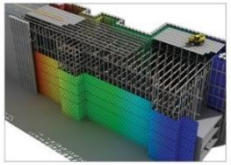
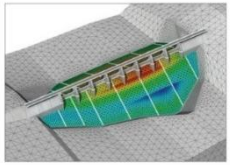
Import Arc Bridge from Civil to GTS NX for Time History Analysis

GTS NX

Geo-Technical analysis System New eXperience





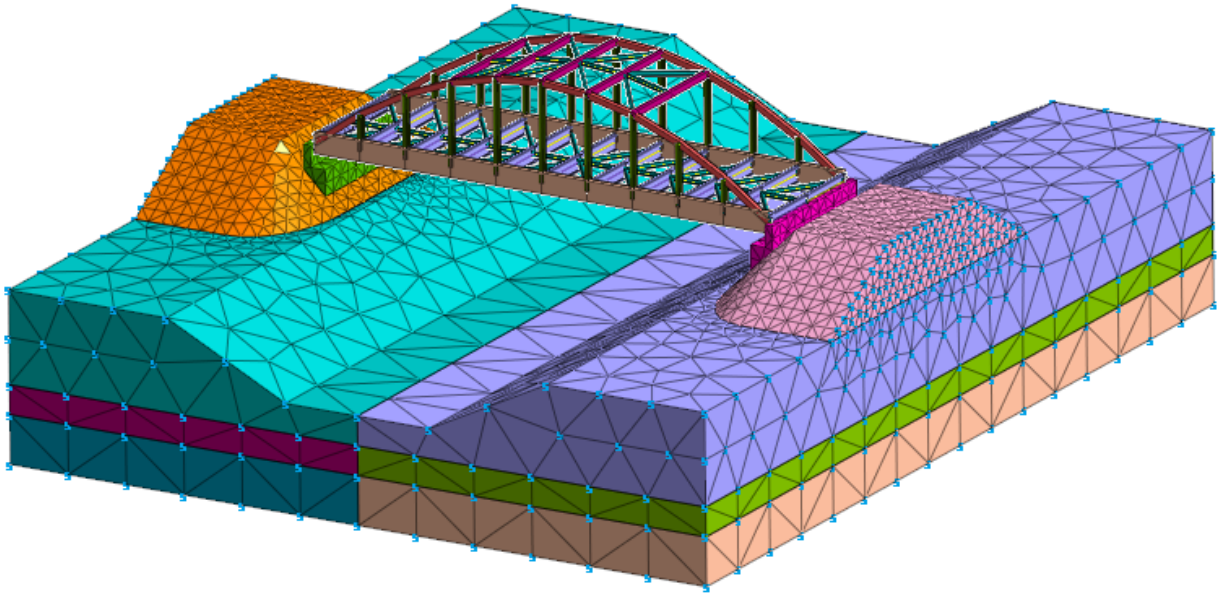
Tutorial by Angel Francisco Martinez

Integrated Solver Optimized for the next generation 64-bit platform

Finite Element Solutions for Geotechnical Engineering

MIDAS

Overview

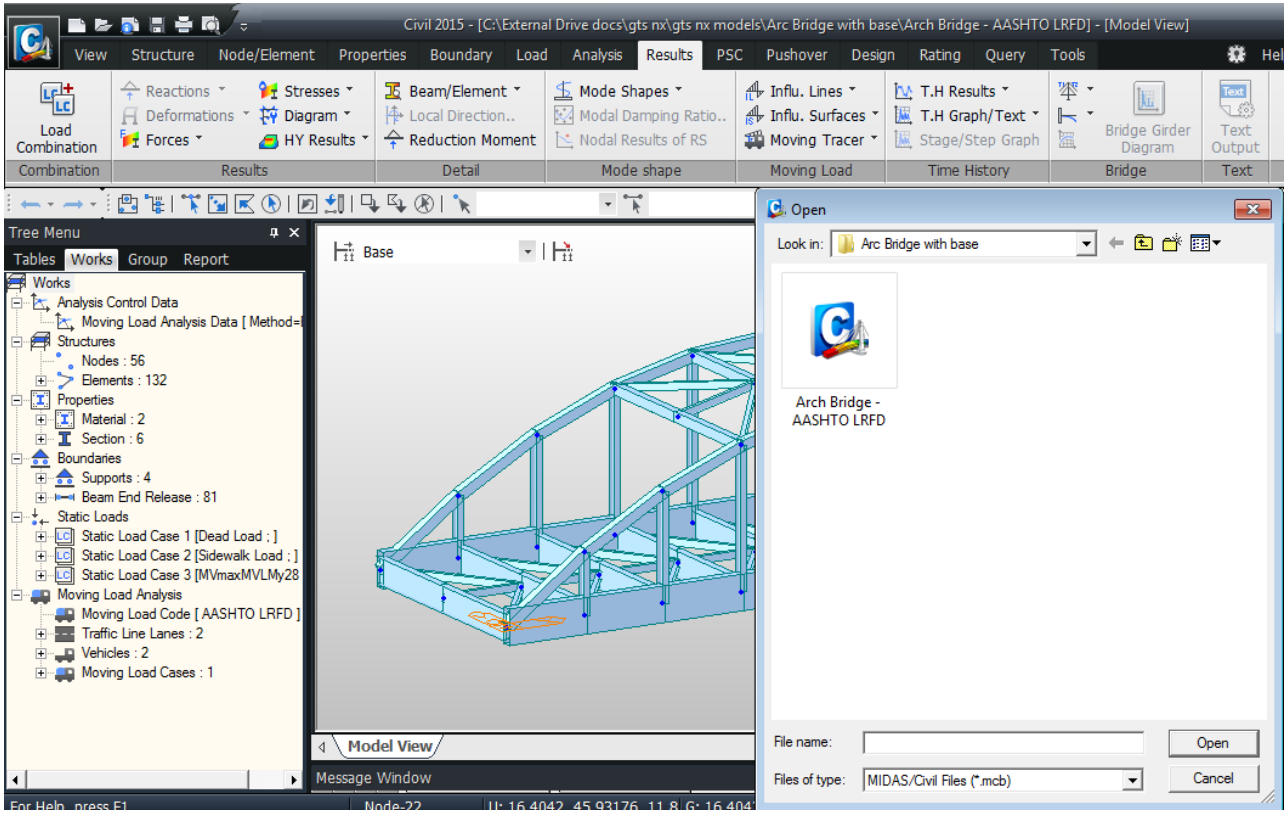


Objectives

1. Show link between midas Civil, specialized structural software and midas GTS NX, specialized geotechnical software.
2. Show preparation and Export from Civil.
3. Show import and ground modeling in GTS NX.
4. Run a consolidation analysis.
5. Check for settlement of bridge foundation.

Step 1. File > Open

GTS NX

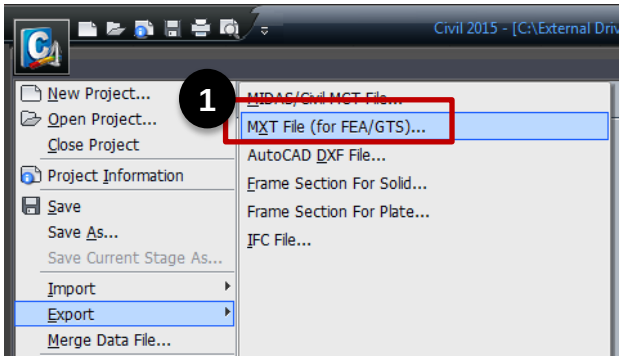


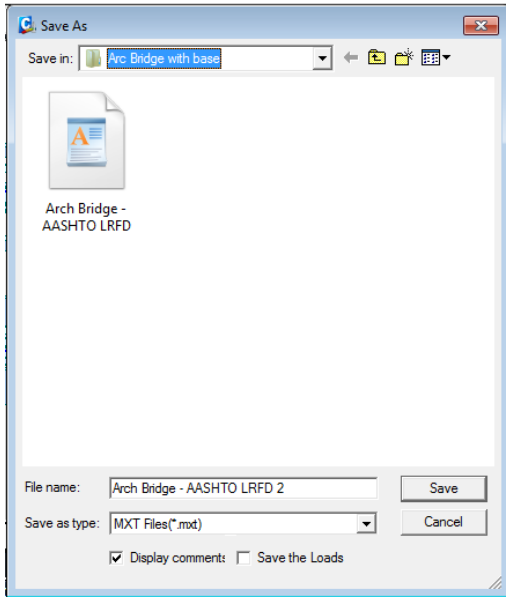
Procedure

1. In midas Civil:
2. Main Menu > File > Open…
3. Double click Arch Bridge.mcb

Step 5. Export

GTS NX





3

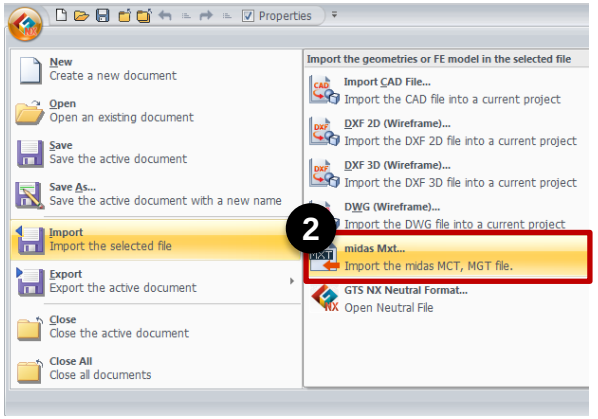
```
;----------------------------------------------------------------------
; MIDAS/Civil Text(MCT) File.
; Date : 2014/12/16
;----------------------------------------------------------------------
--

*VERSION
   8.1.5

*UNIT     ; Unit System
; FORCE, LENGTH, HEAT, TEMPER
   KIPS , FT, BTU, F

*PROJINFO     ; Project Information
   USER=midasit
   ADDRESS=MidasIt

*REBAR-MATL-CODE     ; Rebar Material Code
; CONC_CODE, CONC_MDB, SRC_CODE, SRC_MDB
 ASTM(RC), Grade 60, ASTM(RC), Grade 60

*NODE     ; Nodes
; iNO, X, Y, Z
     1, 0, 0, 0
     2, 16.4041994750656, 0, 11.8110236220472
     3, 32.8083989501312, 0, 20.997375328084
     4, 49.2125984251968, 0, 27.5590551181102
     5, 65.6167979002625, 0, 31.496062992126
     6, 82.0209973753281, 0, 32.8083989501312
     7, 98.4251968503937, 0, 31.496062992126
     8, 114.829396325459, 0, 27.5590551181102
```

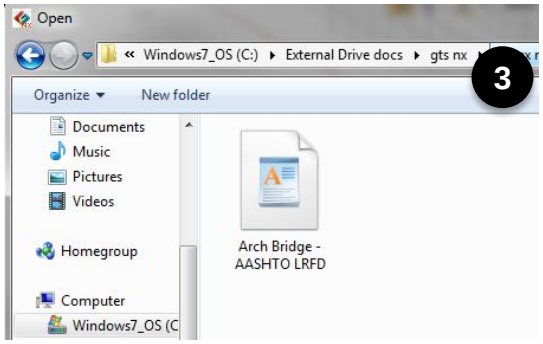
Procedure

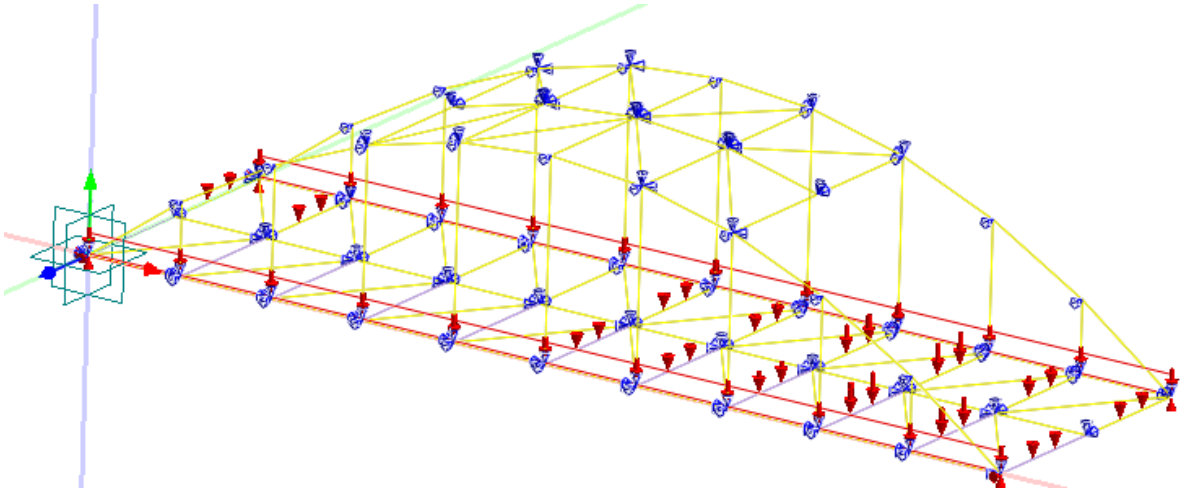
1. Start Menu > Export > MXT
2. Save file
3. Text file opens

Step 6. **Import**

GTS NX





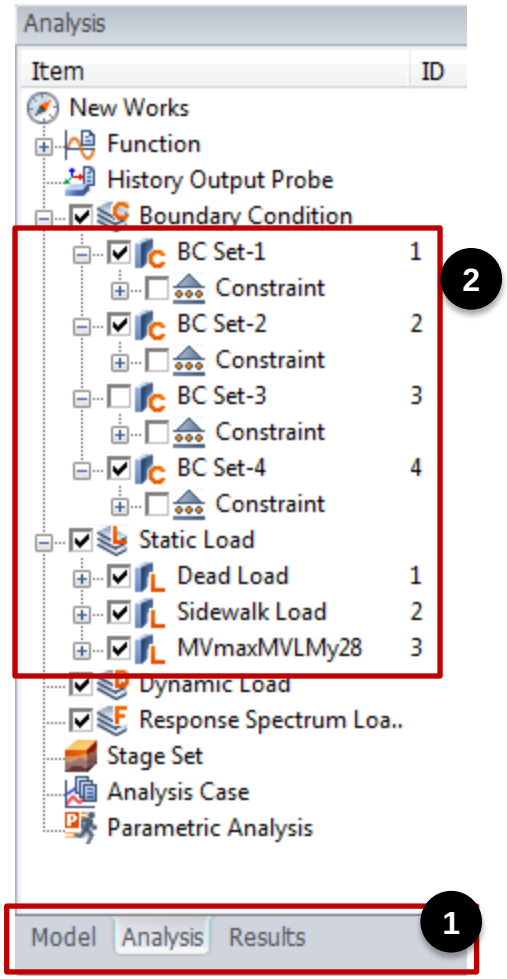


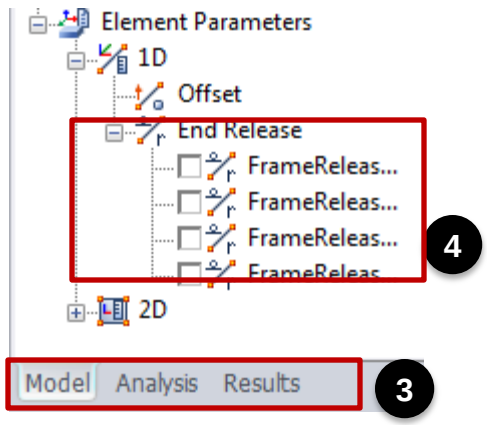
Procedure

1. Open GTS NX
2. Start Menu > Import > midas MXT
3. Select mxt file
4. Click Open

Step 7. Deactivate sets

GTS NX



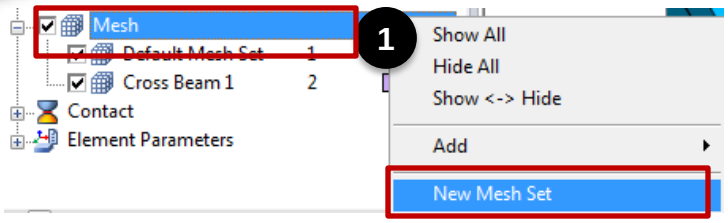


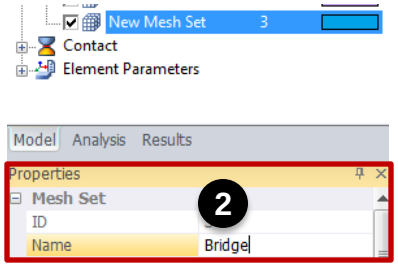
Procedure

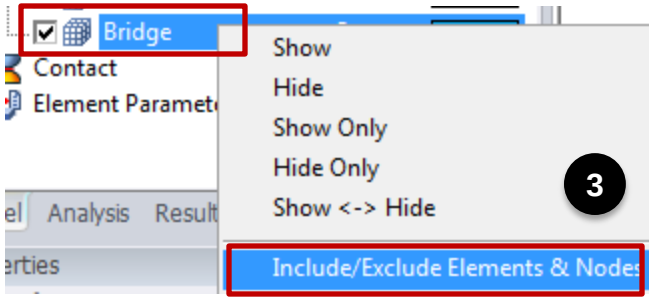
1. Go to Works Tree Menu > Analysis
2. Delete the BC Constraint set. Deactivate Boundary conditions, Static and Dynamic loads
3. Go to Works Tree Menu > Model
4. Deactivate the End Realease
5. Change units to kN, m , s

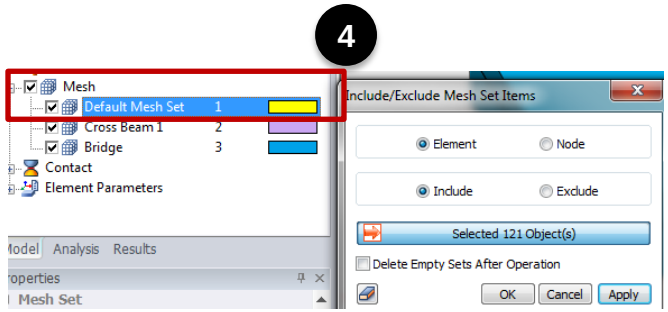
Step 8. Move imported mesh set

GTS NX









Procedure

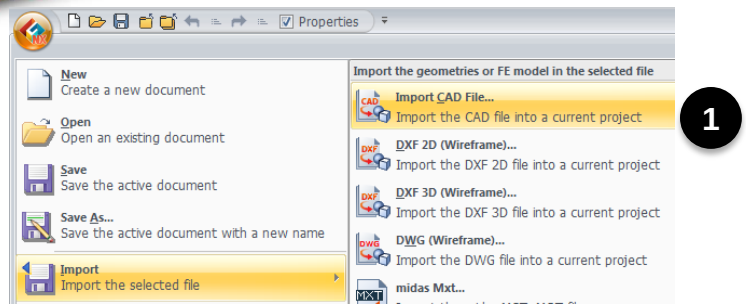
Structure is imported into Defualt Mesh Set, which does not permit editing. We must move the structure into a new mesh set that will allow for modifications.

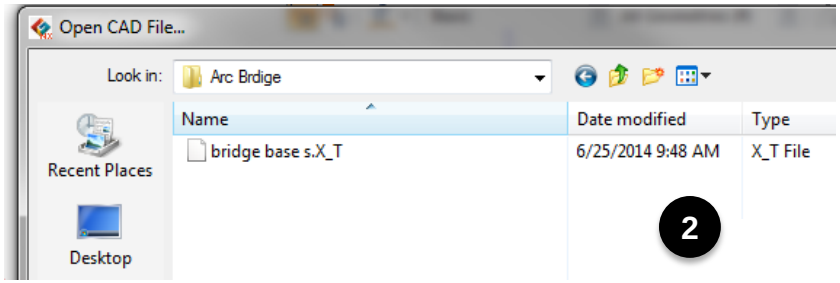
1. Create new mesh set by right clicking over Mesh tab in Work Tree
2. Change name of new mesh in the property window *Name tab*. Click apply.
3. Right click over the building mesh set and select Inlcude / Exclude Elements option.
4. Select Default Mesh Set and Include all 1452 Elements. Click apply

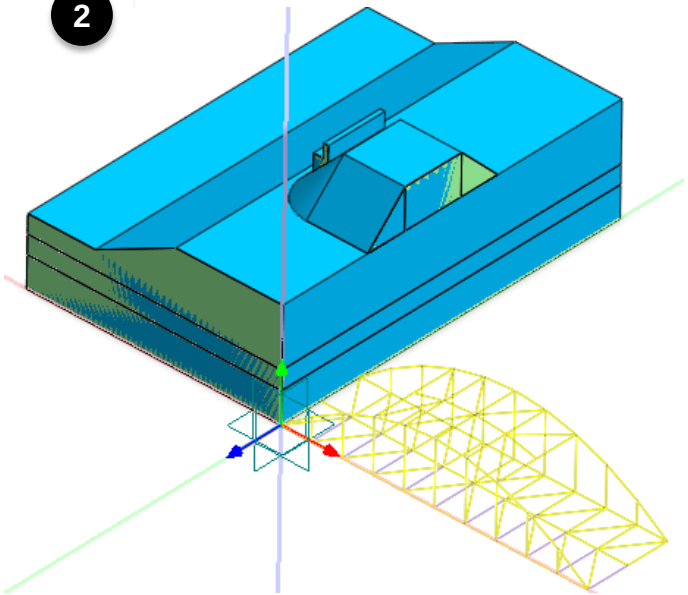
You can now edit the structure and delete sections if needed.

Step 9. Import CAD

GTS NX







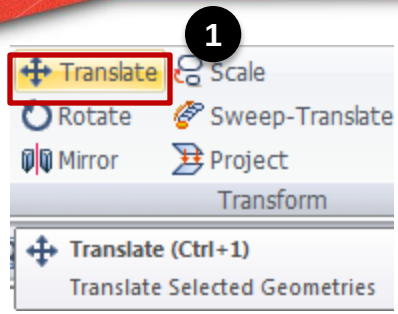
Procedure

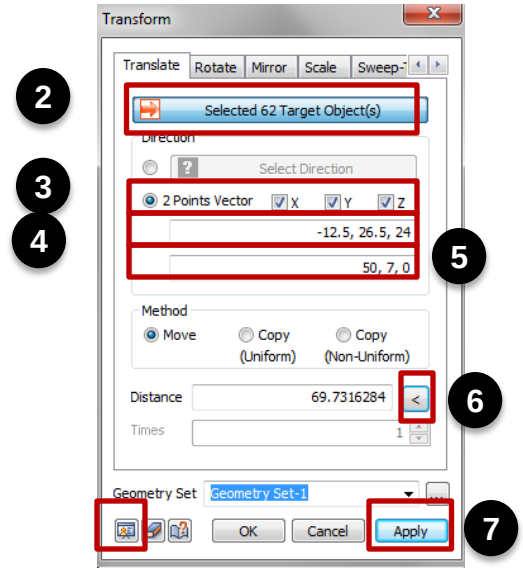
1. Start Menu > Import > CAD File
2. Select X_T file
3. Click Open

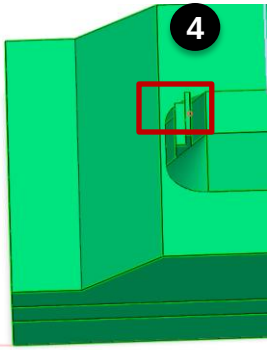
In this case we are importing CAD file of ground geometry From Abutment Foundation Tutorial.
However, GTS NX also has commands to create geometry directly in software similar to other CAD software.

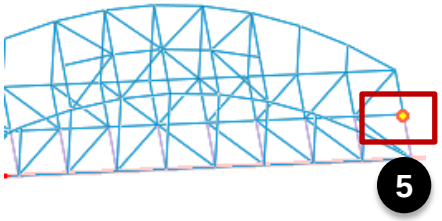
Step 10. **Move CAD**

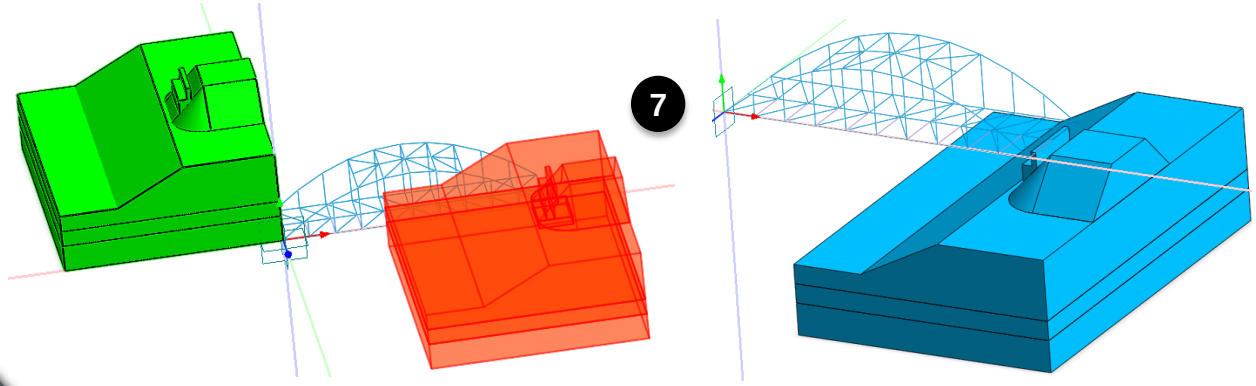
GTS NX











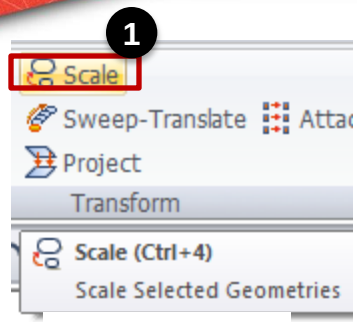
Procedure

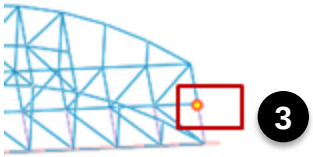
1. Main Menu > Geometry > Translate
2. Select all the geometries
3. Select 2 Points Vector as Direction
4. Select the middle point on the base of the abutment as the 1st point
5. Select the middle node of the right end of the bridge as the 2nd point
6. Click the Distance button. Program will calculate distance automatically.
7. Click Preview, then Apply

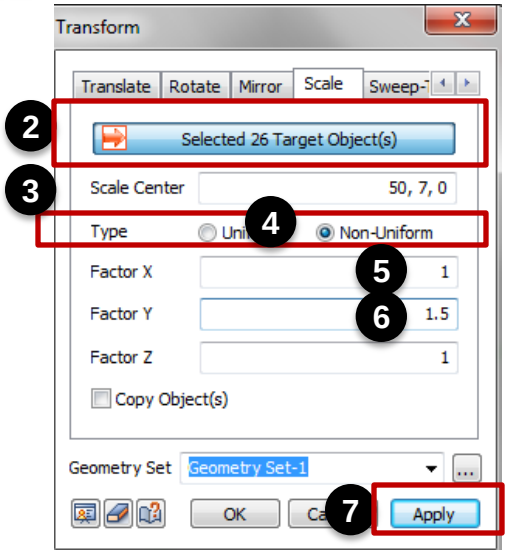
This command translates the selected geometries and lines up the points.

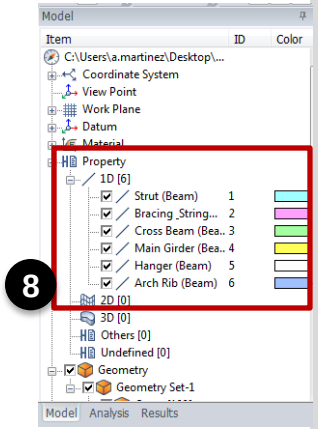
Step 11. Scale CAD

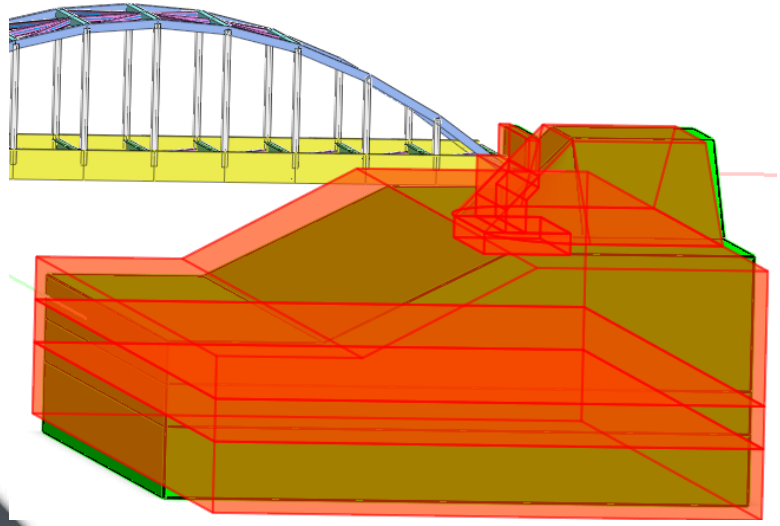
GTS NX

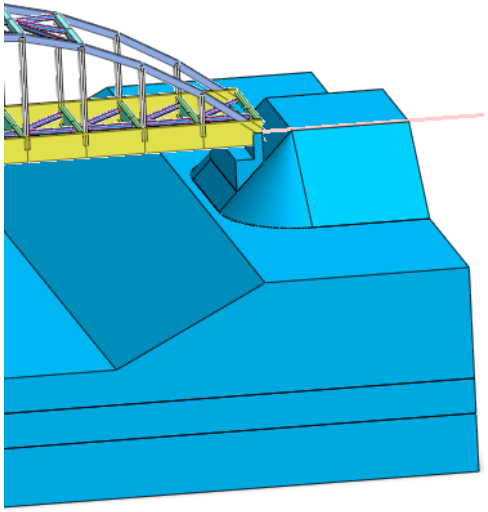












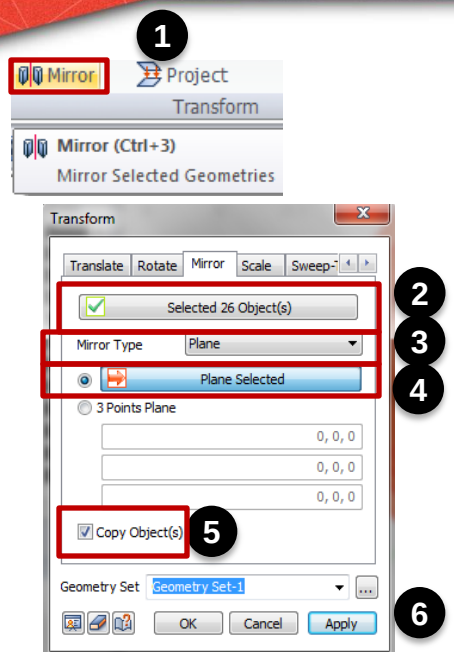
Procedure

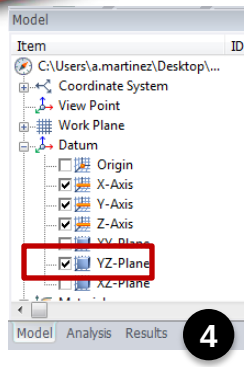
1. Main Menu > Geometry > Scale
2. Select the 6 solid and the 20 pile geometries
3. Select the center of the right end of ten bridge as Scale Center
4. Select Non-Uniform under type
5. Type in 1 for Factor X
6. Type in 1.5 for Factor Y
7. Select Preview then Apply
8. You can activate the 1D strucutral element properties for easier visualization

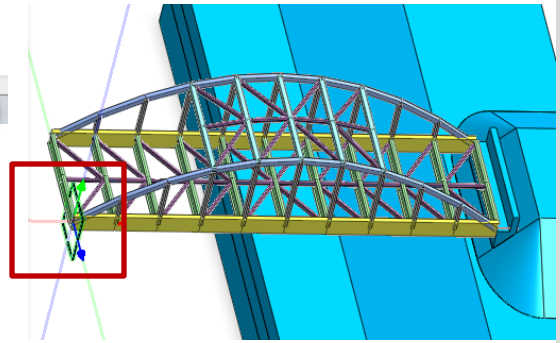
This feature allows you easily scale your geometries to match any structure.

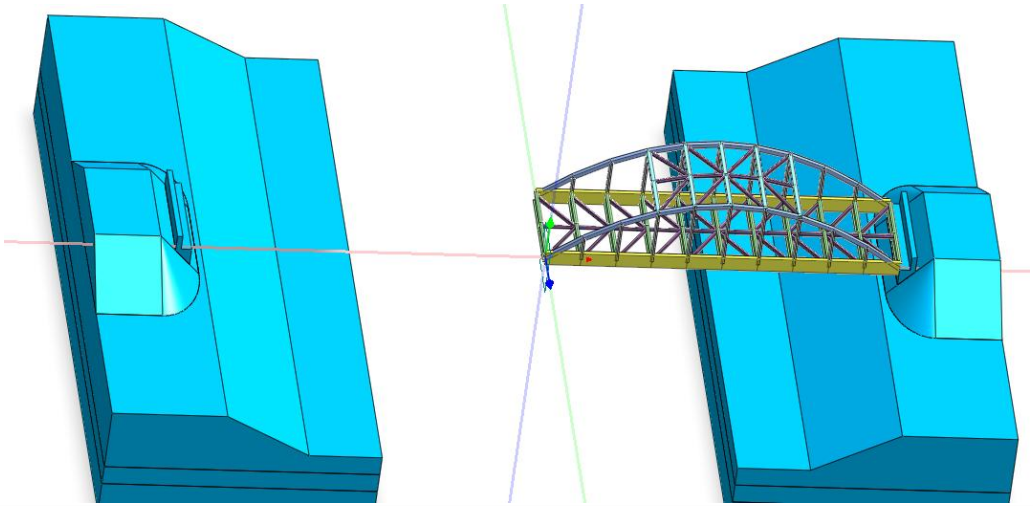
Step 12. **Mirror CAD**

GTS NX







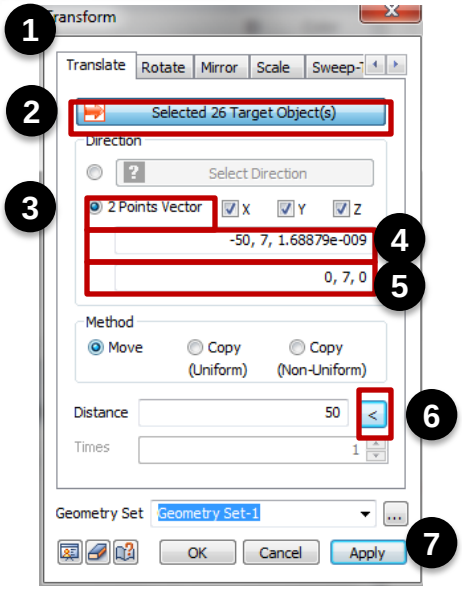


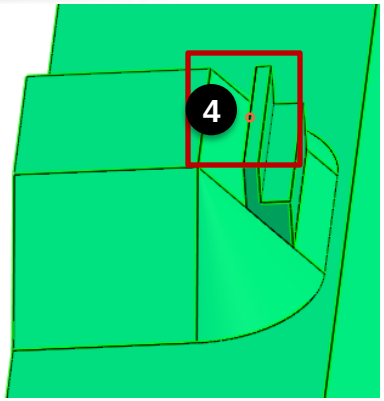
Procedure

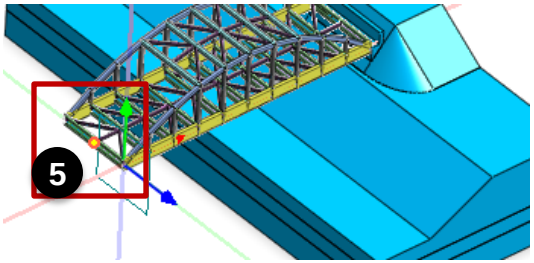
1. Main Menu > Geometry > Transform > Mirror
2. Select the 6 solids and the 20 lines as target object.
3. Select Plane as Mirror type
4. Select the YZ-Plane from the Tree Menu or the Model
5. Activate *Copy Object Box*
6. Press Preview, then Apply

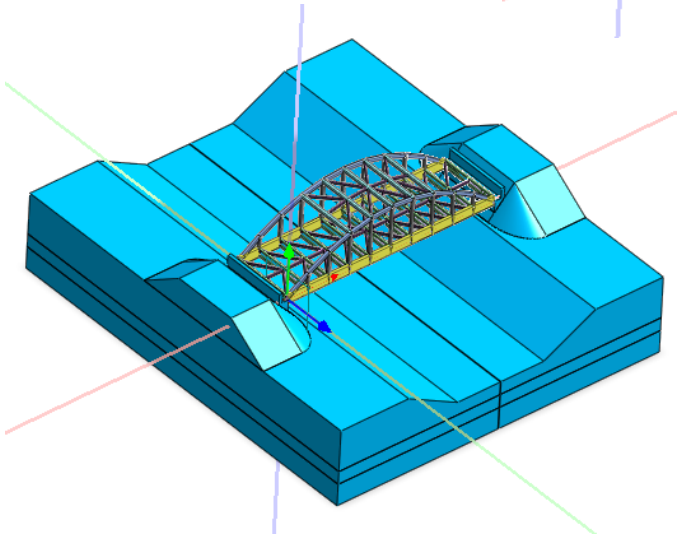
Step 13. Translate CAD

GTS NX







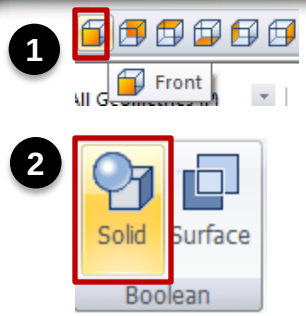


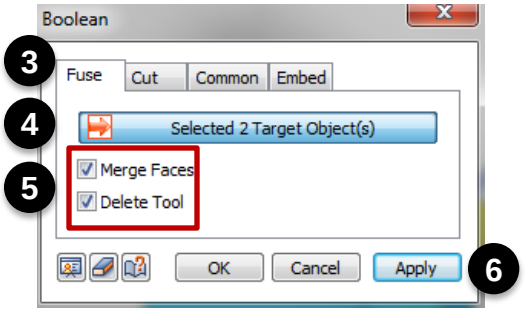
Procedure

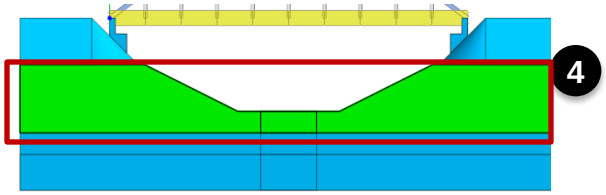
1. Main Menu > Geometry > Transform > Translate
2. Select the 6 solid and 20 lines as target object.
3. Select 2 Points Vector
4. The mid point on the top of the abutment.
5. Select the mid point on the left end of the bridge.
6. Press the < arrow to calculate the distance
7. Press Preview, then Apply

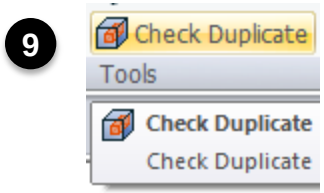
Step 14. Fuse solids

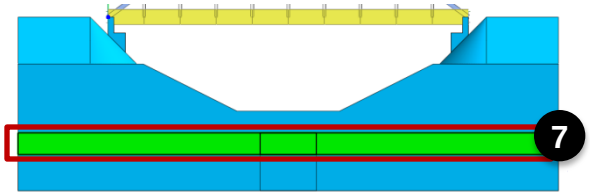
GTS NX

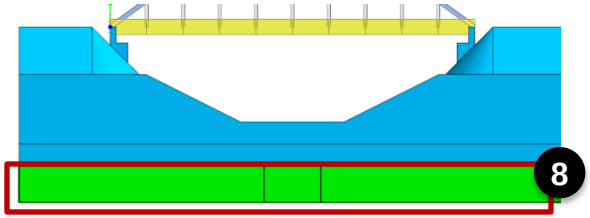


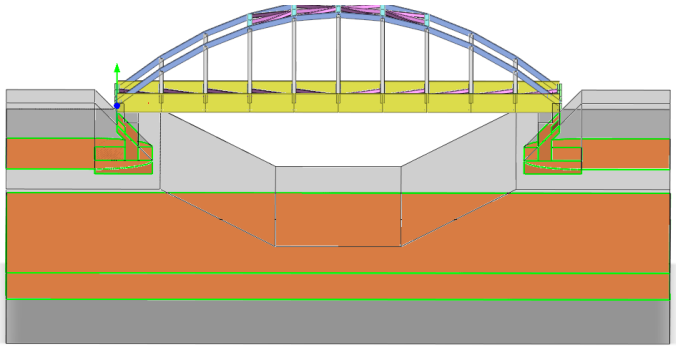












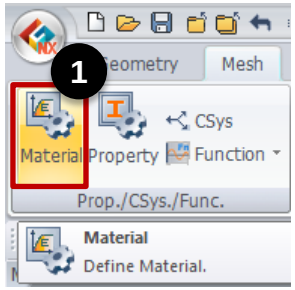
Procedure

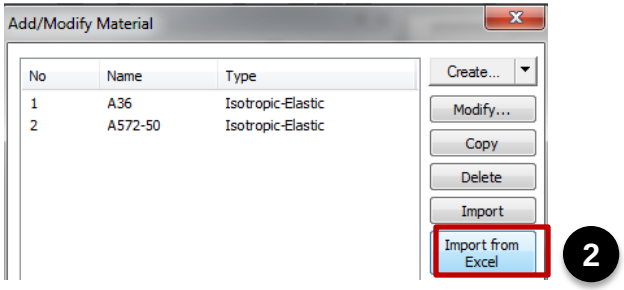
1. Change the view to Front
2. Select Boolean > Solid
3. Select Fuse
4. Select the 2 top ground layer geometries
5. Select Merge Faces and Delete Tool.
6. Press Apply
7. Repeat for the mid ground layers
8. Repeat for bottom ground layers
9. Check the duplicate faces of the new geometries

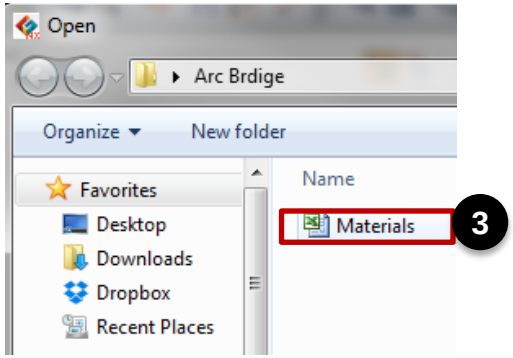
The faces shared by the geometries should light up orange.

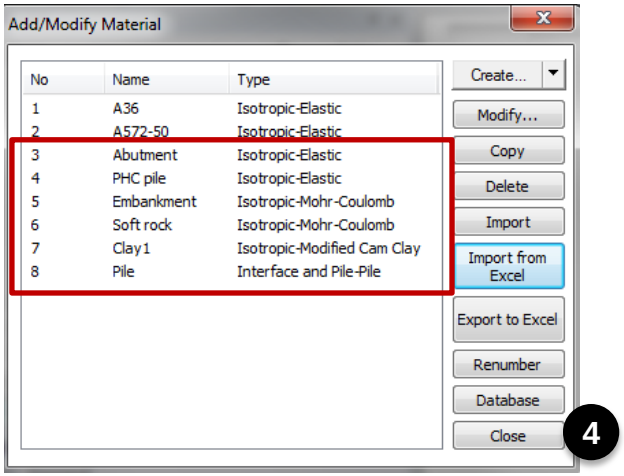
Step 15. Mesh > Material > Import

GTS NX







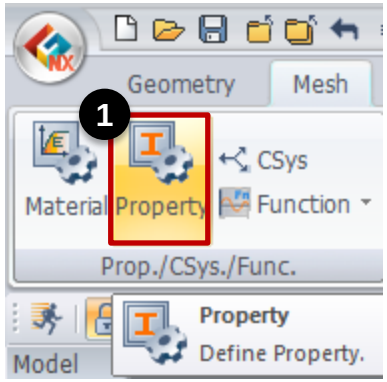


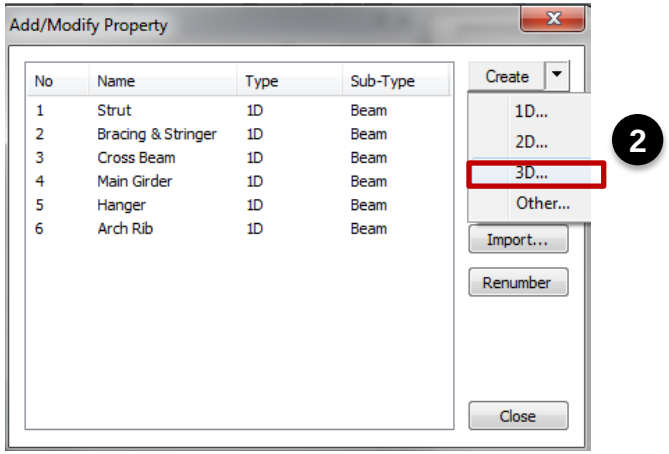
Procedure

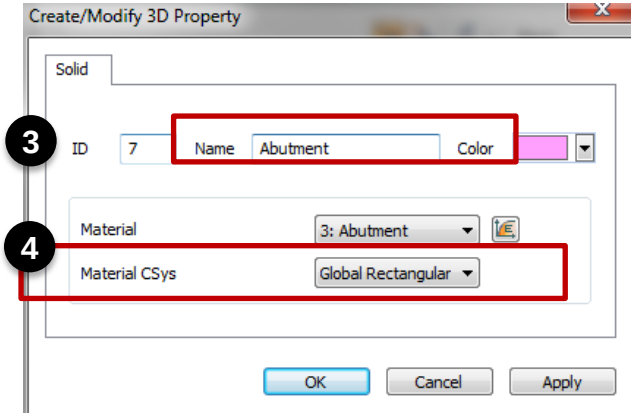
1. Main Menu > Mesh > Material
2. Select Import from Excel
3. Select Excel File: Materials
4. Press Close; 6 new materials were imported.

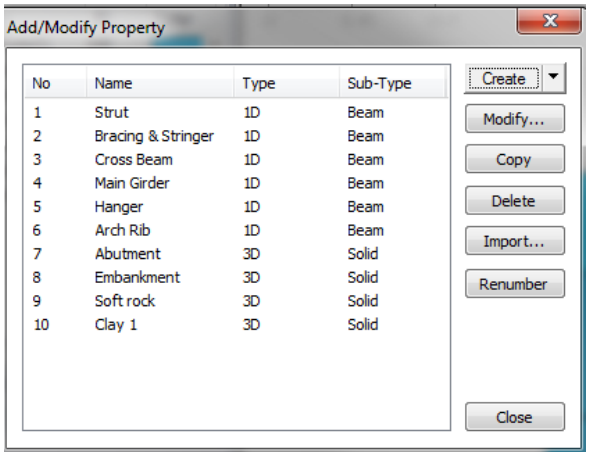
Step 16. Mesh > Property> 3D

GTS NX







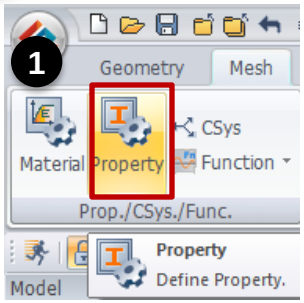


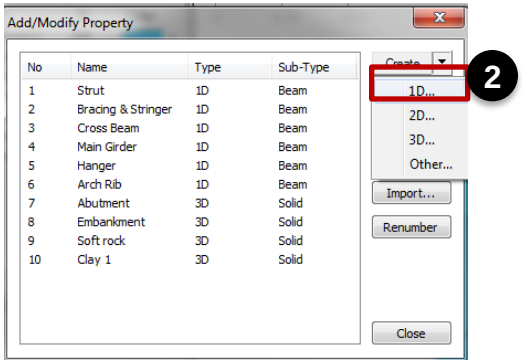
Procedure

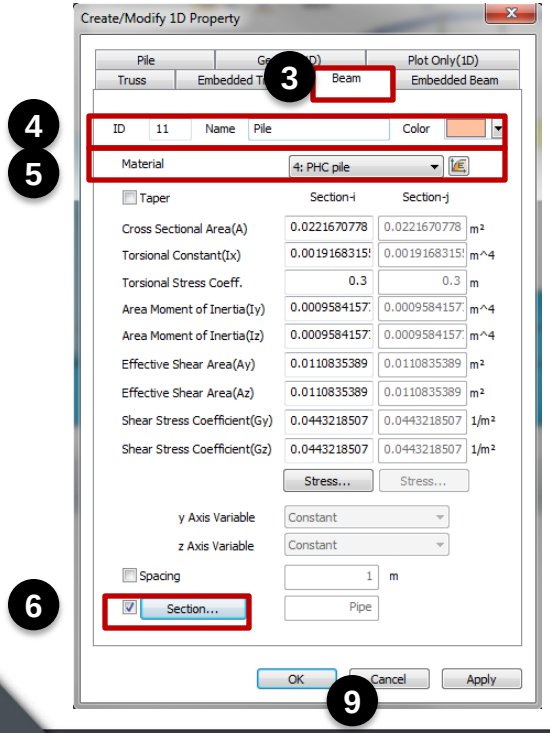
1. Main Menu > Mesh > Property
2. Select Create > 3D
3. Name: Abutment
4. Material: Select Abutment
5. Repeat for Embankment, Clay and Soft Rock

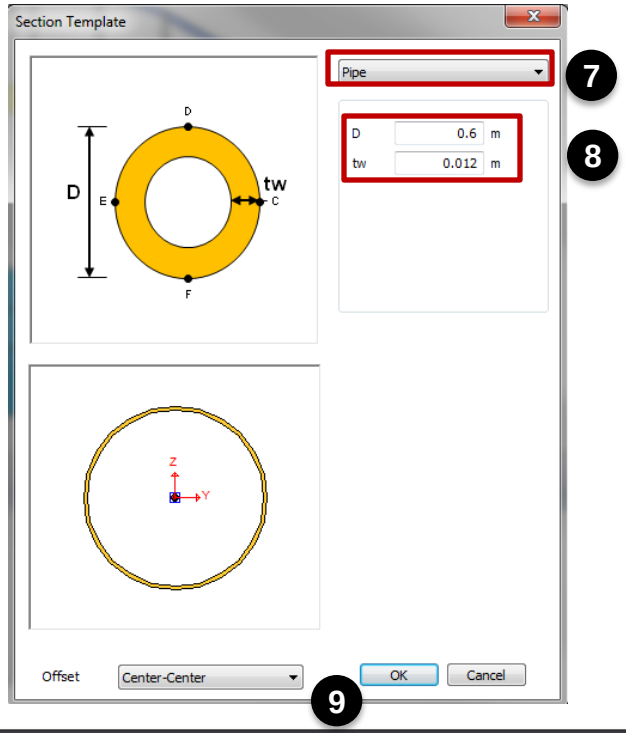
Step 17. Mesh > Property>1D

GTS NX









Procedure

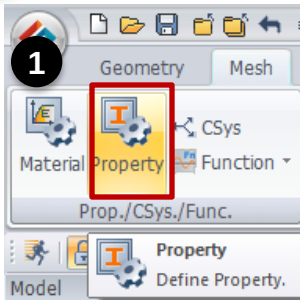
1. Main Menu > Mesh > Property
2. Select Create > 1D
3. Select Beam
4. Name: Pile
5. Material: PHC Pile
6. Activate Section Box
7. Section > Pipe :
8. D: 0.6m
   Tw: 0.012m

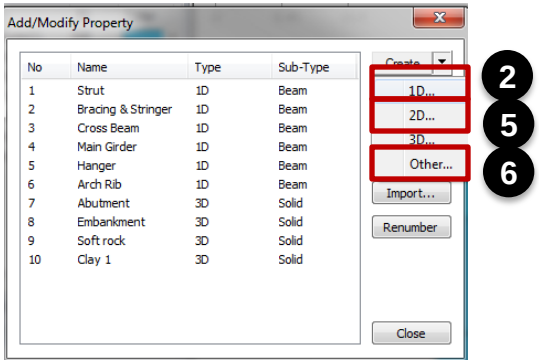
Program will automatically fill in the section properties after giving dimensions.

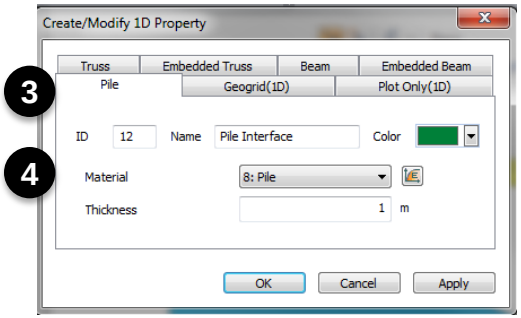
9. Press OK twice to close

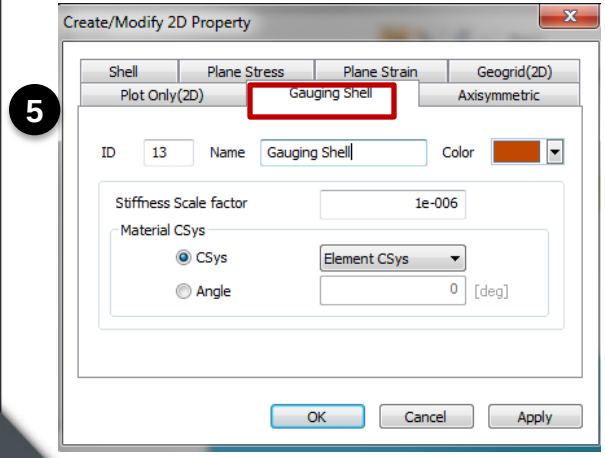
Step 18. Mesh > Property

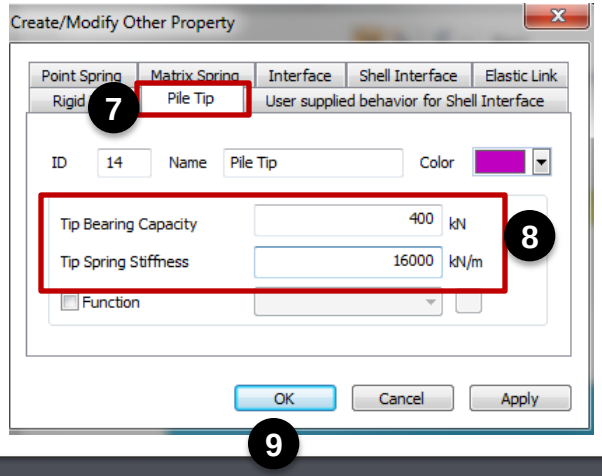
GTS NX











Procedure

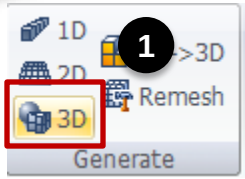
1. Main Menu > Mesh > Property
2. Select Create > 1D
3. Select Pile
4. Insert information shown
5. Repeat for creating material under 2D for Gauging Shell
6. Repeat for creating material under OTHER
7. Select Pile Tip
8. Type in infomration shown:

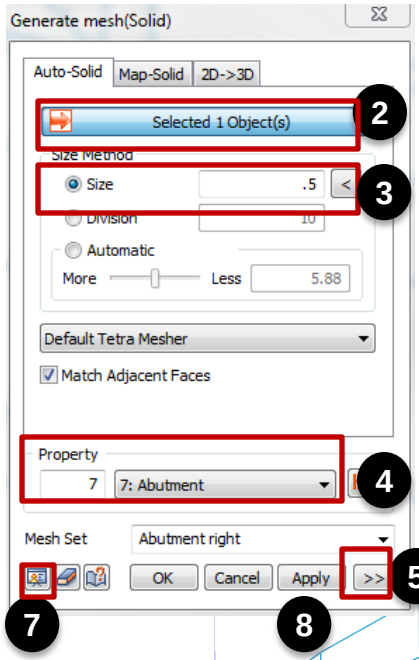
Bearing capacity: 400kN
Spring Stiffness: 16000kN/m

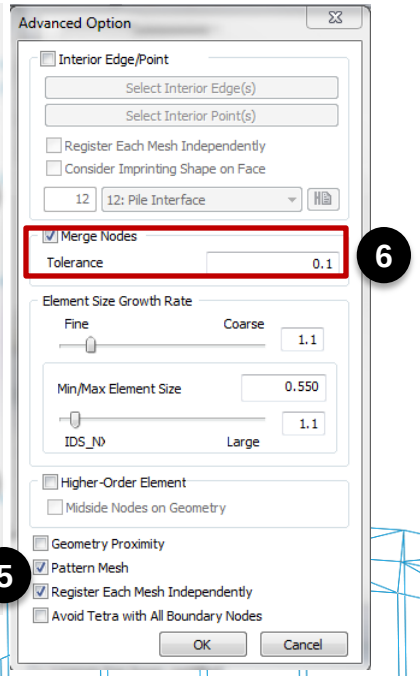
9. Press OK

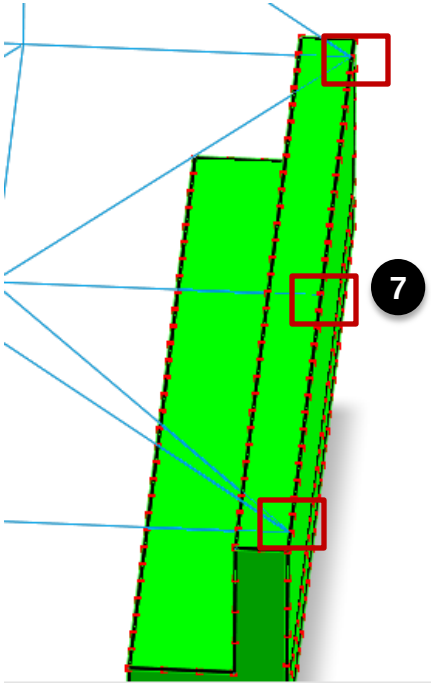
Step 19. **Mesh**

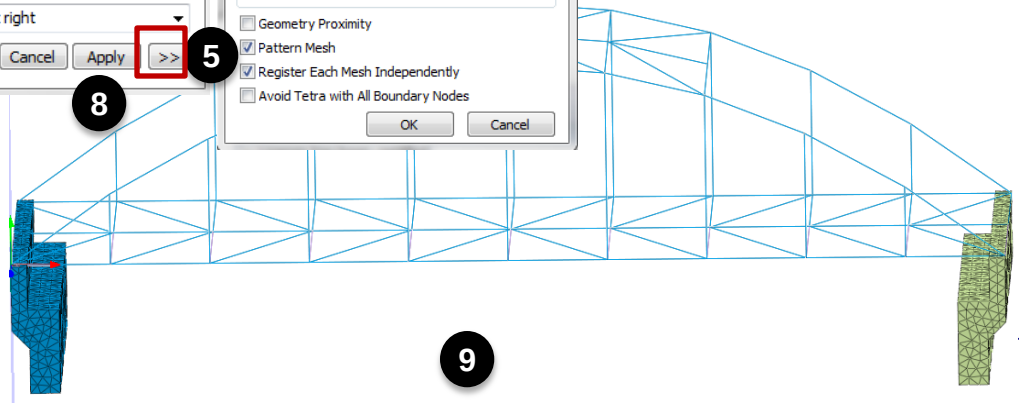
GTS NX









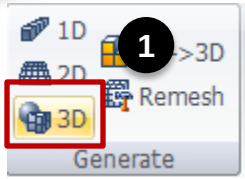


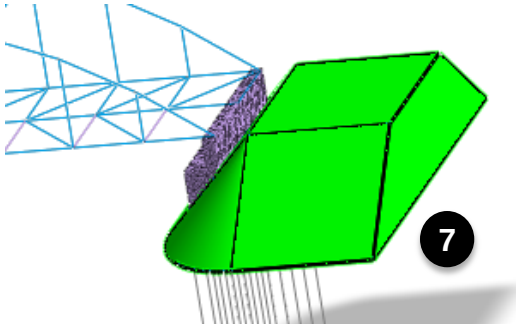
Procedure

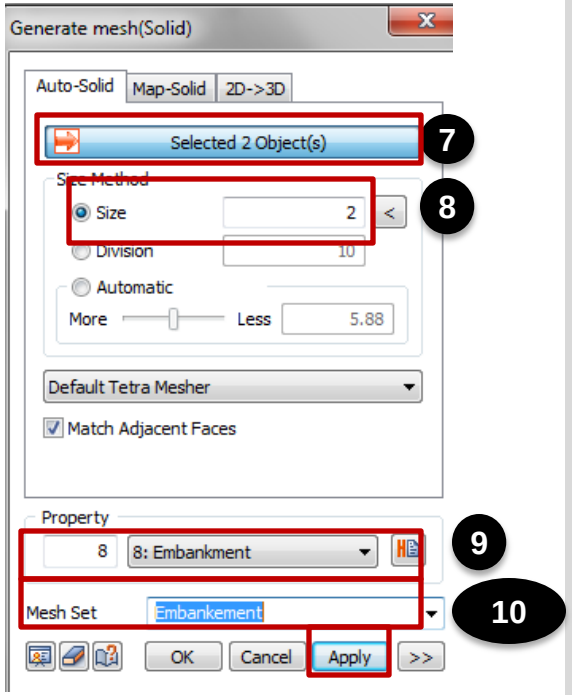
1. Mesh>Generate>3D
2. Select the right abutment only
3. Type in 0.5 in Size
4. Select Abutment from Property
5. Press the >> button to open advanced mesh options
6. Set the tolerance for Merge Nodes at 0.1
7. Press Preview and verify that the nodes of the abutment match up with the 3 points of the bridge.
8. Click apply
9. Repeat for left abutment

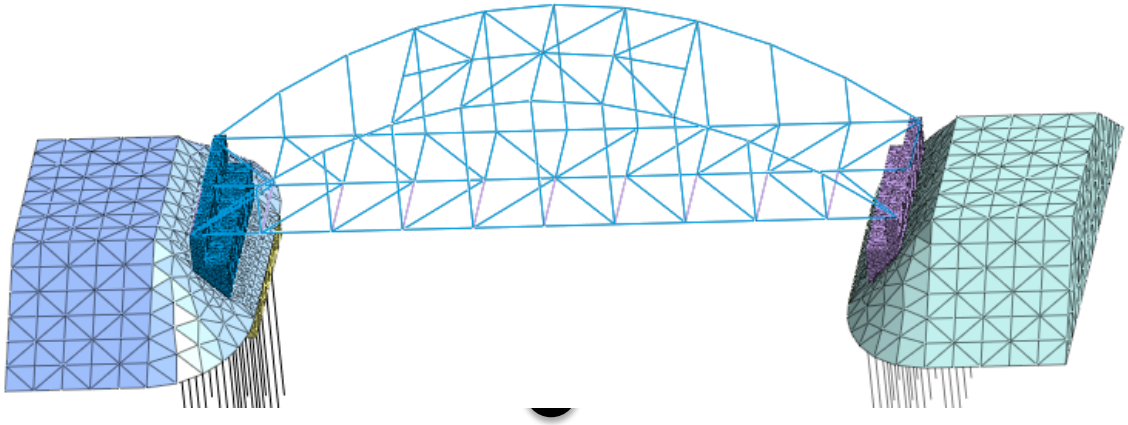
Step 20. **Mesh**

GTS NX







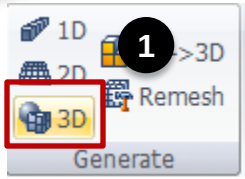


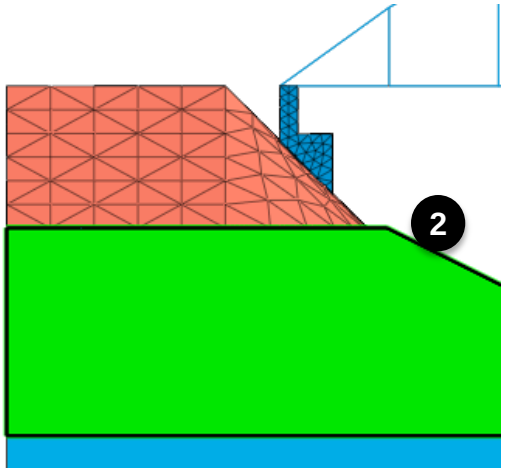
Procedure

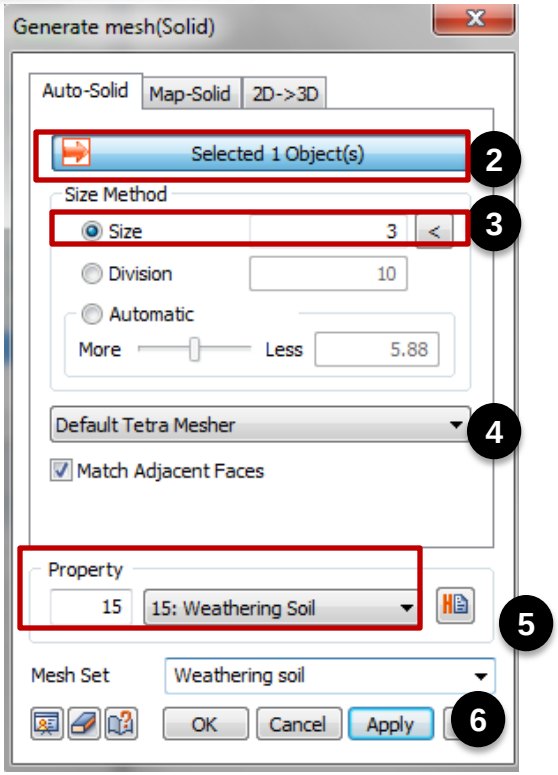
1. Mesh>Generate>3D
2. Select the right and left abutment bases
3. Type in 1 in Size
4. Use Default Tetra Mesher
5. Select Abutment in properties
6. Type Abutment in Mesh Set name then Click Apply
7. Select the left and right embankments
8. Type 2 for size
9. Select Embankement in property
10. Type Embankment in Mest set then Click Apply

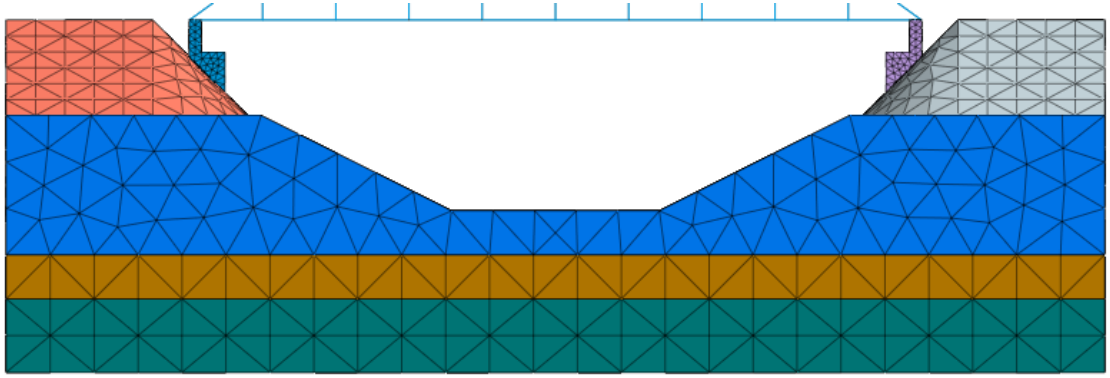
Step 21. **Mesh**

GTS NX







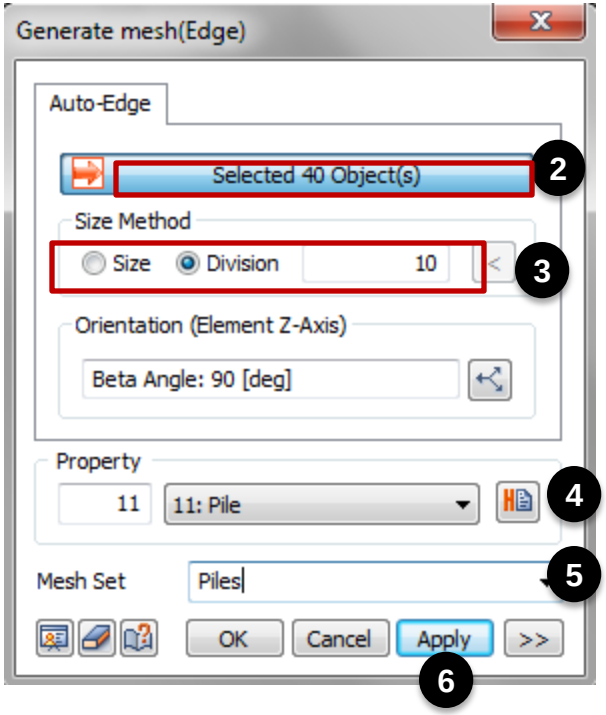


Procedure

1. Mesh>Generate>3D
2. Select the top ground layer
3. Type in 3 in Size
4. Use Default Tetra Mesher
5. Select Weathering soil in properties
6. Click Apply
7. Repeat for Clay middle layer and Soft Rock bottom gound layer

Step 22. **Mesh**

GTS NX



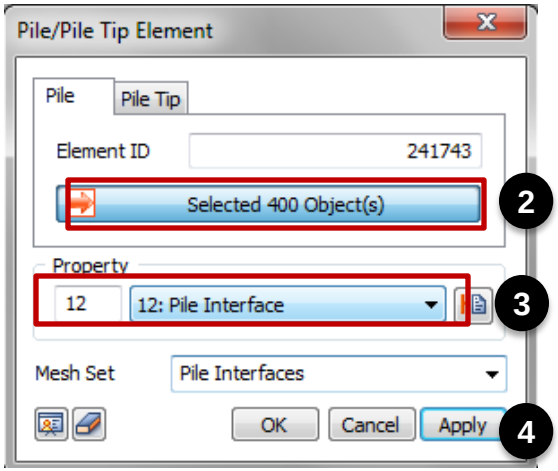
2

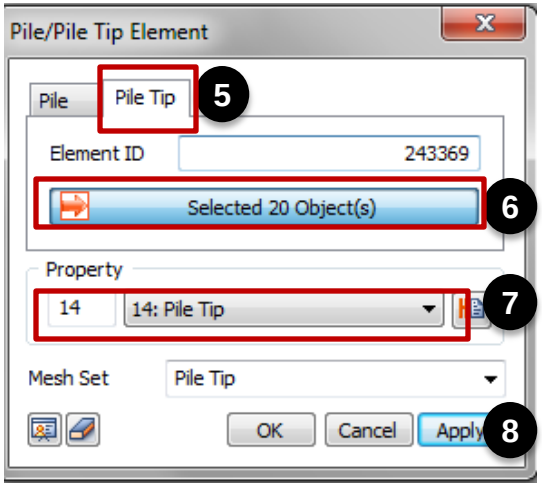
Procedure

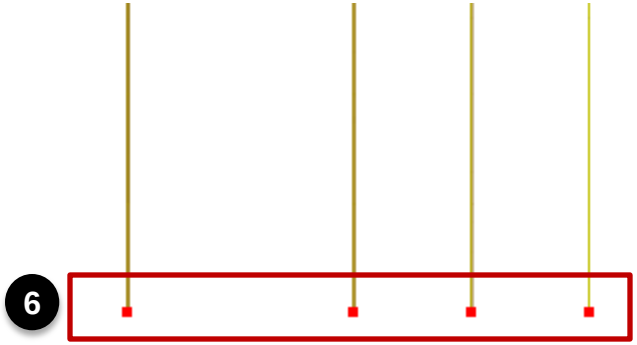
1. Mesh>Generate>1D
2. Select the 40 lines
3. Type in 10 in Division
4. Select Pile from Properties
5. Type Piles in Mesh Set
6. Click Apply

Step 23. Pile Interface and Tips

GTS NX





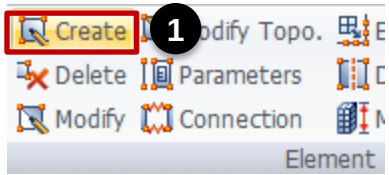


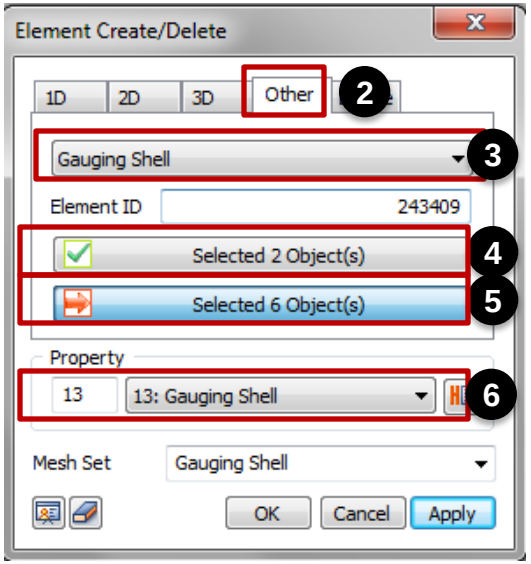
Procedure

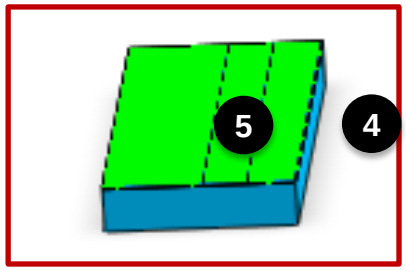
1. Mesh>Element>Pile /PileTip
2. Select the 400 elements of the piles
3. In property select Pile Interface
4. Click Apply
5. Select the Pile Tip Tab
6. Select the 20 pile tip nodes from the left pile sets
7. In property select the Pile Tip
8. Click Apply
9. Repeat for the right piles

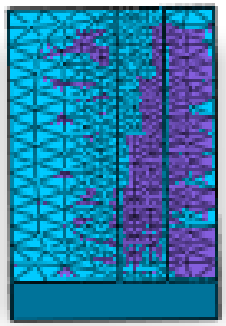
Step 24. Gauging Shell

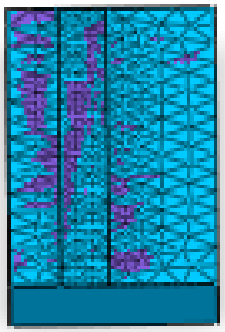
GTS NX











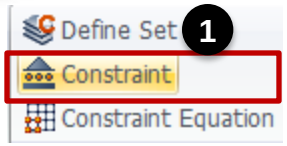
Procedure

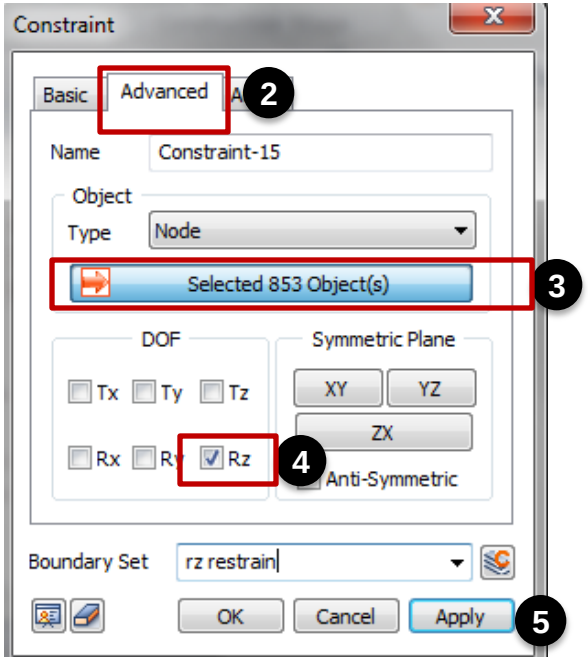
1. Mesh > Element > Create
2. Select the Other Tab
3. Select Gauging Shell from the drop down menu
4. Select the 2 soild geometries of the base of the abutment
5. Select the 6 top faces
6. In property tab select Gauging Shell
7. Click Apply

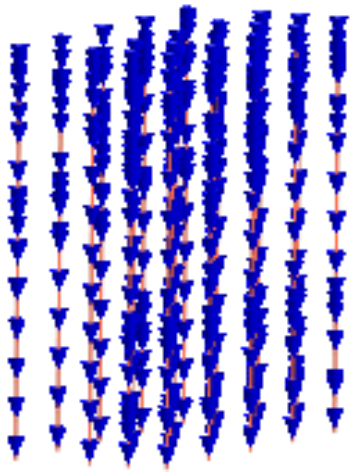
Gauging Shells allow you to obtain strucutral results like forces and moments on a solid object.

Step 25. Boundary Conditions

GTS NX







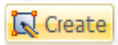
Procedure

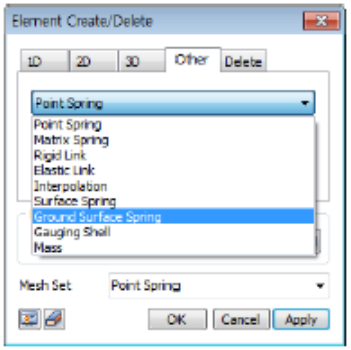
1. Seepage Consolidation Analysis > Boundary > Constrain
2. Select the Advanced Tab
3. Select the Pile Elements
4. Activate Rz, to restrict rotation about Z axis
5. Click Apply

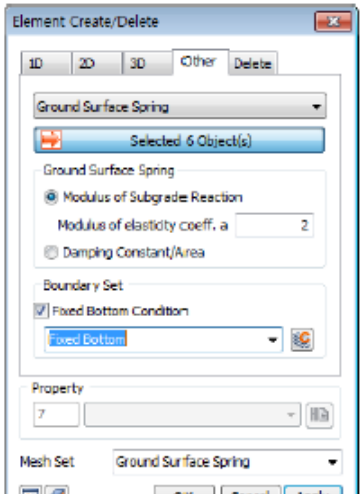
Step 26. **Ground surface spring** (Modulus of subgrade reaction)

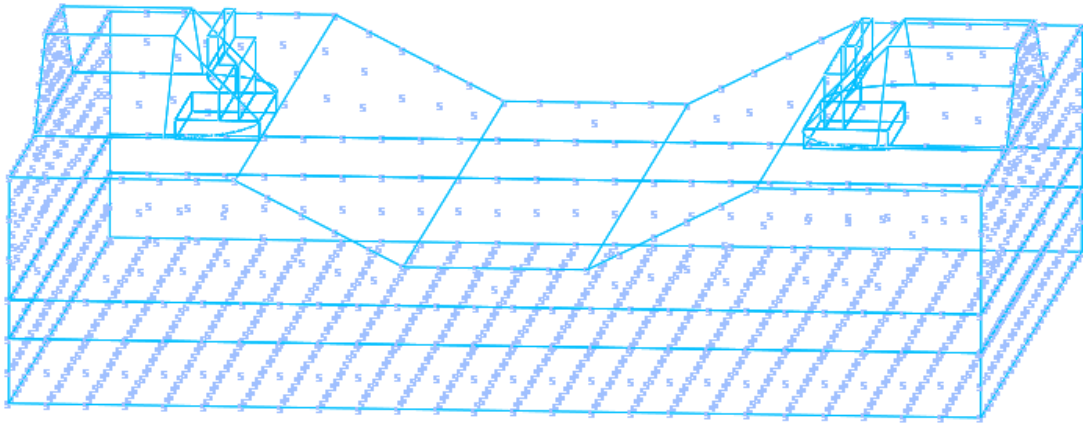
GTS NX

1









• Dynamic shear modulus and poisson's ratio are required to input for the material property

Procedure

Create ground surface spring (Boundary for eigenvalue analysis)

1. Mesh > Element > Create
   - Other > Ground Surface Spring
   - Select all solid elements (Mesh set 9 total)
   - Modulus of elasticity coeff. a = 2
- Check "Fixed Bottom"

(* Modulus of subgrade reaction will be auto calculated based on the following equation(next slide))

Step 27. Ground surface spring (Modulus of subgrade reaction)

GTS NX

- Modulus of subgrade reaction (Auto-calculated)

$$K_H = k_{h0}\left(\frac{B_h}{30}\right)^{-3/4} \qquad k_{h0} = \frac{1}{30}\alpha \cdot E_0$$

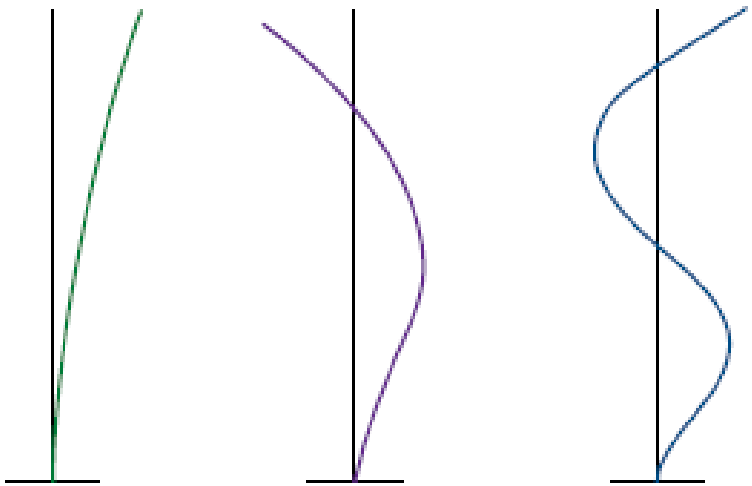
$$K_V = k_{v0}\left(\frac{B_v}{30}\right)^{-3/4} \qquad k_{v0} = \frac{1}{30}\alpha \cdot E_0$$

K_H : Horizontal subgrade reaction (kN/m³)
K_V : Vertical subgrade reaction (kN/m³)
k_{h0} : Hori. Subgrade reaction from PBT (30cm in diameter of steel plate)
k_{v0} : Ver. Subgrade reaction from PBT (30cm in diameter of steel plate)
B_h : $B_h = \sqrt{A_h}$
B_v : $B_v = \sqrt{A_v}$
A_h : Horizontal area of element face (m²)
A_v : Vertical area of element face (m²)
α : Modulus of elasticity coefficient (From table)

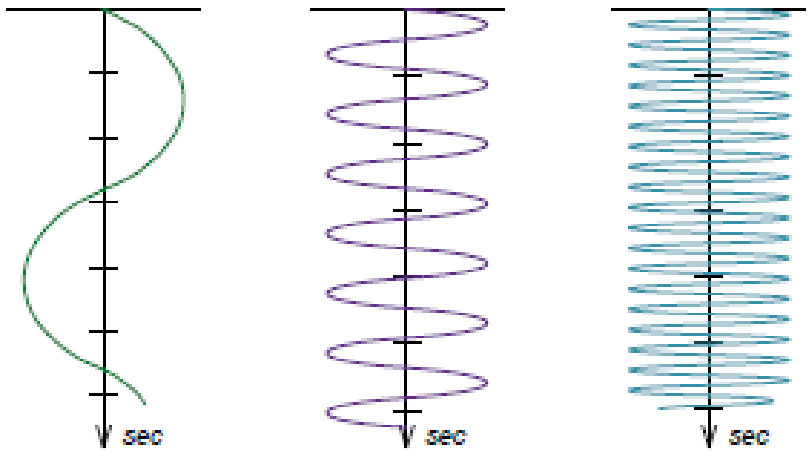
[E_0 vs α]

E_0 (kN/m²)	α	
	Under static	Under seismic
Elastic modulus from PBT	1	2
Measured elastic modulus from boring hole	4	8
Elastic modulus from 1D or 3D compresseion test	4	8
Elastic modulus from SPT using following relation (Eo= 28*N (tonf/m²))	1	2

Step 28. **Ground surface spring** (Modulus of subgrade reaction)



[Natural mode]



$\lambda_1 = 1.87510407$
$T_1 = 1.78702$ sec

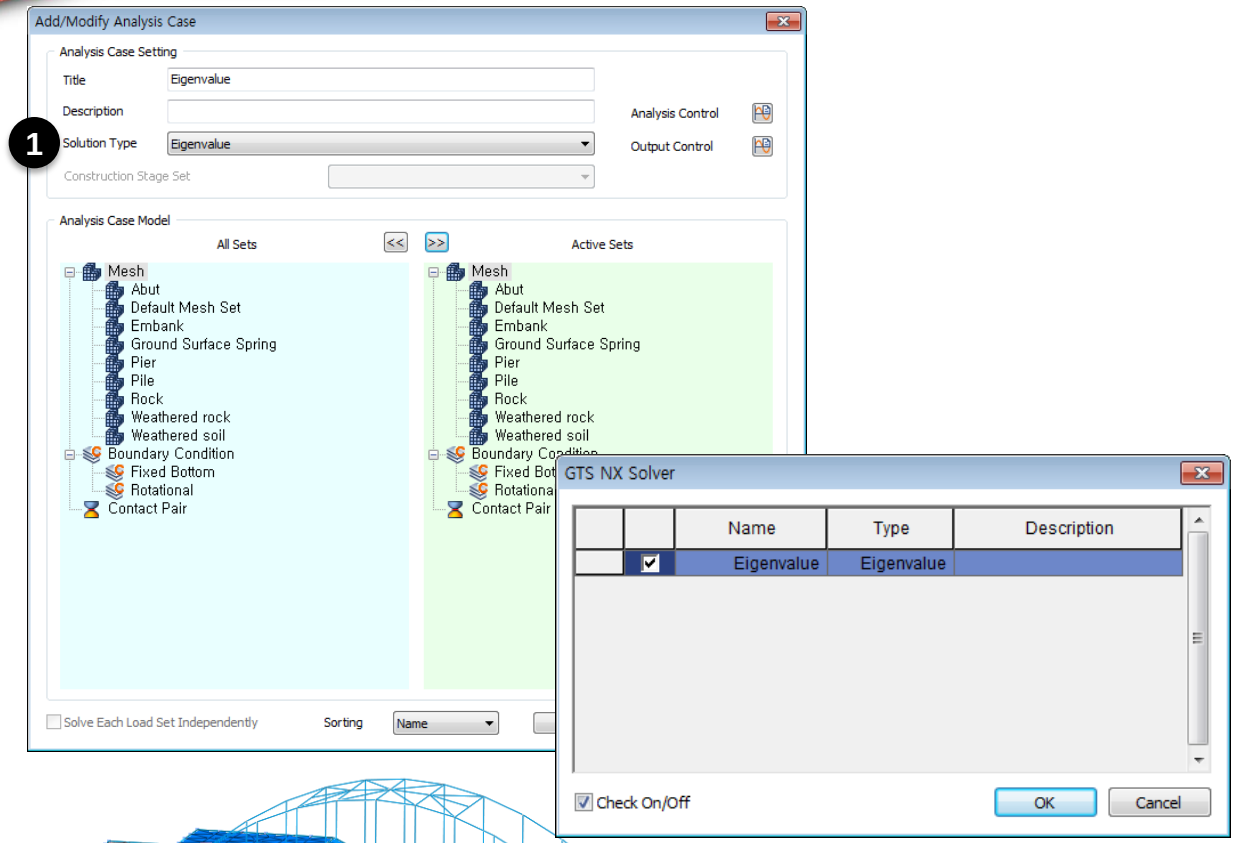
$\lambda_2 = 4.69409113$
$T_2 = 0.28515$ sec

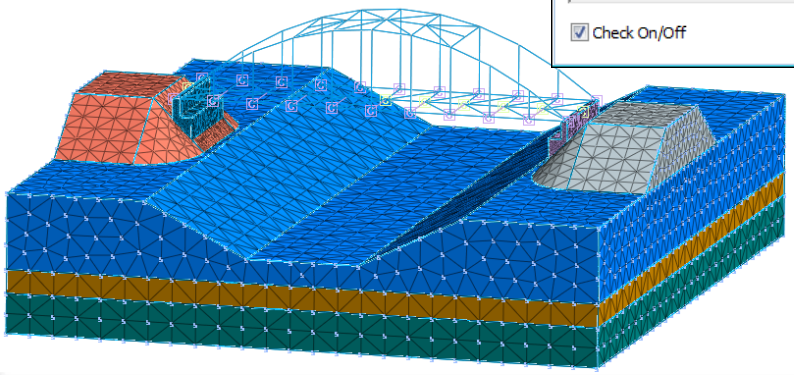
$\lambda_3 = 7.85475744$
$T_3 = 0.10184$ sec

[Natural frequency]

Step 29: **Eigen Value Analysis**

GTS NX





Procedure

Perform eigenvalue analysis to get the natural modes which are related to the max. mass participation.

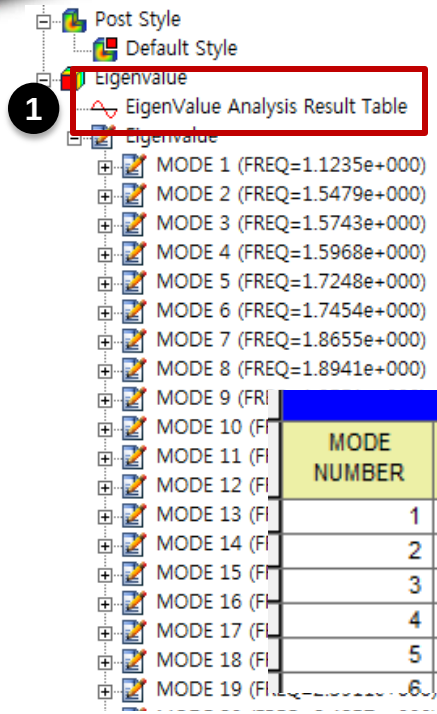
1. Analysis > Analysis Case > General
   - Solution Type : Eigenvalue analysis
   - Activate all mesh sets and boundaries
   - Select Analysis Control
   - Number of Modes : 30
   - Check ‘Sturm Sequence Check’

2. Analysis > Analysis > Perform

The Lanczos resampling method is used for eigenvalue analysis, and this method is appropriate for soil-structure interaction problems. Since this method can cause omission of eigenvalue, using “Sturm sequence check” option is recommended.

Step 30: Eigen Value Analysis

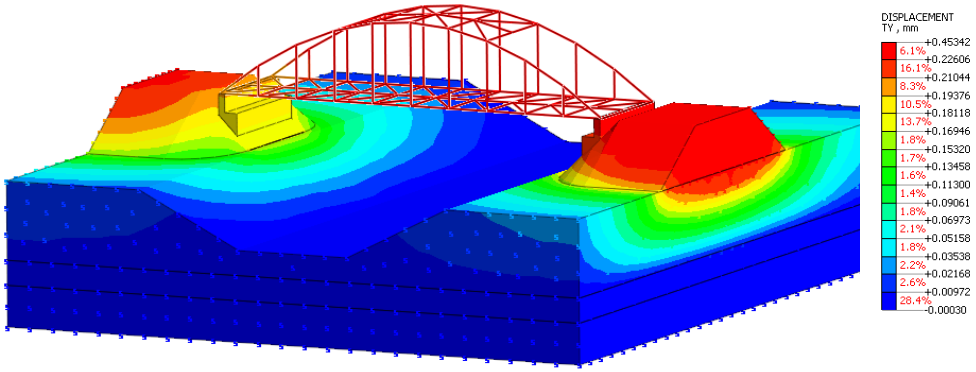
GTS NX



MODE NUMBER	T1	T2
1	0.00%	19.59%
2	0.00%	0.16%
3	0.80%	0.00%
4	0.00%	0.13%
5	0.03%	1.68%

REAL EIGENVA

MODE NUMBER	EIGENVALUE	RADIANS	CYCLES	PERIOD
1	4.356144e+001	6.600109e+000	1.050440e+000	9.519820e-001
2	4.548183e+001	6.744022e+000	1.073344e+000	9.316678e-001
3	9.128010e+001	9.554062e+000	1.520576e+000	6.576455e-001
4	9.595534e+001	9.795680e+000	1.559031e+000	6.414241e-001
5	9.717107e+001	9.857539e+000	1.568876e+000	6.373990e-001
6	9.944497e+001	9.972210e+000	1.587126e+000	6.300697e-001



Procedure

Check the first and second mode which have a high mass participation factor must be considered in the seismic analysis These two modes will be used to calculate Damping effect in analysis.

1. Double click on the “Eigenvalue Analysis Result Table“
   - Check the Mode number with the first and second mass participation along the direction of seismic wave which will be applied.
   - Check Period corresponding each mode number

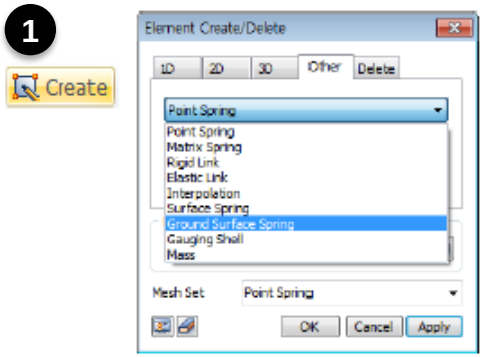
(Period) – (0.951982, 0.6373990)

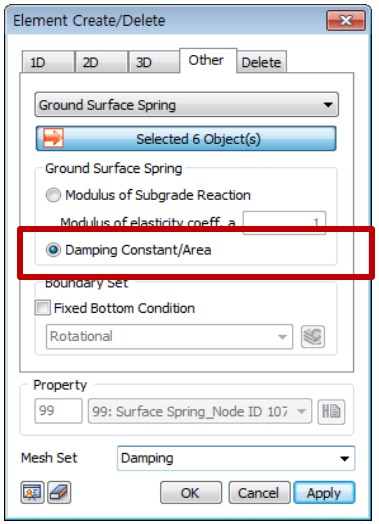
In case that the model does not include bedrock, it is required to apply fixed boundary at the bottom to represent it. If not, it could not be possible to check Natural period.

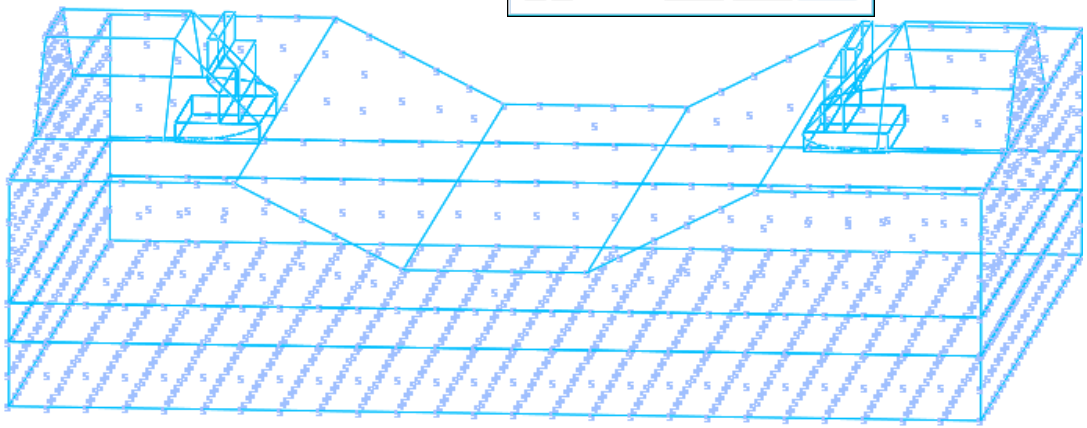
Step 31. Ground surface spring (Damping Constant)

GTS NX

1







Procedure

In order to simulate the radiation damping phenomena at the ground boundary, apply damping boundary.

1. Mesh > Element > Create
   - Other > Ground Surface Spring
   - Select all solid elements (Mesh set 9 total)
   - Selecting Damping Constant / Area

(* Damping Constant will be auto calculated
based on the following equation(next slide))

Step 32: Ground surface spring (Damping Constant)

GTS NX

- Damping constant

Primary wave $$C_p = \rho \cdot A \cdot \sqrt{\frac{\lambda + 2G}{\rho}} = W \cdot A \cdot \sqrt{\frac{\lambda + 2G}{W \cdot 9.81}} = c_p \cdot A$$

Secondary wave $$C_s = \rho \cdot A \cdot \sqrt{\frac{G}{\rho}} = W \cdot A \cdot \sqrt{\frac{G}{W \cdot 9.81}} = c_s \cdot A$$

where, $$\lambda = \frac{\nu \cdot E}{(1+\nu)(1-2\nu)} \qquad G = \frac{E}{2(1+\nu)}$$

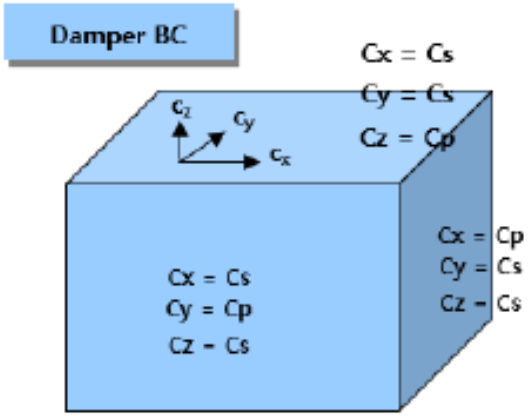
λ : Volumetric modulus (tonf/m^2)

G : Shear modulus (tonf/m^2)

E : Elasticity (tonf/m^2)

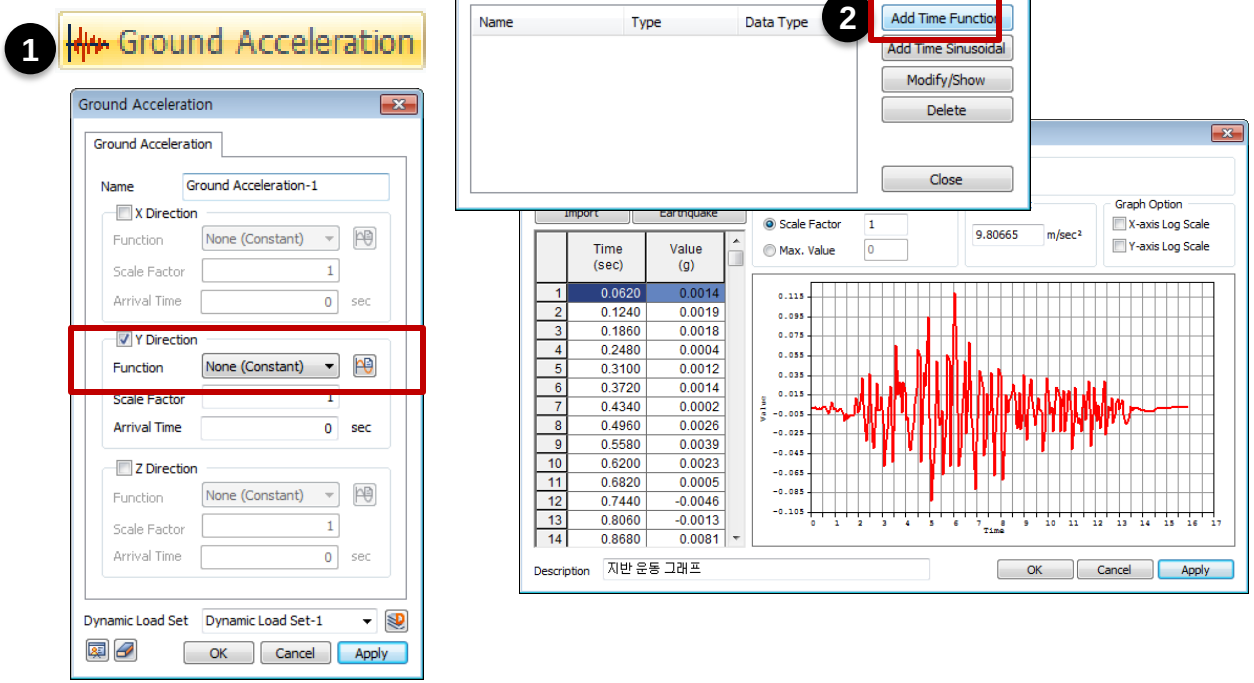
ν : Poisson's ratio

A : Element face area (m^2)

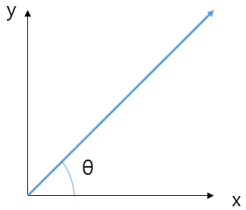


Step 33: Ground acceleration with Direction

GTS NX



- Seismic wave can be classified by direction such as Primary, Secondary and Love wave. Y-direction in this example represent Primary wave which cause a compression and expansion of ground repeatedly.
- In case of defining specified direction in the plane (ex) x-y plane), input scale factor for each direction like below.



➔Ex) Check X and Y direction Input scale factor by following relation X : a · cos θ, y : a · sin θ

Procedure

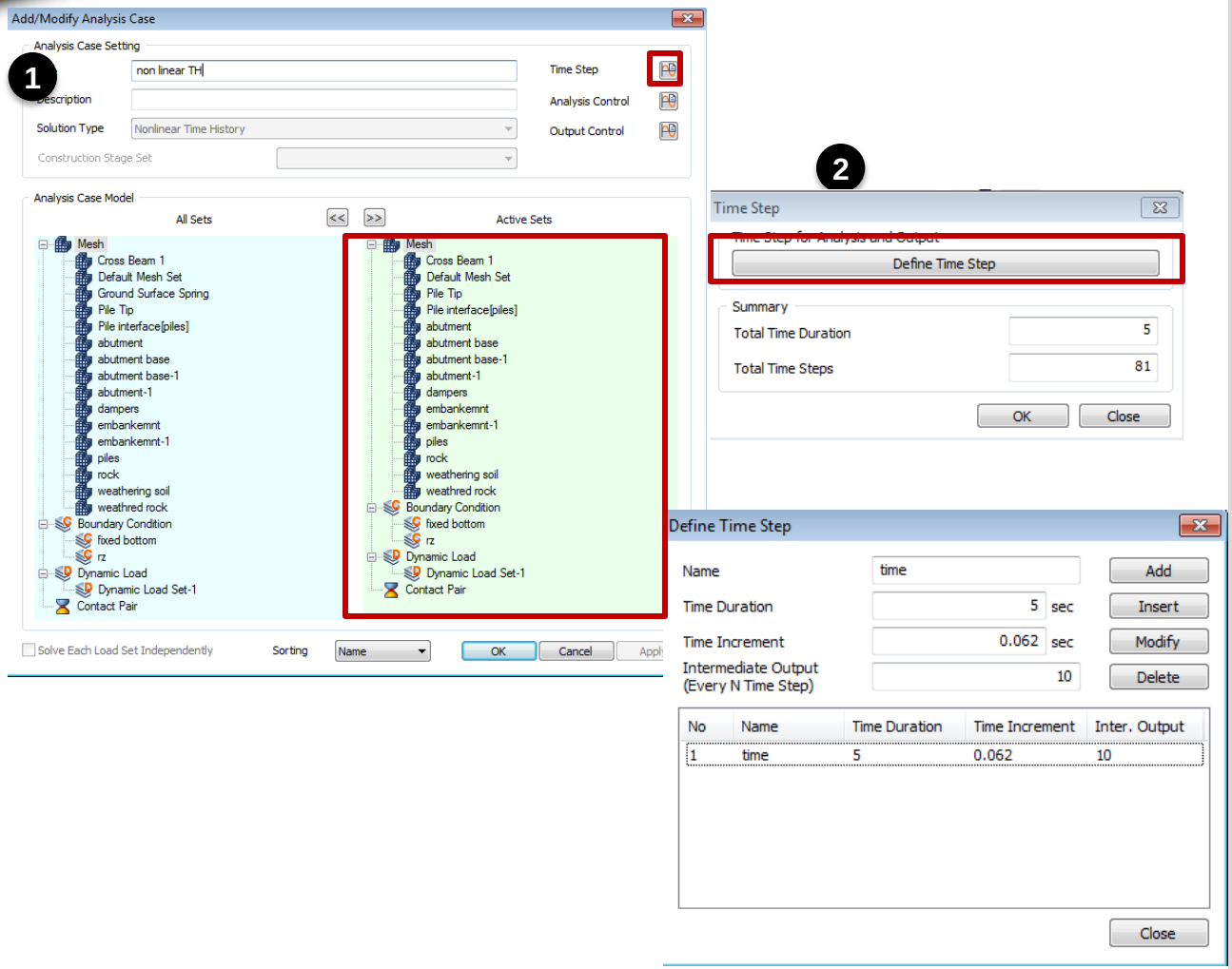
Define ground acceleration with direction

1. Dynamic analysis > Load > Ground Acceleration
   - Check Y - direction
   - Select ( ) button to add time history function
   - Add Time Function
2. Copy Paste the columns from the excel file with tab named “Offunato” in the Name
   - Select Normalized Acceleration

In general, three types of seismic wave such as Offunato, Hachinohe and artificial (or historical earthquake) are used to check the stability to find out the most critical case

Step 34: **Analysis Case (Time Step)**

GTS NX



Procedure

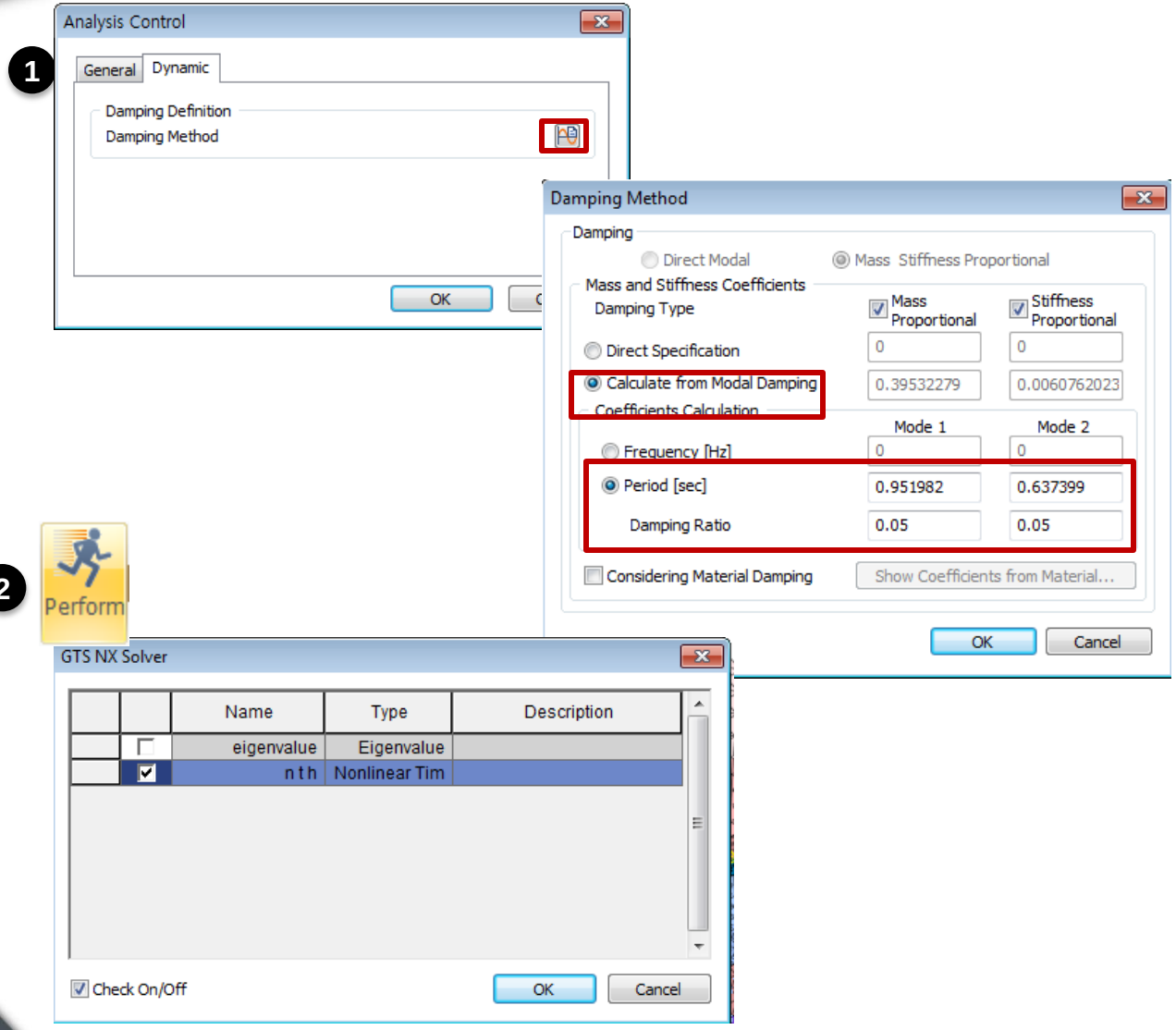
Create Case for Time History

1. Analysis > Analysis Case > General
   - Solution Type : Non Linear Time History
   - Activate all sets excluding 'Ground Surface Spring'

2. Time Step
   - Define Time Step
   - Time Duration : 15.872sec
   - Time Increment : 0.062sec
   - Intermediate Output : 10 (Required to set time step depending on the Time History Load Function(Duration & Increment)

Step 35: Analysis Case (Analysis Control)

GTS NX



Procedure

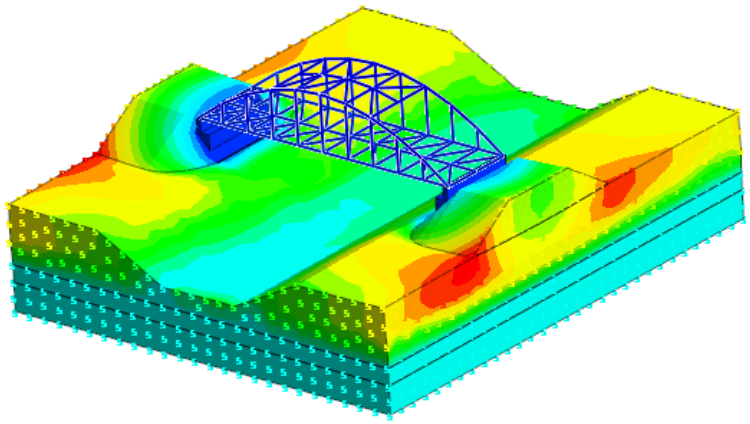
1. Analysis Control
   - Dynamic : Damping Method

   - Select "Calculate from Modal Damping"
   -Select "Period" - Input First and Second mode (0.951982, 0.637399)
   -Damping Ratio : 0.05

2. Analysis > Analysis > Perform

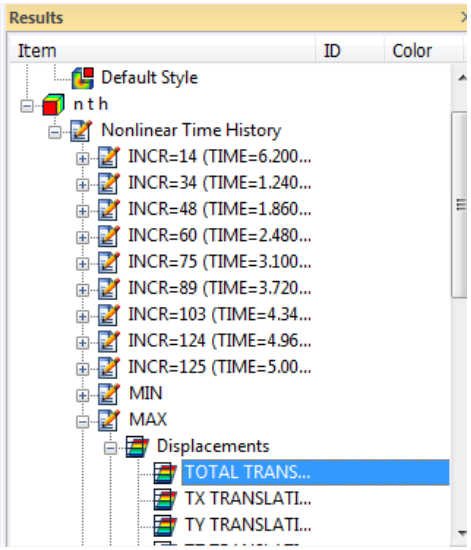
Step 36. **Results**

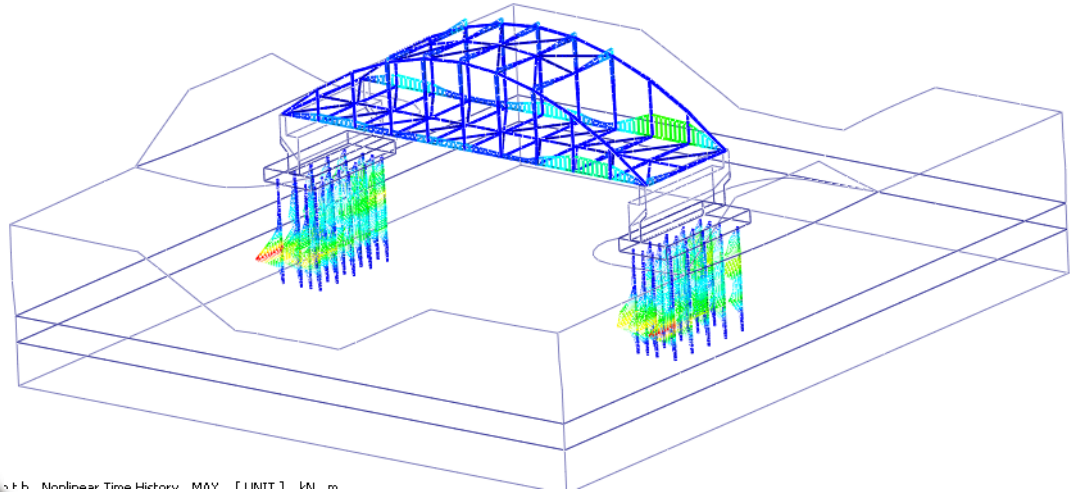
GTS NX



DISPLACEMENT
TOTAL T , cm

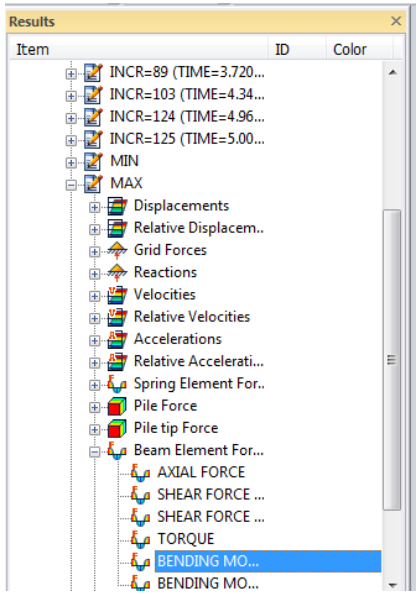
	+14.76850
0.0%	
	+13.86366
0.2%	
	+12.95882
1.0%	
	+12.05397
2.3%	
	+11.14913
3.3%	
	+10.24429
3.4%	
	+9.33945
2.9%	
	+8.43461
2.7%	
	+7.52976
2.5%	
	+6.62492
2.6%	
	+5.72008
31.6%	
	+4.81524
2.8%	
	+3.91039
7.1%	
	+3.00555
37.5%	
	+2.10071
0.1%	
	+1.19587





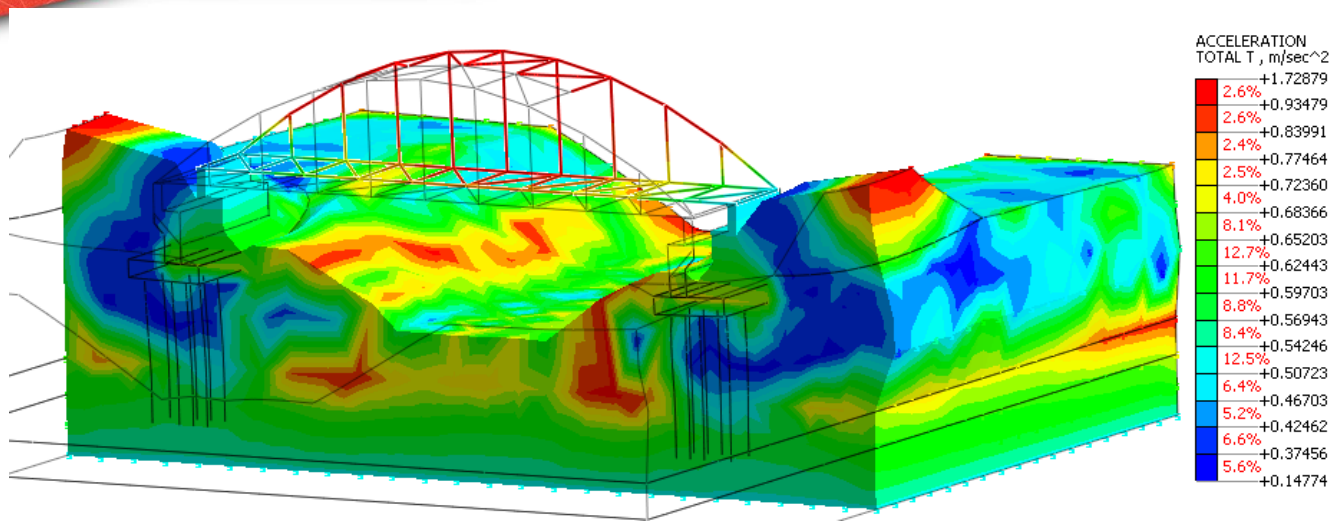
BEAM FORCE
BENDING MMNT Y , kN*m

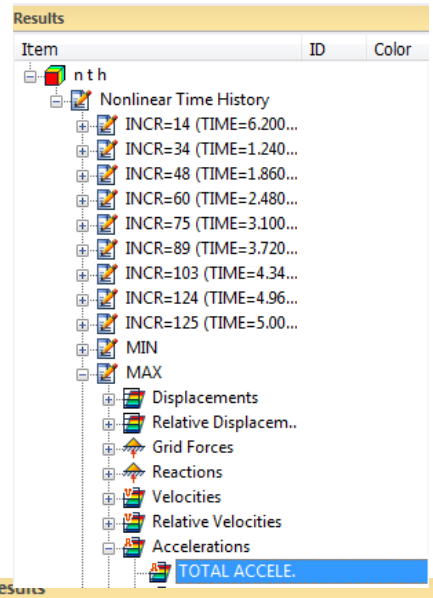
	+388.87799
0.2%	
	+362.73822
0.2%	
	+336.59848
0.2%	
	+310.45871
0.7%	
	+284.31897
1.9%	
	+258.17920
2.8%	
	+232.03946
4.6%	
	+205.89970
6.0%	
	+179.75993
7.6%	
	+153.62018
10.2%	
	+127.48042
11.6%	
	+101.34067
11.0%	
	+75.20090
11.0%	
	+49.06115
11.9%	
	+22.92139
20.0%	
	-3.21837

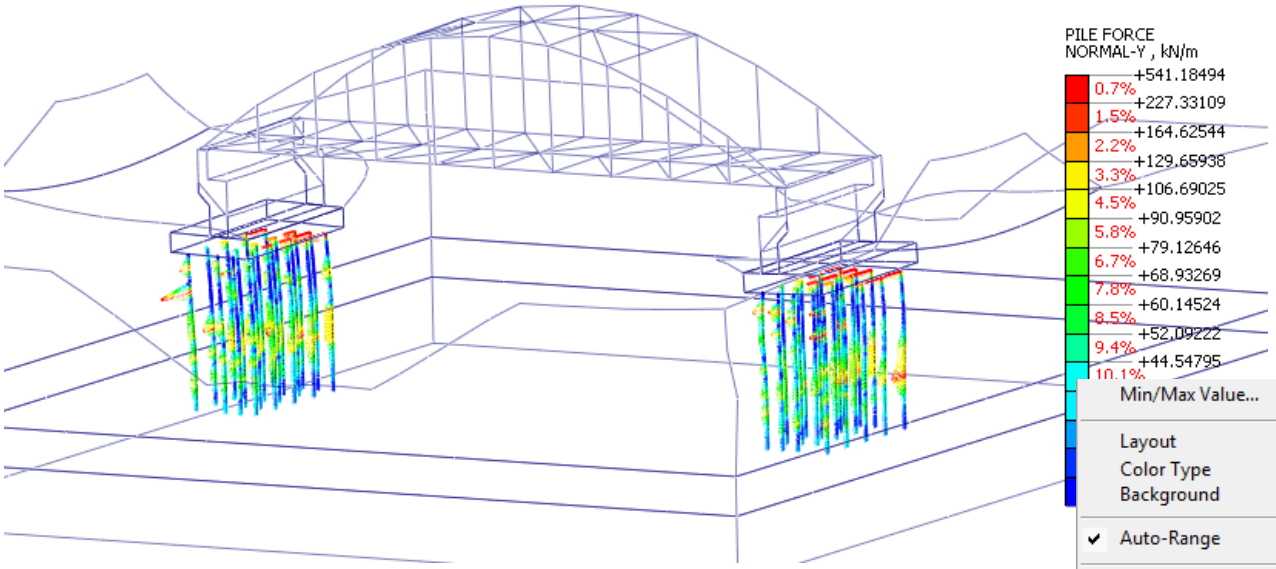


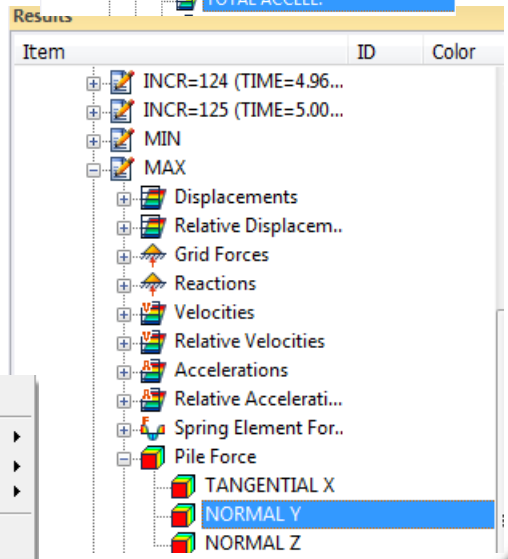
Step 37. **Results**

GTS NX









Thank you

Angel Francisco Martinez
a.martinez@midasuser.com