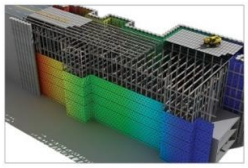
Release Notes

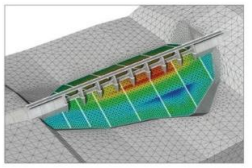
Release Date: August, 2018

Product Ver.: GTSNX 2019(v1.1)

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

Enhancements

1. Analysis

1.1 Construction Stage new analysis types:

- Heat Transfer, Thermal Stress, Seepage-Heat Stress

1.2 Soft Soil Creep – Tensile Strength

1.3 Analysis Setting update

1.4 Interface Relative Displacement result output request

1.5 Nonlinear Time History/SRM - Analysis Case update

2. Pre/Post Processing

2.1 Material/Property/Contact thermal properties

2.2 Layer Control tool

2.3 Result History Output Probes, History Graph

2.4 Flow path

2.5 Slice plane vector

2.6 GTS NX Gen Civil result exchange

2.7 Load Table Import/Export

1. Analysis

1.1 New Heat Transfer, Thermal Stress, Seepage-Thermal Stress analysis types (Construction Stage) *Available upon request

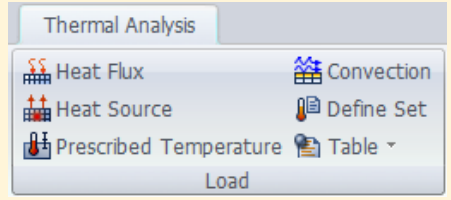
This analysis type can be used to model the thermal changes in the ground due to environmental changes, or due to the construction of facilities, such as buildings or pipelines. In GTS NX thermal energy can be exchanged based on following phenomenological ways: Conduction and Convection.
Thermal analyses (Heat Transfer, Thermal Stress, Seepage-Thermal Stress) are available as the steady state analysis and transient analysis (time dependent).

After the heat transfer analysis, results such as temperature distribution, temperature gradient, heat flow direction / size are printed.

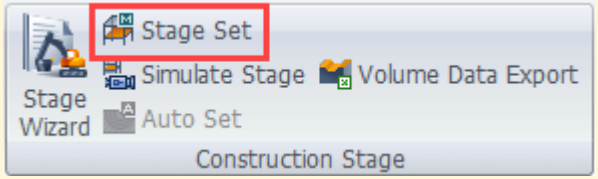
Thermal stress and thermal deformation due to generated/exchanged heat can be obtained from Thermal Stress analysis (thermo-mechanical study).

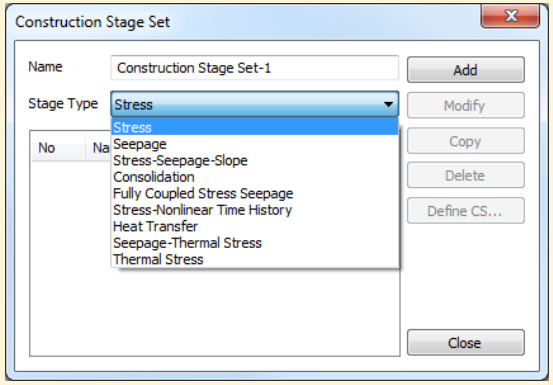
Seepage -Thermal Stress analysis allows for hydro thermo mechanical simulation.
Both structural analysis results and seepage / heat transfer analysis results are output.

Analyses are available from Construction Stage define window.

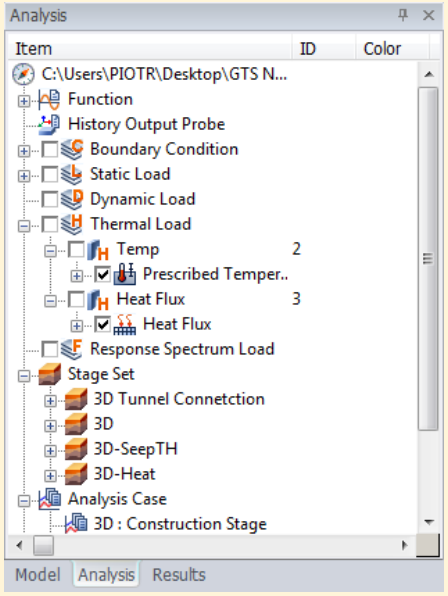


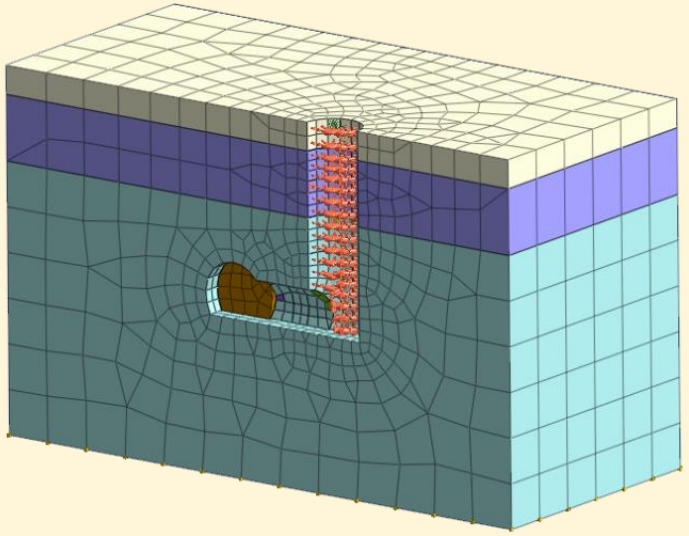
[Available thermal loads]





[Construction Stage Set window]

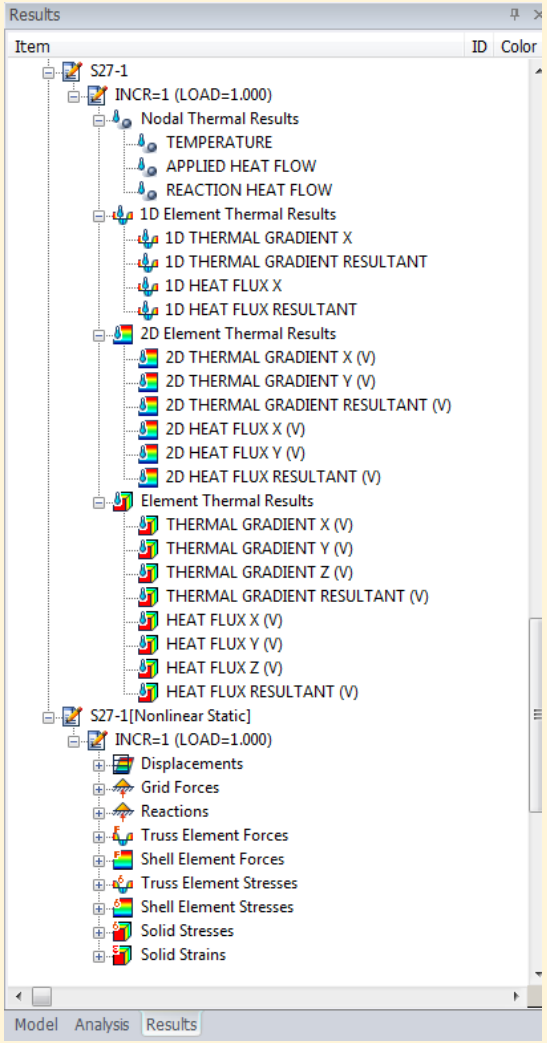




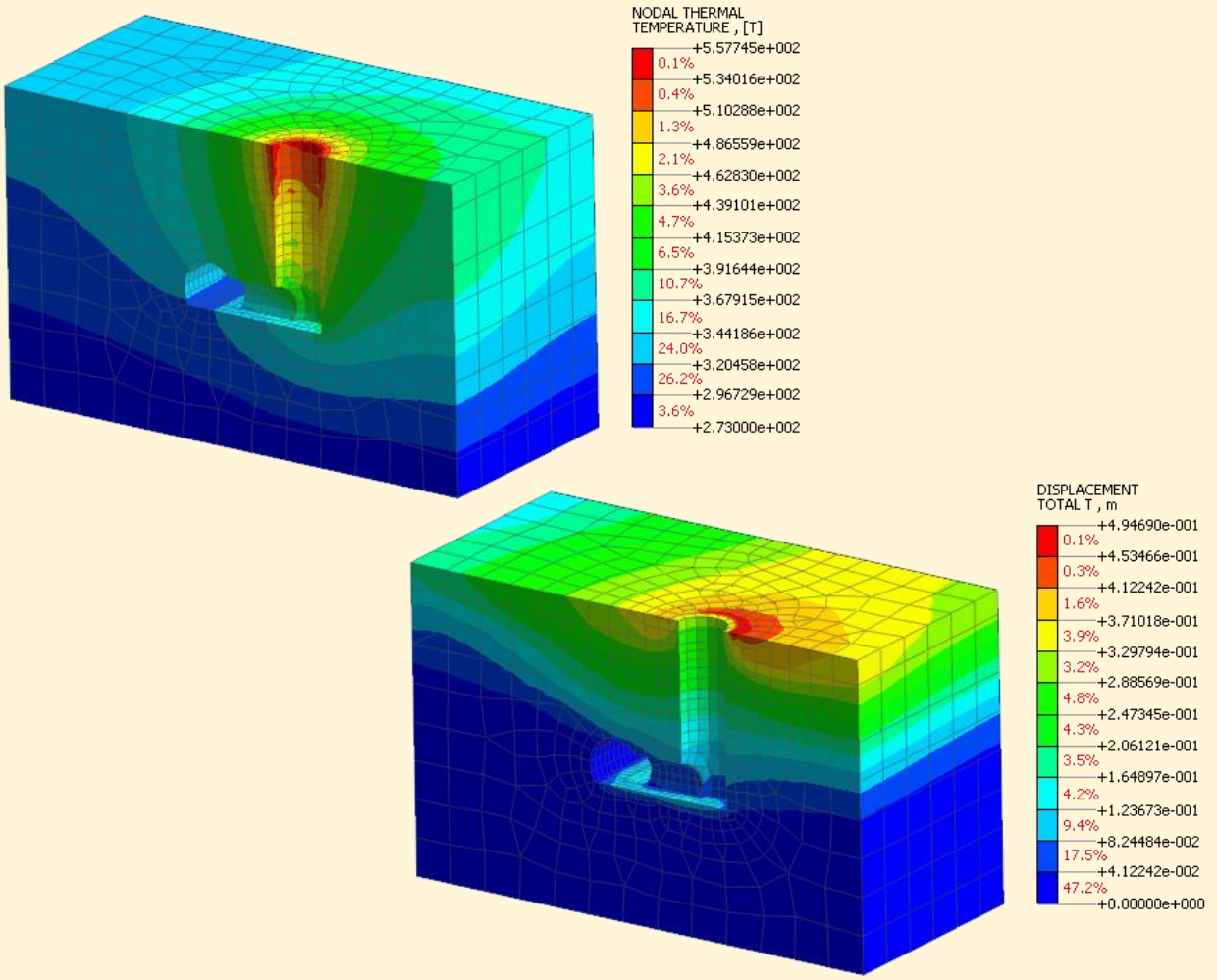
[Model with applied thermal loads]

1. Analysis

1.1 New Heat Transfer, Thermal Stress, Seepage-Thermal Stress analysis types (Construction Stage) *Available upon request



[Every CS output combines Heat Transfer and Structural results]



[Thermal Stress analysis results for lining thermal expansion analysis]

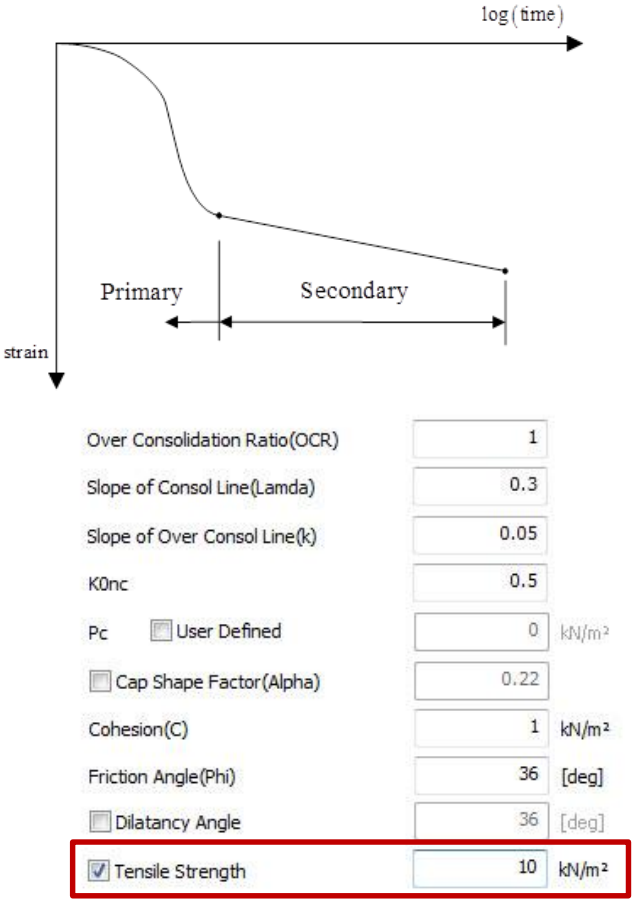
1. Analysis

1.2 Soft Soil Creep – Tensile Strength

- Tensile stress can be obtained by various conditions, such as the heaving of neighboring ground due to embankment load during consolidation or uplift due to excavation. To overcome the material model limits and increase the applicability, Tensile Strength option has been added.

Following is the summary of parameters for the Soft Soil Creep model

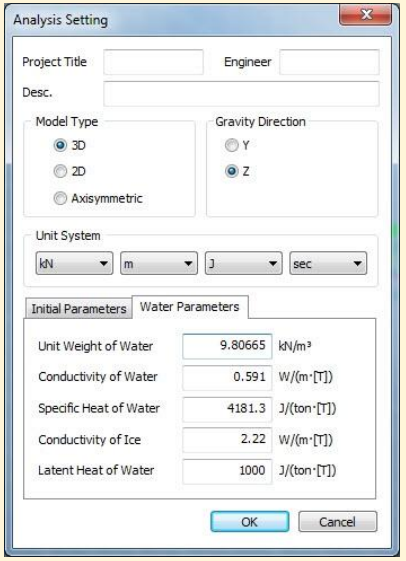
Parameter	Description	Reference value (kN, m)
Soil stiffness and failure		
λ	Compression index	Cc / 2.303
κ	Swelling index	Cs / 2.303 (Cc / 5 for a rough estimation)
μ	Creep index	Cc / 20 for a rough estimation
c	Cohesion	Failure parameter as in MC model
φ	Friction angle	Failure parameter as in MC model
ψ	Dilatancy angle	0
Tensile Strength	Cut off value for tensile hydrostatic pressure	-
Advanced parameters (Recommend to use Reference value)		
KNC	Ko for normal consolidation	1-sinφ (< 1)
Cap yield surface		
OCR / Pc	Over Consolidation Ratio / Pre-overburden pressure	When entering both parameters, Pc has the priority of usage
α	Cap Shape Factor (scale factor of pre consolidation stress)	from KNC (Auto)



1. Analysis

1.3 Analysis Settings

- Initial Parameters and Water Parameters has been moved to different tabs.
- Additional Water Parameters has been added for thermal analyses purposes: Conductivity of Water/Ice, Specific Heat of Water, Latent Heat of Water
- Units of Energy (cal, kcal, J, Btu, kJ) has been added at Unit System toolbar.



[Analysis Setting window]

1.4 Strain - Interface Relative Displacement result output request

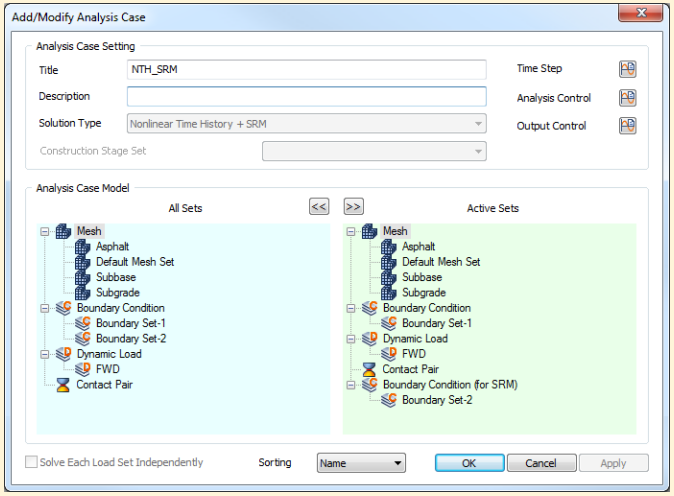
- Interface Relative Displacement result will be printed when Strain option will be checked (Output Control).

It is the element result according to the traction concept of the interface element, and all the Total / Plastic results are made to follow the 'Strain' option.

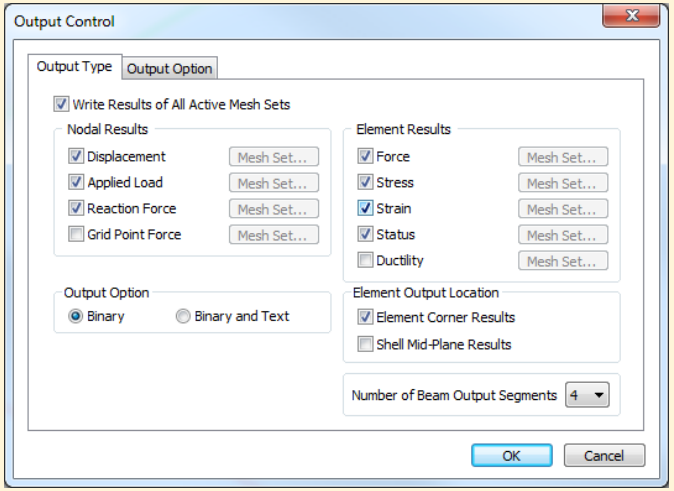
1.5 Nonlinear Time History/SRM - Analysis Case update

- Two separate Boundary Conditions can be used independently for Nonlinear Time History with/without SRM analysis.

- Nonlinear Time History: only the boundary conditions defined in Boundary Condition are reflected in the analysis.
- SRM: Boundary Condition + Boundary Condition (for SRM)



[NTH Analysis with separate SRM boundary condition]

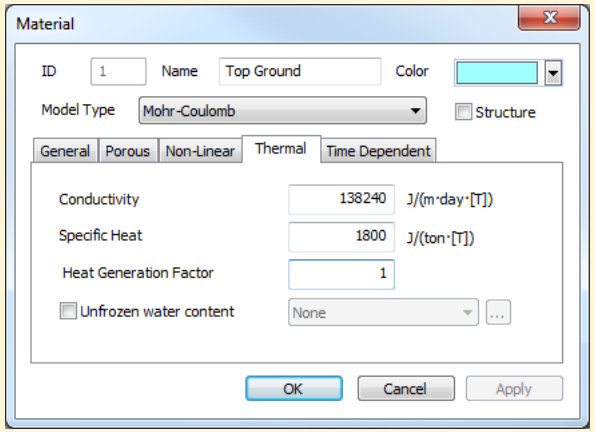


[Strain option for Inreface Relative Displacement result]

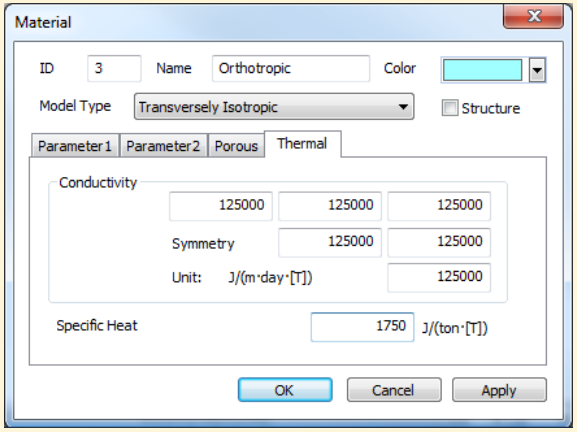
2. Pre/Post Processing

2.1 Material/Property/Contact thermal properties

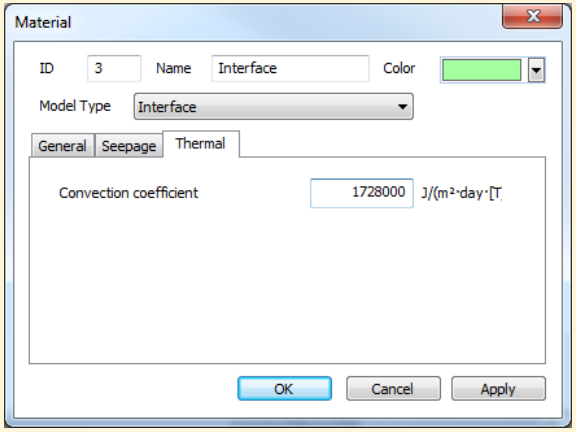
- Thermal properties has been added to material definitions (Isotropic, Orthotropic, Interface/Pile), element (Rigid Link, Elastic Link) and contact properties.



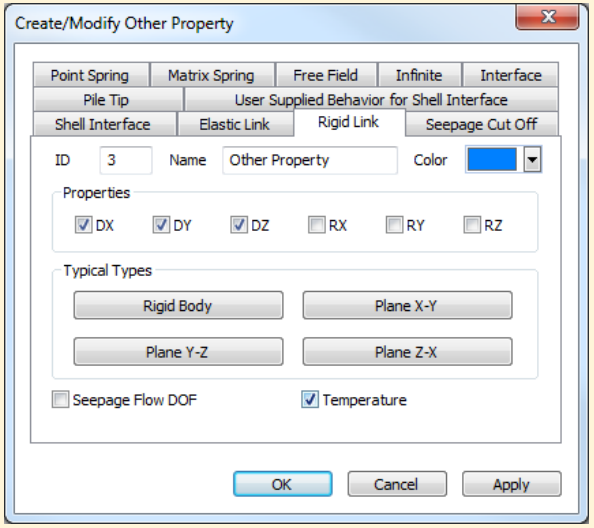
[Isotropic material thermal input]



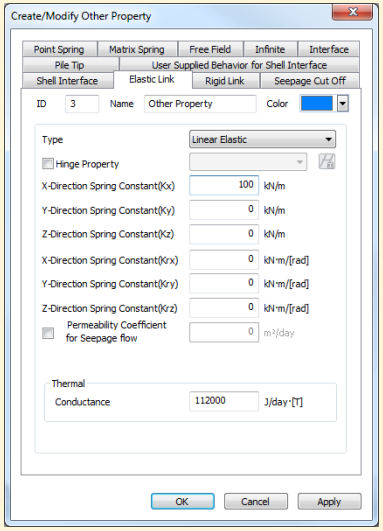
[Orthotropic material thermal input]



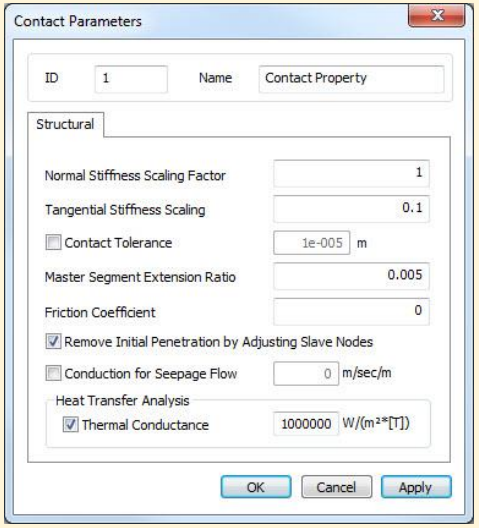
[Interface/Pile material thermal input]



[Rigid link property card]



[Elastic link property card]

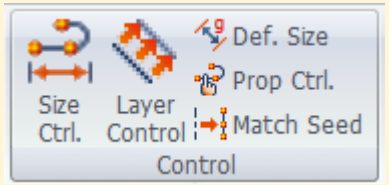


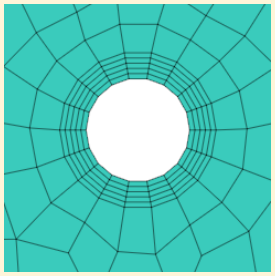
[Contact property card]

2. Pre/Post Processing

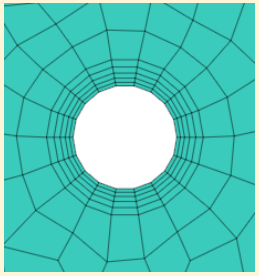
2.2 Layer Control tool

- This tool creates several layers of mesh around holes (circular closed shapes) to obtain more accurate stress concentration result.

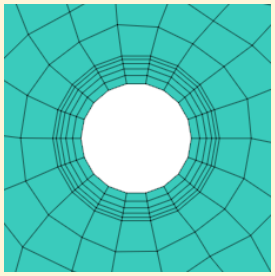




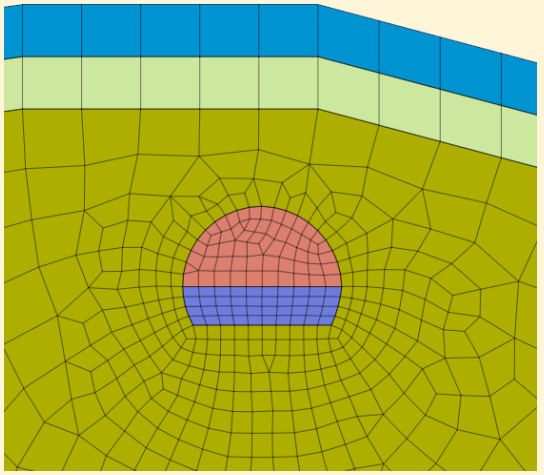
Layer Growth Rate=1



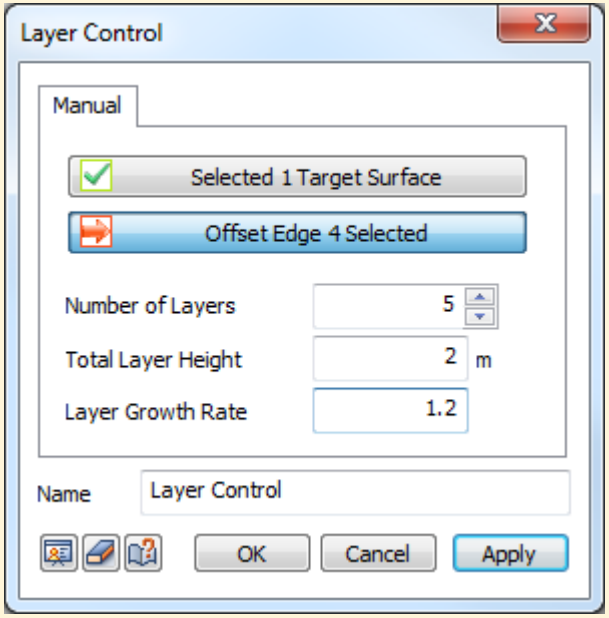
Layer Growth Rate=1.2



Layer Growth Rate=0.8



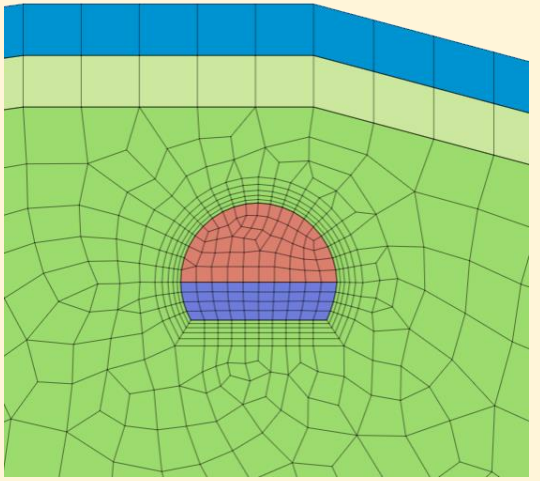
[Tunnel shape – mesh without layer control]



Number of Layers
Specify the number of layers to be offset (minimum value 1).

Total Layer Height
Specifies the height of the total number of boundary layers.

Layer Growth Rate
Proportionally adjusts the height value as the layer advances when the number of boundary layers is 2 or more.



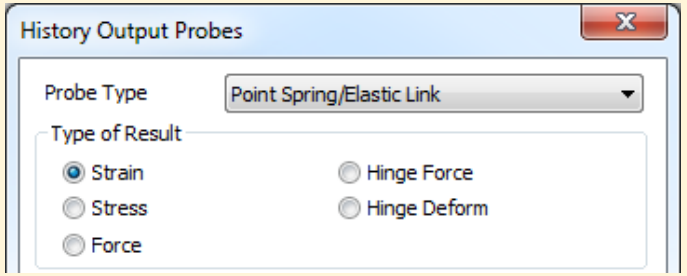
[Mesh created with Layer Control tool]
5 layers, 2m total layer height, 1.2 growth rate factor

2. Pre/Post Processing

2.3 Improvements in History Output Control: Probes and Graph

Analysis > History > History Output Probes

- New Probe Type has been added: Spring/Elastic Link
- New Result Type has been added: Hinge Force, Hinge Deform

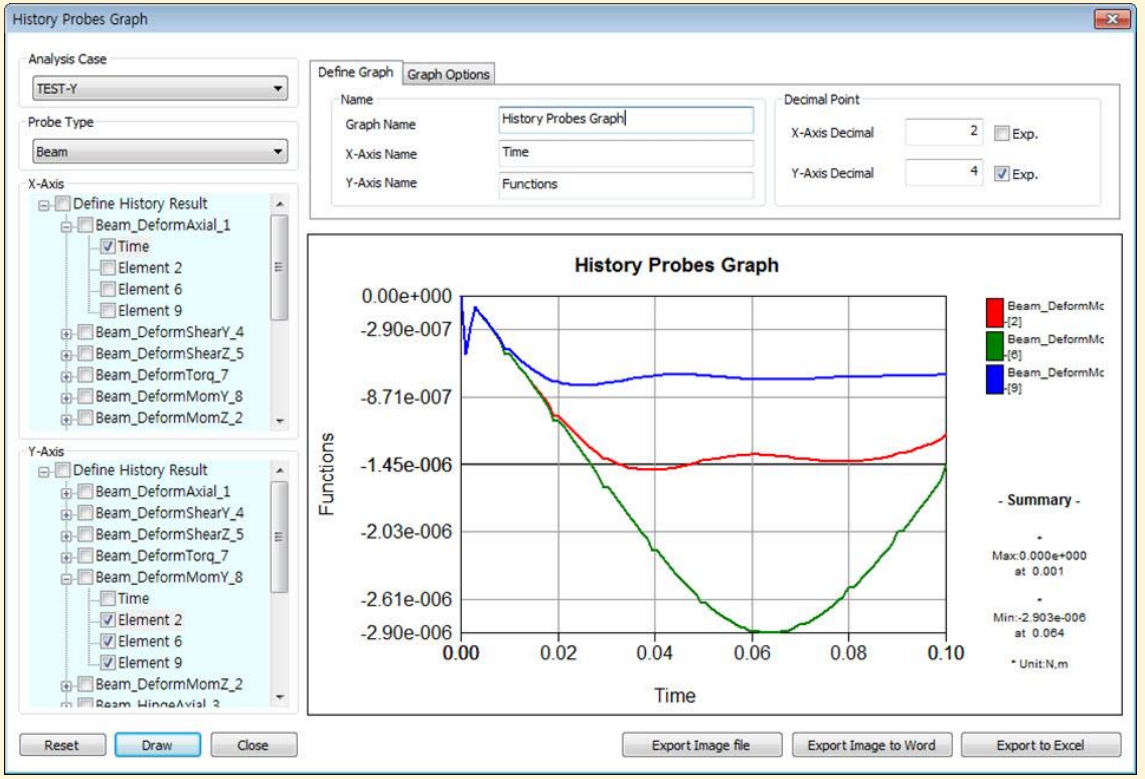


Probe Type	Type of Result
Truss/Em Truss/Geogrid(1D)	Hinge Force
	Hinge Deform
Beam/Embedded Beam	Hinge Force
	Hinge Deform
Spring/Elastic Link	Hinge Force
	Hinge Deform
	Strain
	Stress
	Force

[History Output Probes]

Result > Special Post > History > Graph

- History Probes Graph allows for customizing displayed data for both X and Y axes.



[History Result graph]

2. Pre/Post Processing

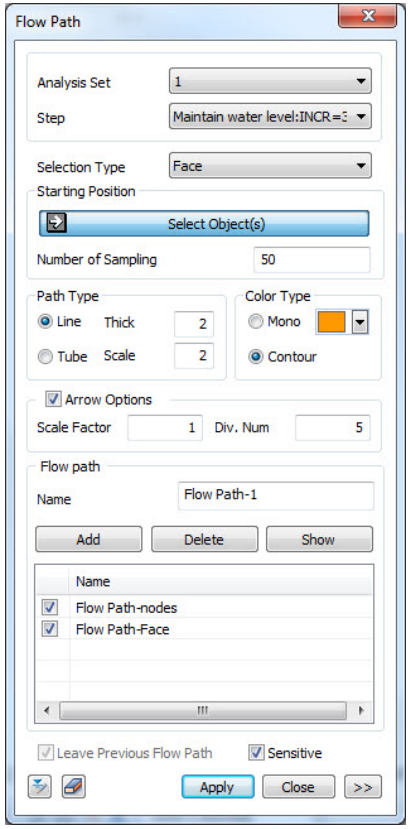
2.4 Flow Path

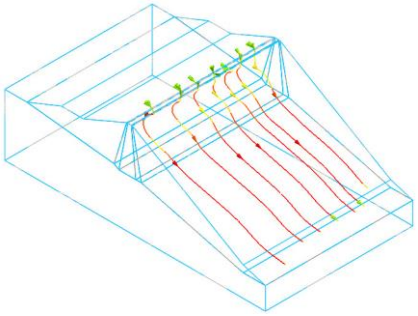
- Multi-path selection tool has been added. Now, based on selection method (Node/Face) the flow lines can be stored and displayed independently.

2.5 Slice Plane Vector

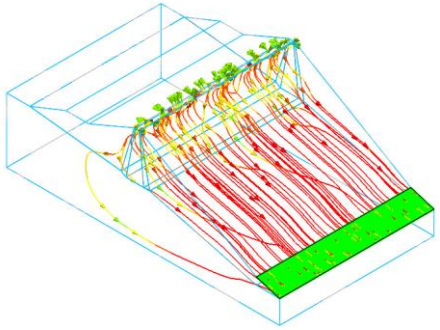
Tool has been equipped with new option allowing control of vector display on your model. Number of displayed vectors can be controlled by Show Uniform option.

▪ **Tools > Special Post > Seepage > Flow Quantity**



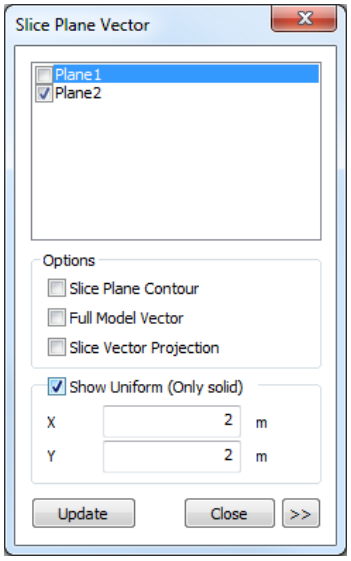


[Nodal based flow paths – Set 1]

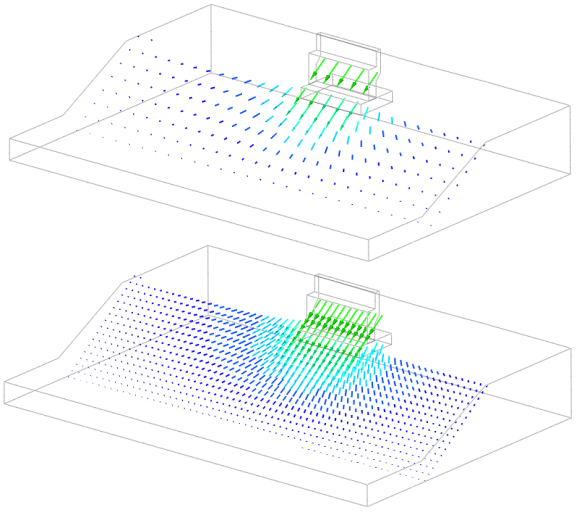


[Face based flow paths – Set 2]

▪ **Result > General> Vector > Slice Plane Vector**



[Show Uniform (only Solid)] - Controls display of vector density on the screen. This option is model size sensitive - it is recommended to check dimensions of the model (geometry and/or element size) to insert reasonable input.



[Slice Plane Vector control]

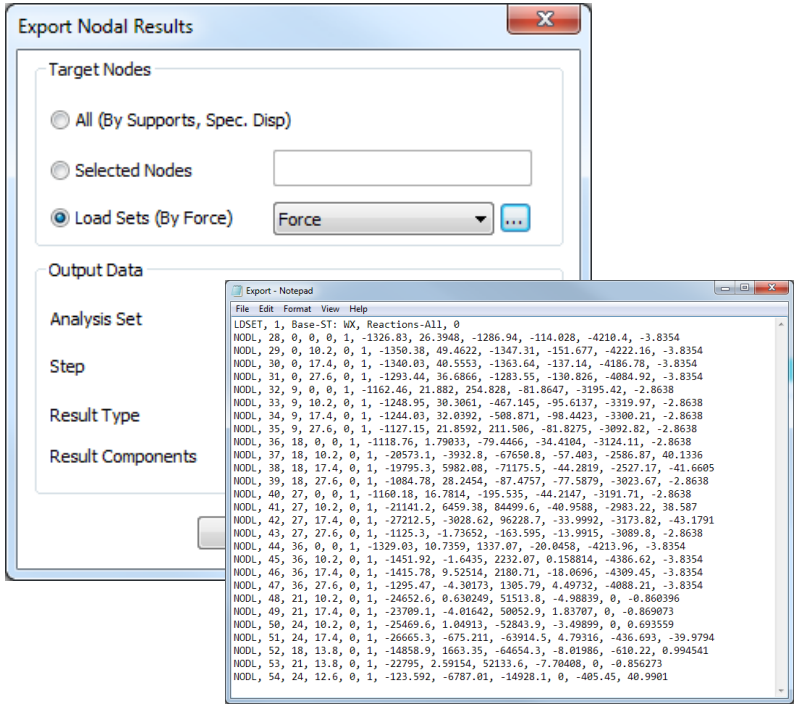
2. Pre/Post Processing

2.6 Export Nodal Results by Load Set

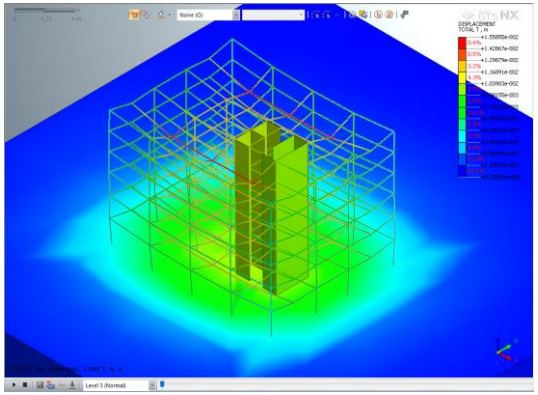
Export Nodal Results feature from GTSNX model to midas Gen / Civil allows for outputting the results of the reaction / displacement results calculated from the existing constraint locations as well as the force-displacement results of the nodes subjected to the nodal load.

Load Sets (by Force) - reaction force / displacement will be exported at only the nodes where Load Set has been defined (as Force type only) according to the analysis set and step chosen by the user. User defined choice for specific load set is allowed.

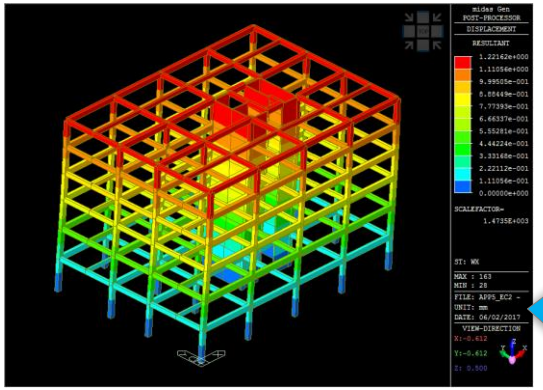
- ***Home > Export > Export Nodal Results (*.txt)…***



[Export results in .txt format]



Analysis Results from GTS NX



Analysis Results from Gen/Civil

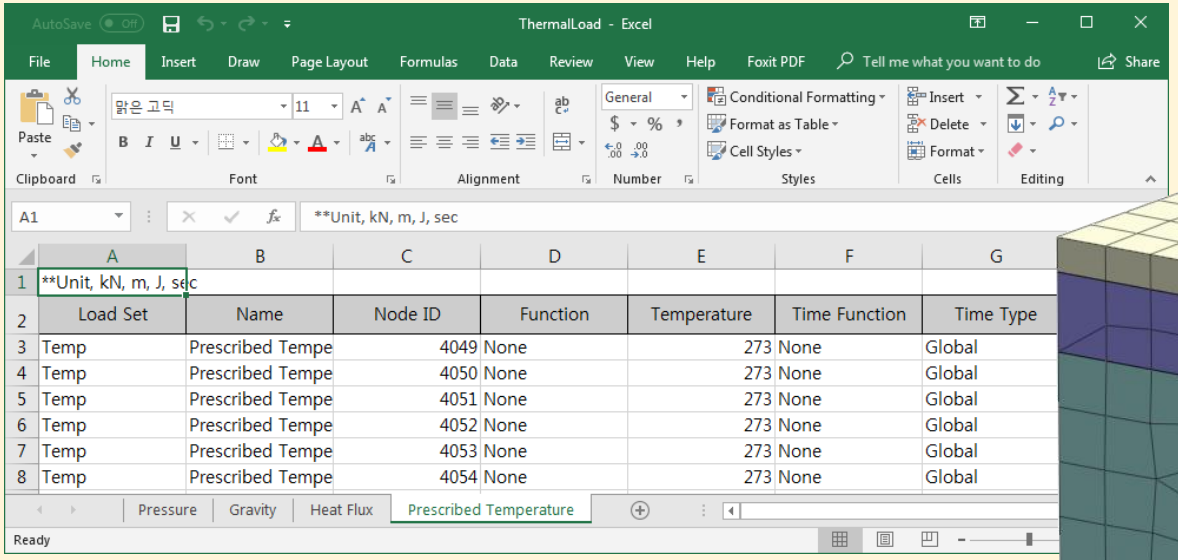
The 'force-displacement' type will simultaneously output the value of the nodal load considered in the load set and the displacement result calculated at the corresponding node after the analysis and will be loaded in Gen / Civil as Multi-Linear Point Spring property.

2. Pre/Post Processing

2.7 Load Table import/export for thermal analyses

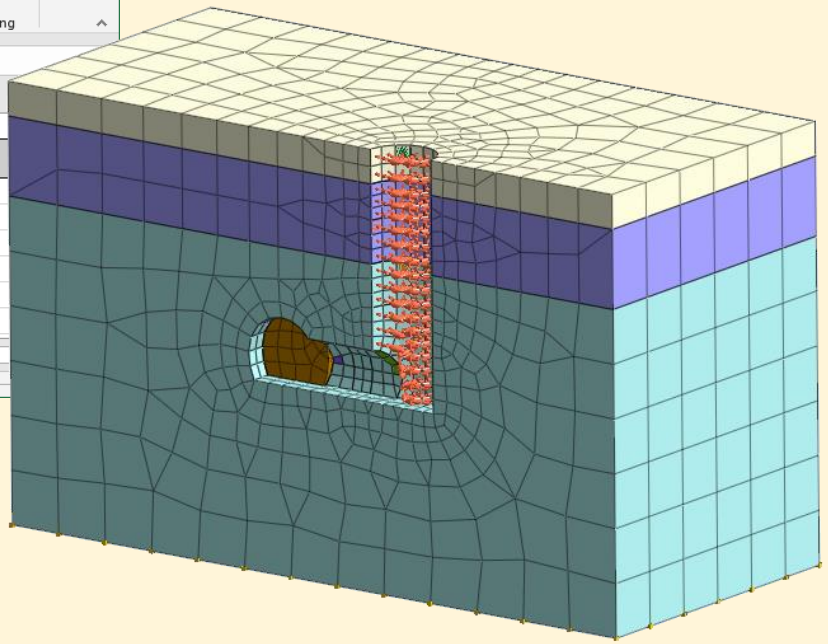
Cards for thermal loads has been added to manage large amount of data.
Users can import the amounts of load sets from excel file and export defined load sets (node/element number, magnitude, and direction) as well.

The sample of table for load sets (Load Table Sample.xlsx) can be found in the installation folder. (ex. C:\Program Files\MIDAS\GTS NX\Sample)



	A	B	C	D	E	F	G
1	**Unit, kN, m, J, sec						
2	Load Set	Name	Node ID	Function	Temperature	Time Function	Time Type
3	Temp	Prescribed Tempe	4049	None	273	None	Global
4	Temp	Prescribed Tempe	4050	None	273	None	Global
5	Temp	Prescribed Tempe	4051	None	273	None	Global
6	Temp	Prescribed Tempe	4052	None	273	None	Global
7	Temp	Prescribed Tempe	4053	None	273	None	Global
8	Temp	Prescribed Tempe	4054	None	273	None	Global

[Thermal Load Set to import]



[Imported Temperatures and Heat Fluxes]