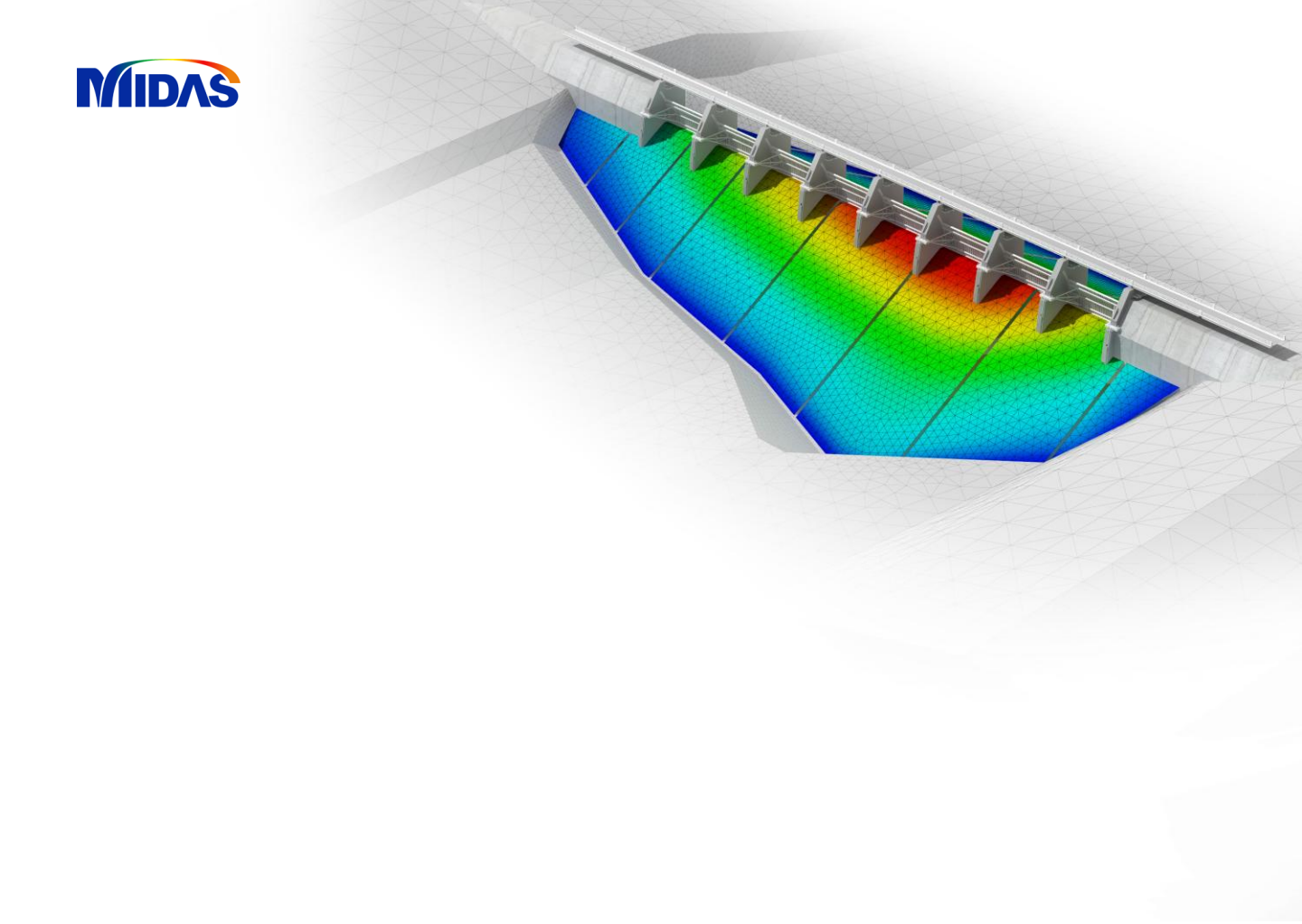
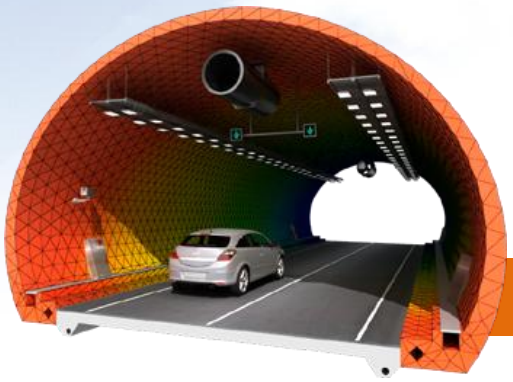


midas **GTS 2010** v1.2 Release Notes

Next Generation Solution for Geotechnical and Tunnel Engineering



Enhancements

▪ Pre Processing

▪ Analysis

▪ Post Processing

Pre-Processing

1. Create Interface

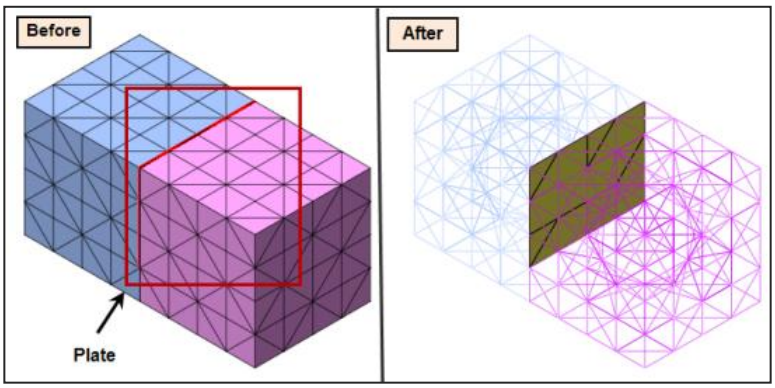
Modified Contents

- Users can control the directional component, i.e. Element Coordinate System (ECS), of interface elements that are created using “From Plate” method.
- Automatically generates rigid link elements between nodes with full control in assigning degrees of freedom. (Create Rigid Link Element : 3D: DX, DY, DZ and 2D: DX, DY)

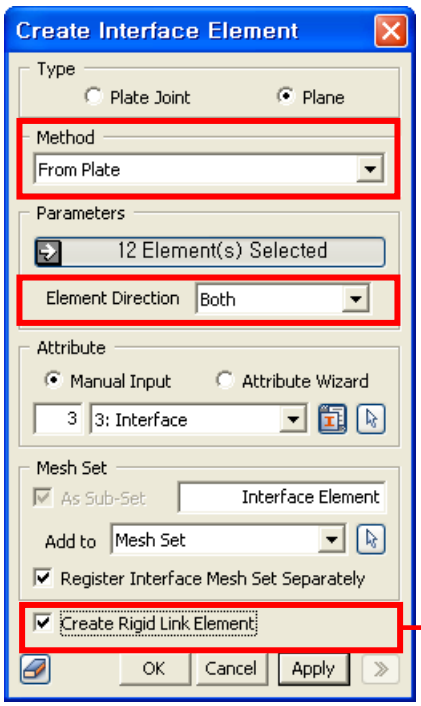
Effects & Usage

- This function can be used to model soil to structural interaction for a wall that is embedded between two soil elements.

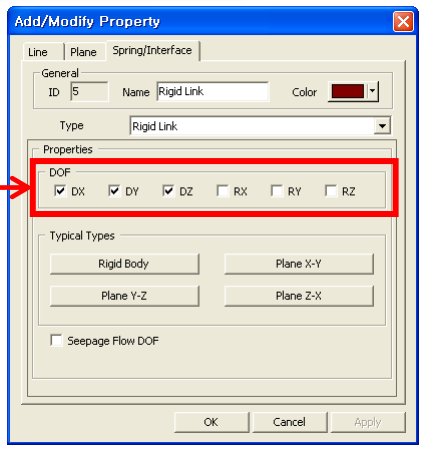
▪ *Model* > ***Element*** > *Create Interface*



Interface Generation



Create Interface – From Plate



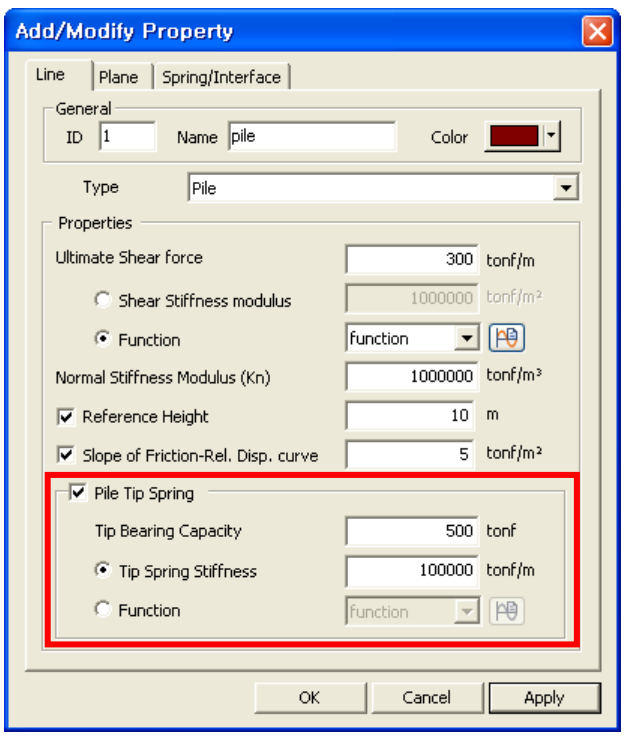
DOF assignment for Rigid Links

2. Pile / Pile Tip mesh set

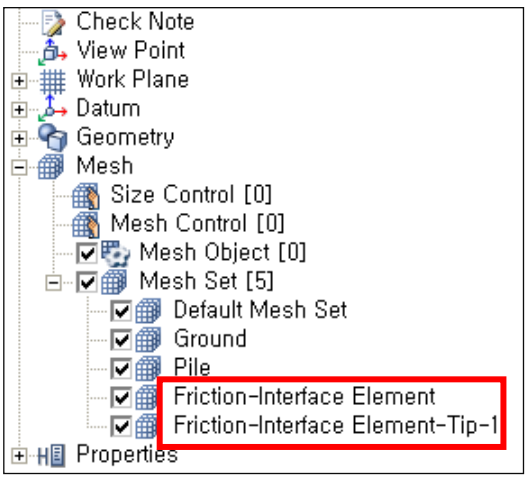
Upgrade Contents

- Activation of “Pile Tip Spring” generates two independent mesh sets: Pile tip springs and pile elements.
- In order to reflect the Slope of Friction-Rel Disp. Curve values, the reference height is referenced from the global coordinate system (GCS).
- The graph below shows variation of shear stiffness at reference height with a user defined Slope of Friction-Rel. Disp. Curve (relative displacement – friction curve) parameter being activated. The diagram below shows the shear stiffness as a function of depth.
- Midas GTS v1.2 automatically creates an additional activated pile tip mesh set(s), i.e. Construction Stages, for model files that were created in previous versions of midas GTS with activated pile tip spring(s). (Pile tip springs that are manually deactivated/removed also require corresponding mesh sets to be deactivated/removed, i.e. Construction Stages.)

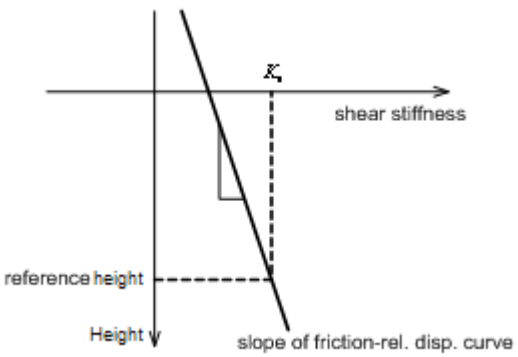
▪ *Model* > ***Property*** > ***Property(Pile)***



Pile Interface Property



Pile Interface Mesh Set



Variation of shear stiffness at reference height

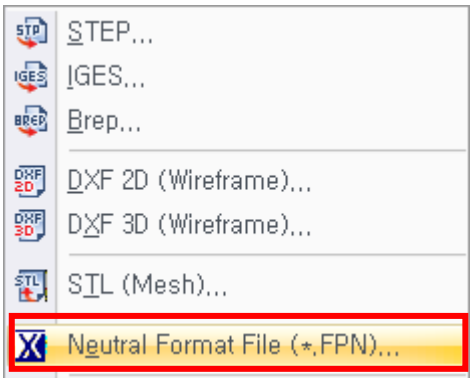
3. Text file format

Upgrade Contents

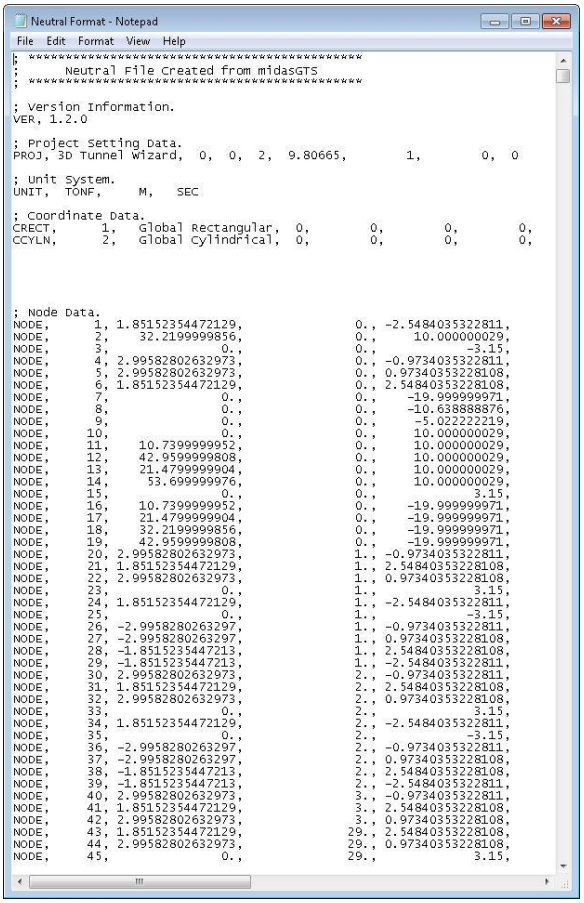
- Neutral Format File *.FPN files contains the following information about a model: midas GTS version number, Geometry, Attribute, Material, Property, Boundary, Load, Construction stage and Analysis types. Neutral Format Files can be viewed in any generic text reader only and imported/exported.

▪ *File* > ***Import, Export*** > *Neutral Format File(*.FPN)*

Export



Import



Neutral Text File

Text file

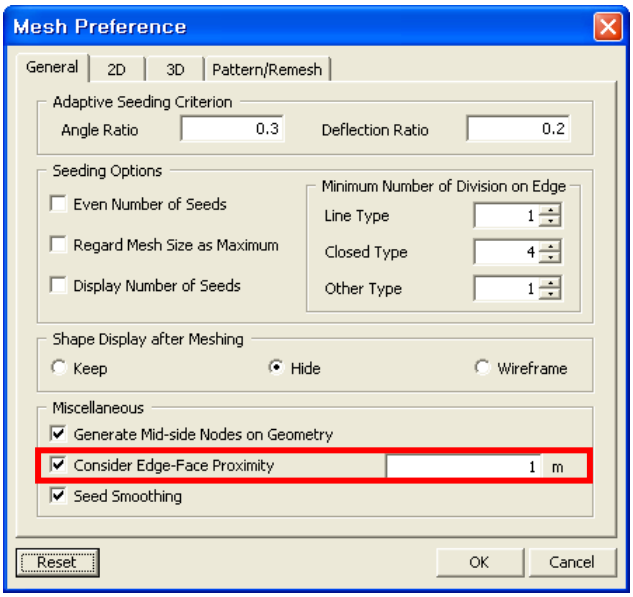
Input Data File

4. Mesh Preference

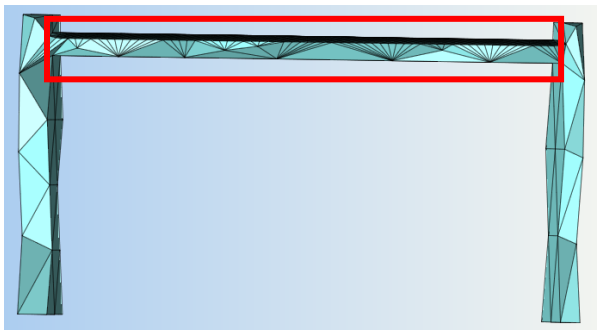
Modified Contents

- Mesh Preference options have been organized into tabs. This allows users to easily navigate between the following options: General, 2D, 3D, and Pattern/Remesh.
- Consider Edge-Face Proximity matches seeding definitions from edges to faces within a predefined proximity.

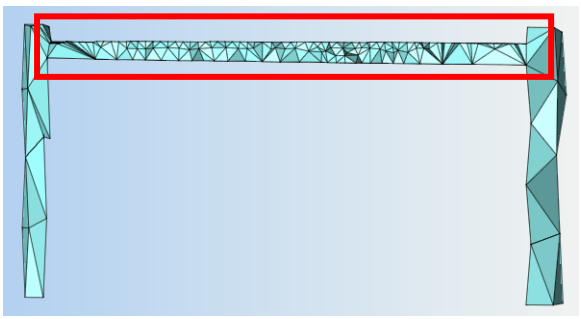
▪ *Mesh* > *Mesh Preference*



Mesh Preference Interface



Proximity - OFF



Proximity - ON

Analysis

1. User Supplied Material

Modified Contents

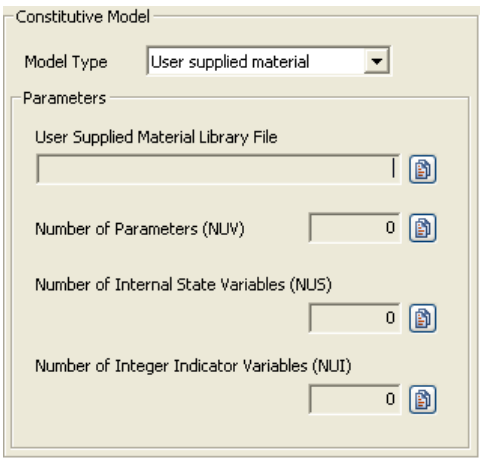
- User Supplied Material allows users to develop Fortran material models for nonlinear elastic and plastic behavior.

2. D-min model

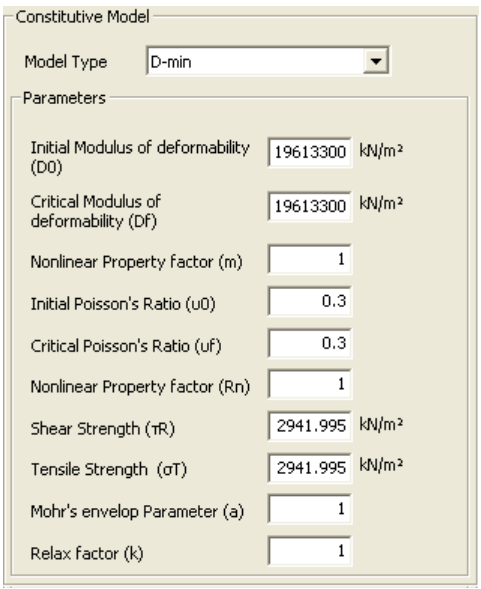
Upgrade Contents

- The D-min model was developed by Hayashi, Hibino in Japan Central Research Institute of Electric Power Industry (CRIEPI).
- D-min model assumes linear elastic characteristics where the material stiffness is updated incrementally in every construction stage while the material stiffness remains constant during each stage.

▪ *Model* > ***Property*** > *Material*



User supplied material



D-min model

3. Initial Prestress Input

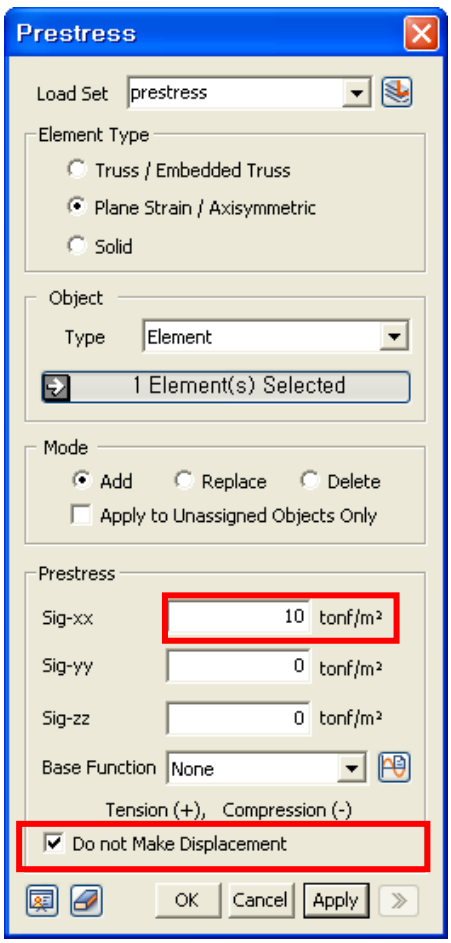
Upgrade Contents

- The option to 'Do not Make Displacement' clears any deformation when assigning a prestress load set to an element.

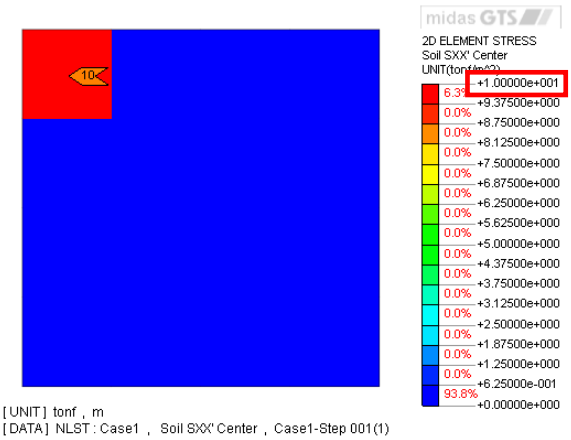
Effects & Usage

- Applying an initial prestress to an element can have two effects: either remain static without any stress reduction or recalculate initial conditions to find the equilibrium stress states. (Similar experimental conditions and analysis results can be simulated)

▪ *Model* > ***Load*** > *Prestress*



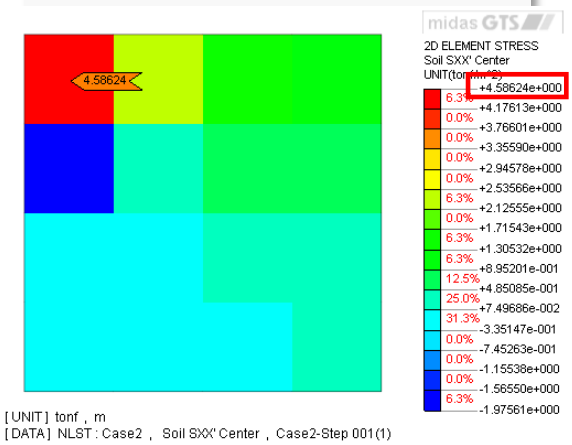
Prestress Interface



[UNIT] tonf , m
[DATA] NLST : Case1 , Soil SXX' Center , Case1-Step 001(1)

X-direction : Sxx

'Do not Make Displacement (Activated)



[UNIT] tonf , m
[DATA] NLST : Case2 , Soil SXX' Center , Case2-Step 001(1)

X-direction : Sxx

'Do not Make Displacement (Deactivated)

4. Flow Interface

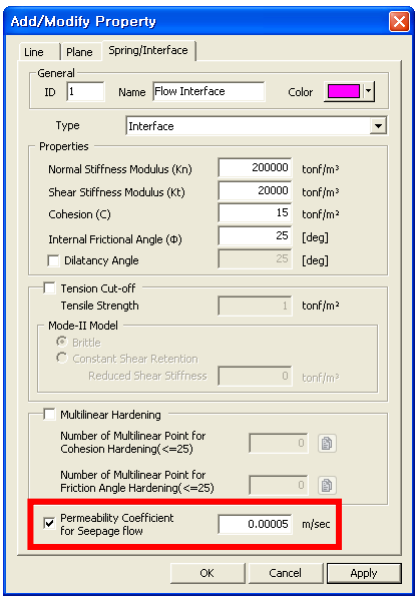
Upgrade Contents

- Interface Elements can be defined with a permeability parameter to allow flow.
  (If the interface element nodes are not separated, an error will occur upon calling the solver. Users must ensure that the nodes have been properly separated.)
- Simulate the flow between two nodes and head boundary conditions using elastic and rigid links. Elastic and rigid link elements can be modeled and assigned either a permeability coefficient for seepage flow and seepage flow DOF.

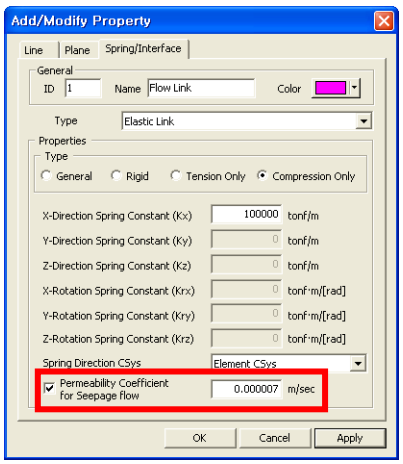
Effects & Usage

- Simulate soil to structure interaction by modeling interface elements with option to consider groundwater.

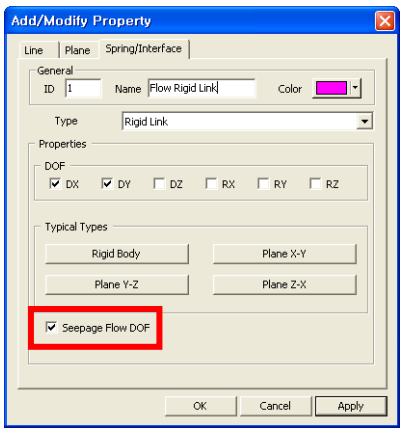
▪ *Model* > ***Property*** > ***Property*** > *Interface , Rigid Link , Elastic Link*



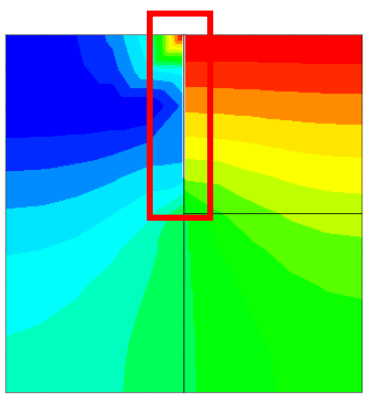
Interface Dialog

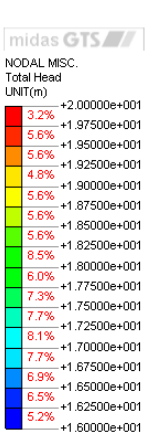


Elastic Link Dialog



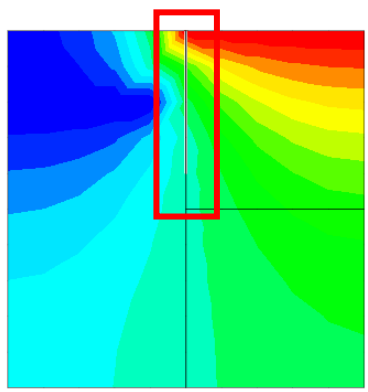
Rigid Link Dialog

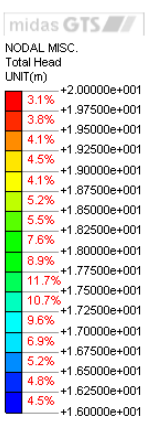




[UNIT] tonf , m
[DATA] SGSS : SS , Total Head , SS

Flow Interface - OFF





[UNIT] tonf , m
[DATA] SGSS : SS , Total Head , SS

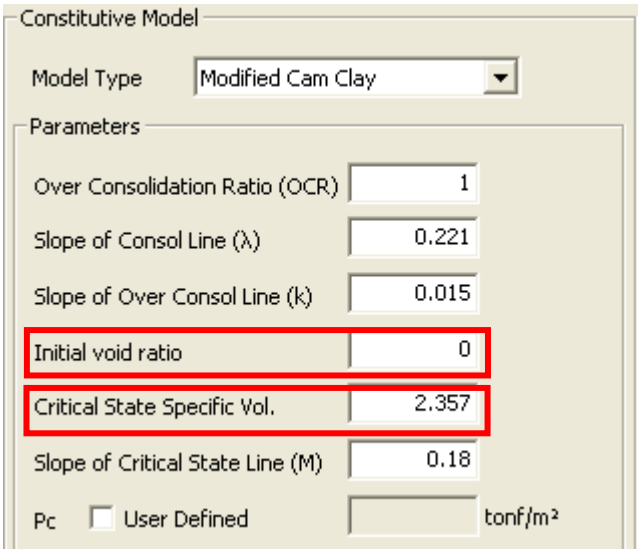
Flow Interface - ON

5. Analysis Type Modified Cam Clay model

Modified Contents

- When using Modified Cam-Clay for analysis types non-linear, construction stage non-linear, and seepage analysis the input parameter “Initial void ratio” is required.
- Analysis types other than the aforementioned cases, only the “Critical State Specific Vol” input parameter is required.

▪ *Model* > ***Attribute*** > ***Material*** > ***Modified Cam Clay***



Modified Cam Clay model

	Initial void ratio	Critical State Specific Vol.
Analysis Type	• Nonlinear Static • Construction Stage (Construction, Steady State, Transient) • Seepage (Steady State) • Seepage (Transient)	• Linear Static • Construction Stage (Consolidation) • Consolidation • Slope Stability (SRM, SAM) • Eigenvalue • Response Spectrum • Time History(Linear)

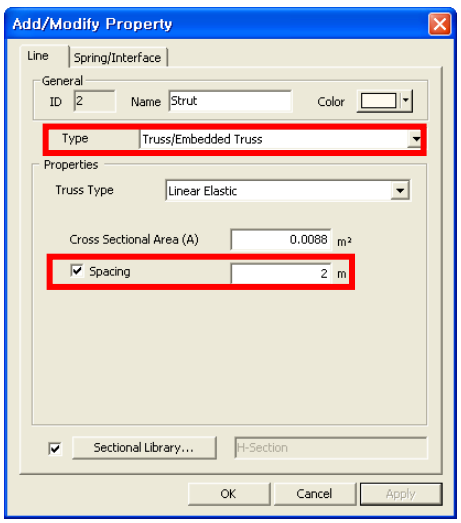
Parameter definition for Analysis Types

1. Spacing input parameter : Beam, Truss/Embedded Truss

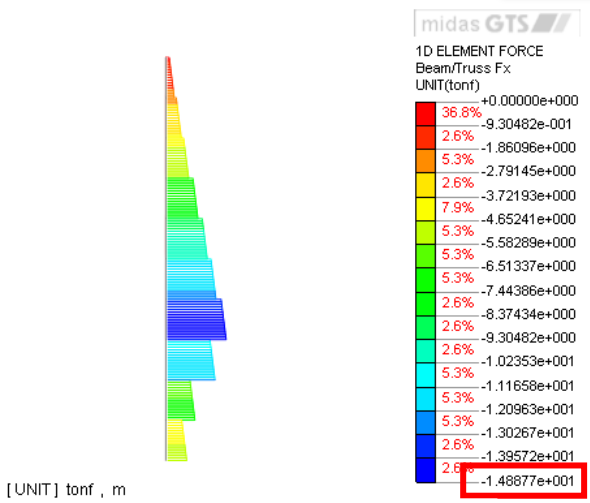
Modified Contents

- In 2D modeling, spacing can be considered for Beam, Truss/Embedded Truss elements. Results are automatically outputted under their respective label. (Spacing does not apply to 3D modeling)

▪ *Model* > ***Property*** > ***Property*** > *Truss/Embedded Truss, Beam/Nonlinear Beam*

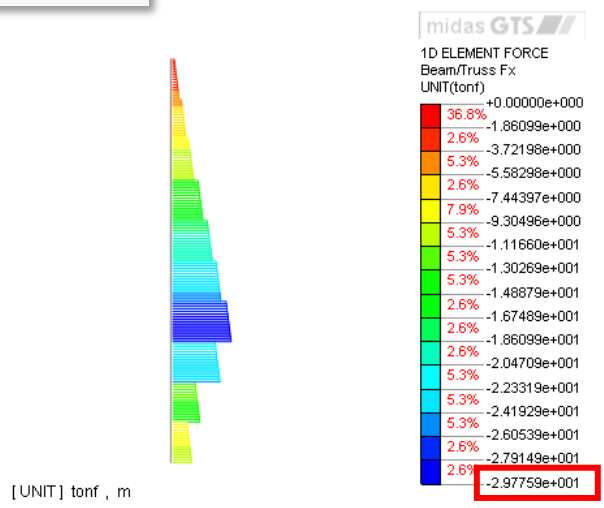


Property Dialog



No Spacing – Beam Fx

- Beam Fx : (Max #) x (Spacing)
  = -14.89 x 2 = **-29.78 tonf**



Spacing – Beam Fx

- Beam Fx : Max #
  = **-29.78 tonf**

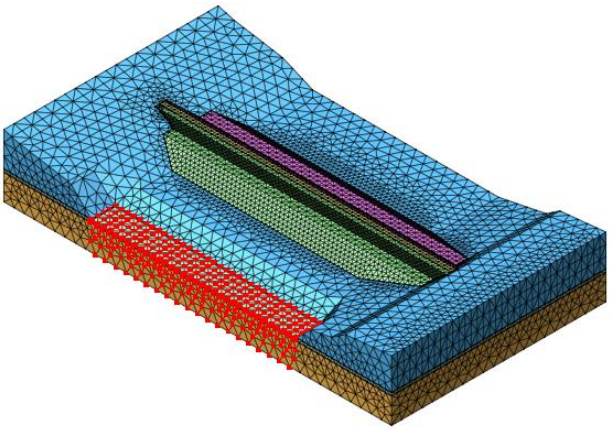
Post-processing

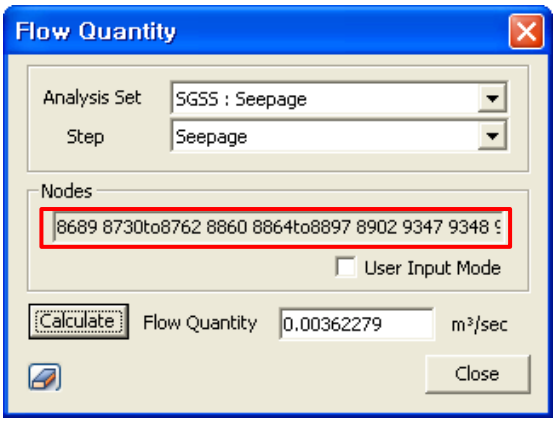
1. Flow Quantity

Modified Contents

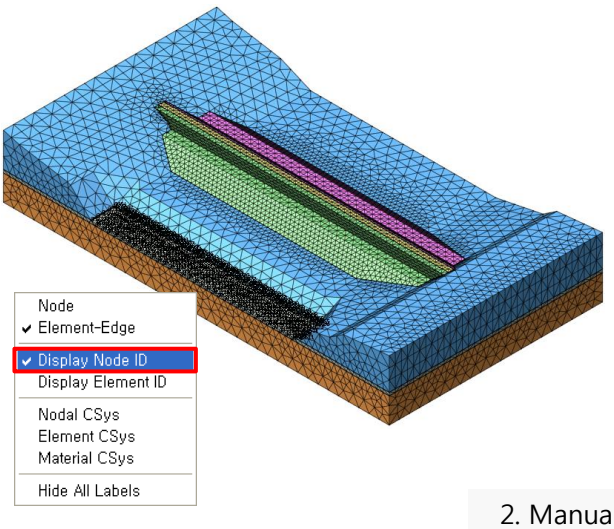
- After performing a seepage analysis, users can manually enter Node ID to confirm the flow quantity values.

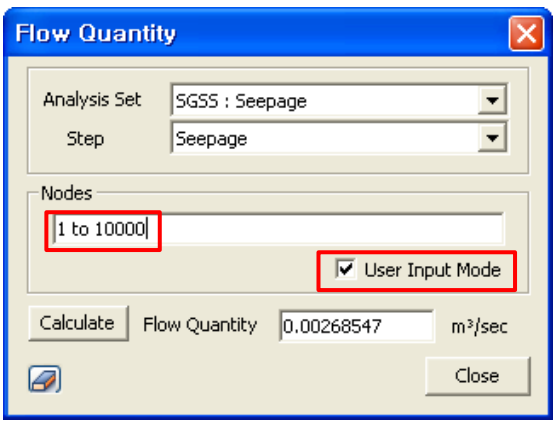
▪ *Result* > ***Seepage Result*** > *Flow Quantity*





1. Node Selection Method





2. Manual Selection