed rock' meshes.

▶Generated mesh



Tip

Meshes transfer model's geometrical shape to the finite element analysis solver. Therefore, the accuracy of analysis result depends on quality of the meshes. Generally better result can be obtained when the mesh size is smaller and the mesh shape is closer to regular polygon/regular polyhedron shape. In other words, rectangle/hexahedron elements are generally better than triangle/tetrahedron elements. But in case that rectangle meshes can't be properly generated according to model's complex geometry, it is much better to use triangle elements rather than low quality rectangle elements.

GTS NX provides 'Hybrid Mesher (hexahedron centered)' to optimize the mesh quality. And to perform 2D and 3D slope stability analysis, strength reduction method (SRM) is used.

Because when low order elements are used, the rigidity of the model is relatively higher than when high order elements are used, the safety factor can be calculated to be higher. Therefore stability analysis based on strength reduction method provides good quality of result when using high order elements. But when using low order elements, inappropriate results might come out depending on model.

In conclusion, it is recommended to use high order elements when performing slope stability analysis based on strength reduction method.

Section 5
Analysis Setting

5.1. Setting Load Condition

* : Static/Slope Analysis > Load > Self Weight

Gravity is calculated automatically by multiplying the inputted unit weight of the ground, the structure geometry and the acceleration of gravity. It can be easily set by inputting a scale factor of direction. The default value of the gravity direction is defined.

- Set the [Name] as 'Gravity-1' and enter 'Self weight' as the [Load set] name.
- Put scale factor '-1' at direction of gravity acceleration 'Gz'.
- Click [OK].

5.2 Setting Boundary Conditions

* : Static/Slope Analysis > Boundary > Constraint

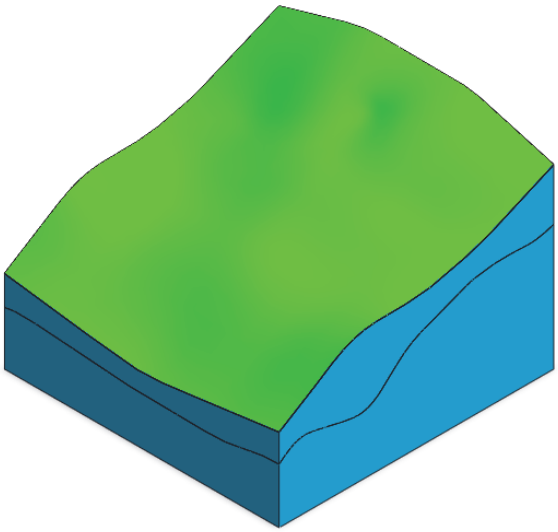
This process sets boundary conditions against internal deformation or rotation based on GCS. Automatically set displacement constraints at the left/right/bottom parts of the model according to GCS.

- In the [Auto] tab, type name and boundary set name.
- Click [OK].

* : Static/Slope Analysis > Water Level

This process sets boundary to simulate saturated status of the ground surface. In GTS NX, water level can be defined in 3D space using function.

- In Work Tree > Geometry, mouse right click to select [Show].
- In the [Face] tab, select ground surface as target object. .
- Enter interval to '5' .
- Name 'Water level' and Click [OK].



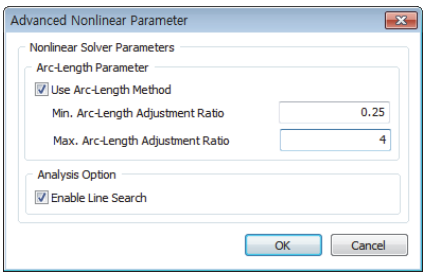
5.3 Setting Analysis Case

* : Analysis > Analysis Case > General

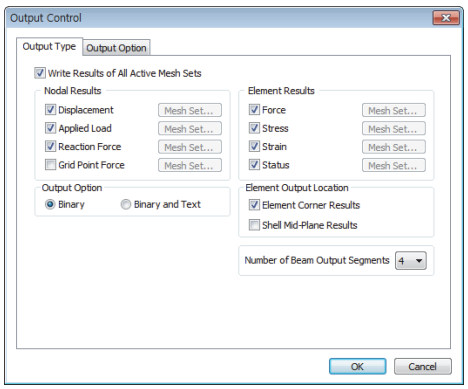
This process sets analysis method and model data for the analysis. The analysis and output types could be controlled using advanced options.

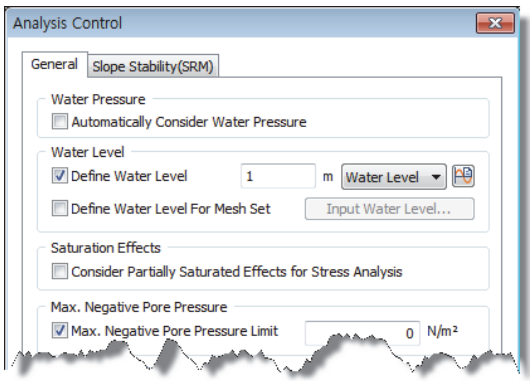
- Input name as 'Dry period'.
- Select solution type to 'Slope Stability (SRM)'.
- In Analysis Control > Slope Stability (SRM) tab, select [Advanced Nonlinear Parameter]. Check [Use Arc-Length Method] and input value as shown in the following image.
- Move all mesh sets, boundary conditions and load sets to [Active set].
- In [Output Control], check 'Strain' at [Output type] tab.
- Click [Apply].
- In the same way, create analysis case for 'Rainy period'.
- For rainy period, check [Define water level] in Analysis Control > General. Input '1', and select 'Water Level'.

▶Nonlinear analysis option



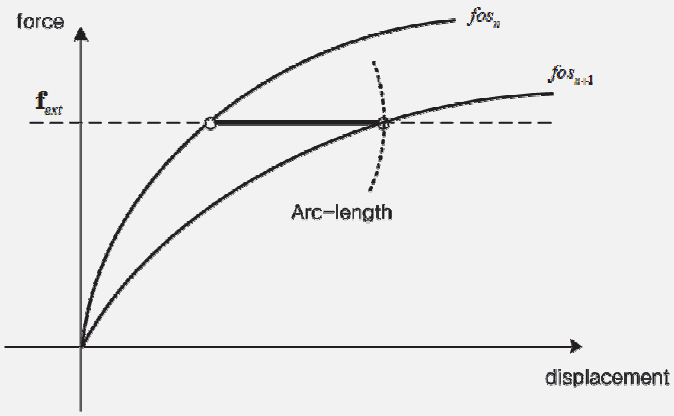
▶Output Control(Check strain)
▶▶Analysis(Define water level)





Tip

Major difference between strength reduction method using Arc-length and standard strength reduction method is the method of increasing/decreasing safety factor. Standard strength reduction method calculates safety factor of next stage using the increment assigned by user. When it is used on very stable models or unstable models without engineer's estimation, the safety factor varies uniformly and inefficient calculation can be performed.
On the other hand, Arc-length calculates the increment of safety factor according to convergence speed of last step. More reliable safety factor can be obtained.



<Increment of safety factor according to Arc-length>

5.5 Perform Analysis

* : Analysis > Analysis Case > Perform
  - Perform analysis.

Tip

During the analysis, you can check the calculation process in real-time. Messages such as whether the results converge or not, warnings and errors can be checked through [Output Window].
The model is automatically saved before being analyzed. The information about analysis is saved in *.OUT file in the same place where the model file is saved.
When analyzing a model, the results can be controlled by Analysis > Analysis Case > General > Output Control option. And if you set the output option as [Binary and Text], the results of nodes

Section 6
Results and Generate Reports

You can verify displacement, stress in the Result Tree after the analysis. All the results can be plotted as contour, table, and graph. In this tutorial, the main result items which need to be checked are listed below.

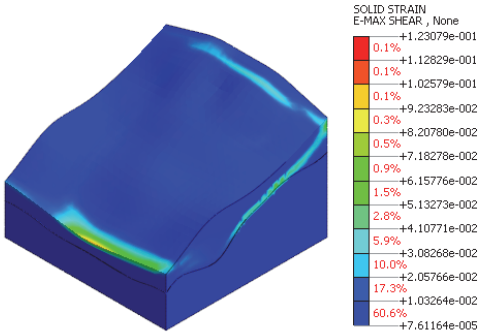
- Safety factor/Destroyed shape
- Review through 'Clipping plane'

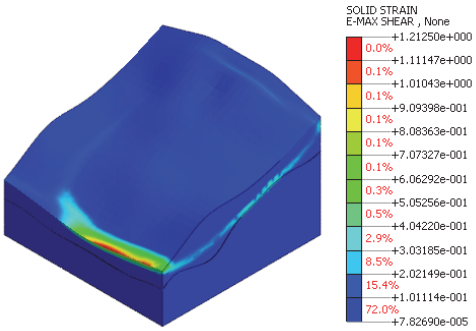
6.1 Safety Factor/Destroyed Shape

If you perform strength reduction method, it is possible to verify the minimum safety factor in the Result Tree.
Destroyed shape is verified at 'Max. Shear Strains' of ground at the stage where safety factor is plotted.

- Select the last stage to verify results (the stage which shows minimum safety factor) > Solid Strains > E-MAX SHEAR in the Work Tree > Results > Dry period > Slope stability analysis.

▶Maximum shear strain (Dry period)
▶▶Maximum shear strain (Rain period)





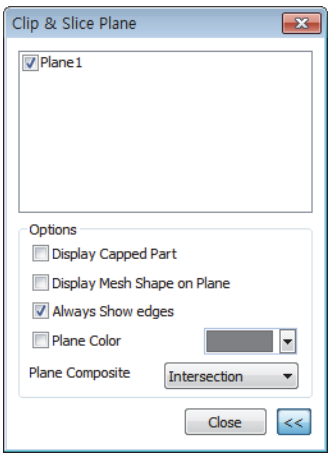
6.2 Verify with Clipping Plane

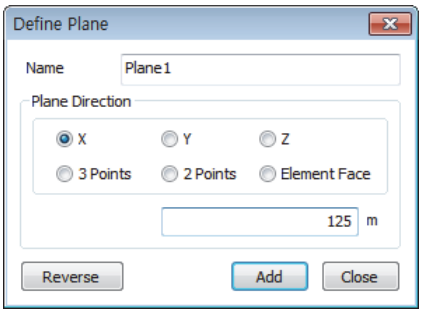
In GTS NX, you can see the result at any cross section by dividing model with "clipping plane" tool.

- Select stage to verify result (stage which shows minimum safety factor) > Solid Strains > E-MAX SHEAR in the Work Tree > Results > Dry period > Slope stability analysis.

- Select Clipping plane ( ) in the 'Advanced View Toolbar'. In [Define Plane] set plane direction to 'X' and distance to '125(m)'. Select [Add] to create 'plane1'.

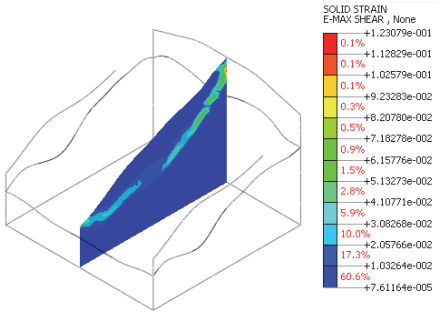
▶Clip&Slice plane option
▶▶Define plane

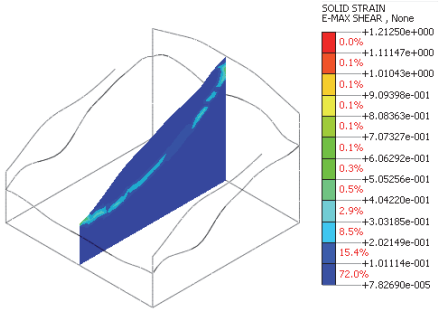


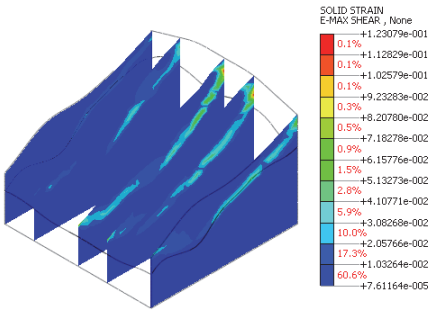
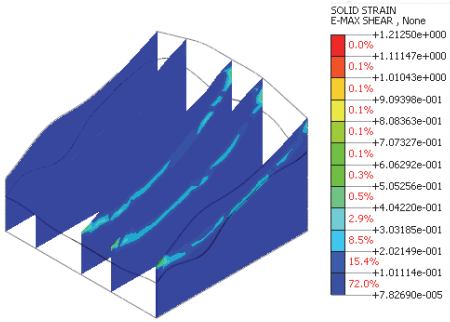


- Uncheck [Display Capped Part] option. In Result > General > contour, set contour type to 'Fringe'.

▶Break down plane(Dry period)
▶▶Break down plane(Rainy period)





- In 'Clip & slice Plane', Click >> button. You can see destroyed shape in specific locations by defining several planes.

▶Multi break down plane (Dry period)
▶▶Multi break down plane (Rainy period)