

Release Notes

Release Date: August, 2018

Product Ver.: GTSNX 2019(v1.1)

GTS W Geo-Technical analysis System New eXperience





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





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



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



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.



[Construction Stage Set window]





[Model with applied thermal loads]



*Available upon request

+4.94690e-001

+4.53466e-001

+4.12242e-001

+3.71018e-001

+3.29794e-001

+2.88569e-001

+2.47345e-001

-+2.06121e-001

+1.64897e-001

-+1.23673e-001

+8.24484e-002

+4.12242e-002

+0.00000e+000

1. Analysis

Results Item

Ē

÷

•

1.1 New Heat Transfer, Thermal Stress, Seepage-Thermal Stress analysis types (Construction Stage)



[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.

			-	log(time	e)
Parameter	Description	Reference value (kN, m)			•
	Soil stiffness and failure				
λ	Compression index	Cc / 2.303			
к	Swelling index	Cs / 2.303 (Cc / 5 for a rough estimation)			
μ	Creep index	Cc / 20 for a rough estimation	Primary Second	ary	
С	Cohesion	Failure parameter as in MC model	strain	•	
φ	Friction angle	Failure parameter as in MC model	_ ▼		
Ψ	Dilatancy angle	0	Over Consolidation Ratio(OCR)	1	
ensile Strength	Cut off value for tensile hydrostatic pressure	-	Slope of Consol Line(Lamda)	0.3	
	Advanced parameters (Recommend to use Re	ference value)	Slope of Over Consol Line(k)	0.05	
KNC	Ko for normal consolidation	1-sinφ (< 1)	K0nc	0.5	
	Cap vield surface	-	Pc 🛛 User Defined	0	kN/m
			Cap Shape Factor(Alpha)	0.22	
OCR / Pc	OCR / Pc Ver Consolidation Ratio / When entering bo	When entering both parameters,	Cohesion(C)	1	kN/m
	Pre-overburden pressure	Pc has the priority of usage	Friction Angle(Phi)	36	[deg
α	Cap Shape Factor (scale factor of pre consolidation stress)	from KNC (Auto)	Dilatancy Angle	36	[deg
	, ,		Topsile Strength	10	LNU

GTS NX 2019 Enhancements

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.

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)

Analysis Case Se	tting						
Title	NTH_SRM					Time Step	M
Description						Analysis Control	P
Solution Type	Nonlinear Time Histor	y + SRM			-	Output Control	P
Construction Sta	ige Set				-		
Analysis Case Mo	del		<<	>>	Active 5	iets	
Mesh Aspi Aspi Subi Subi Subi Subi Subi Subi Subi Sub	halt base grade y Condition ndary Set-1 ndary Set-2 Load D Pair			Mesh Mesh De De Su Su Su Su Su Su Su Su Su Su	phalt draut Mesh Set bbgrade ary Condition undary Set-1 ic Load VD YD X Pair ary Condition (for SF undary Set-2	RM)	

[NTH Analysis with separate SRM boundary condition]

roject Title	Engineer	
esc.		
Model Type	Gravity Direction	1
() 3D	© Y	
© 2D	🔘 Z	
Axisymmetric		
Unit System	▼] [] ▼] [se	ec 🔻
Unit System	J Se Se	2C ▼
Unit System Initial Parameters W Unit Weight of Wate Conductivity of Wate	J J se ter Parameters 9.80665 kN/m 0.591 W/r	ec ▼ 1 ³ n·[T])
Unit System Initial Parameters W Unit Weight of Wate Conductivity of Wate Specific Heat of Wate Specific Heat of Wate	J See	ec ▼ 1 ³ n·[T]) m·[T])
Unit System KN (KN (M)	J J Se Se	ec ▼ 1 ³ n·[T]) n·[T])

[Analysis Setting window]

Output Control	×
Output Type Output Option	
Write Results of All Active Mesh Sets	
Nodal Results	Element Results
Displacement Mesh Set	Force Mesh Set
Applied Load Mesh Set	Stress Mesh Set
Reaction Force Mesh Set	Strain Mesh Set
Grid Point Force Mesh Set	Status Mesh Set
	Ductility Mesh Set
Output Option	Element Output Location
Binary	Element Corner Results
	Shell Mid-Plane Results
	Number of Beam Output Segments 4
	OK Cancel

[Strain option for Inreface Relative Displacement result]



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.

Material X					
ID 1 Name Top Groun	nd Color 🔽				
Model Type Mohr-Coulomb					
General Porous Non-Linear Therr	nal Time Dependent				
Conductivity	138240 J/(m·day·[T])				
Specific Heat	1800 J/(ton·[T])				
Heat Generation Factor	1				
Unfrozen water content					
OK Cancel Apply					

[Isotropic material thermal input]

Cre	Create/Modify Other Property						
	Point Spring Matrix Spring		Spring	Free Field	Infinite	Interface	
	Pile Tip Us		User S	User Supplied Behavior for Shell Interface			
	Shell I	nterface	Ela	stic Link	Rigid Link	Seepa	age Cut Off
	ID	3	Name	Other F	Property	Color	
	Prope	erties					
	V	DX	V DY	V DZ	RX	RY	RZ
	Typic	al Type	,				
	Rigid Body					Plane X-Y	
	Plane Y-Z Plane Z-X						
	Seepage Flow DOF Temperature						
				(ок 📃 🖂	Cancel	Apply

[Rigid link property card]

iterial			l	x		
ID 3 I	Name Orthotropic	0	plor	•		
Model Type Transversely Isotropic Structure						
Parameter1 Param	neter2 Porous Th	ermal				
Conductivity	125000	125000	125000			
	Symmetry	125000	125000			
	Unit: J/(m·day·[T])	125000			
Specific Heat 1750 J/(ton·[T])						
	ОК	Can	icel Appl	у		

[Orthotropic material thermal input]

Pile Tip	User Si	upplied Behavior fo	or Shell Interface
Shell Interface	Elastic Link	Rigid Link	Seepage Cut Of
ID 3	Name Other P	roperty	Color
Туре		Linear Elastic	•
I Hinge Proper	ty		× /4
X-Direction Sprin	g Constant(Kx)	100	kN/m
Y-Direction Sprin	g Constant(Ky)	0	kN/m
Z-Direction Sprin	g Constant(Kz)	0	kN/m
X-Direction Sprin	g Constant(Krx)	0	kN·m/[rad]
Y-Direction Sprin	g Constant(Kry)	0	kN·m/[rad]
Z-Direction Sprin	g Constant(Krz)	0	kN·m/[rad]
Permeabilit for Seepag	y Coefficient e flow	0	m²/day
Thermal Conductance		112000	J/day·[T]
Thermal Conductance		112000	J/day·[T]

[Elastic link property card]

х Material ID 3 Name Interface Color • Interface Model Type • General Seepage Thermal 1728000 J/(m2·day·[T Convection coefficient ОК Cancel Apply

[Interface/Pile material thermal input]

ID 1	Name	Contact Property	
Structural			
Normal Stiffne	ess Scaling Factor		1
Tangential Sti	ffness Scaling		0.1
Contact To	olerance	1e-005 m	
Master Segme	ent Extension Ratio	0.	005
Friction Coeff	icient		0
Remove Ir	nitial Penetration by A	djusting Slave Nodes	
Conduction	n for Seepage Flow	0 m/sec/m	
Heat Transf	er Analysis		
🚺 Therma	l Conductance	1000000 W/(m2*[T])

[Contact property card]

MIDAS

GTS NX 2019 Enhancements

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]

Layer Control	×					
Manual						
Selected 1 Target Surface						
Offset Edge 4 Selected						
Number of Layers	5 🛋					
Total Layer Height	2 m					
Layer Growth Rate 1.2						
Name 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



2.3 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

History Output Pro	bes 🛛 🔍
Probe Type	Point Spring/Elastic Link 🔻
Type of Result	
Strain	Hinge Force
Stress	Hinge Deform
Force	

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

Result > Special Post > History > Graph

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



[History Output Probes]

[History Result graph]

GTS NX 2019 Enhancements

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.



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)...

oort Nodal Results		
Target Nodes		
All (By Supports, S)	ipec. Disp)	
Selected Nodes		
Load Sets (By Ford)	re) Force 🔻 🛄	
Output Data		
	Export - Notepad	-
Analysis Set	File Edit Format View Help LDSET, 1, Base-ST: MX, Reactions-All, 0 NODL, 20, 0, 0, 0, 1, -1326.83, 26.3948, -1286.94, -114.028, -4210.4, -3.8354 NODL, 20, 0, 0, 0, 1, -1326.83, 26.3948, -1286.94, -114.028, -4210.4, -3.8354	
Step	NULL, 29, 0, 10.2, 0, 1, -1590.38, 49.4022, -1347.31, -131.677, -4222.10, -3.8354 NODL, 30, 0, 17.4, 0, 1, -1340.83, 40.5553, -1365.64, -137.14, -4186.78, -3.8354 NODL, 31, 0, 27.6, 0, 1, -1293.44, 36.6866, -1283.55, -130.826, -4884.92, -3.8354 NODL, 32, 9, 0, 0, 1, -1162.46, 21882, 254, 828, -81.8647, -3195.42, -2.8638	
Result Type	NODL, 33, 9, 10.2, 0, 1, -1248.95, 30.3061, -467.145, -95.6137, -3319.97, -2.6638 NODL, 34, 9, 17.4, 0, 1, -1244.03, 32.0392, -508.871, -98.4423, -3300.21, -2.6638 NODL, 35, 9, 27.6, 0, 1, -1127.15, 21.8592, 211.506, -81.6275, -309.282, -2.8638	
Result Components	NOUL, 36, 18, 6, 6, 1, -1118.76, 1.70433, -73.4466, -34.4144, -3124.11, -2.8638 NOUL, 37, 18, 10.2, 6, 1, -245731, -3932.8, -67568, -57.403, -526.67, 461.1366 NOUL, 38, 18, 197.4, 6, 1, -137953, 5932.48, -71175, -44.2139, -2527.17, -41.6609 NOUL, 39, 27, 2, 6, 6, 11.463, 37, 62.3244, 565, 557, -71775, 712, 723, 723, 724, 724, 724, 724, 724, 724, 724, 724	
	MODL, 46, 36, 102, 9 1, -125, 105, 107, 107, 2232, 207, 015, 5814, -4366, 62, -3, 8354 MODL, 46, 36, 17, 4, 0, 1, -1451, 78, 9, 52514, 2180, 71, -18, 6866, -4309, 45, -3, 8354 MODL, 46, 36, 17, 4, 0, 1, -1451, 78, 9, 52514, 2180, 71, -18, 6866, -4309, 45, -3, 8354 MODL, 47, 36, 27, 6, 0, 1, -1254, 74, -4, 30173, 1305, 79, 44, 9732, -4088, 21, -3, 8354	
	100L, 48, 21, 102, 0, 1, 24622, 6, 0.632249, 51513.8, -4.98839, 0, -0.869396 100L, 49, 21, 17-4, 0, 1, 27083, 1, -4.01642, 59052, 9, 1.83767, 0, -0.859373 100L, 59, 24, 102, 0, 1, 25463, 6, 1.04913, -52843.9, -3.49899, 0, 0.695559 100L, 51, 24, 17-4, 0, 1, 26665.3, -675,211, -639145, 5, 4.79316, -36.6693, -39.9734	
	NODL, 52, 18, 13.8, 0, 1, -14858.9, 1663.35, -64654.3, -8.01986, -610.22, 0.994541 NODL, 53, 21, 13.8, 0, 1, -22795, 2.59154, 52133.6, -7.70408, 0, -0.856273	

[Export results in .txt format]



Analysis Results from Gen/Civil

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)

