

# Release Note

---

Release Date : November 2022

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



ADVANCED NONLINEAR AND DETAIL ANALYSIS

New Paradigm in Advanced Structural Analysis

# 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**

Probe Results
✕

Entity Type

Node

Element

Color

Tag Color

Text Color

Value

Exponential

Decimal Point 4

Tag Type

←

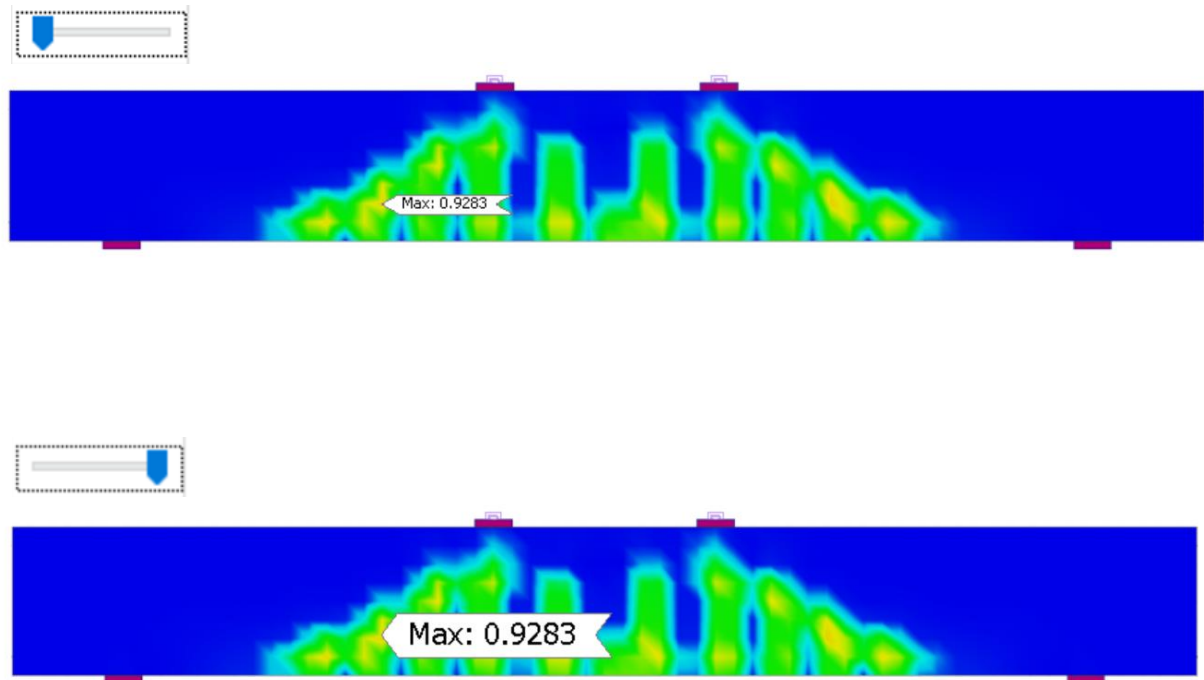
**Size** ←

Results

Show	Type	ID	Value
<input checked="" type="checkbox"/>	Node	933	Max: 0.9283
<input type="checkbox"/>			

Max
Min
Abs Max
Clear All

Min/Max Value of Each Part Close



## 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**

**Analysis Control** ✕

**General**

Initial Temperature

Initial Temperature By Value  [T]

Eigenvectors

Number of Modes

Frequency Range of Interest

Lowest   Highest   
Unit: [Cycle]/ sec

Sturm Sequence Check

Mass Parameters

Coupled Mass Calculation

**Modal Damping Ratio**

Calculate Strain Energy Proportional Damping Ratio

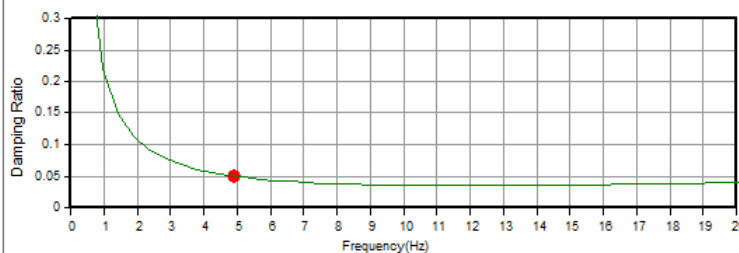
Modal Damping Ratio

Eigenvalue Analysis Control

**Strain Energy Proportional Modal Damping Ratio** ✕

Output Data

Analysis Set



Mode No.	Frequency (Hz)	Period (sec)	M.P.M. X (%)	M.P.M. Y (%)	M.P.M. Z (%)	Modal Damping Ratio
1	4.90548	0.20385	0.00000	61.30717	0.00000	0.05000
2	4.90548	0.20385	61.30717	0.00000	0.00000	0.05000
3	29.48173	0.03392	19.32665	0.00000	0.00000	0.05000
4	29.48173	0.03392	0.00000	19.32665	0.00000	0.05000
5	76.59575	0.01306	0.00000	0.00000	80.72382	0.05000
6	78.00845	0.01282	6.72997	0.00107	0.00000	0.05000
7	78.00845	0.01282	0.00107	6.72997	0.00000	0.05000

Calculation of Mass and Stiffness Coefficients

Damping Option  Mass Proportional  Stiffness Proportional

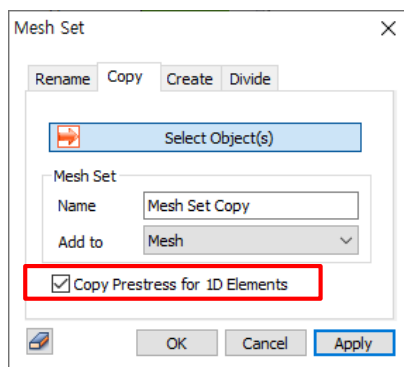
		Mode No.	Frequency(Hz)	Damping Ratio	Calc. Alpha Beta	C=Alpha*M + Beta*K
Mode 1	Mode 2	4.90548	0.05			Alpha
Mode 2	Mode 3	29.4817	0.05		Beta	0.000462832

Modal Damping Ratio

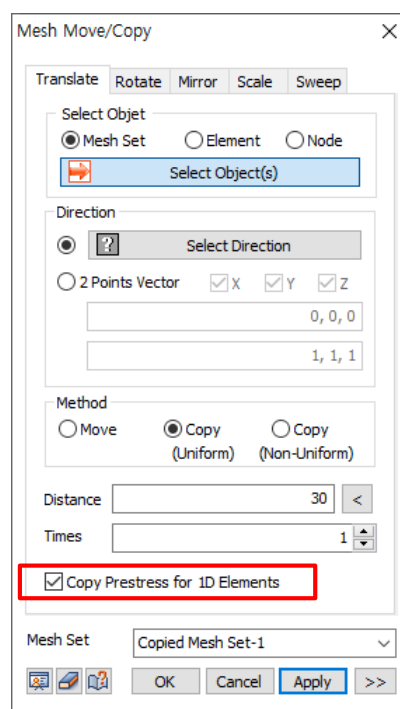
### 3. Prestress copy function when copying 1D mesh/elements

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

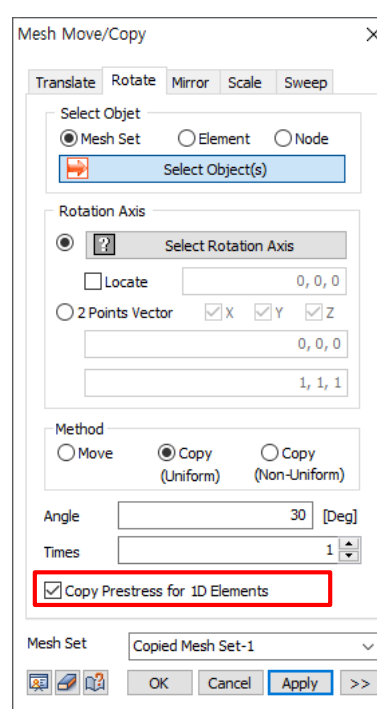
- Mesh > Mesh Set > Copy**
- Mesh > Transform > Translate / Rotate / Mirror**



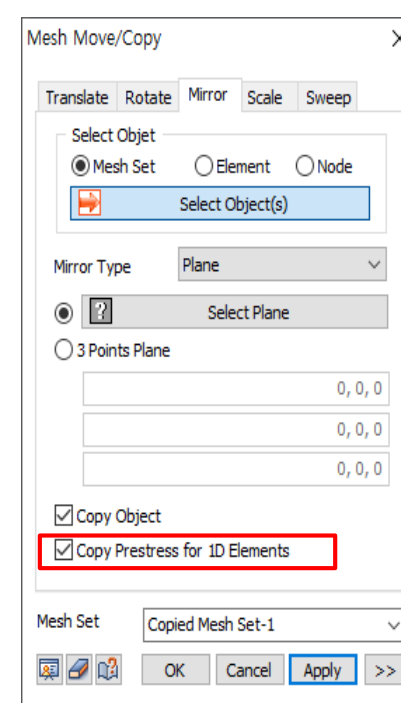
Copy



Translate



Rotate

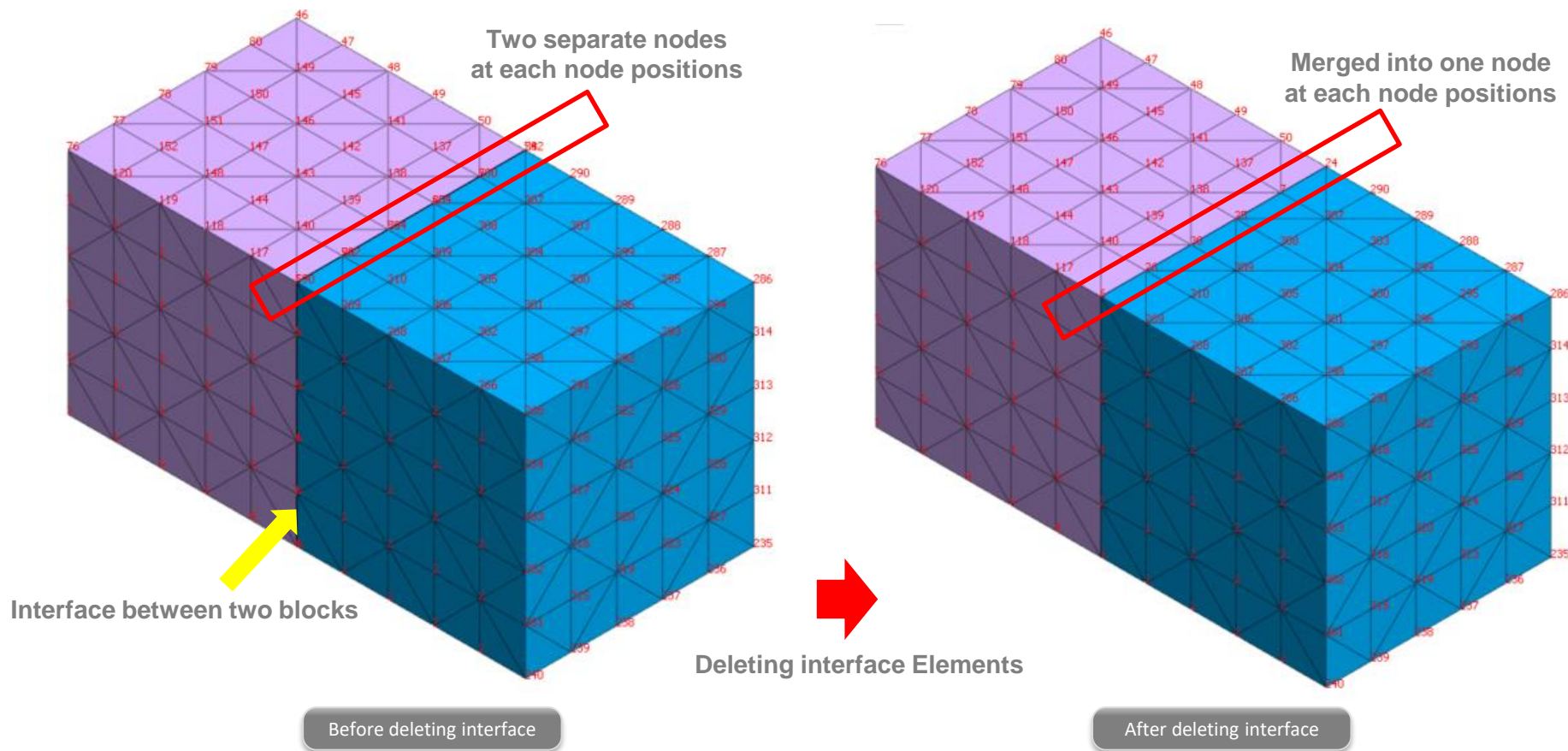


Mirror

#### 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**

Result Combination

New Set: New Combination

From Results

Analysis Set: K0\_0.5

Data: Harden invert support:INCR+

Step	Factor	Scale Factor
Invert support:INCR...	1	1.0
Harden invert support...	1	

Buttons: Add, Delete, Delete All

Combination Type

Linear Comb.  Envelope

Exact (highlighted), Simple Add, Exact

Buttons: OK, Cancel, Apply

Combination Dialog Box

### Simple Add

Calculate normal stresses  $\sigma_{xx}$ ,  $\sigma_{yy}$ ,  $\sigma_{zz}$  and shear stresses  $\tau_{xy}$ ,  $\tau_{yz}$ ,  $\tau_{xz}$  for load case 1 and 2.

Calculate von Mises stresses  $\sigma_{v1}$  for load case 1 and  $\sigma_{v2}$  for load case 2.

Add von Mises stresses  $\sigma_{v1} + \sigma_{v2}$  for load combination.

### Exact

Calculate normal stresses  $\sigma_{xx}$ ,  $\sigma_{yy}$ ,  $\sigma_{zz}$  and shear stresses  $\tau_{xy}$ ,  $\tau_{yz}$ ,  $\tau_{xz}$  for load case 1 and 2.

Add normal stresses and shear stresses for load combination.  
 $\sigma_{xx1} + \sigma_{xx2}$ ,  $\sigma_{yy1} + \sigma_{yy2}$ ,  $\sigma_{zz1} + \sigma_{zz2}$ ,  $\tau_{xy1} + \tau_{xy2}$ ,  $\tau_{yz1} + \tau_{yz2}$ ,  $\tau_{xz1} + \tau_{xz2}$

Calculate von Mises stress  $\sigma_v$  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

#### Simple Average

Calculate normal stresses  $\sigma_{xx}$ ,  $\sigma_{yy}$ ,  $\sigma_{zz}$  and shear stresses  $\tau_{xy}$ ,  $\tau_{yz}$ ,  $\tau_{xz}$  for element 1 and 2.

Calculate von Mises stresses  $\sigma_{v1}$  for element 1 and  $\sigma_{v2}$  for element 2.

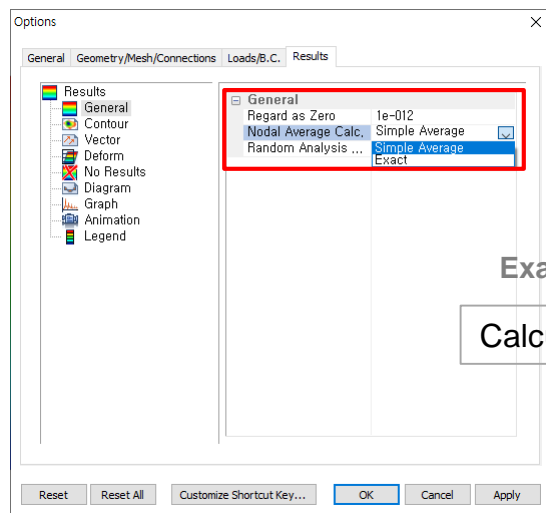
Take simple average of  $\sigma_{v1}$  and  $\sigma_{v2}$  for nodal average.

#### Exact

Calculate normal stresses  $\sigma_{xx}$ ,  $\sigma_{yy}$ ,  $\sigma_{zz}$  and shear stresses  $\tau_{xy}$ ,  $\tau_{yz}$ ,  $\tau_{xz}$  for element 1 and 2.

Take average of normal stresses and shear stresses for nodal average.  
 $\sigma_{xx\_avg}$ ,  $\sigma_{yy\_avg}$ ,  $\sigma_{zz\_avg}$ ,  $\tau_{xy\_avg}$ ,  $\tau_{yz\_avg}$ ,  $\tau_{xz\_avg}$

Calculate von Mises stress  $\sigma_v$  using the average of normal stresses and shear stresses.



Options Dialog Box



## 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

Output Data

Analysis Set KO\_0.5

Result Type Beam Element Forces

Results All

Step: Results

- Crown support: INCR=1 (LOAD=1.000): Beam E
- Harden crown support: INCR=1 (LOAD=1.000):
- Invert excavation: INCR=1 (LOAD=1.000): Bear
- Invert support: INCR=1 (LOAD=1.000): Beam E
- Harden invert support: INCR=1 (LOAD=1.000):

< >

Select All Unselect All

Order

Step  Node/Element

Object

Node  Element

Element Result Extraction

User Defined

Select Object 1168

Sort X Y Z  Ascending

Maximum  Minimum  Abs. Max

Only Show Node/Element

Extraction Position in Element

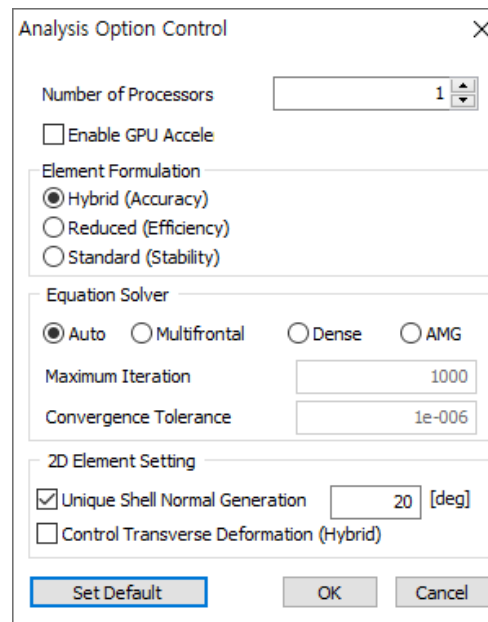
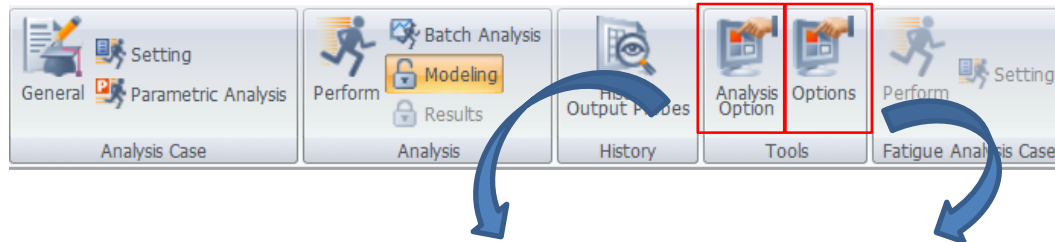
- Node 1
- Node 1
- Node 2
- Node 1 (Average)
- Node 2 (Average)
- Node 1/4
- Node 2/4
- Node 3/4

Extract Results

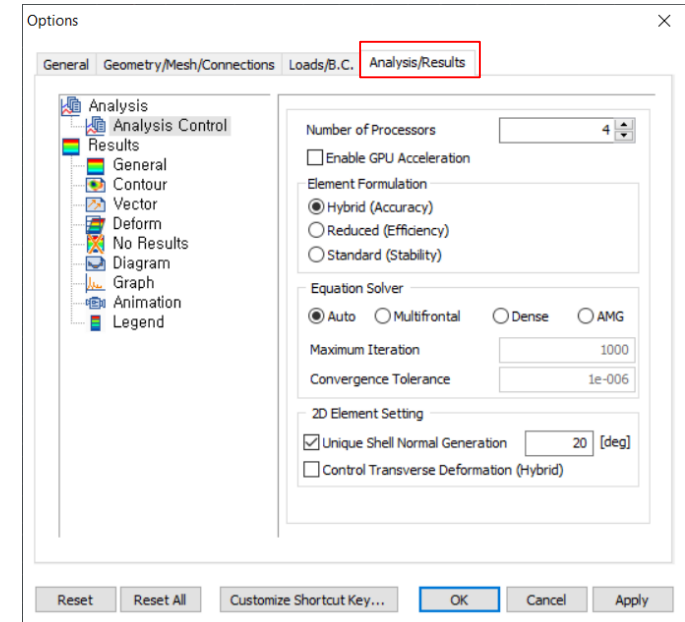
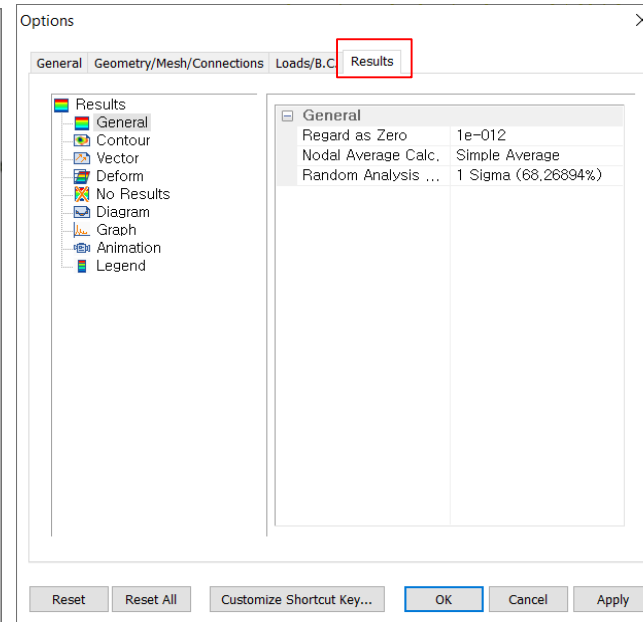
## 8. Separation of analysis option from general options

- 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**

Create/Modify Function ×

Function Name:

Spectrum Data Type:  Normalized Accel  Acceleration  Velocity  Displacement

Design Spectrum...

Period (sec)	Spectral Data
0	0.10204
0.1	0.22959
0.12	0.2551
0.2	0.2551
0.3	0.2551
0.36	0.2551
0.4	0.2551
0.5	0.2551
0.6	0.2551
0.7	0.21866
0.8	0.19133
0.9	0.17007
1	0.15306

Scaling:  Scale Factor   Max. Value  g

Damping Ratio:

Graph Option:  X-axis Log Scale  Y-axis Log Scale

Input Unit:

Description: TAIWAN(2022) : General, Design, SoilType1, I=1.00, ay=1.00, R=1.6

OK Cancel Apply

Design Spectrum

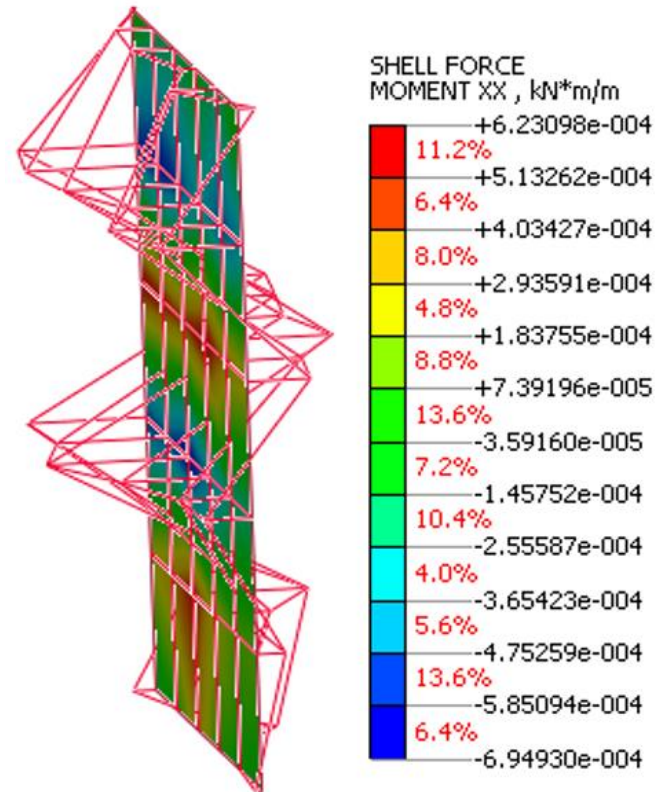
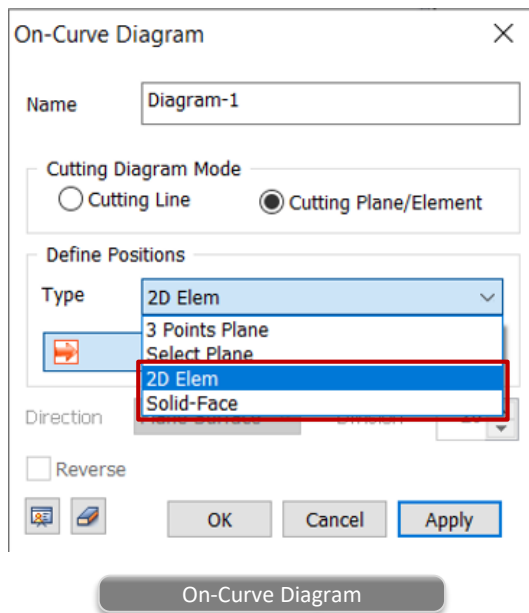
- KDS(41-17-00:2019)
- KDS(41-17-00:2019)
- KDS(17-10-00:2018)
- KBC(2009)
- KBC(2005)
- KBC(2016)
- Korea(Arch. 2000)
- Korea(Arch. 1992)
- Korea(Bridge)
- IBC2000(ASCE7-98)
- UBC(1997)
- UBC 88-94
- NBC(1995)
- Eurocode-8(2004)
- Eurocode-8(1996) Design
- Eurocode-8(1996) Elastic
- China(GB/T 51408-2021)
- China(JTG/T 2231-01-2020)
- China(GB50011-2019)
- China(CJJ 166-2011)
- China(GB50011-2010)
- China(GB50111-2006)
- China(GB50011-2001)
- China Shanghai(DGJ08-9-2003)
- China(JTJ004-89)
- China(JTG/T B02-01-2008)
- China(GBJ11-87)
- Japan(Bridge2017)
- Japan(Arch. 2000)
- Japan(Bridge2002)
- Taiwan(2022)
- Taiwan(2006)
- TaiwanBrg(89) Horizontal
- TaiwanBrg(89) Vertical
- IS1893(2002)
- IS1893(2016)
- NSR-10
- P100-1(2013)
- NTC2018
- DPWH-LRFD BSDS(2013)
- AS 5100.2(2017)
- IRC:SP:114-2018

- 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.

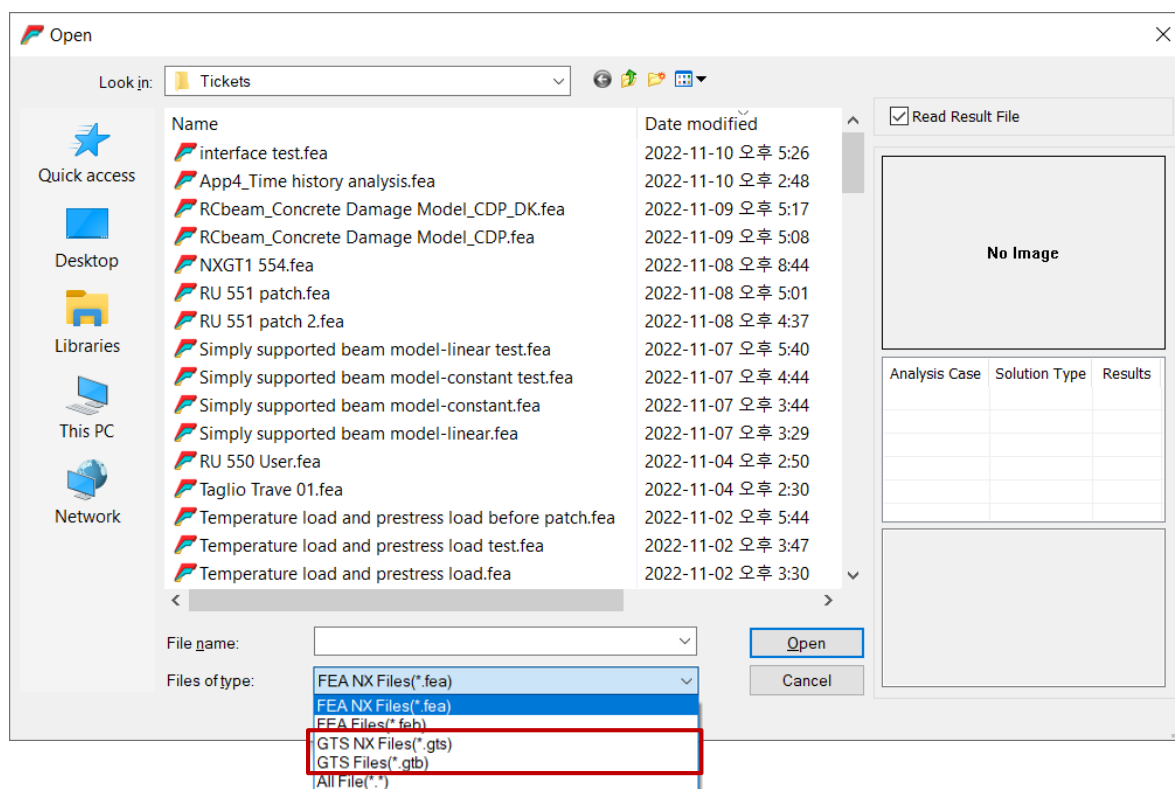
- **Result > Advanced > Cutting Diag. > Cutting Plane/Element**



## 11. Direct opening of GTS and GTS NX model files

- GTS (\*.gts) 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