Release Note

Release Date : November 2022

Product Ver. : FEA NX 2023 (v1.1)

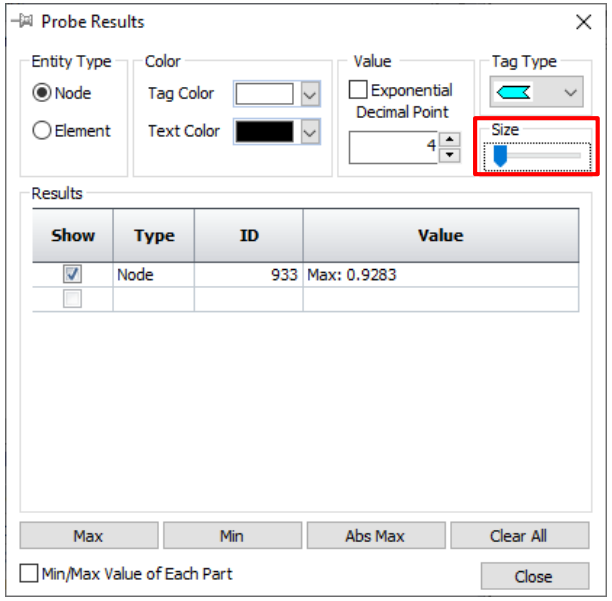
Enhancements

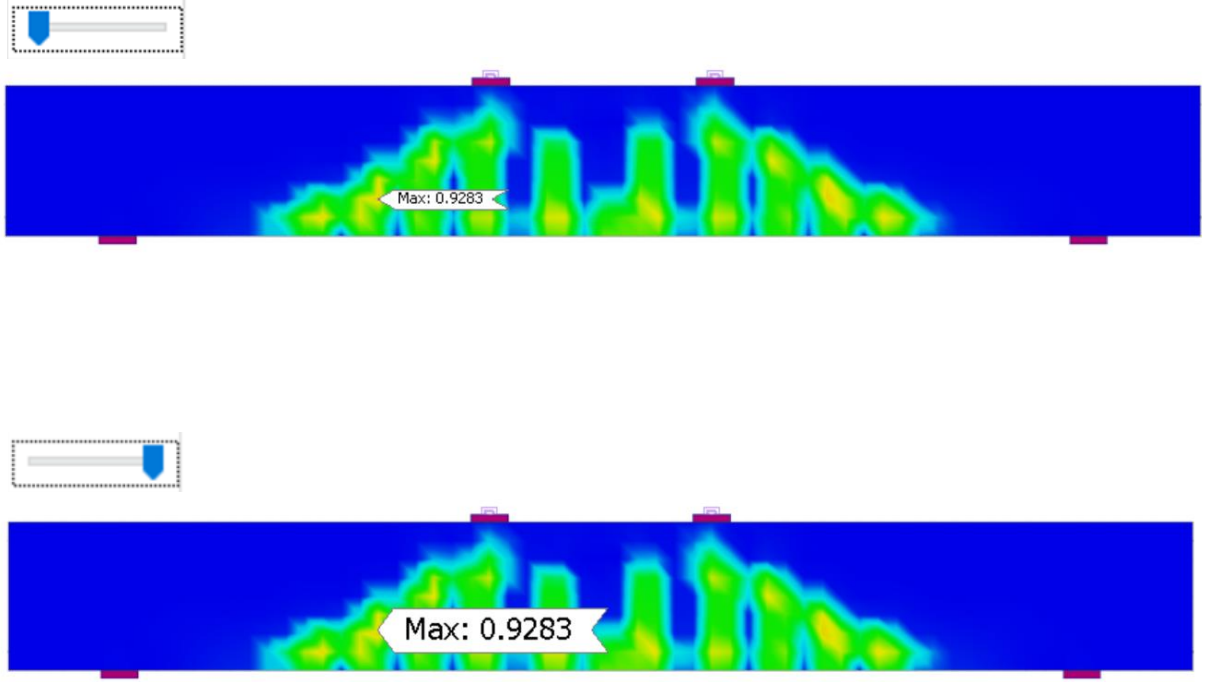
1. Result tag font size adjustment
2. Damping ratio output for each mode of eigenvalue analysis
3. Prestress copy function when copying 1D mesh/elements
4. Automatic merging of nodes when deleting interface elements
5. Exact method to calculate von Mises stress and principal stress for load combinations
6. Exact method to calculate von Mises stress and principal stress for nodal average calculation
7. More result extraction positions for beam elements
8. Separation of analysis option from general options
9. Design Spectrum for seismic analysis
10. On-Curve Diagram function extension
11. Direct opening of GTS and GTS NX model files

1. Result tag font size adjustment

- You can adjust the text size of the result tag. Adjustment is possible in steps 1 to 5.

▪ ***Result > Advanced > Probe***

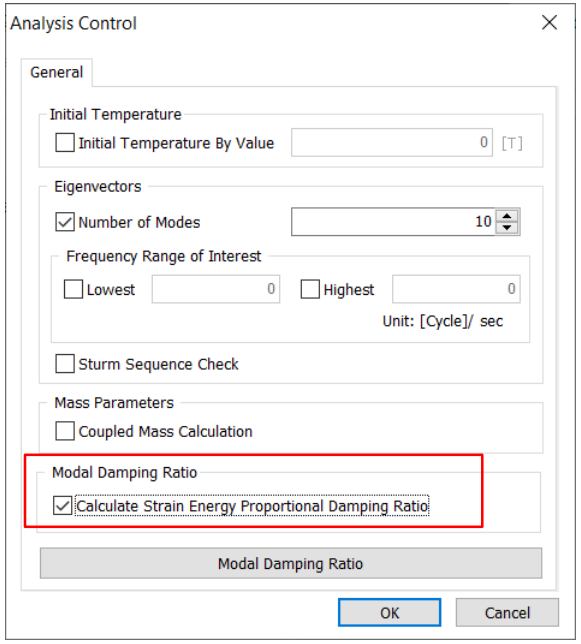




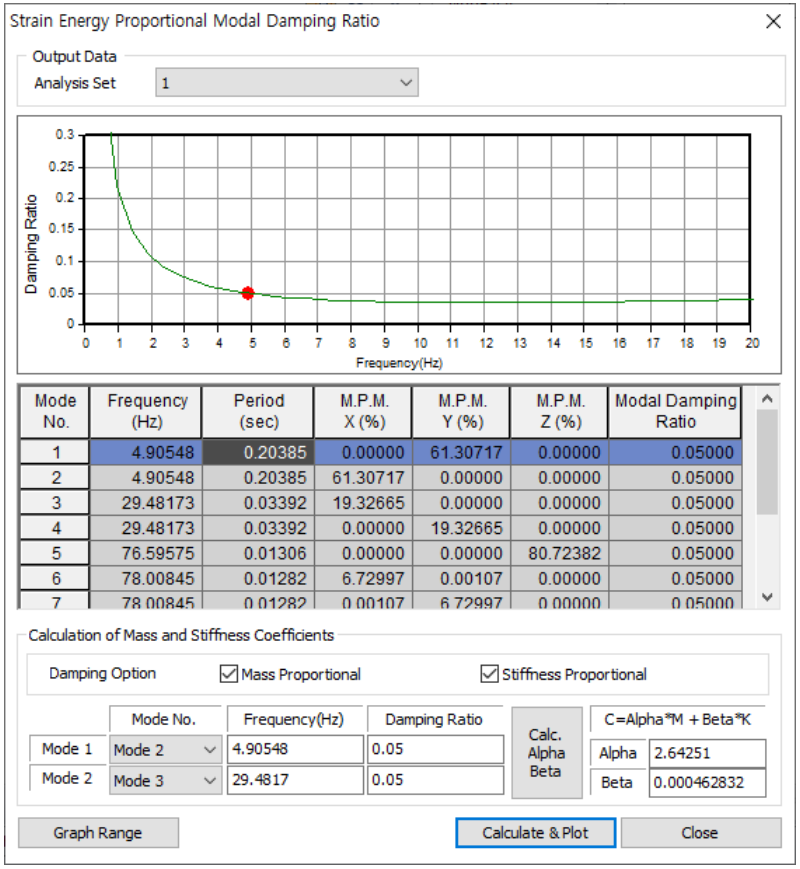
2. Damping ratio output for each mode of eigenvalue analysis

- Eigenvalue analysis provides damping ratios for each mode based on the strain energy of the structure.
- This can be used to obtain modal damping ratios in the structure with different materials or damping devices.

- ***Analysis > Analysis Case > General > Solution Type: Eigenvalue > Analysis Control***
- ***Result > Advanced > Others > Modal Damping Ratio***



Eigenvalue Analysis Control

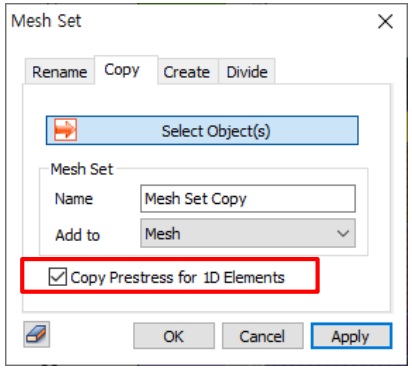


Modal Damping Ratio

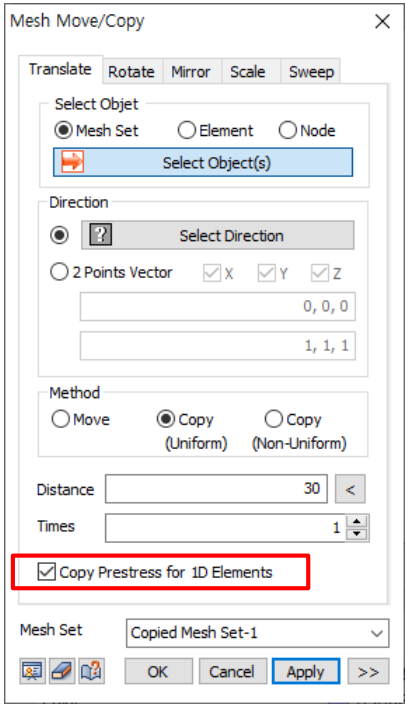
3. Prestress copy function when copying 1D mesh/elements

- When you copy 1D elements to which prestress is entered, the prestress load can also be copied together.

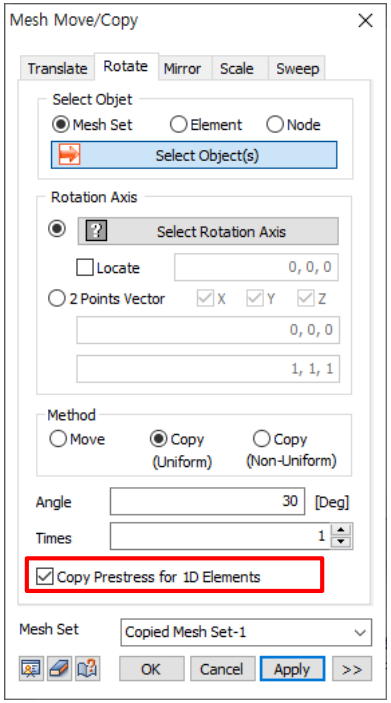
- ***Mesh > Mesh Set > Copy***
- ***Mesh > Transform > Translate / Rotate / Mirror***



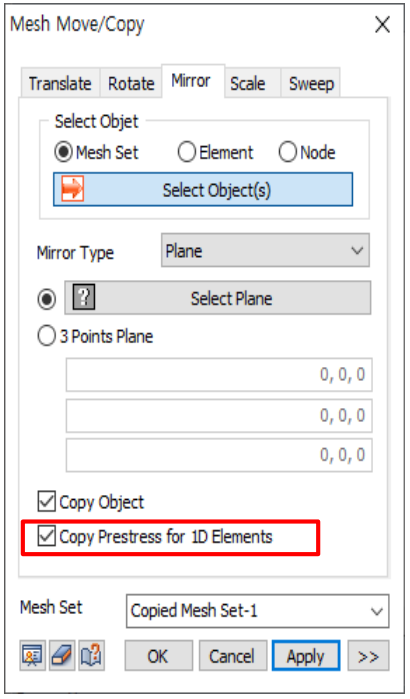
Copy



Translate



Rotate

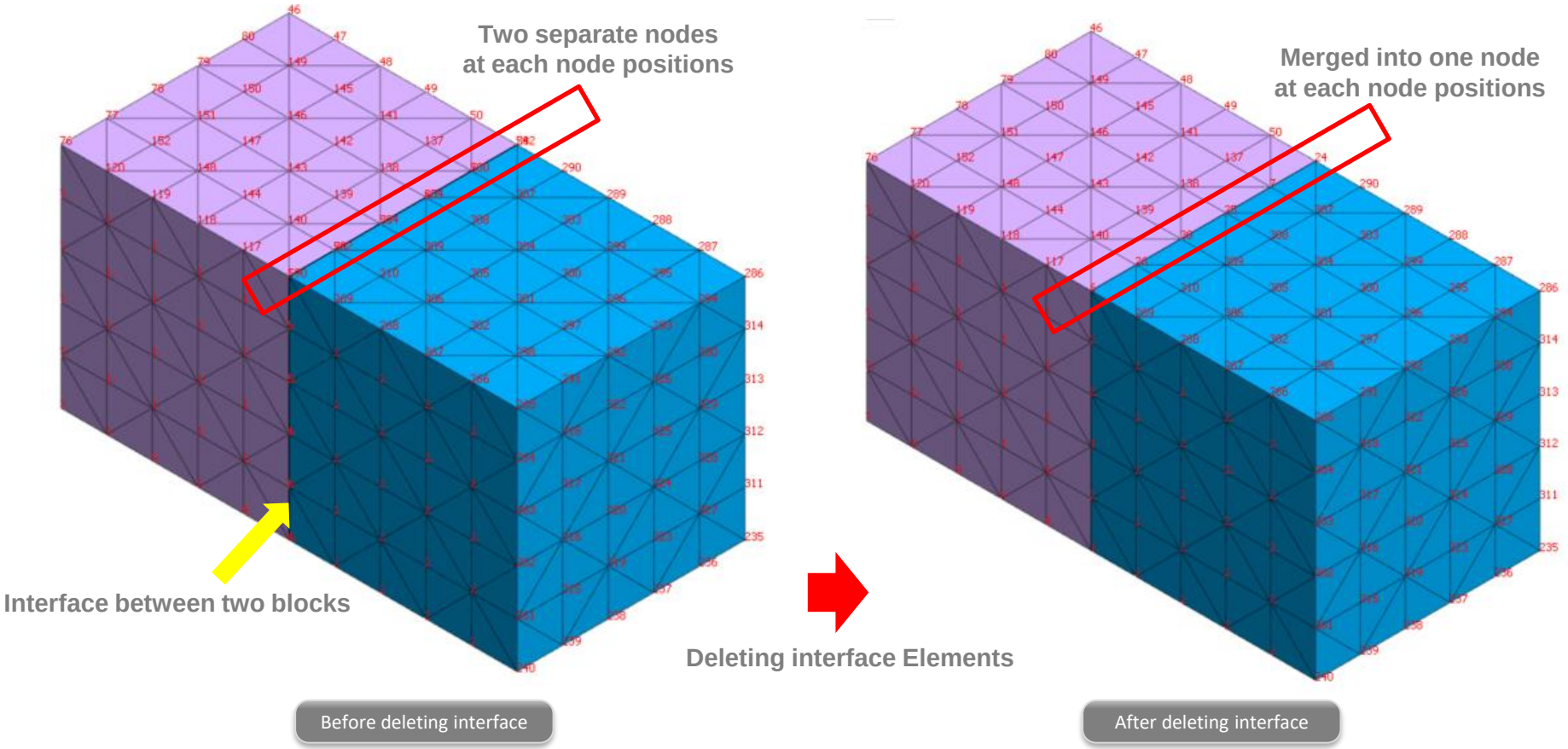


Mirror

4. Automatic merging of nodes when deleting interface elements

- When an interface element is created, it automatically creates two separate nodes at each nodes on the interface. In the previous version, even if the interface was deleted, the separated nodes were maintained, so the user had to merge the separated nodes manually. For the convenience of the user, the function has been changed to automatically merge the separated nodes when the interface element is deleted.

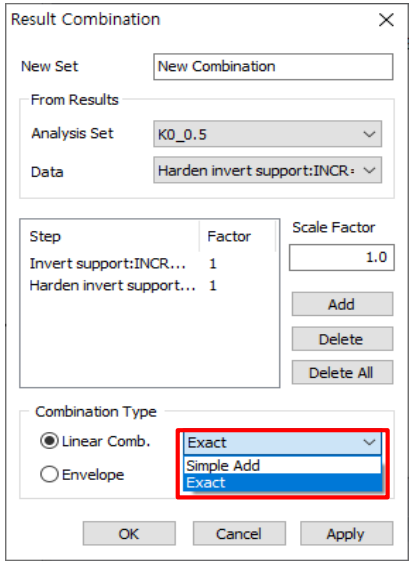
▪ ***Works Tree > Model > Mesh > Delete interface mesh set***



5. Exact method to calculate von Mises stress and principal stress for load combinations

- Von Mises stresses and principal stresses have no direction, while normal stresses and shear stresses have their own directions.
- When we combine two separate load cases to calculate von Mises stresses or principal stresses, the 'Exact' method is implemented.

▪ ***Result > Result > Combination***



Combination Dialog Box

Simple Add

Calculate normal stresses σxx, σyy, σzz and shear stresses τxy, τyz, τxz for load case 1 and 2.

↓

Calculate von Mises stresses σv1 for load case 1 and σv2 for load case 2.

↓

Add von Mises stresses σv1+ σv2 for load combination.

Exact

Calculate normal stresses σxx, σyy, σzz and shear stresses τxy, τyz, τxz for load case 1 and 2.

↓

Add normal stresses and shear stresses for load combination.
σxx1+σxx2, σyy1+σyy2, σzz1+σzz2, τxy1+τxy2, τyz1+τyz2, τxz1+τxz2

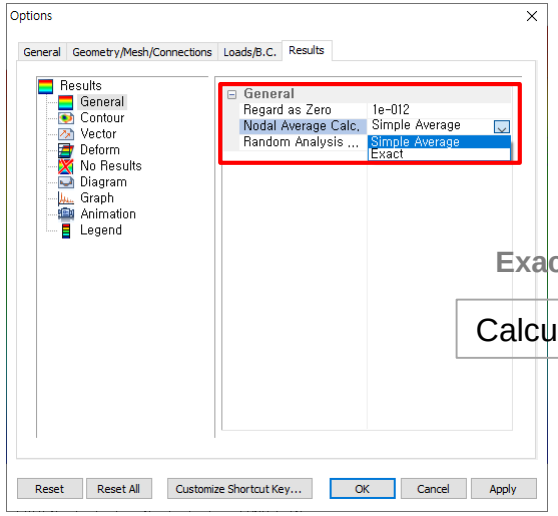
↓

Calculate von Mises stress σv using the sum of normal stresses and shear stresses.

6. Exact method to calculate von Mises stress and principal stress for nodal average calculation

- Von Mises stresses and principal stresses have no direction, while normal stresses and shear stresses have their own directions.
- When we check von Mises stresses or principal stresses with the 'Nodal Average' option, the 'Exact' method is implemented.

▪ ***Analysis > Tools > Option***



Options Dialog Box

Simple Average

Calculate normal stresses σxx, σyy, σzz and shear stresses τxy, τyz, τxz for element 1 and 2.

↓

Calculate von Mises stresses σv1 for element 1 and σv2 for element 2.

↓

Take simple average of σv1 and σv2 for nodal average.

Exact

Calculate normal stresses σxx, σyy, σzz and shear stresses τxy, τyz, τxz for element 1 and 2.

↓

Take average of normal stresses and shear stresses for nodal average.
σxx_avg, σyy_avg, σzz_avg, τxy_avg, τyz_avg, τxz_avg

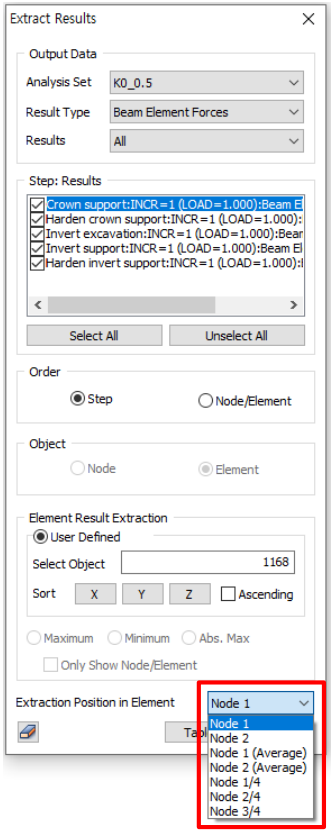
↓

Calculate von Mises stress σv using the average of normal stresses and shear stresses.

7. More result extraction positions for beam elements

- Previously, in the case of beam elements, only the results for I and J stages could be extracted, but the result extraction location has been added so that the results can be extracted according to the [number of output segments of beam elements] set in Analysis Case > Result Control.

▪ ***Result > Advanced > Extract***

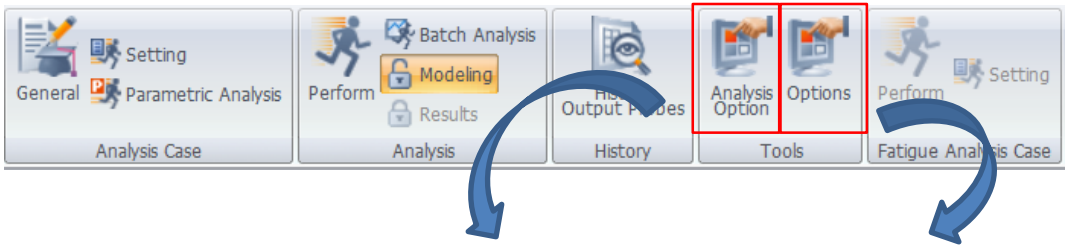


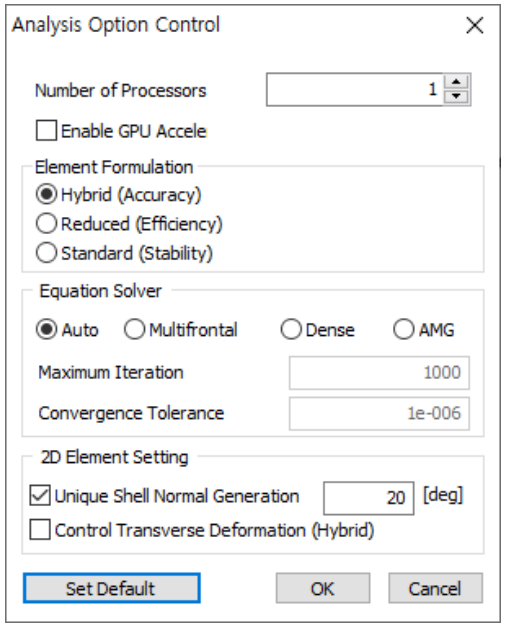
Extract Results

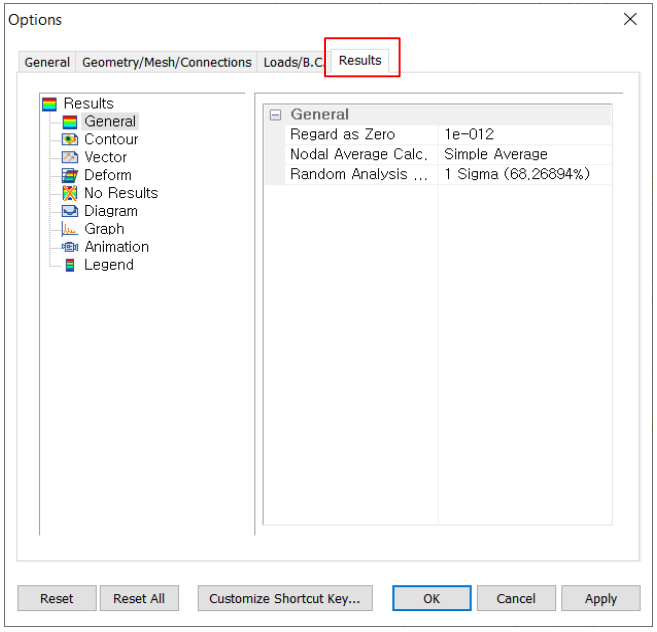
8. Separation of analysis option from general options

- General options and analysis options are separated. Previously, the analysis options were not saved in the model file despite the analysis results depend on the options.
- Now, the analysis options are saved in the individual model file.

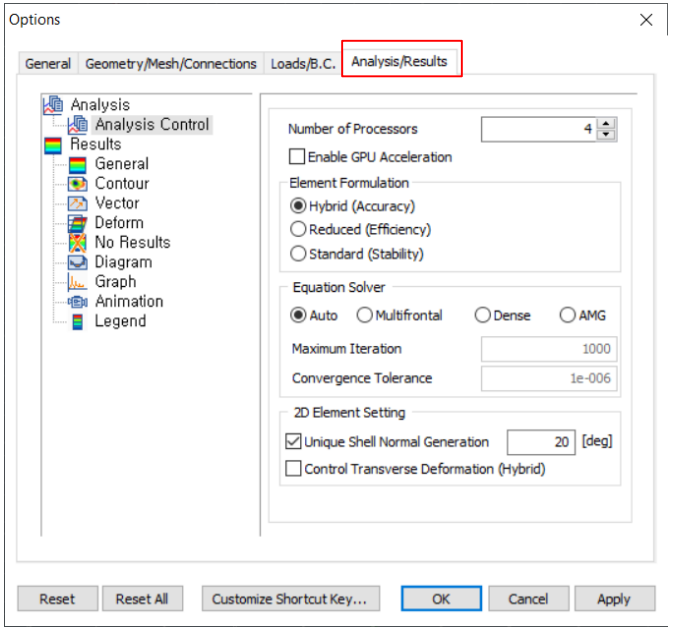
▪ ***Analysis > Tools > Analysis Option***







FEA NX 2023 (v1.1)

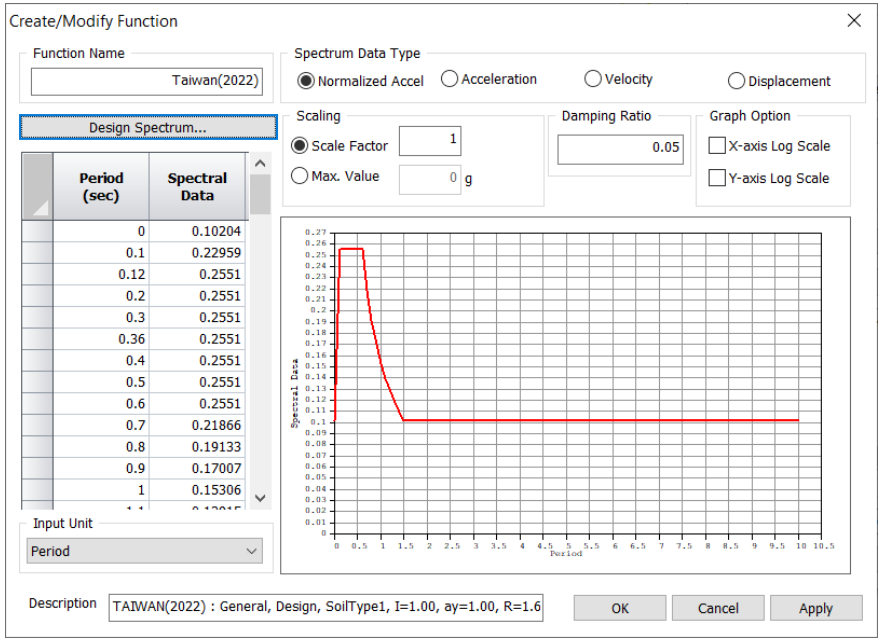


Previous version

9. Design Spectrum for seismic analysis

- More latest international design spectrum functions are added.

▪ ***Dynamic Analysis > Load > Response Spectrum***



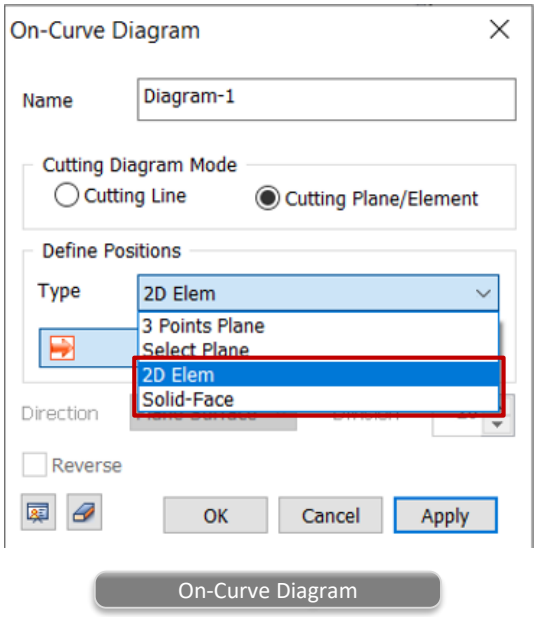
Design Spectrum

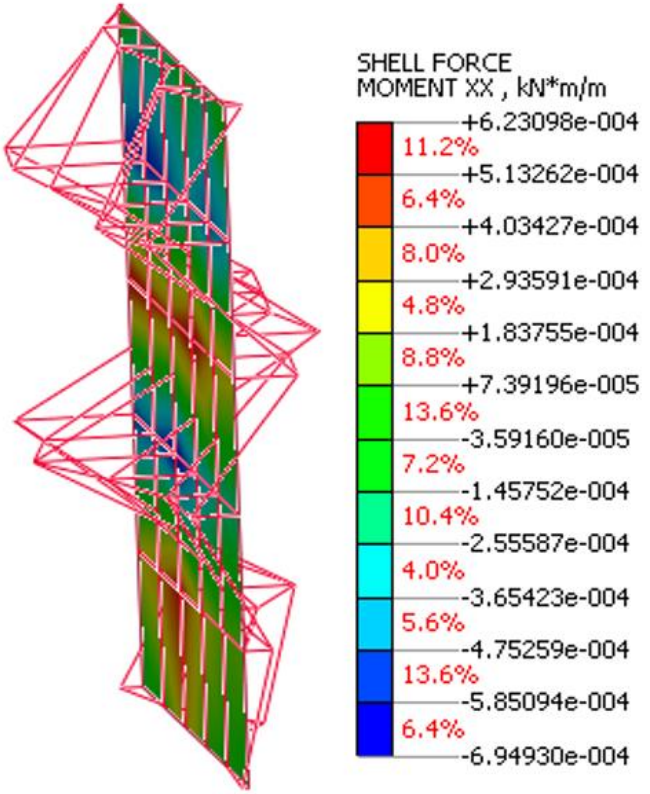
KBC 2016
China GB/T 51408-2021
China JTG/T 2231-01-2020
China GB 50011-2019
China CJJ 166-2011
Japan Bridge 2017
Taiwan 2022
IS 1893 2016
NSR-10
P 100-1 2013
NTC 2018
DPWH-LRFD BSDS 2013
AS 5100.2 2017
IRC:SP:114-2018

10. On-Curve Diagram function extension

- The function type of On-Curve Diagram has been extended so that diagrams are drawn like the contour values of 2D elements. The tendency of member force of shell element can be easily grasped in diagram form.

▪ ***Result > Advanced > Cutting Diag. > Cutting Plane/Element***

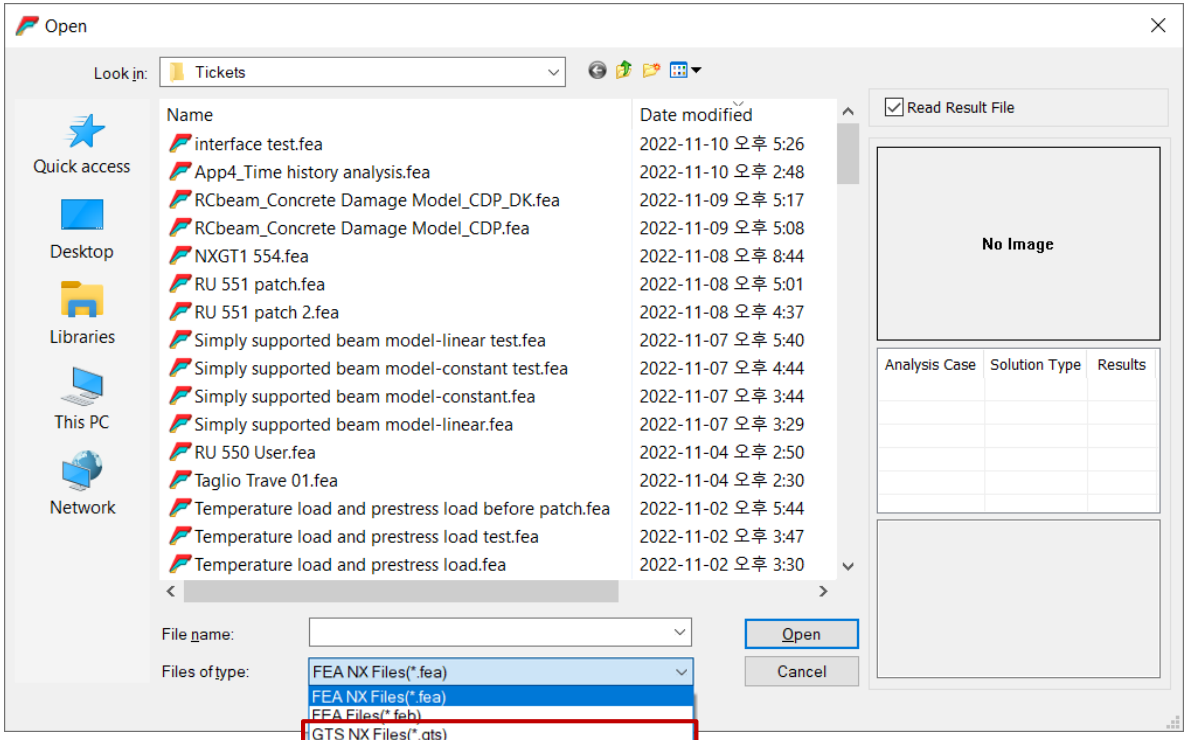




11. Direct opening of GTS and GTS NX model files

- GTS (*.gtb) and GTS NX (*.gts) model files can be opened directly without changing the extension of the file to fea.

▪ ***File > Open***



File Open Dialog Box