

The background features a complex network of interconnected nodes and lines, resembling a mesh or a molecular structure. The nodes are small circles, and the lines are thin, connecting the nodes in a non-uniform pattern. The overall color scheme is a gradient from green on the left to orange and red on the right.

GTS INX

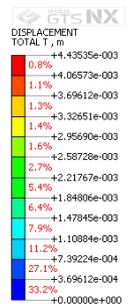
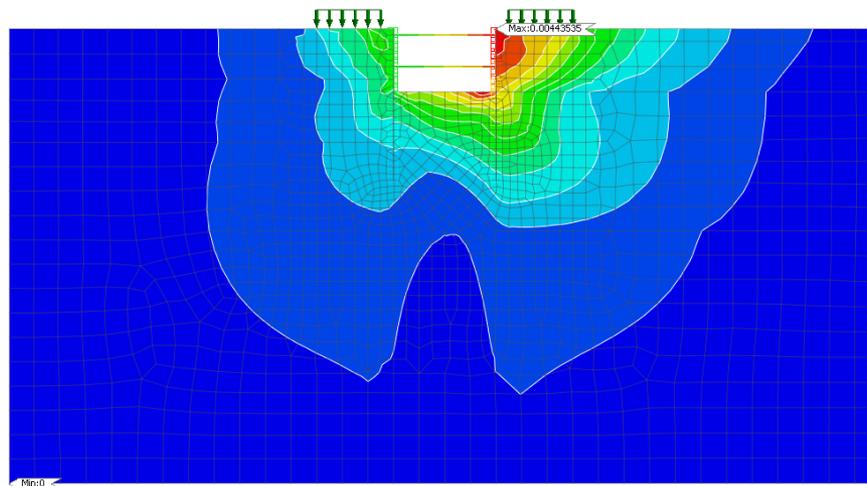
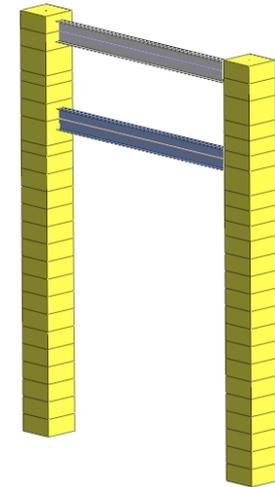
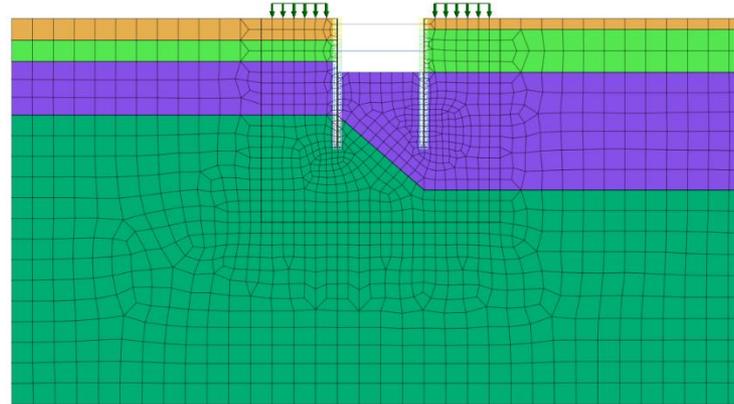
New eXperience of Geo-Technical analysis System

Earth Retaining Stabilizing System

MIDAS

Procedure

- This tutorial is aiming on performing the analysis with 3 stages of excavation including retaining wall and strutting.
- Surcharge and undrained condition are considered in this analysis.
- This example was prepared based on the most general excavation work in the real practice.



[DATA] Excavation, FEL, INCR=1 (LOAD=1.000), [UNIT] kN, m



Ground

[unit : kN, m]

Name	Fill	Fluvial Clay	Old Alluvium (C)	Old Alluvium (A)
Material	Isotropic	Isotropic	Isotropic	Isotropic
Model Type	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb
General				
Elastic Modulus (E) [kN/m²]	10,000	13,000	133,333	400,000
Poisson's Ratio (ν)	0.25	0.25 (0.495)	0.25	0.25
Unit Weight (γ) [kN/m³]	18	19	20	21
Ko	0.5	1.0	0.5	0.5
Porous				
Unit Weight (Saturated) [kN/m³]	18	19	20	21
Drainage Parameters	Drained	Undrained	Drained	Drained
Non-Linear				
Cohesion (c) [kN/m²]	1	40	5	20
Frictional Angle (Φ) [deg]	30	0	30	35

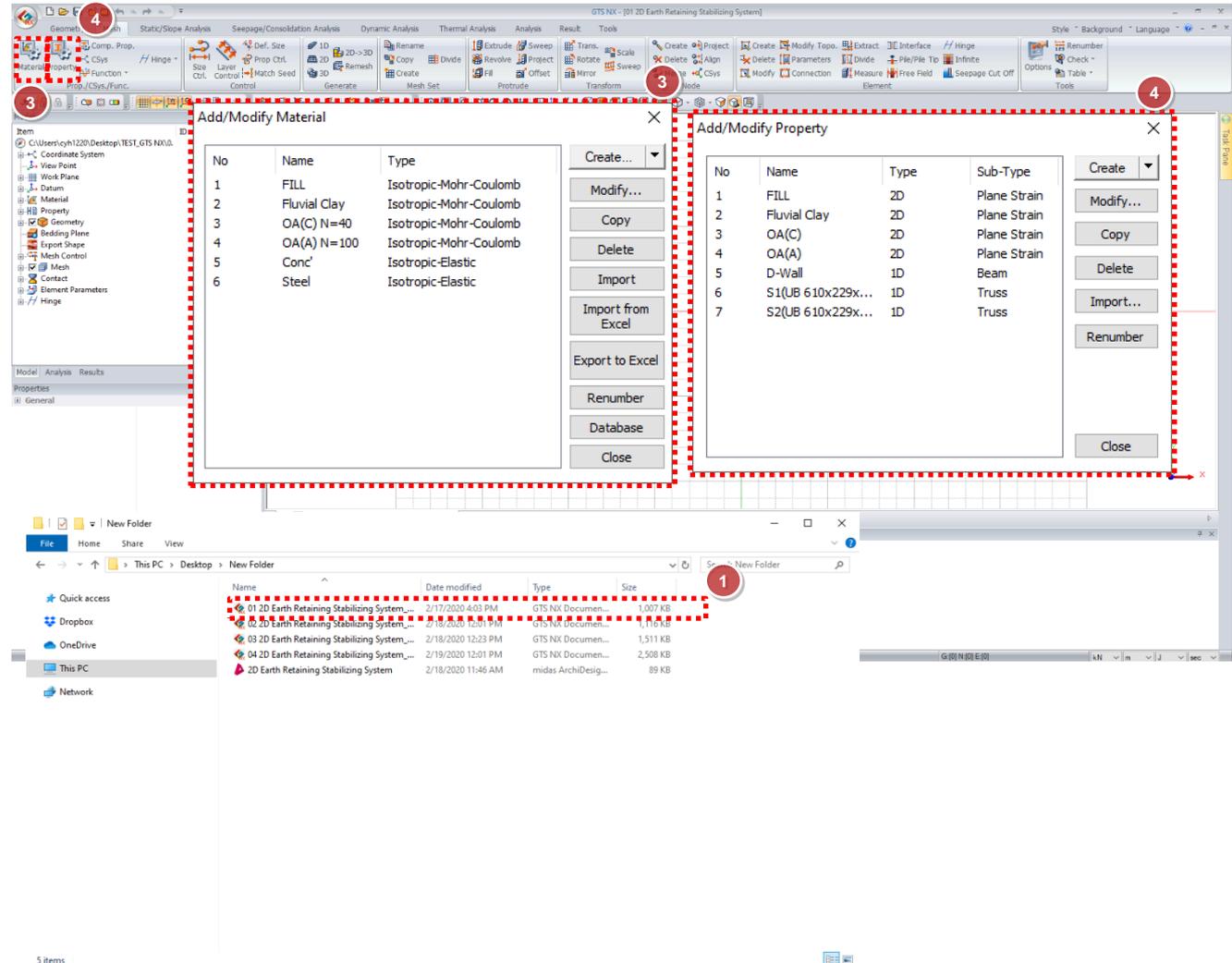
Structure

[unit : kN, m]

Name	Conc'	Steel
Material	Isotropic	Isotropic
Model Type	Elastic	Elastic
Elastic Modulus (E) [kN/m ²]	28,000,000	205,000,000
Poisson's Ratio (ν)	0.15	0.15
Unit Weight (γ) [kN/m ³]	24	74

Procedure

- 1 Open [2D Earth Retaining Stabilizing System_Start] file
- 2 Save as another folder (Ctrl + Shift + s)
- 3 Check existing materials (Mesh > Prop./CSys./Func. > Material)
- 4 Check existing properties (Mesh > Prop./CSys./Func. > Property)



The screenshot displays the GTS NX software interface with two dialog boxes open: 'Add/Modify Material' and 'Add/Modify Property'. The 'Add/Modify Material' dialog contains the following table:

No	Name	Type
1	FILL	Isotropic-Mohr-Coulomb
2	Fluvial Clay	Isotropic-Mohr-Coulomb
3	OA(C) N=40	Isotropic-Mohr-Coulomb
4	OA(A) N=100	Isotropic-Mohr-Coulomb
5	Conc'	Isotropic-Elastic
6	Steel	Isotropic-Elastic

The 'Add/Modify Property' dialog contains the following table:

No	Name	Type	Sub-Type
1	FILL	2D	Plane Strain
2	Fluvial Clay	2D	Plane Strain
3	OA(C)	2D	Plane Strain
4	OA(A)	2D	Plane Strain
5	D-Wall	1D	Beam
6	S1(UB 610x229x...	1D	Truss
7	S2(UB 610x229x...	1D	Truss

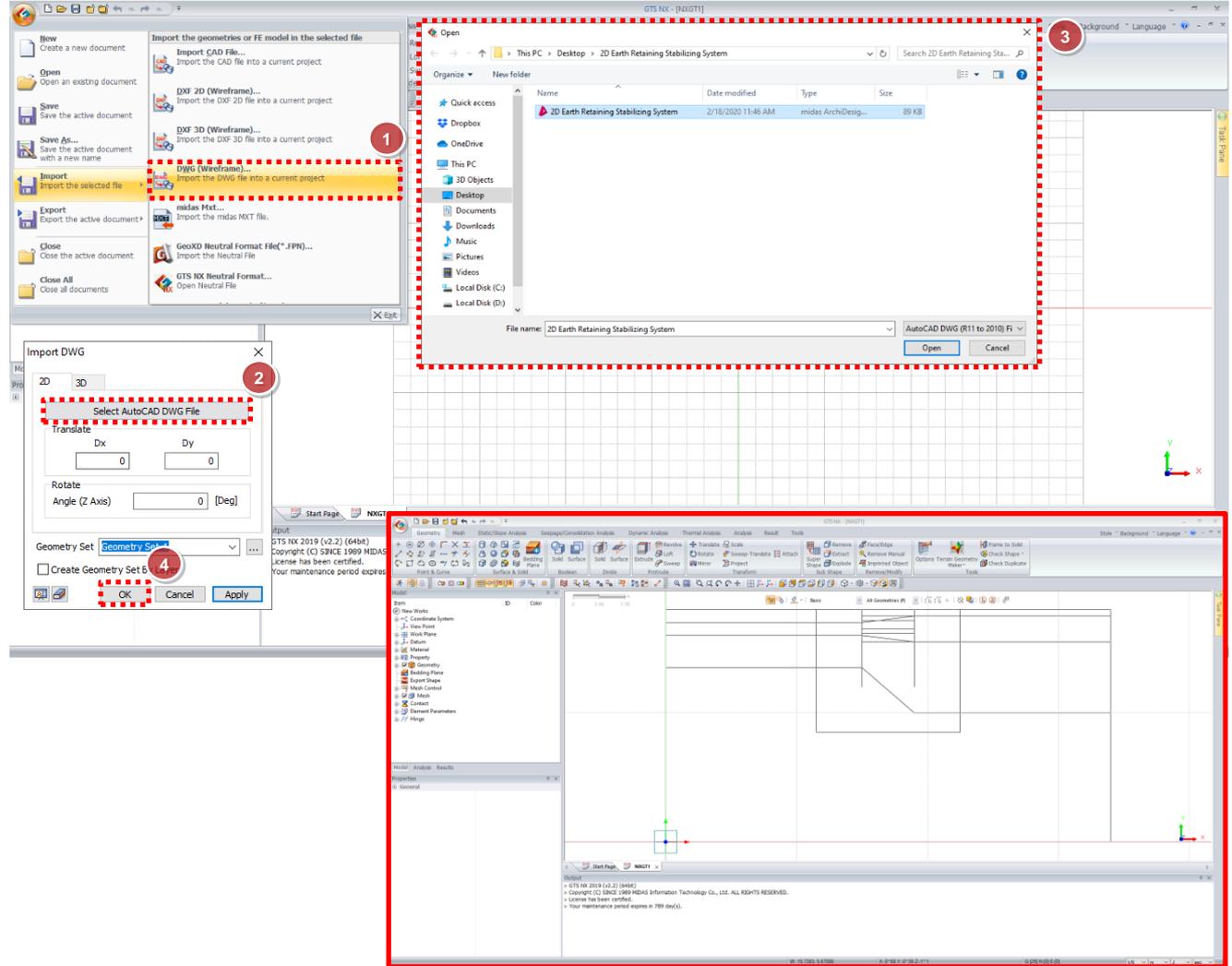
At the bottom, a Windows File Explorer window shows a folder named 'New Folder' on the Desktop. It contains five files:

Name	Date modified	Type	Size
01 2D Earth Retaining Stabilizing System...	2/17/2020 4:03 PM	GTS NX Document...	1,007 KB
02 2D Earth Retaining Stabilizing System...	2/18/2020 12:01 PM	GTS NX Document...	1,511 KB
03 2D Earth Retaining Stabilizing System...	2/18/2020 12:23 PM	GTS NX Document...	1,511 KB
04 2D Earth Retaining Stabilizing System...	2/19/2020 12:01 PM	GTS NX Document...	2,508 KB
2D Earth Retaining Stabilizing System	2/18/2020 11:46 AM	midas ArchiDesig...	89 KB

03 Geometry Modeling

Procedure

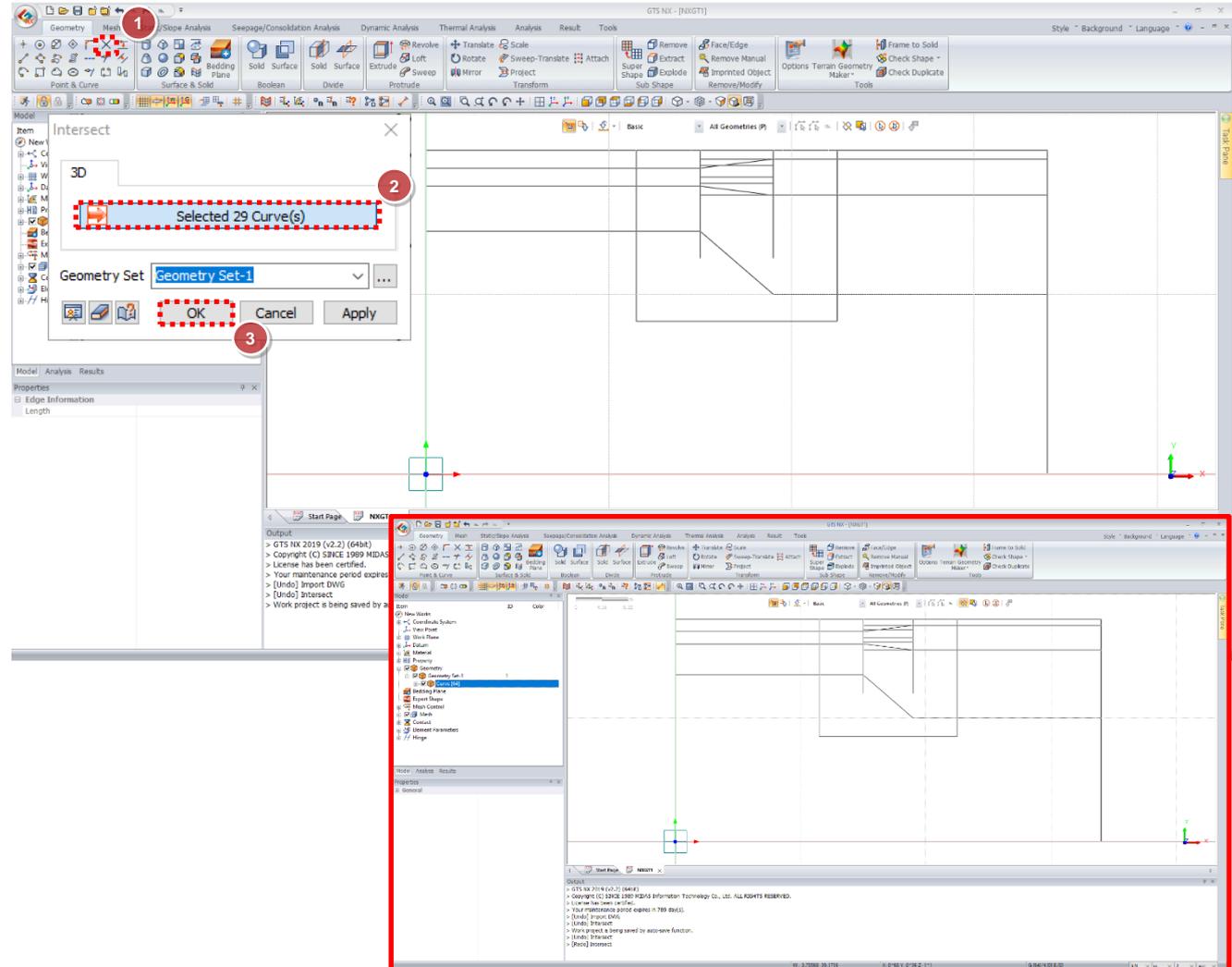
- 1 Click [DWG(Wireframe)]
(File > Import > DWG (Wireframe))
- 2 Click [Select AutoCAD
DWG File]
- 3 Select “2D Earth
Retaining Stabilizing
System”
Click [Open]
- 4 Click [OK]



Procedure

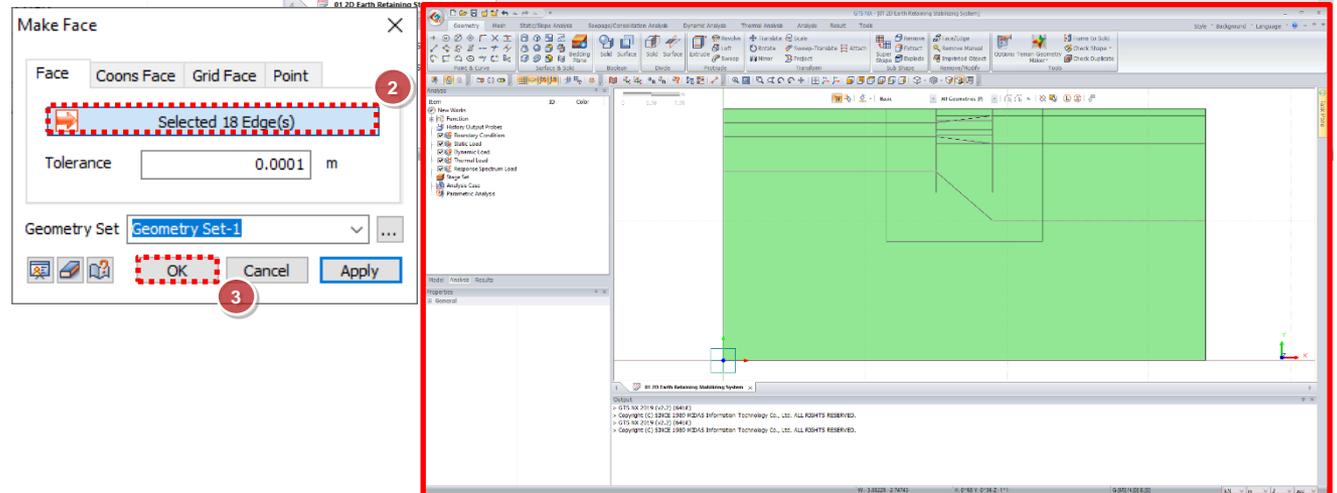
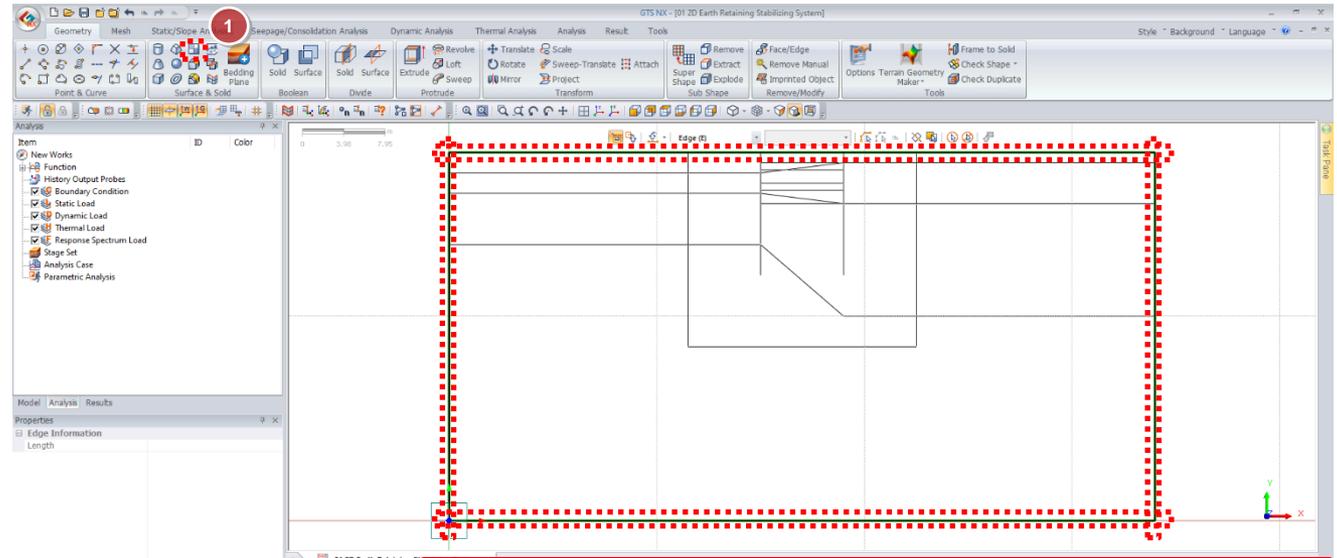
- 1 Click [Intersect]
(Geometry > Point & Curve > Intersect)
- 2 Click [Select Curve(S)]
(29 curves)
Select All lines (Ctrl + a)
- 3 Click [OK]

•Check the curves are divided to 64 curves



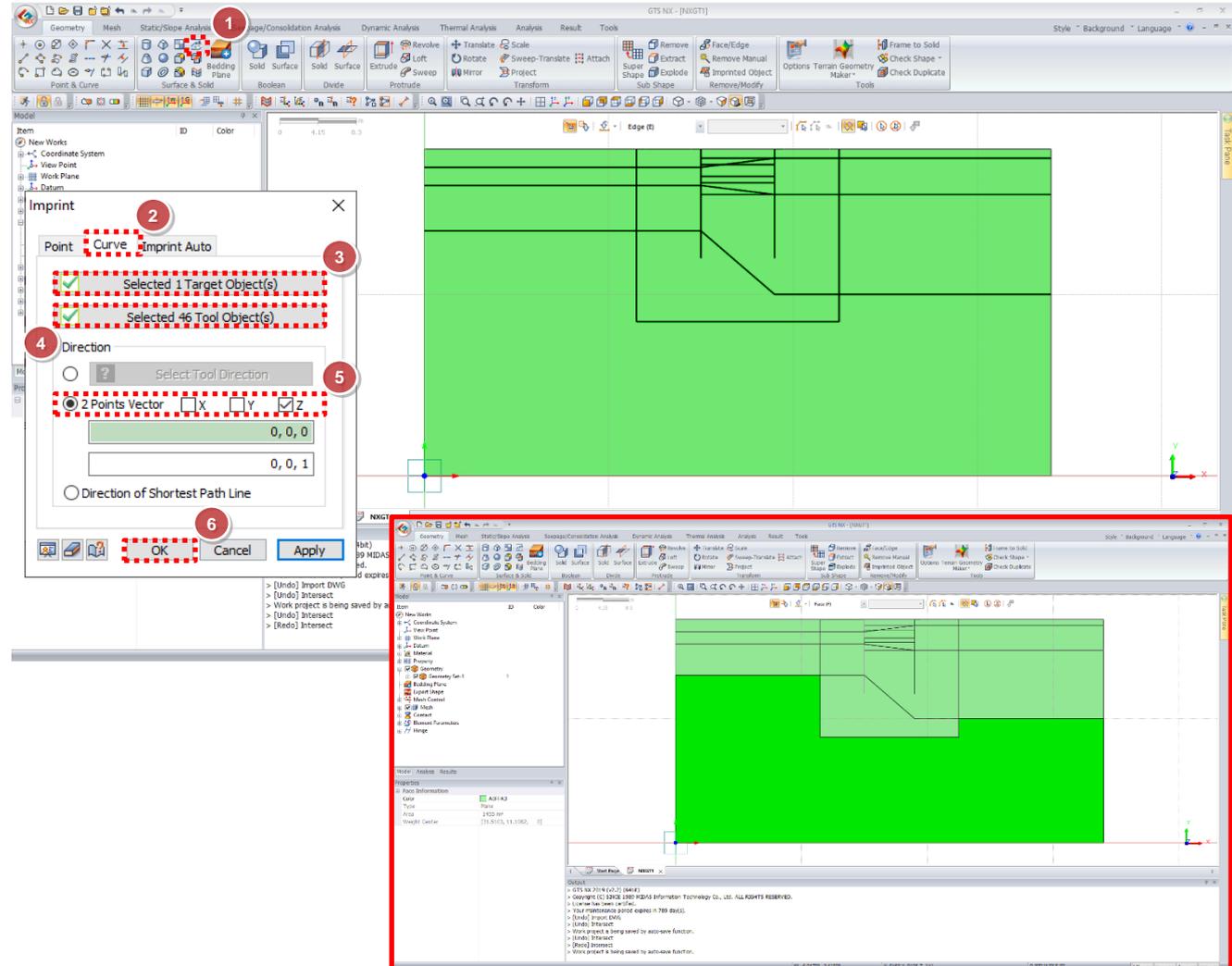
Procedure

- 1 Click [Make Face]
(Geometry > Surface & Solid)
 - 2 Click [Select Edge(S)]
Select outer lines
(18 edges)
 - 3 Click [OK]
- Check the curves are divided to 64 curves



Procedure

- 1 Click [Imprint]
(Geometry >Surface & Solid > Imprint)
- 2 Move to [Curve] tab
- 3 Select Area (Ctrl + a)
(1 area)
- 4 Select inner curves
(46 curves)
- 5 Click [2 Points Vector]
and “check on z direction”
only
- 6 Click [OK]



Procedure

- 1 Click [Prop Ctrl.]
 - 2 Move to [Face] tab
 - 3 Select faces for Fill layer
 - 4 Change property to "Fill"
 - 5 Click [Apply]
- Do it repeatedly step 3 to step 5 until step 13
- 14 Click [OK]

The screenshot displays the GTS NX software interface with four 'Property Control' dialog boxes open, each showing the 'Face' tab. The dialog boxes are numbered 1 through 14, corresponding to the procedure steps:

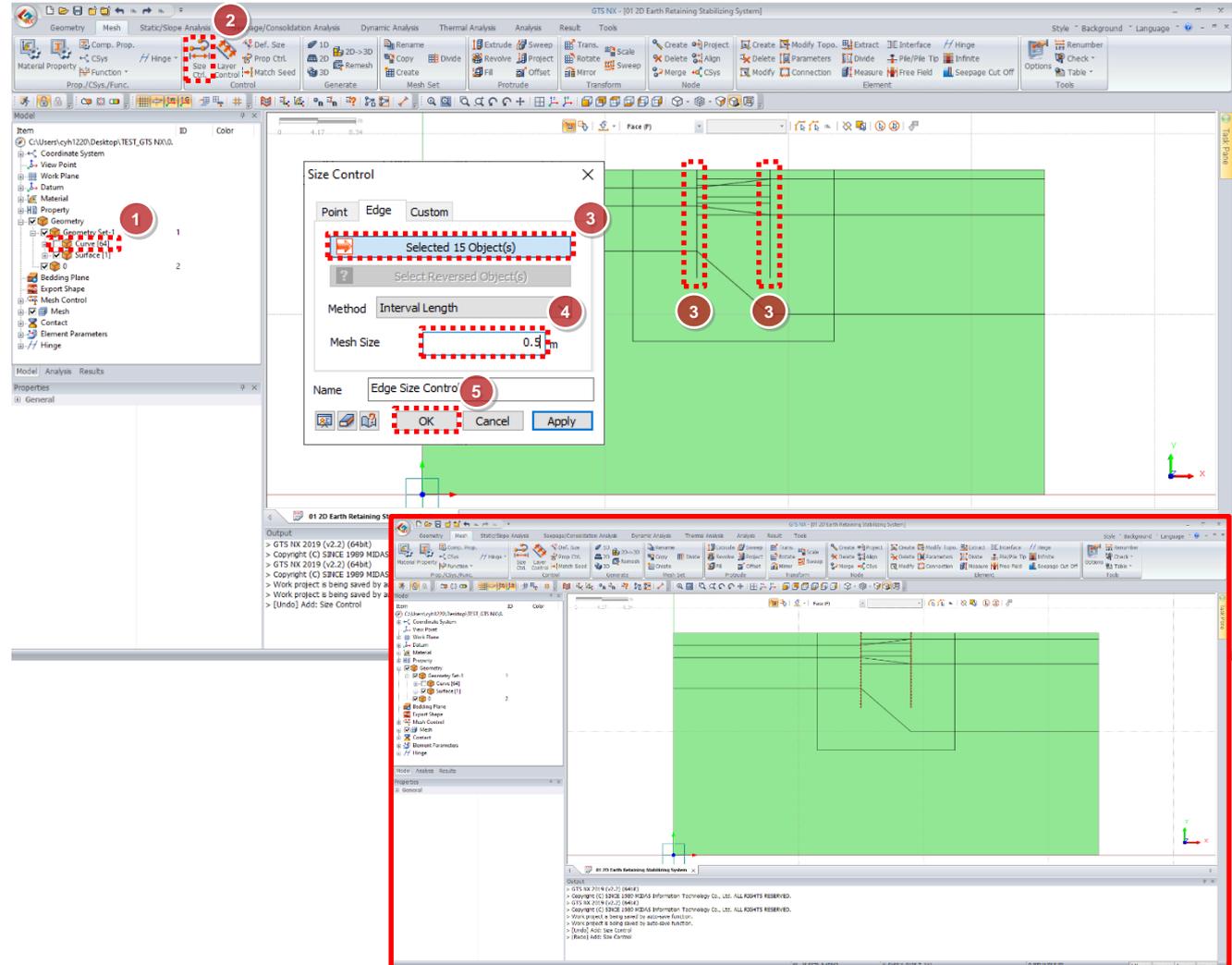
- 1: Click [Prop Ctrl.]
- 2: Move to [Face] tab
- 3: Select faces for Fill layer (Selected 7 Object(s))
- 4: Change property to "Fill" (Property: 1: FILL)
- 5: Click [Apply]
- 6: Select faces for Fluvial Clay layer (Selected 8 Object(s))
- 7: Change property to "Fluvial Clay" (Property: 2: Fluvial Clay)
- 8: Click [Apply]
- 9: Select faces for OA(C) layer (Selected 5 Object(s))
- 10: Change property to "OA(C)" (Property: 3: OA(C))
- 11: Click [Apply]
- 12: Select faces for OA(A) layer (Selected 2 Object(s))
- 13: Change property to "OA(A)" (Property: 4: OA(A))
- 14: Click [OK]

Below the dialog boxes, a cross-section diagram of a retaining wall and soil is shown. The layers are labeled with red dashed lines and text:

- Fill
- Fluvial Clay
- OA(C)
- OA(A)

Procedure

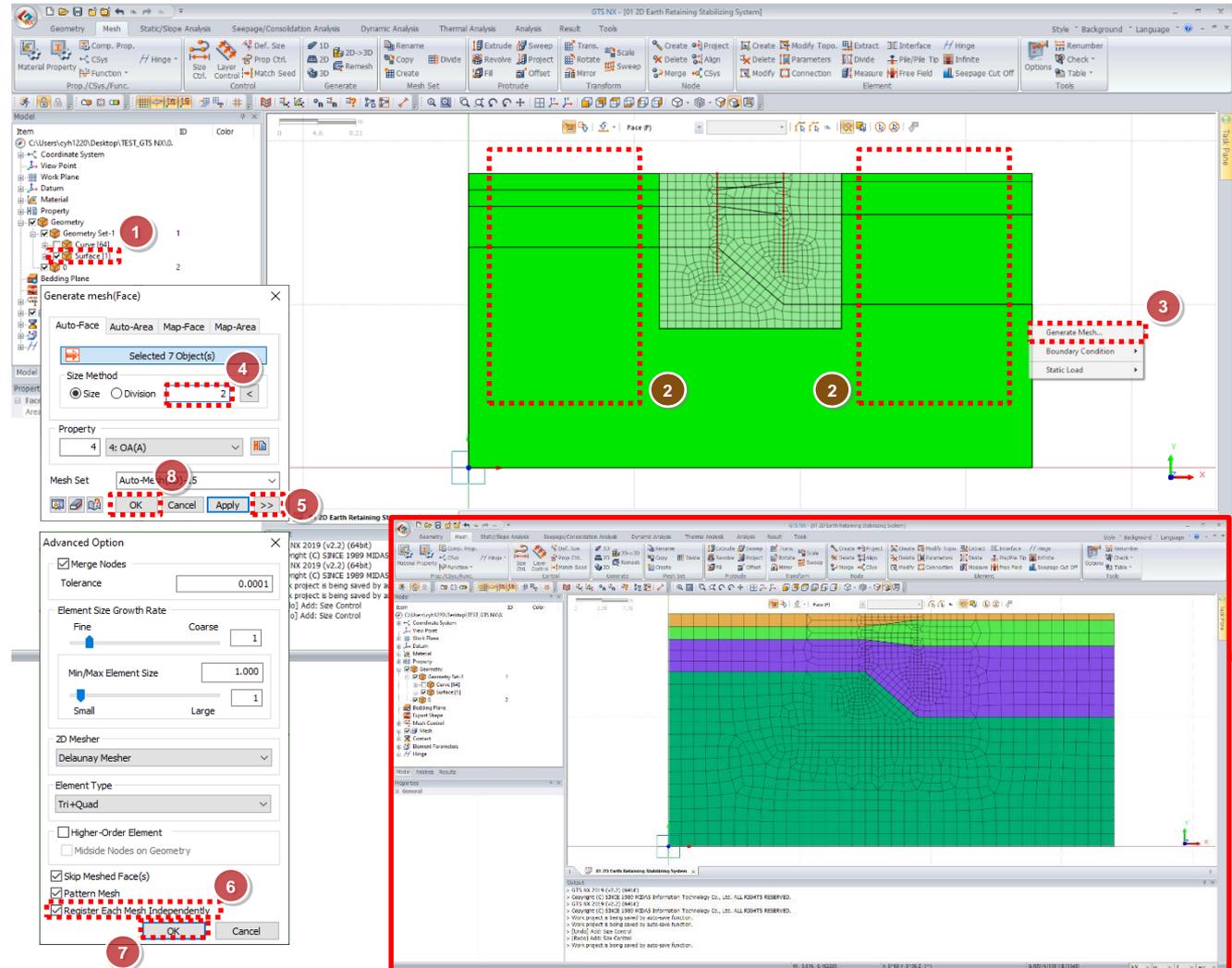
- 1 Check off "Curve [64]" under Geometry tree
- 2 Click [Size Ctrl.] (Mesh > Control > Size Ctrl.)
- 3 Select edges of retaining wall part (15 edges)
- 4 Mesh Size: 0.5m
- 5 Click [OK]



04 Mesh Generation

Procedure

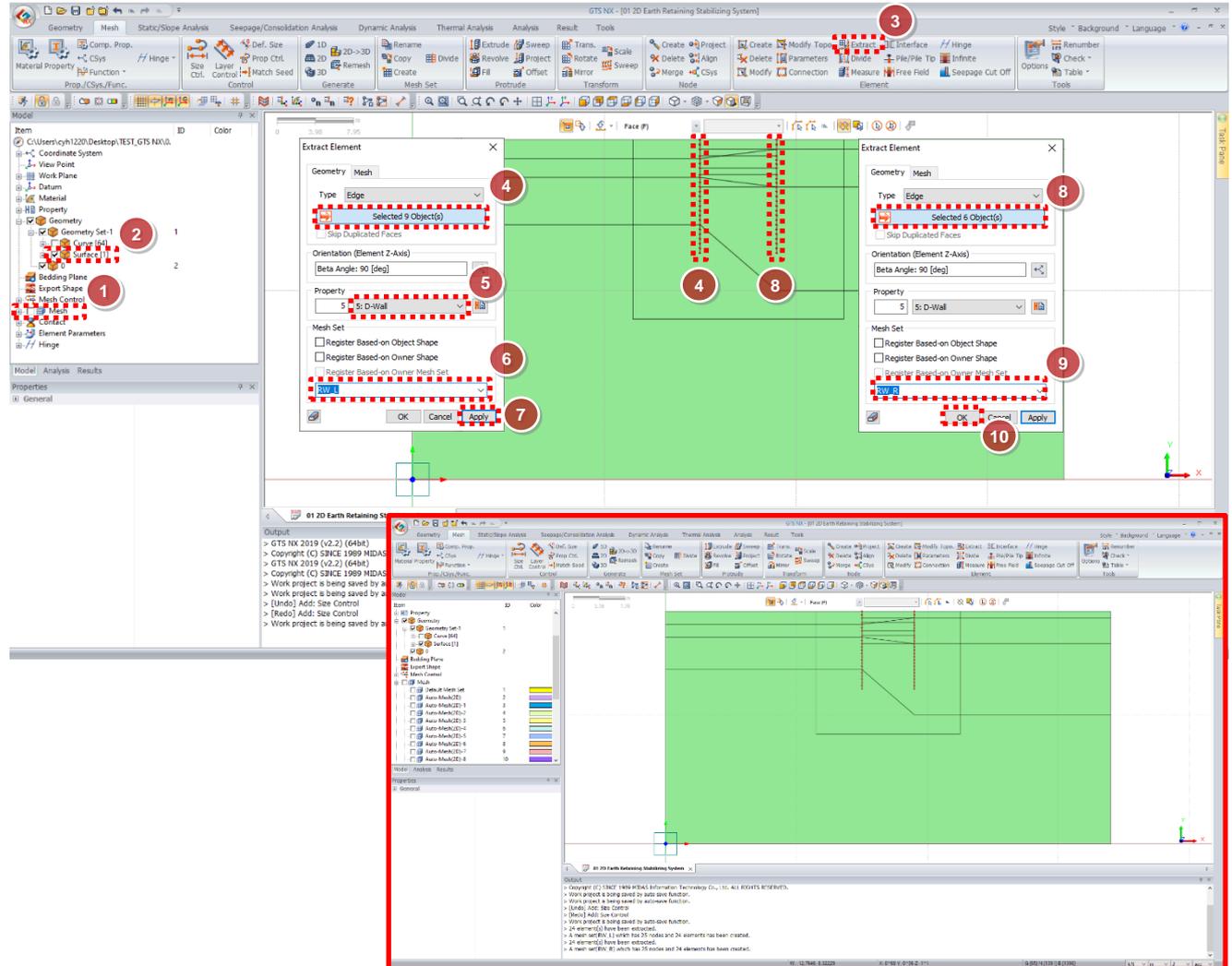
- 1 Check off "Surface [1]" and check on "Surface [1]" under Geometry tree
- 2 Select faces (7 faces)
- 3 Right Click on the selected face and Click [Generate Mesh]
- 4 Size: 2
- 5 Click [>>]
- 6 Check on "Register Each Mesh Independently"
- 7 Click [OK]
- 8 Click [OK]



04 Mesh Generation

Procedure

- 1 Check off “Mesh”
- 2 Check off “Surface [1]” and check on “Surface [1]” under Geometry tree
- 3 Click [Extract] (Mesh > Element > Extract)
- 4 Select edges of left wall (9 edges)
- 5 Change property to “D-Wall”
- 6 Mesh set name: RW_L
- 7 Click [Apply]
- 8 Select edges of right wall (6 edges)
- 9 Mesh set name: RW_R
- 10 Click [OK]



04 Mesh Generation

Procedure

- 1 Check on "Mesh"
- 2 Check off "Geometry"
- 3 Click [Create]
(Mesh > Element > Create)
- 4 Change property to "S1"
Click start and end points
Mesh set: S1
Click [Apply]
- 5 Change property to "S2"
Click start and end points
Mesh set: S2
Click [OK]

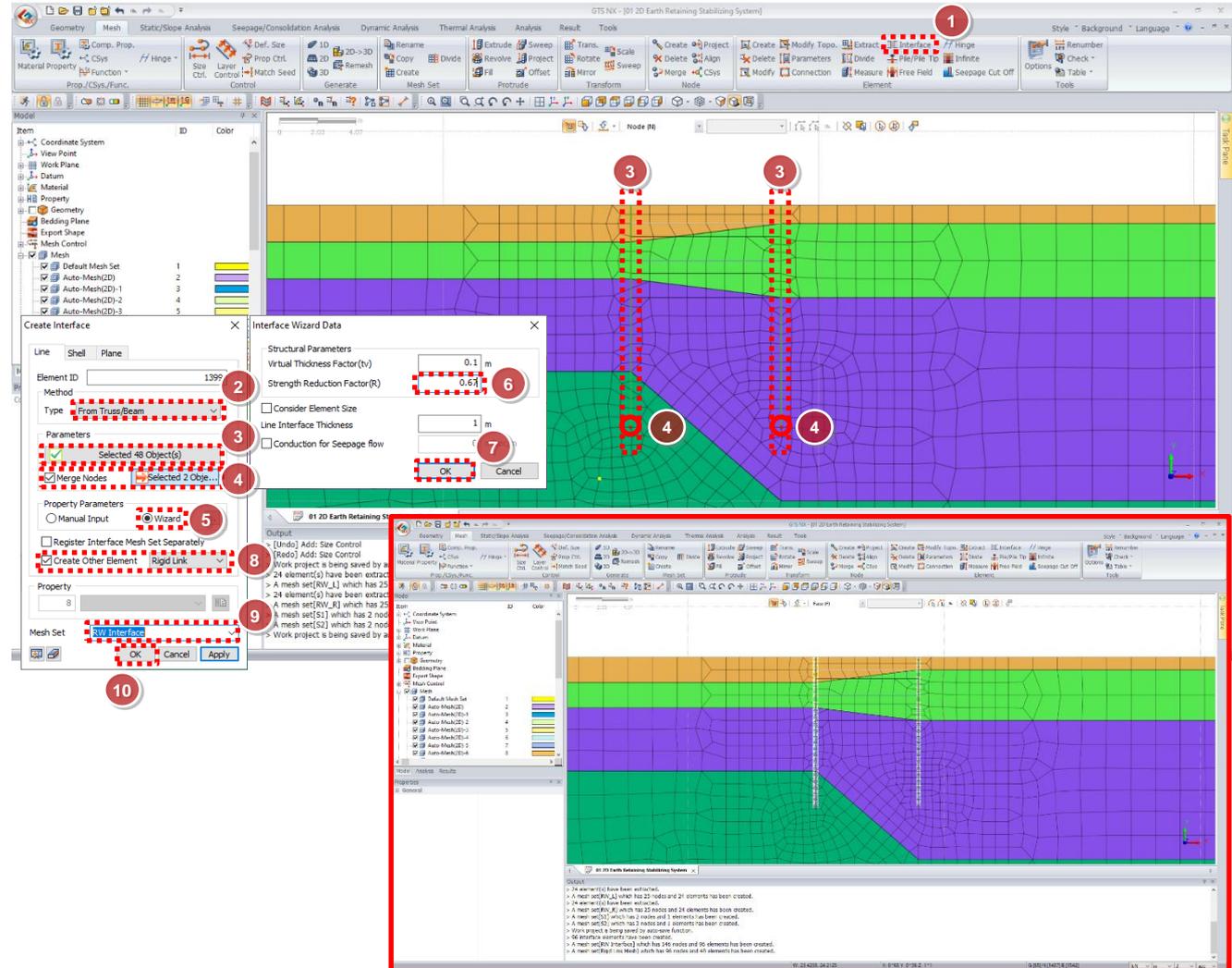
- Check with mesh tree, Check on only RWs, S1 and S2.

The screenshot shows the GTS NX software interface for mesh generation. The main window displays a 2D mesh of an earth retaining system. A red dashed box highlights the 'Element Create/Delete' dialog boxes for steps 4 and 5. Step 4 shows the dialog for element 1397 with Mesh Set S1. Step 5 shows the dialog for element 1398 with Mesh Set S2. A red box highlights the mesh tree on the right, showing elements 13, 14, 15, 16, 17, 18, 19, 20, 21, 22, 23, 24, 25, 26, 27, 28, 29, 30, 31, 32, 33, 34, 35, 36, 37, 38, 39, 40, 41, 42, 43, 44, 45, 46, 47, 48, 49, 50, 51, 52, 53, 54, 55, 56, 57, 58, 59, 60, 61, 62, 63, 64, 65, 66, 67, 68, 69, 70, 71, 72, 73, 74, 75, 76, 77, 78, 79, 80, 81, 82, 83, 84, 85, 86, 87, 88, 89, 90, 91, 92, 93, 94, 95, 96, 97, 98, 99, 100, 101, 102, 103, 104, 105, 106, 107, 108, 109, 110, 111, 112, 113, 114, 115, 116, 117, 118, 119, 120, 121, 122, 123, 124, 125, 126, 127, 128, 129, 130, 131, 132, 133, 134, 135, 136, 137, 138, 139, 140, 141, 142, 143, 144, 145, 146, 147, 148, 149, 150, 151, 152, 153, 154, 155, 156, 157, 158, 159, 160, 161, 162, 163, 164, 165, 166, 167, 168, 169, 170, 171, 172, 173, 174, 175, 176, 177, 178, 179, 180, 181, 182, 183, 184, 185, 186, 187, 188, 189, 190, 191, 192, 193, 194, 195, 196, 197, 198, 199, 200, 201, 202, 203, 204, 205, 206, 207, 208, 209, 210, 211, 212, 213, 214, 215, 216, 217, 218, 219, 220, 221, 222, 223, 224, 225, 226, 227, 228, 229, 230, 231, 232, 233, 234, 235, 236, 237, 238, 239, 240, 241, 242, 243, 244, 245, 246, 247, 248, 249, 250, 251, 252, 253, 254, 255, 256, 257, 258, 259, 260, 261, 262, 263, 264, 265, 266, 267, 268, 269, 270, 271, 272, 273, 274, 275, 276, 277, 278, 279, 280, 281, 282, 283, 284, 285, 286, 287, 288, 289, 290, 291, 292, 293, 294, 295, 296, 297, 298, 299, 300, 301, 302, 303, 304, 305, 306, 307, 308, 309, 310, 311, 312, 313, 314, 315, 316, 317, 318, 319, 320, 321, 322, 323, 324, 325, 326, 327, 328, 329, 330, 331, 332, 333, 334, 335, 336, 337, 338, 339, 340, 341, 342, 343, 344, 345, 346, 347, 348, 349, 350, 351, 352, 353, 354, 355, 356, 357, 358, 359, 360, 361, 362, 363, 364, 365, 366, 367, 368, 369, 370, 371, 372, 373, 374, 375, 376, 377, 378, 379, 380, 381, 382, 383, 384, 385, 386, 387, 388, 389, 390, 391, 392, 393, 394, 395, 396, 397, 398, 399, 400, 401, 402, 403, 404, 405, 406, 407, 408, 409, 410, 411, 412, 413, 414, 415, 416, 417, 418, 419, 420, 421, 422, 423, 424, 425, 426, 427, 428, 429, 430, 431, 432, 433, 434, 435, 436, 437, 438, 439, 440, 441, 442, 443, 444, 445, 446, 447, 448, 449, 450, 451, 452, 453, 454, 455, 456, 457, 458, 459, 460, 461, 462, 463, 464, 465, 466, 467, 468, 469, 470, 471, 472, 473, 474, 475, 476, 477, 478, 479, 480, 481, 482, 483, 484, 485, 486, 487, 488, 489, 490, 491, 492, 493, 494, 495, 496, 497, 498, 499, 500, 501, 502, 503, 504, 505, 506, 507, 508, 509, 510, 511, 512, 513, 514, 515, 516, 517, 518, 519, 520, 521, 522, 523, 524, 525, 526, 527, 528, 529, 530, 531, 532, 533, 534, 535, 536, 537, 538, 539, 540, 541, 542, 543, 544, 545, 546, 547, 548, 549, 550, 551, 552, 553, 554, 555, 556, 557, 558, 559, 560, 561, 562, 563, 564, 565, 566, 567, 568, 569, 570, 571, 572, 573, 574, 575, 576, 577, 578, 579, 580, 581, 582, 583, 584, 585, 586, 587, 588, 589, 590, 591, 592, 593, 594, 595, 596, 597, 598, 599, 600, 601, 602, 603, 604, 605, 606, 607, 608, 609, 610, 611, 612, 613, 614, 615, 616, 617, 618, 619, 620, 621, 622, 623, 624, 625, 626, 627, 628, 629, 630, 631, 632, 633, 634, 635, 636, 637, 638, 639, 640, 641, 642, 643, 644, 645, 646, 647, 648, 649, 650, 651, 652, 653, 654, 655, 656, 657, 658, 659, 660, 661, 662, 663, 664, 665, 666, 667, 668, 669, 670, 671, 672, 673, 674, 675, 676, 677, 678, 679, 680, 681, 682, 683, 684, 685, 686, 687, 688, 689, 690, 691, 692, 693, 694, 695, 696, 697, 698, 699, 700, 701, 702, 703, 704, 705, 706, 707, 708, 709, 710, 711, 712, 713, 714, 715, 716, 717, 718, 719, 720, 721, 722, 723, 724, 725, 726, 727, 728, 729, 730, 731, 732, 733, 734, 735, 736, 737, 738, 739, 740, 741, 742, 743, 744, 745, 746, 747, 748, 749, 750, 751, 752, 753, 754, 755, 756, 757, 758, 759, 760, 761, 762, 763, 764, 765, 766, 767, 768, 769, 770, 771, 772, 773, 774, 775, 776, 777, 778, 779, 780, 781, 782, 783, 784, 785, 786, 787, 788, 789, 790, 791, 792, 793, 794, 795, 796, 797, 798, 799, 800, 801, 802, 803, 804, 805, 806, 807, 808, 809, 810, 811, 812, 813, 814, 815, 816, 817, 818, 819, 820, 821, 822, 823, 824, 825, 826, 827, 828, 829, 830, 831, 832, 833, 834, 835, 836, 837, 838, 839, 840, 841, 842, 843, 844, 845, 846, 847, 848, 849, 850, 851, 852, 853, 854, 855, 856, 857, 858, 859, 860, 861, 862, 863, 864, 865, 866, 867, 868, 869, 870, 871, 872, 873, 874, 875, 876, 877, 878, 879, 880, 881, 882, 883, 884, 885, 886, 887, 888, 889, 890, 891, 892, 893, 894, 895, 896, 897, 898, 899, 900, 901, 902, 903, 904, 905, 906, 907, 908, 909, 910, 911, 912, 913, 914, 915, 916, 917, 918, 919, 920, 921, 922, 923, 924, 925, 926, 927, 928, 929, 930, 931, 932, 933, 934, 935, 936, 937, 938, 939, 940, 941, 942, 943, 944, 945, 946, 947, 948, 949, 950, 951, 952, 953, 954, 955, 956, 957, 958, 959, 960, 961, 962, 963, 964, 965, 966, 967, 968, 969, 970, 971, 972, 973, 974, 975, 976, 977, 978, 979, 980, 981, 982, 983, 984, 985, 986, 987, 988, 989, 990, 991, 992, 993, 994, 995, 996, 997, 998, 999, 1000.

04 Mesh Generation

Procedure

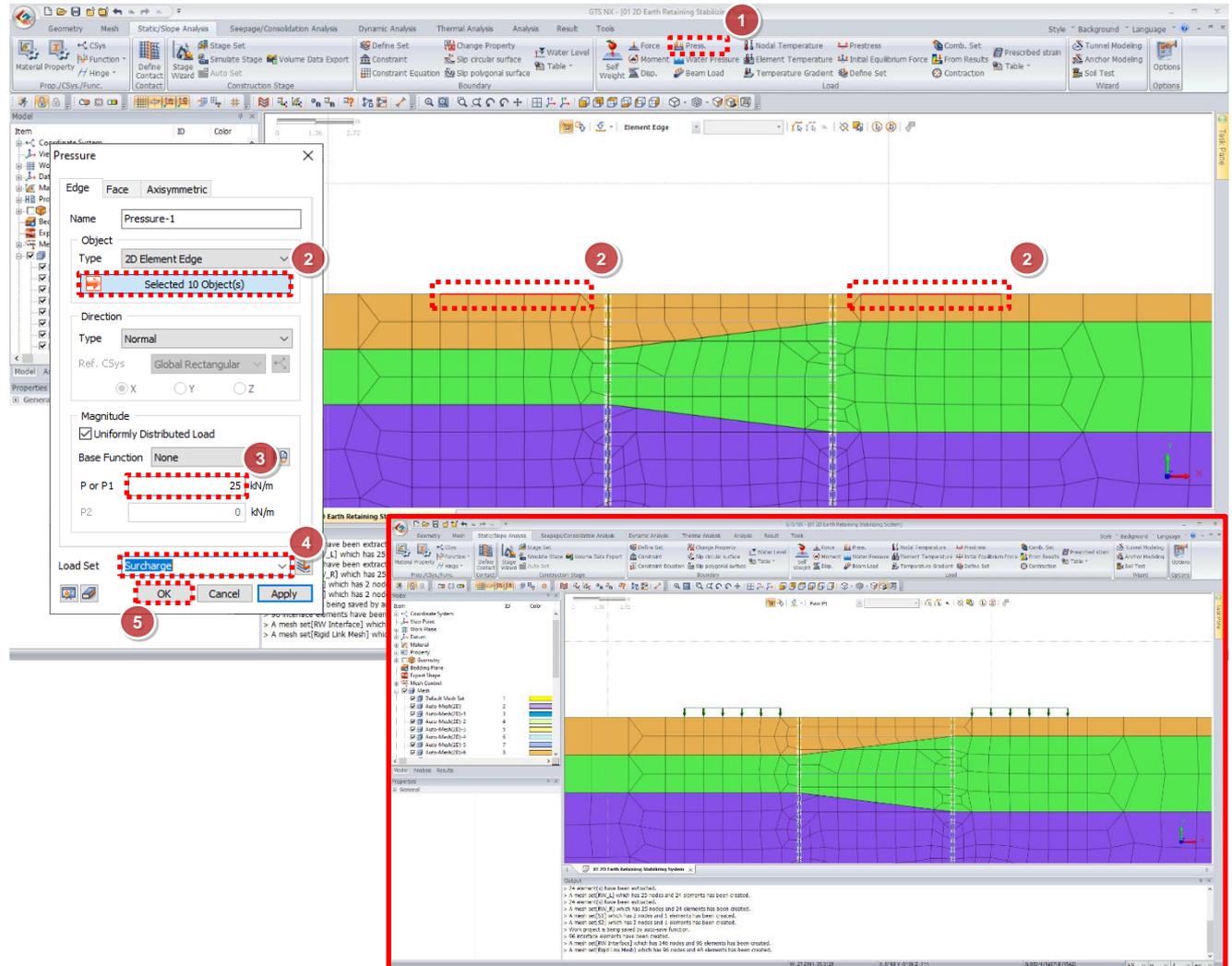
- 1 Click [Interface]
(Mesh > Element > Interface)
- 2 Type: From Truss/Beam
- 3 Select RWs
(48 objects)
- 4 Check on "Merge Nodes"
Select end points of RWs
(2 points)
- 5 Check on "Wizard"
- 6 Strength Reduction
Factor: 0.67
- 7 Click [OK]
- 8 Check on "Create Other
Element"
Select "Rigid Link"
- 9 Mesh set: RW Interface
- 10 Click [OK]



05 Load & Boundary Condition

Procedure

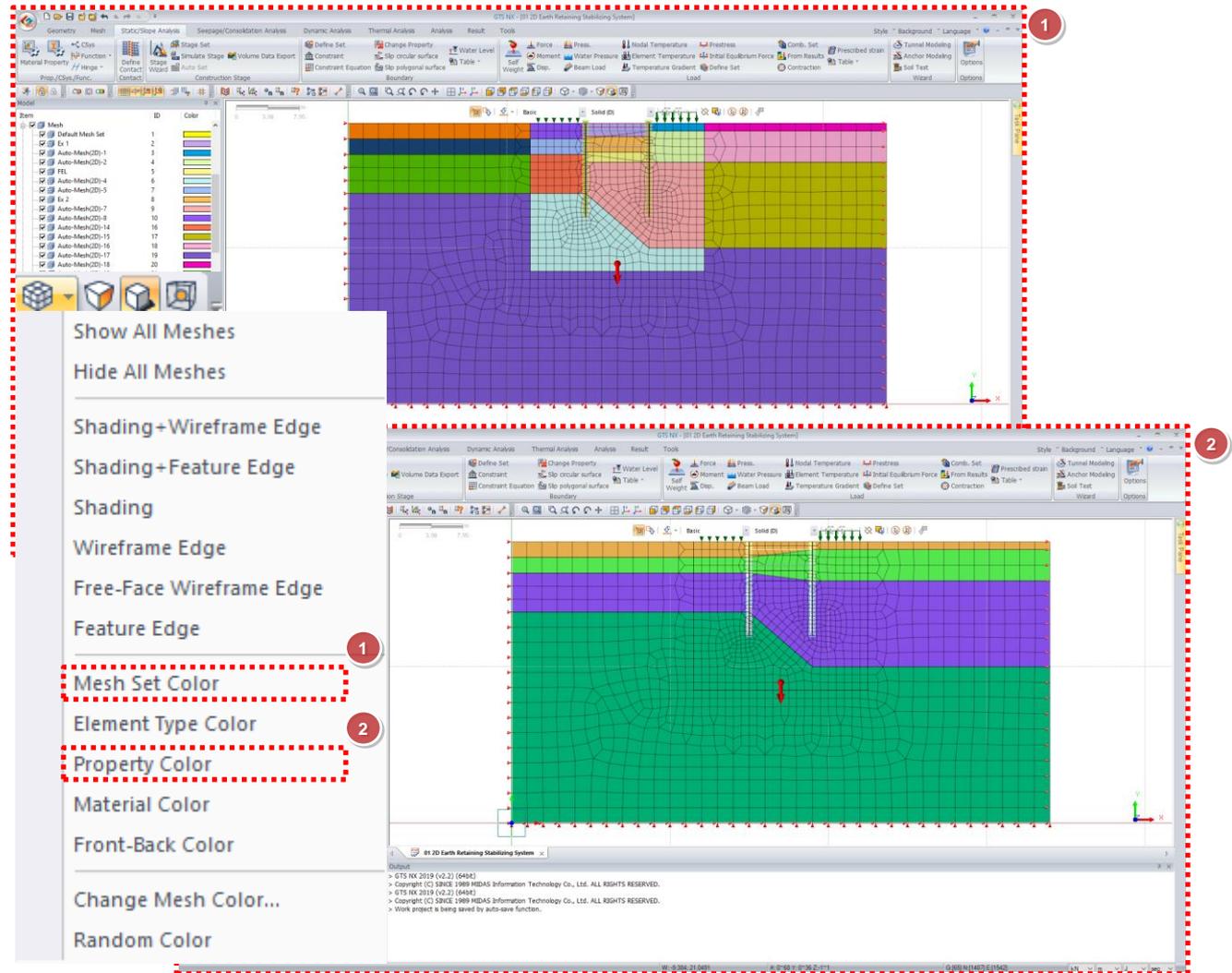
- 1 Click [Press.]
(Static/Slope Analysis > Load > Press.)
- 2 Select Surcharge parts
(10 objects)
- 3 P or P1: 25
- 4 Load Set: Surcharge
- 5 Click [OK]



05 Load & Boundary Condition

Procedure

- Using “Display Mode” to select “Mesh Set Color” or “Property Color”



The image shows two screenshots of the GTS NX software interface. The top screenshot, labeled '1', shows the 'Display Mode' menu with 'Mesh Set Color' selected. The bottom screenshot, labeled '2', shows the 'Display Mode' menu with 'Property Color' selected. Both screenshots show a 3D model of an earth retaining system with a mesh overlay. The mesh is colored according to the selected display mode. The software interface includes a toolbar with various analysis and display tools, and a 'Mesh' panel on the left side.

1

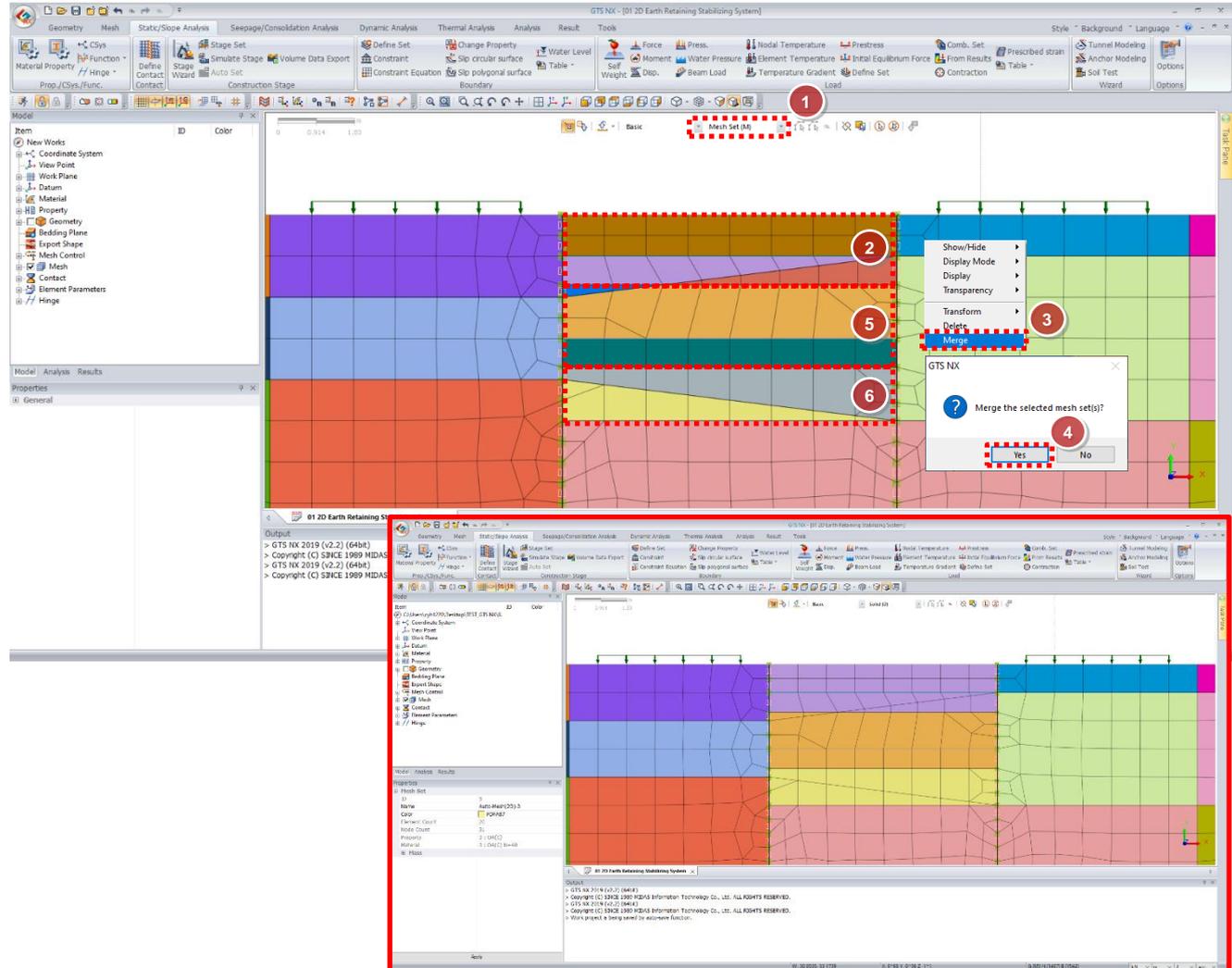
2

Show All Meshes
Hide All Meshes
Shading+Wireframe Edge
Shading+Feature Edge
Shading
Wireframe Edge
Free-Face Wireframe Edge
Feature Edge
Mesh Set Color
Element Type Color
Property Color
Material Color
Front-Back Color
Change Mesh Color...
Random Color

05 Load & Boundary Condition

Procedure

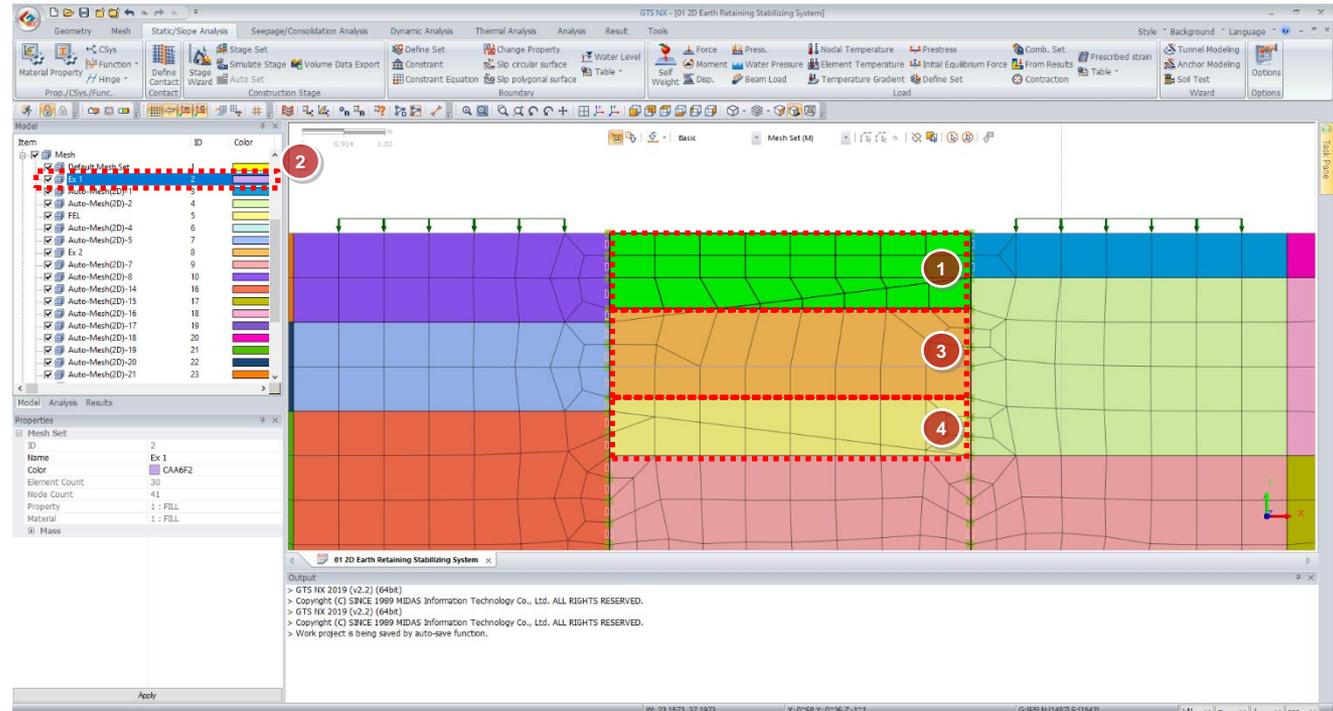
- 1 Press "M" key to change selection method to Mesh Set
- 2 Select 1st excavation part
- 3 Right Click and Click [Merge]
- 4 Click [OK]
- 5 Select 2nd excavation part
Do it repeatedly step 3 and 4
- 6 Select final excavation level part
Repeat step 3 and 4



05 Load & Boundary Condition

Procedure

- 1 Select 1st excavation part
- 2 Press “F2” key to change mesh set name under mesh tree
Name: Ex 1
- 3 Repeat step 1&2
Name: Ex 2
- 4 Repeat step 1&2
Name: FEL



05 Load & Boundary Condition

Procedure

- 1 Click [Self Weight]
(Static/Slope Analysis > Load > Self Weight)
- 2 Load Set: SW
- 3 Click [OK]

Gravity

Name: Gravity-1

Reference Object

Type: Coordinate

Ref. CSys: Global Rectangular

Components

Gx: 0

Gy: -1

Gz: 0

Spatial Distribution

Base Function: None

Load Set: SW

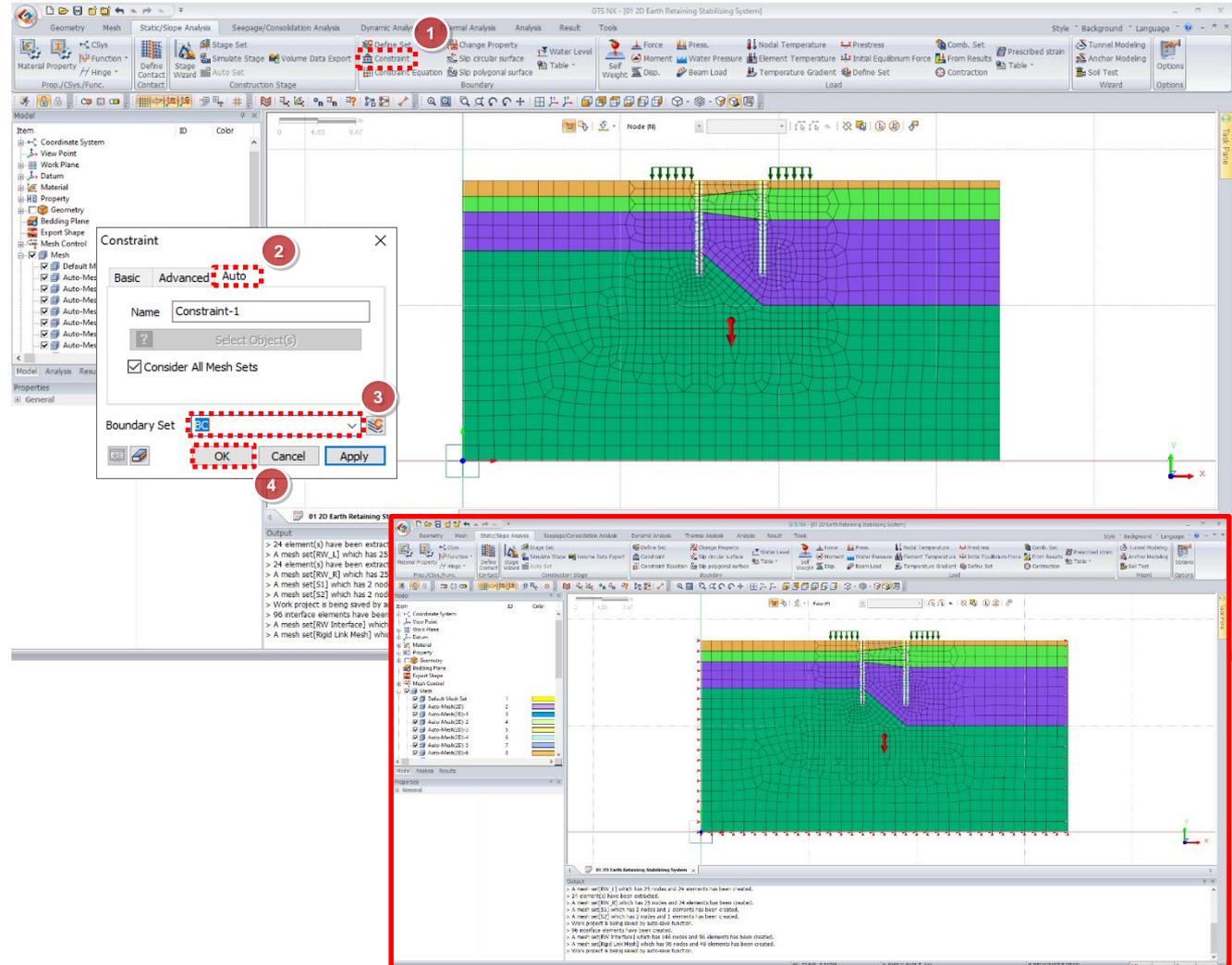
OK Cancel Apply

24 element(s) have been extracted
> A mesh set[RV_1] which has 25 elements has been extracted
> A mesh set[RV_R] which has 25 elements has been extracted
> A mesh set[S1] which has 2 nodes has been created
> Work project is being saved by GTS NX
> A mesh set[RV_Interface] which has 96 interface elements have been created
> A mesh set[igid Link Head] which has 24 elements has been created

05 Load & Boundary Condition

Procedure

- 1 Click [Constraint]
(Static/Slope Analysis > Boundary > Constraint)
- 2 Move to "Auto"
- 3 Boundary Set: BC
- 4 Click [OK]



Output

- > 24 element(s) have been extracted
- > A mesh set[RW_1] which has 25 elements has been created.
- > 24 element(s) have been extracted
- > A mesh set[RW_R] which has 25 elements has been created.
- > A mesh set[S1] which has 2 nodes has been created.
- > Work project is being saved by the user.
- > A mesh set[S2] which has 2 nodes has been created.
- > 96 interface elements have been extracted
- > A mesh set[RW_Interface] which has 96 elements has been created.
- > A mesh set[igid Link Head] which has 2 nodes has been created.

Model

Item	ID	Color	Order
Coordinate System			0
View Point			1
Work Plane			2
Datum			3
Material			4
Property			5
Geometry			6
Bedding Plane			7
Export Shape			8
Mesh Control			9
Auto-Mesh			10
Auto-Mesh			11
Auto-Mesh			12
Auto-Mesh			13
Auto-Mesh			14
Auto-Mesh			15
Auto-Mesh			16
Auto-Mesh			17
Auto-Mesh			18
Auto-Mesh			19
Auto-Mesh			20
Auto-Mesh			21
Auto-Mesh			22
Auto-Mesh			23
Auto-Mesh			24
Auto-Mesh			25
Auto-Mesh			26
Auto-Mesh			27
Auto-Mesh			28
Auto-Mesh			29
Auto-Mesh			30
Auto-Mesh			31
Auto-Mesh			32
Auto-Mesh			33
Auto-Mesh			34
Auto-Mesh			35
Auto-Mesh			36
Auto-Mesh			37
Auto-Mesh			38
Auto-Mesh			39
Auto-Mesh			40
Auto-Mesh			41
Auto-Mesh			42
Auto-Mesh			43
Auto-Mesh			44
Auto-Mesh			45
Auto-Mesh			46
Auto-Mesh			47
Auto-Mesh			48
Auto-Mesh			49
Auto-Mesh			50
Auto-Mesh			51
Auto-Mesh			52
Auto-Mesh			53
Auto-Mesh			54
Auto-Mesh			55
Auto-Mesh			56
Auto-Mesh			57
Auto-Mesh			58
Auto-Mesh			59
Auto-Mesh			60
Auto-Mesh			61
Auto-Mesh			62
Auto-Mesh			63
Auto-Mesh			64
Auto-Mesh			65
Auto-Mesh			66
Auto-Mesh			67
Auto-Mesh			68
Auto-Mesh			69
Auto-Mesh			70
Auto-Mesh			71
Auto-Mesh			72
Auto-Mesh			73
Auto-Mesh			74
Auto-Mesh			75
Auto-Mesh			76
Auto-Mesh			77
Auto-Mesh			78
Auto-Mesh			79
Auto-Mesh			80
Auto-Mesh			81
Auto-Mesh			82
Auto-Mesh			83
Auto-Mesh			84
Auto-Mesh			85
Auto-Mesh			86
Auto-Mesh			87
Auto-Mesh			88
Auto-Mesh			89
Auto-Mesh			90
Auto-Mesh			91
Auto-Mesh			92
Auto-Mesh			93
Auto-Mesh			94
Auto-Mesh			95
Auto-Mesh			96
Auto-Mesh			97
Auto-Mesh			98
Auto-Mesh			99
Auto-Mesh			100

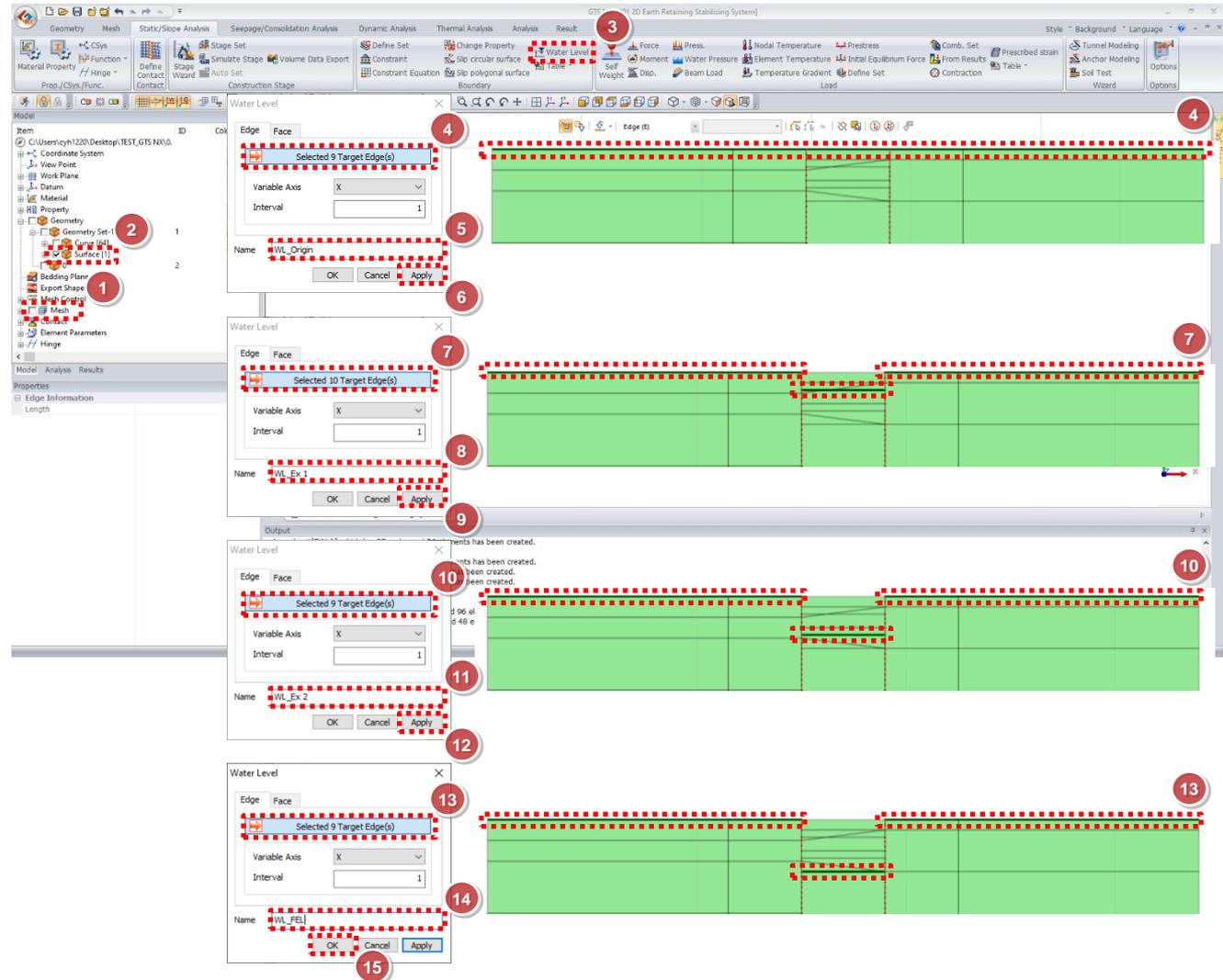
Model Analysis Results

General

05 Load & Boundary Condition

Procedure

- 1 Check off "Mesh"
- 2 Check on "Surface [1]"
- 3 Click [Water Level]
(Static/Slope Analysis > Boundary > Water Level)
- 4 Select water level lines
- 5 Name: WL_Origin
- 6 Click [Apply]
- Repeat step 4 to step 6 until step 14
- 15 Click [OK]



GTS NX

New eXperience of Geo-Technical analysis System

Thank you!

MIDAS

The background features a complex network of interconnected nodes and lines, resembling a mesh or a molecular structure. The nodes are small circles, and the lines are thin, connecting the nodes in a non-uniform pattern. The overall color scheme is a gradient from green on the left to orange and red on the right.

GTS INX

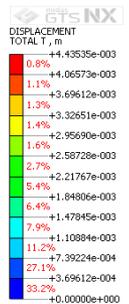
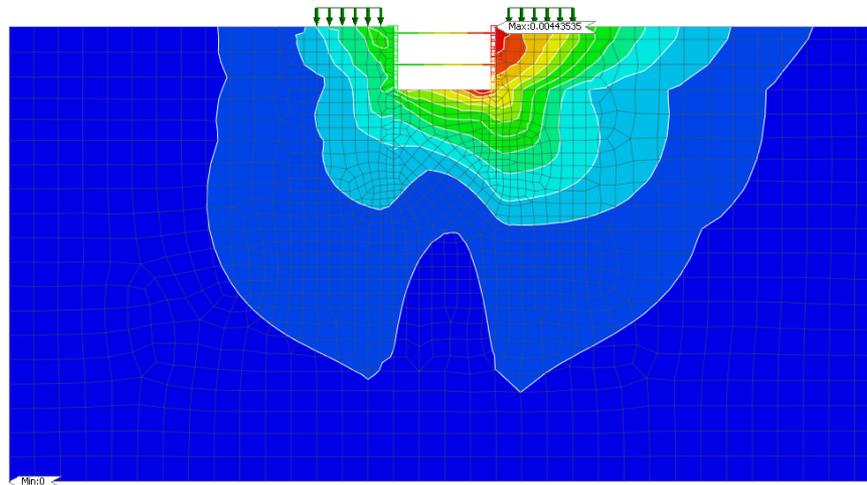
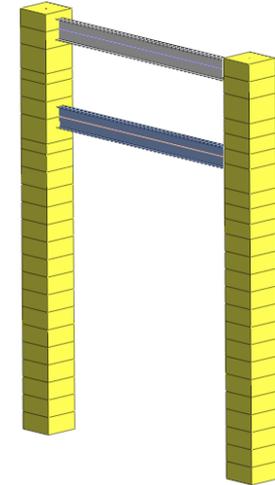
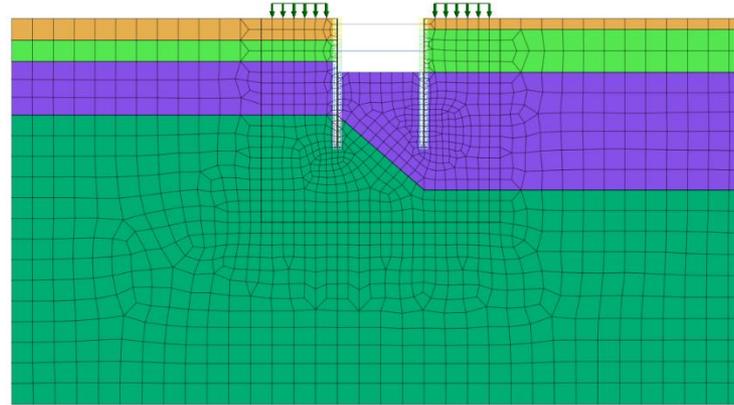
New eXperience of Geo-Technical analysis System

Earth Retaining Stabilizing System

MIDAS

Procedure

- This tutorial is aiming on performing the analysis with 3 stages of excavation including retaining wall and strutting.
- Surcharge and undrained condition are considered in this analysis.
- This example was prepared based on the most general excavation work in the real practice.



[DATA] Excavation, FEL, INCR=1 (LOAD=1.000), [UNIT] kN, m



Ground

[unit : kN, m]

Name	Fill	Fluvial Clay	Old Alluvium (C)	Old Alluvium (A)
Material	Isotropic	Isotropic	Isotropic	Isotropic
Model Type	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb
General				
Elastic Modulus (E) [kN/m²]	10,000	13,000	133,333	400,000
Poisson's Ratio (ν)	0.25	0.25 (0.495)	0.25	0.25
Unit Weight (γ) [kN/m³]	18	19	20	21
Ko	0.5	1.0	0.5	0.5
Porous				
Unit Weight (Saturated) [kN/m³]	18	19	20	21
Drainage Parameters	Drained	Undrained	Drained	Drained
Non-Linear				
Cohesion (c) [kN/m²]	1	40	5	20
Frictional Angle (Φ) [deg]	30	0	30	35

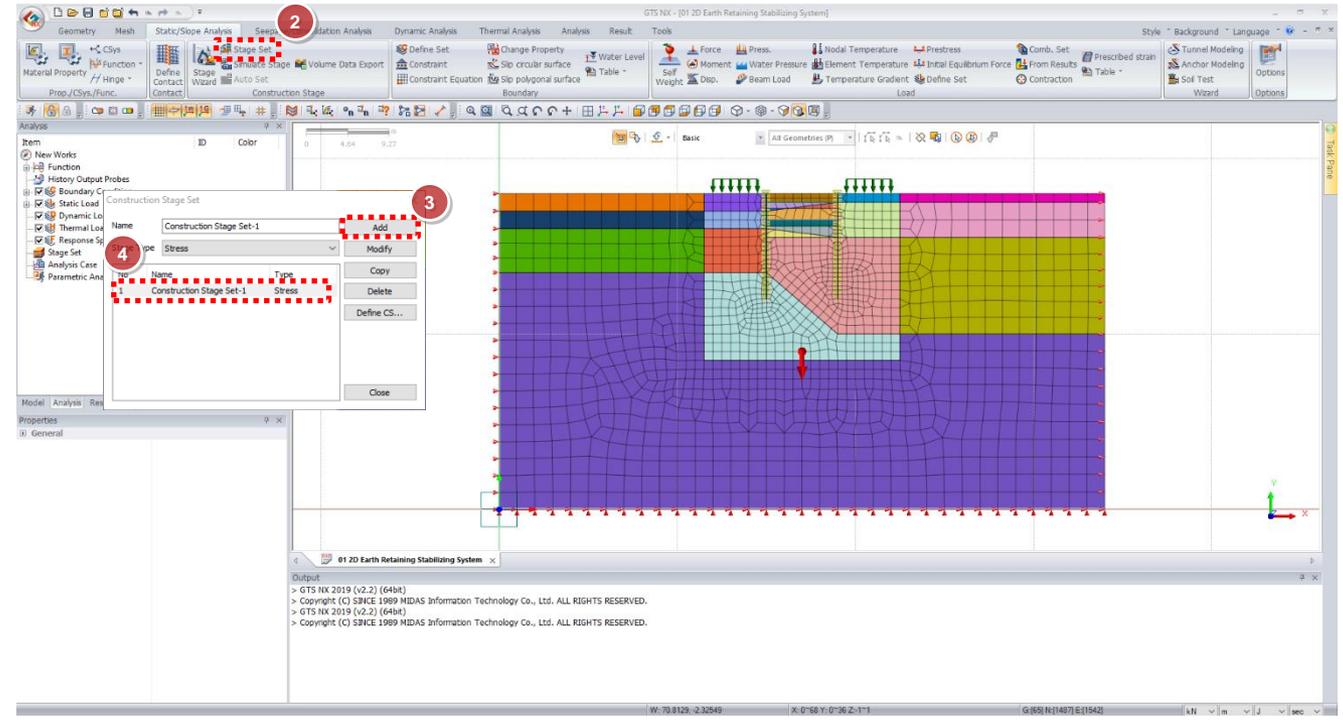
Structure

[unit : kN, m]

Name	Conc'	Steel
Material	Isotropic	Isotropic
Model Type	Elastic	Elastic
Elastic Modulus (E) [kN/m ²]	28,000,000	205,000,000
Poisson's Ratio (ν)	0.15	0.15
Unit Weight (γ) [kN/m ³]	24	74

Procedure

- 1 Check off geometry and check on mesh
- 2 Click [Stage Set] (Static/Slope Analysis > Construction Stage > Stage Set)
- 3 Click [Add]
- 4 Double click [Construction stage set-1]



06 Analysis Setting

Procedure

- 1 Stage Name: Initial Phase
- 2 Move(Drag & Drop) the Data from “Set Data” to “Activated Data” as shown in figure
- 3 Check on “Define Water Level For Global” Key in “1m” Select “WL_Origin”
- 4 Check on “Clear Displacement”
- 5 Check on “Analysis Control” and Click “Analysis Control”
- 6 Check on “Allow Undrained Material Behavior”
- 7 Click [OK]
- 8 Click [Save]
- 9 Click [New]

Procedure

- 1 Stage Name: RW + Surchage
- 2 Move(Drag & Drop) the Data from “Set Data” to “Activated Data” and “Deactivated Data” as shown in figure
- 3 Check on “Clear Displacement”
- 4 Check on “Analysis Control” and Click “Analysis Control”
- 5 Check on “Allow Undrained Material Behavior”
- 6 Click [OK]
- 7 Click [Save]
- 8 Click [New]

Procedure

- 1 Stage Name: Ex 1
- 2 Move(Drag & Drop) the Data from “Set Data” to “Deactivated Data” as shown in figure
- 3 Check on “Define Water Level For Global” Key in “1m” Select “WL_Ex 1”
- 4 Check on “Analysis Control” and Click “Analysis Control”
- 5 Check on “Allow Undrained Material Behavior”
- 6 Click [OK]
- 7 Click [Save]
- 8 Click [New]

The image shows a sequence of steps in the GTS NX software interface:

- Define Construction Stage Dialog:**
 - Step 1: Stage Name is set to "Ex 1".
 - Step 2: Data from "Set Data" (Mesh, Boundary Condition, Static Load, Contact) is moved to "Deactivated Data".
 - Step 3: "Define Water Level For Global" is checked, with "1 m" and "WL_Ex 1" selected.
 - Step 4: "Analysis Control" is checked.
 - Step 7: The "Save" button is highlighted.
- Analysis Control Dialog:**
 - Step 5: "Allow Undrained Material Behavior" is checked under the "Undrained Condition" section.
 - Step 6: The "OK" button is highlighted.
- Main Model View:** Shows a 3D cross-section of a soil structure with a mesh overlay, representing the analysis setup.

Procedure

- 1 Stage Name: S1
- 2 Move(Drag & Drop) the Data from “Set Data” to “Activated Data” as shown in figure
- 3 Check on “Analysis Control” and Click “Analysis Control”
- 4 Check on “Allow Undrained Material Behavior”
- 5 Click [OK]

The image shows a sequence of steps in the GTS NX software interface:

- Step 1:** In the 'Define Construction Stage' dialog, the 'Stage Name' is set to 'S1'.
- Step 2:** Data from the 'Set Data' list (including Mesh, Boundary Condition, Static Load, and Contact) is moved to the 'Activated Data' list.
- Step 3:** The 'Analysis Control' checkbox is checked in the 'Define Construction Stage' dialog.
- Step 4:** In the 'Analysis Control' dialog, the 'Allow Undrained Material Behavior' checkbox is checked.
- Step 5:** The 'OK' button is clicked in the 'Analysis Control' dialog.
- Step 6:** The 'Save' button is clicked in the 'Define Construction Stage' dialog.
- Step 7:** The 'New' button is clicked in the 'Define Construction Stage' dialog.

The main 3D view shows a cross-section of a dam with a mesh applied to the structure.

Procedure

- 1 Stage Name:
Ex 2
- 2 Move(Drag & Drop) the Data from “Set Data” to “Deactivated Data” as shown in figure
- 3 Check on “Define Water Level For Global”
Key in “1m”
Select “WL_Ex 2”
- 4 Check on “Analysis Control”
and Click “Analysis Control”
- 5 Check on “Allow Undrained Material Behavior”
- 6 Click [OK]
- 7 Click [Save]
- 8 Click [New]

Procedure

- 1 Stage Name: S2
- 2 Move(Drag & Drop) the Data from “Set Data” to “Activated Data” as shown in figure
- 3 Check on “Analysis Control” and Click “Analysis Control”
- 4 Check on “Allow Undrained Material Behavior”
- 5 Click [OK]
- 6 Click [Save]
- 7 Click [New]

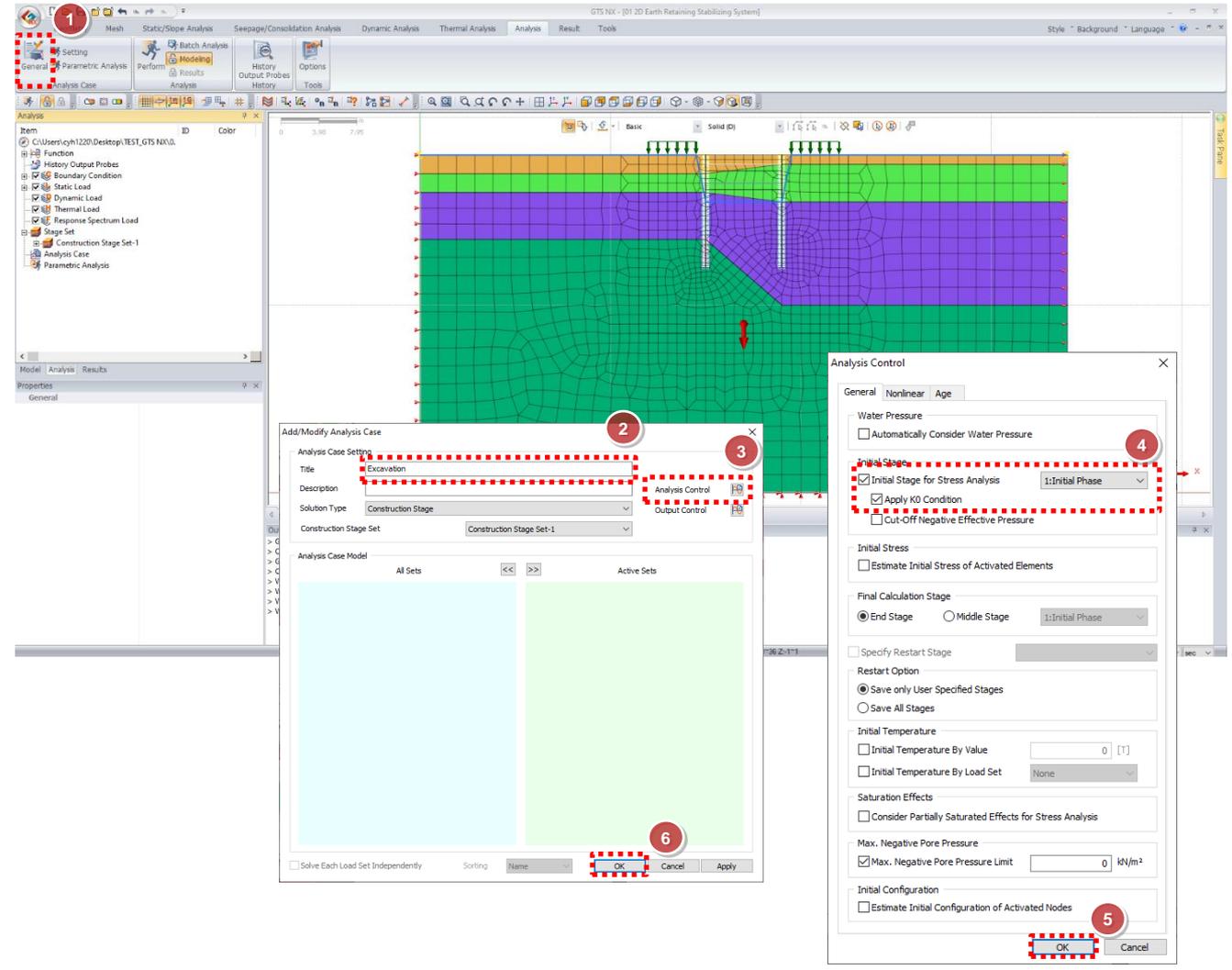
Procedure

- 1 Stage Name: FEL
- 2 Move(Drag & Drop) the Data from “Set Data” to “Deactivated Data” as shown in figure
- 3 Check on “Define Water Level For Global” Key in “1m” Select “WL_FEL”
- 4 Check on “Analysis Control” and Click “Analysis Control”
- 5 Check on “Allow Undrained Material Behavior”
- 6 Click [OK]
- 7 Click [Save]
- 8 Click [Close]

06 Analysis Setting

Procedure

- 1 Click [General]
(Analysis > Analysis Case > General)
- 2 Title: Excavation
- 3 Click [Analysis Control]
- 4 Check on “Initial Stage for Stress Analysis”
Check on “Apply K0 Condition”
- 5 Click [OK]
- 6 Click [OK]



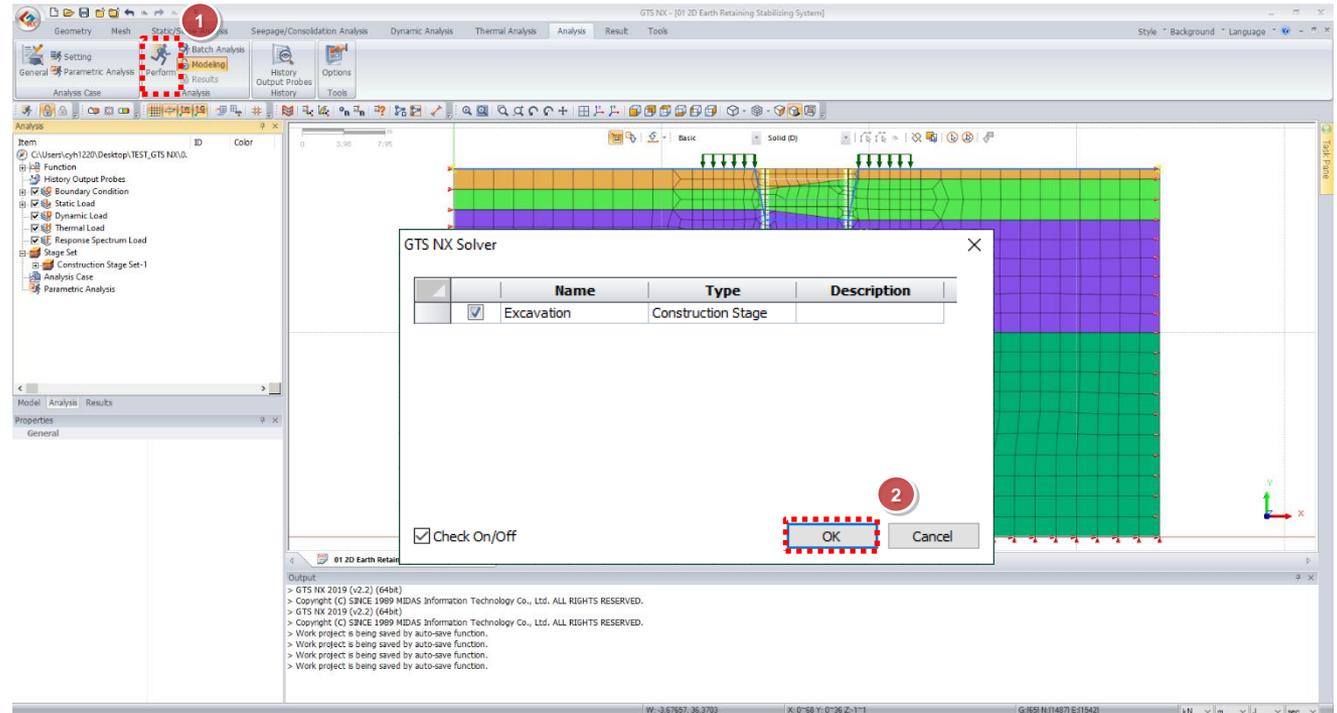
The screenshot shows the GTS NX software interface for setting up an analysis case. The main window displays a cross-section of an excavation with a mesh. Two dialog boxes are open: 'Add/Modify Analysis Case' and 'Analysis Control'. Red dashed boxes and numbered circles (1-6) highlight specific steps in the procedure:

- 1: Click on the 'General' tab in the 'Add/Modify Analysis Case' dialog.
- 2: Enter 'Excavation' in the 'Title' field.
- 3: Click on the 'Analysis Control' button in the 'Add/Modify Analysis Case' dialog.
- 4: Check the 'Initial Stage for Stress Analysis' checkbox in the 'Analysis Control' dialog.
- 5: Click the 'OK' button in the 'Analysis Control' dialog.
- 6: Click the 'OK' button in the 'Add/Modify Analysis Case' dialog.

06 Analysis Setting

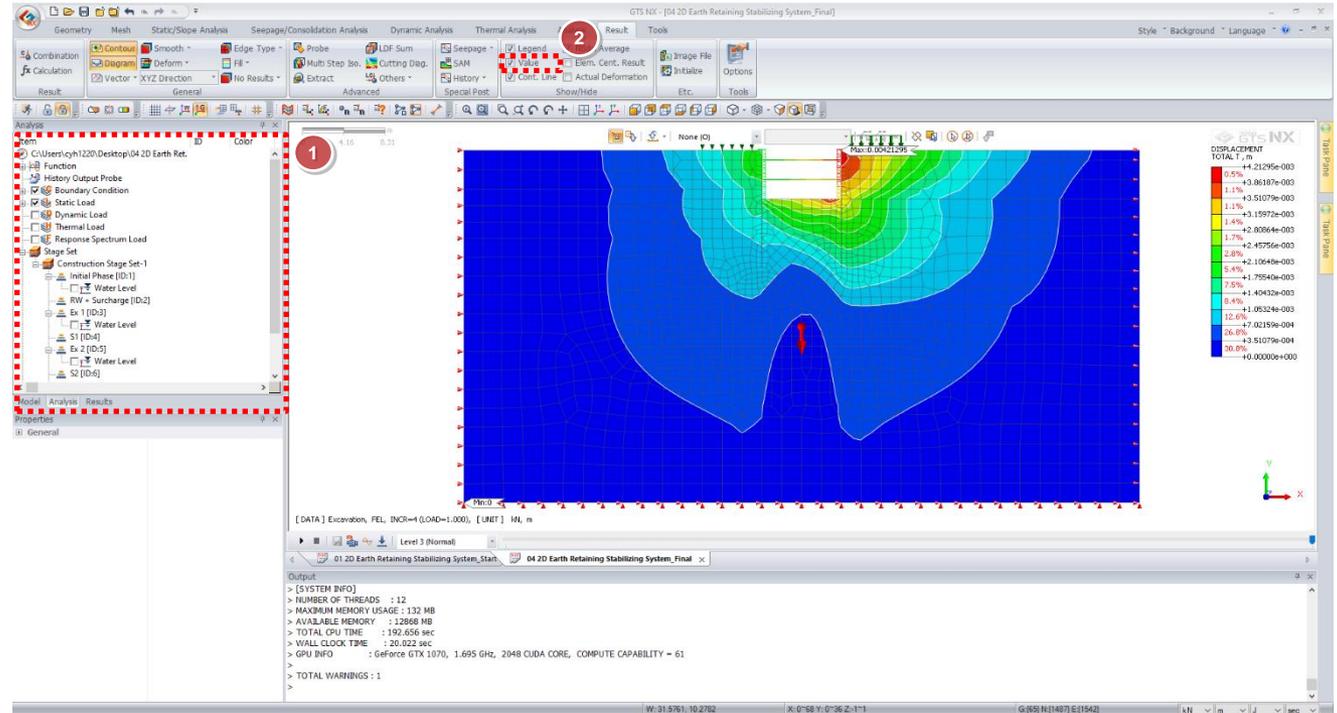
Procedure

- 1 Click [Perform]
- 2 Click [OK]



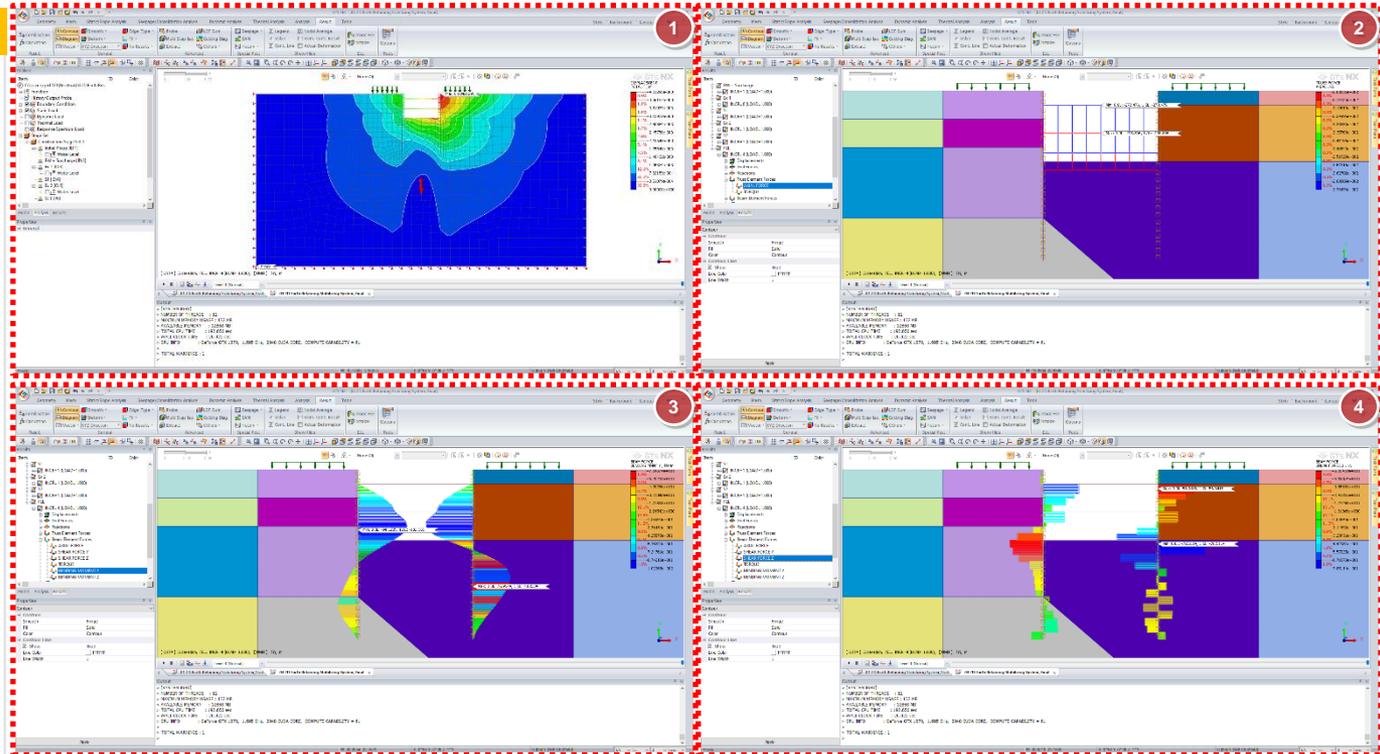
Procedure

- 1 The results from each stages can be found in the “Results” tab
- 2 Check on “Value” can be found maximum and minimum value directly on the mesh



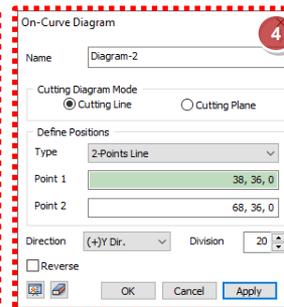
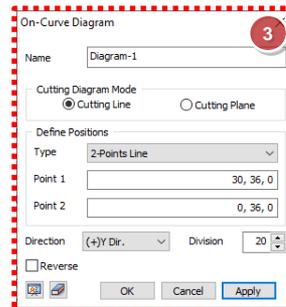
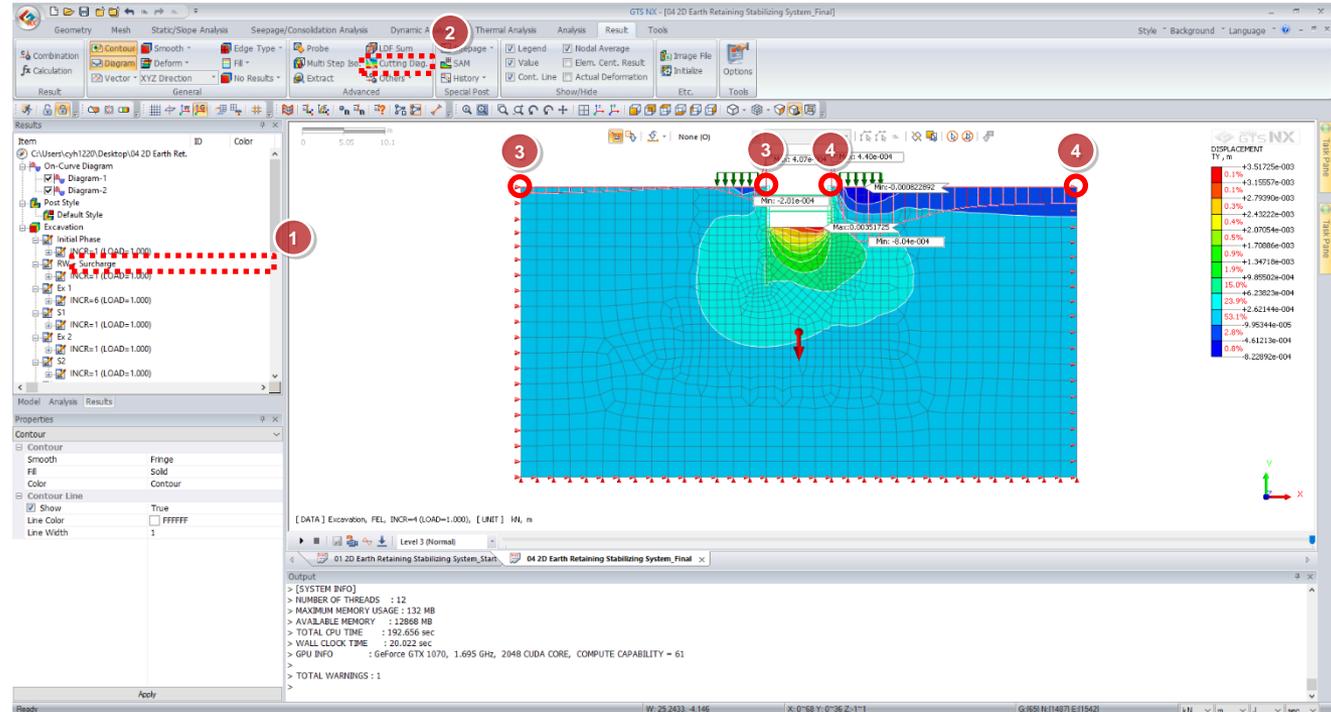
Procedure

- 1 Displacement of ground
- 2 Axial Force from struts
- 3 Bending moment of RWs
- 4 Shear Force of RWs



Procedure

- 1 Select "TY TRANSLATION (V)"
- 2 Click [Cutting Diag.] (Result > Advanced > Cutting Diag.)
- 3 Select start & end nodes
Direction: (+)Y Dir.
Click [Apply]
- 4 Select start & end nodes
Direction: (+)Y Dir.
Click [OK]



Procedure

- 1 Right Click on Diagram-1
 - 2 Click [Show Table]
 - 3 Verify the values from Diagram-1
- Table can be copied to spread sheet directly with "ctrl + c & ctrl + v"
 - Increase the division from "On-Curve Diagram" function can be increase data of table.

The screenshot shows the GTS NX software interface. The main window displays a finite element analysis of an earth retaining system. The mesh is color-coded by displacement, with a legend on the right showing values from -8.22892e-004 to -1.51725e-003. A context menu is open over the 'Excavation' element, with 'Show Table' selected. A secondary window shows the resulting data table for the excavation element.

Item	ID	Color	Value
Excavation	1	1	0.00000000
Excavation	2	2	0.00000000
Excavation	3	3	0.00000000
Excavation	4	4	0.00000000
Excavation	5	5	0.00000000
Excavation	6	6	0.00000000
Excavation	7	7	0.00000000
Excavation	8	8	0.00000000
Excavation	9	9	0.00000000
Excavation	10	10	0.00000000
Excavation	11	11	0.00000000
Excavation	12	12	0.00000000
Excavation	13	13	0.00000000
Excavation	14	14	0.00000000
Excavation	15	15	0.00000000
Excavation	16	16	0.00000000
Excavation	17	17	0.00000000
Excavation	18	18	0.00000000
Excavation	19	19	0.00000000
Excavation	20	20	0.00000000
Excavation	21	21	0.00000000
Excavation	22	22	0.00000000
Excavation	23	23	0.00000000
Excavation	24	24	0.00000000
Excavation	25	25	0.00000000
Excavation	26	26	0.00000000
Excavation	27	27	0.00000000
Excavation	28	28	0.00000000
Excavation	29	29	0.00000000
Excavation	30	30	0.00000000

GTS NX

New eXperience of Geo-Technical analysis System

Thank you!

MIDAS