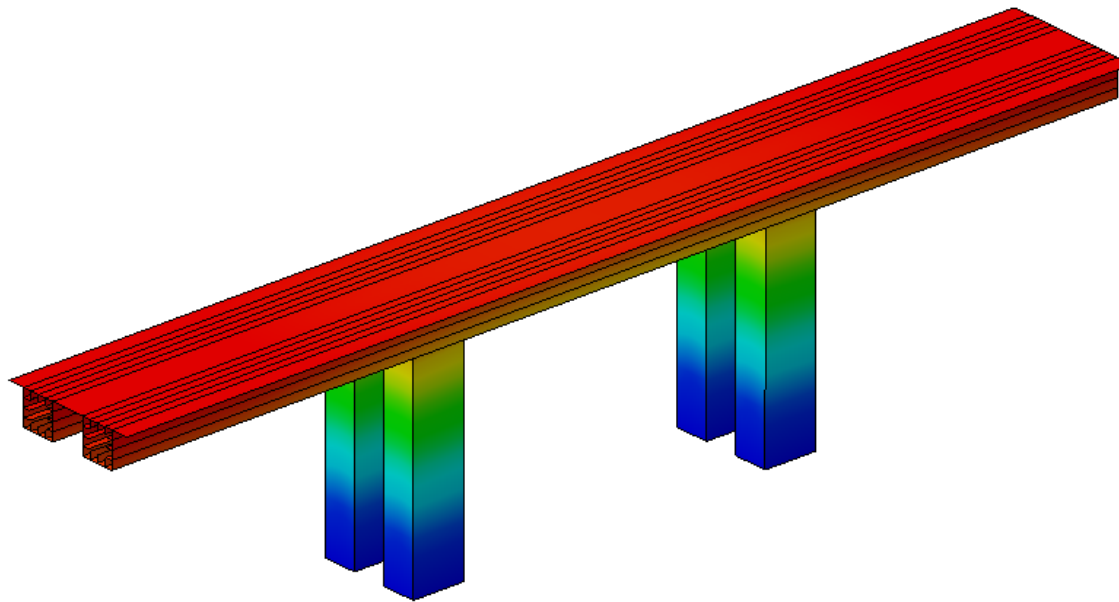


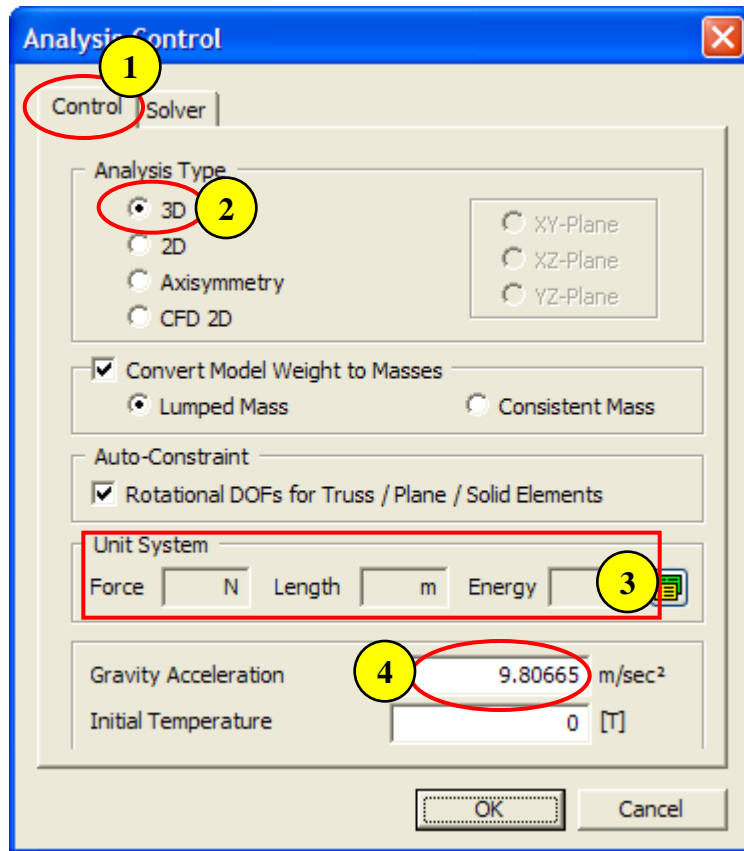
RS-1. Analysis of a Steel Box Bridge



Overview

- 3-D Response Spectrum Analysis
- Model
 - Unit : N, m
 - Isotropic Elastic Material
 - Plate, Solid Element
- Load & Boundary Conditions
 - Response Spectrum Analysis
 - : Response Spectrum Functions
 - : Response Spectrum Load Set
 - Constraint
- Result Evolution
 - Deformation
 - 2D Element Principal Stress
 - 3D Element Principal Stress

Step 1.



1. Analysis > Analysis Control – “Control” tab

2. Analysis Type : 3D

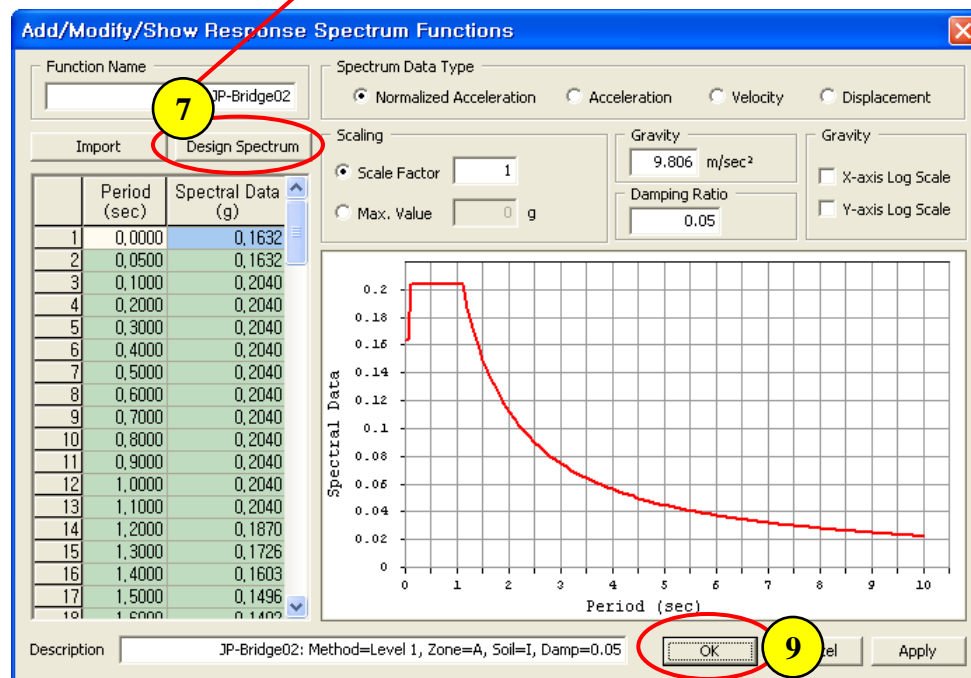
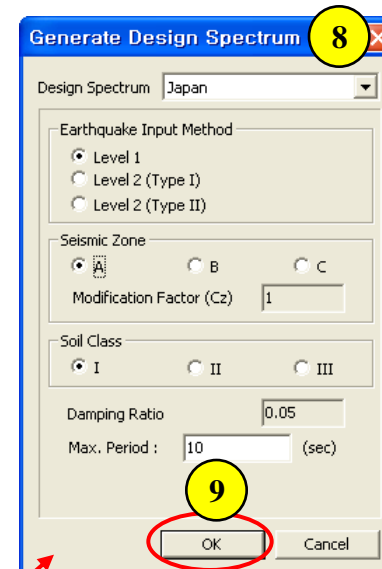
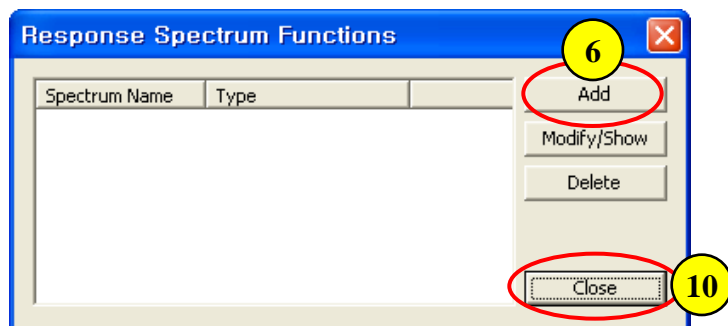
3. Unit : N , m

4. Click [OK] Button

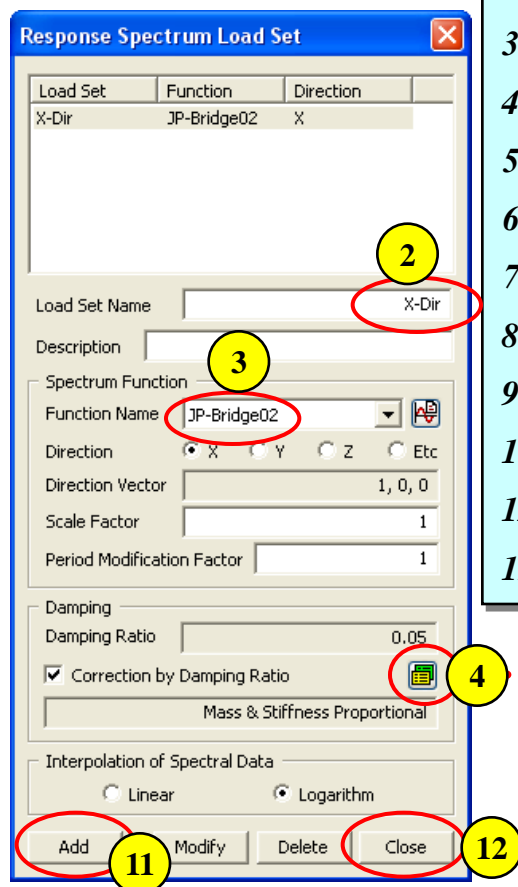
Analysis Control Dialog is automatically activated at startup.


Step 2.

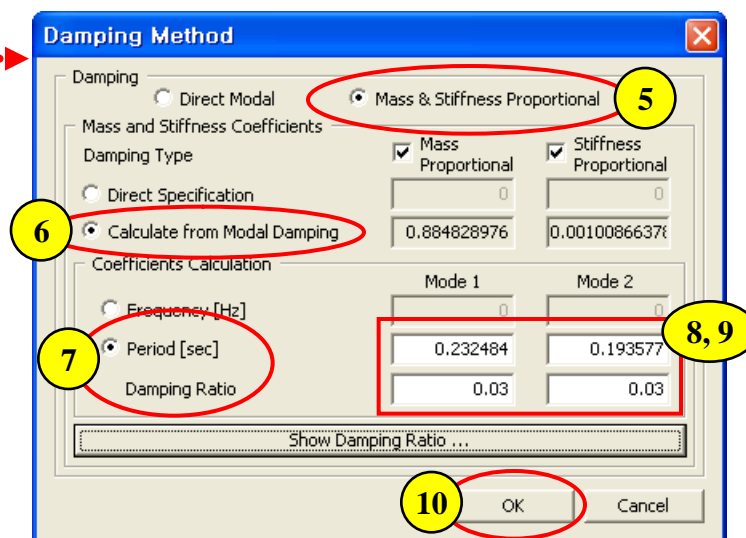
1. File > Open ...
2. Select "TH-1. Analysis of a Steel Box Bridge" File
3. Select "Analysis" Toolbar
4. Click "Pre-Mode"
5. Analysis > Response Spectrum Analysis
> Response Spectrum Functions ...
6. Click [Add] Button
7. Click [Design Spectrum] Button
8. Use Default Setting
9. Click [OK] Button
10. Click [Close] Button



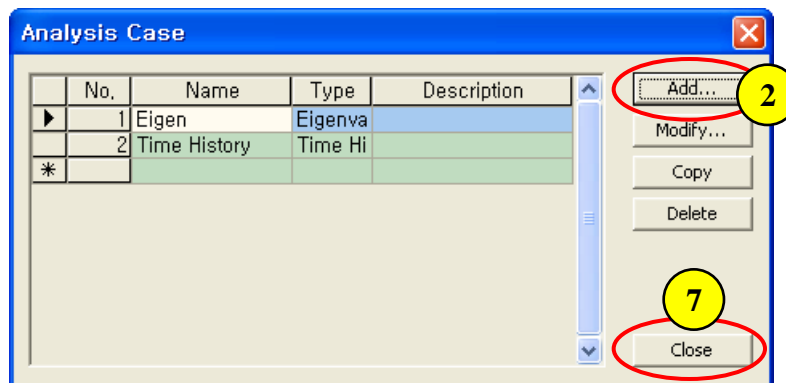
Step 3.



1. Analysis > Response Spectrum Analysis > Response Spectrum Load Set ...
2. Load Set Name : X-Dir
3. Select "JP-Bridge02" for Function Name
4. Click  Button of Damping
5. Check on "Mass & Stiffness Proportional"
6. Check on "Calculate from Modal Damping"
7. Check on "Period [sec]" for Coefficients Calculation"
8. Input Period of Mode 1 and 2 from Eigenvalue Analysis (same as Time History Analysis)
9. Damping Ratio : 0.03
10. Click [OK] Button
11. Click [Add] Button
12. Click [Close] Button



Step 4.



1. Analysis > Analysis Case ...

2. Click [Add] Button

3. Name : Response_X-Dir

4. Analysis Type : Response Spectrum

5. Drag & Drop "Load > X-Dir" to "Used" Window

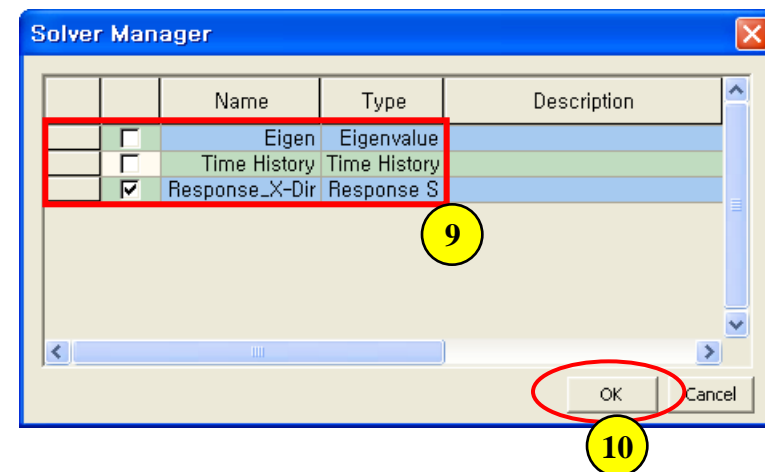
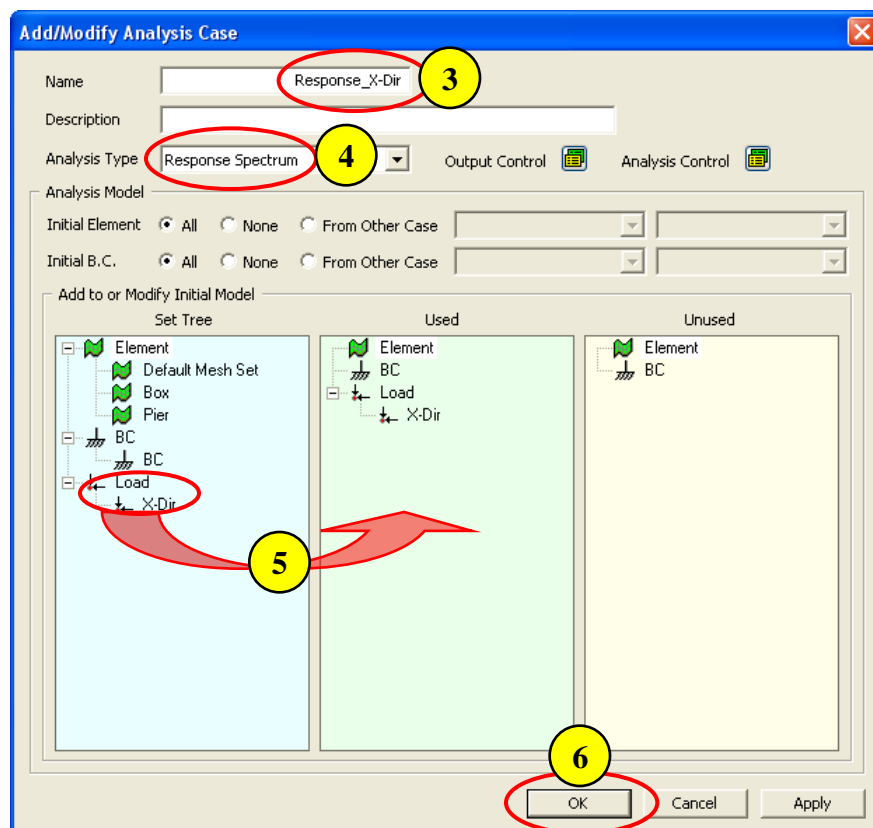
6. Click [OK] Button

7. Click [Close] Button

8. Analysis > Solve ...

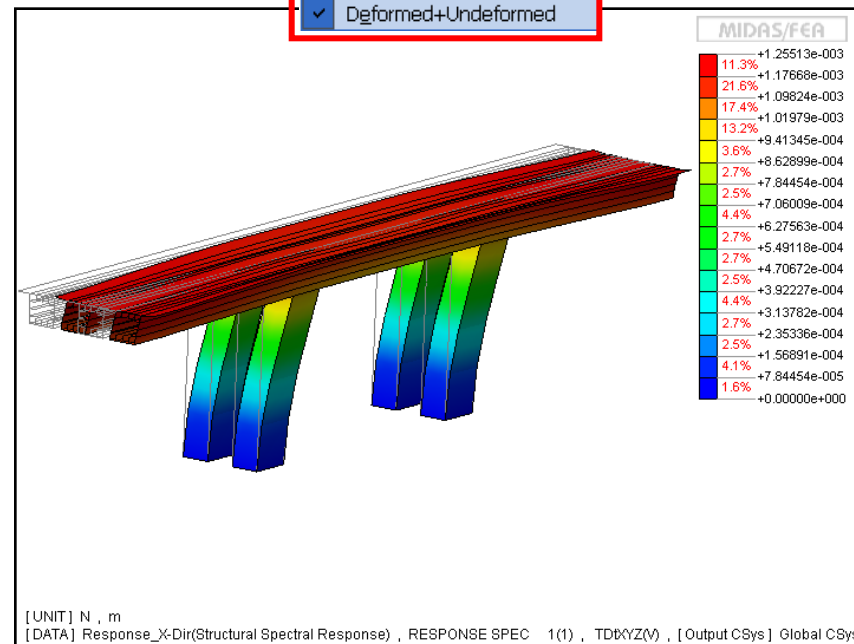
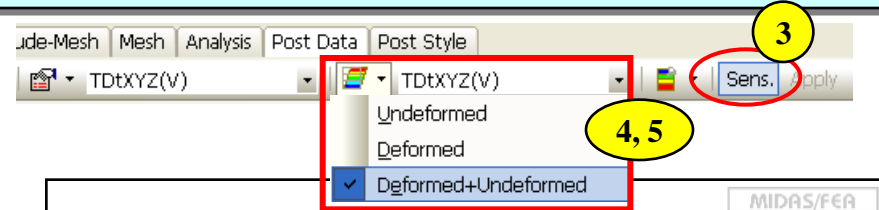
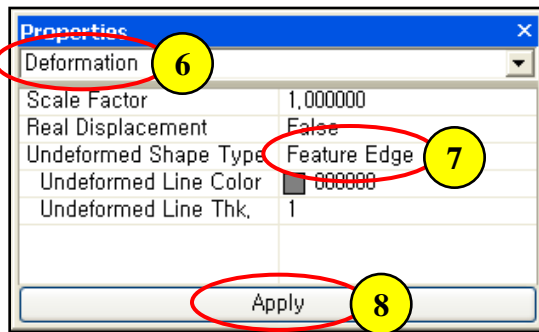
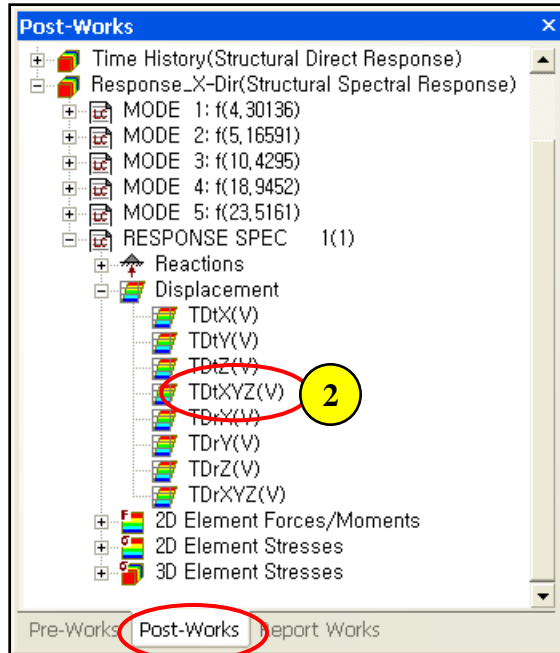
9. Check off "Eigen" & "Time History"

10. Click [OK] Button

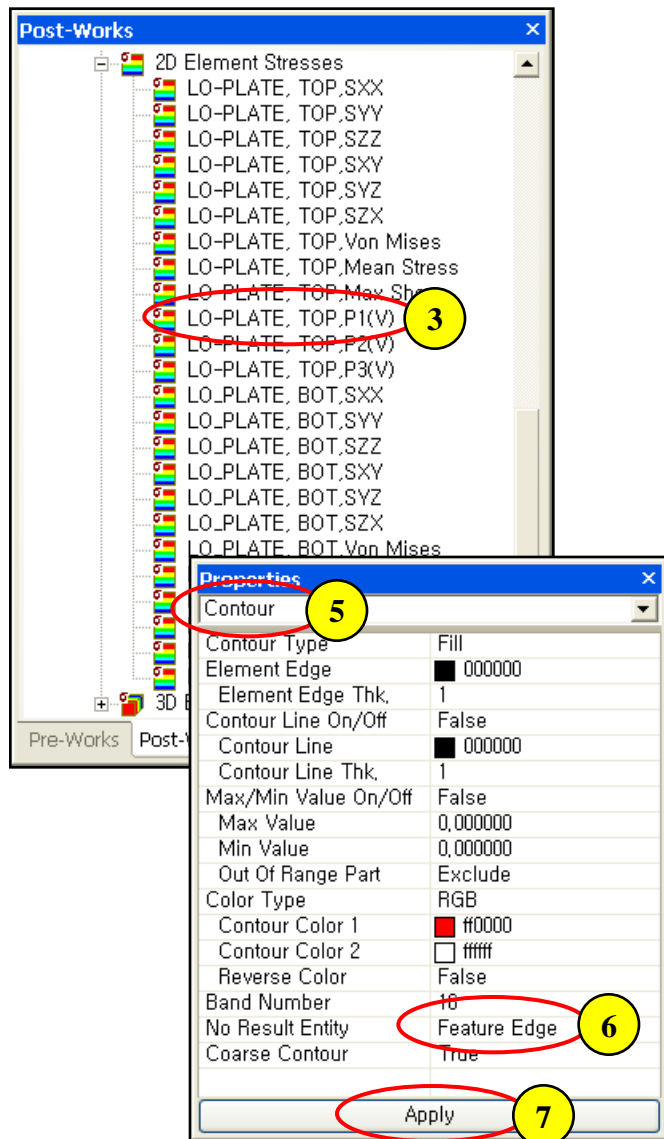


Step 5.

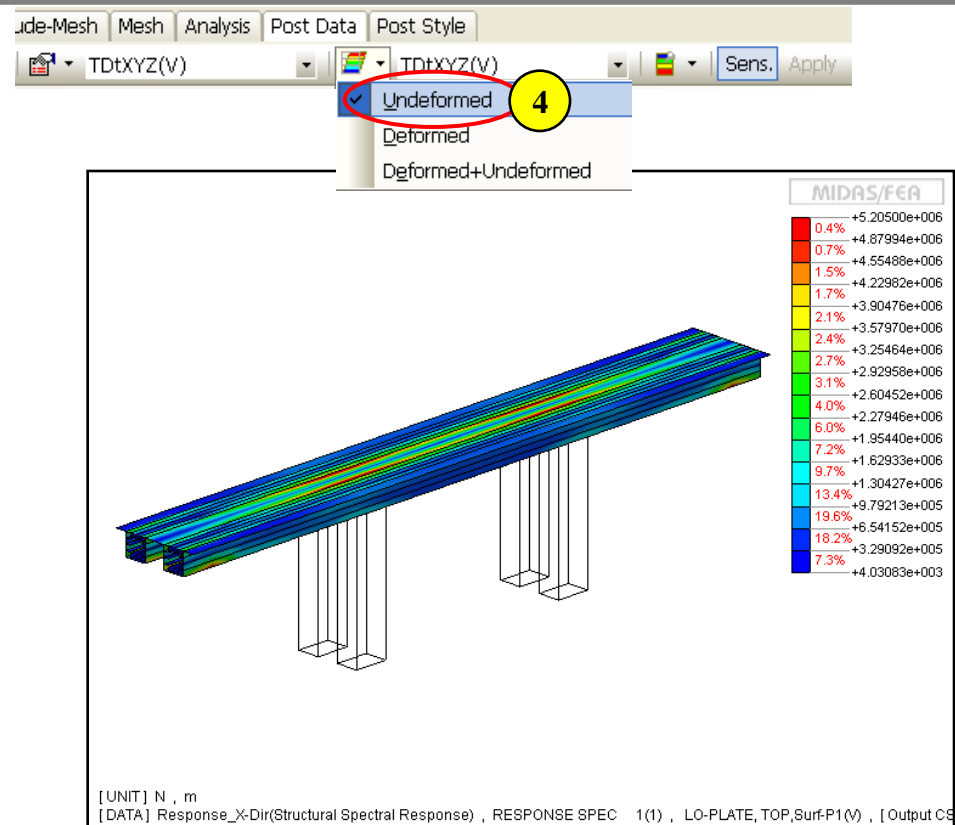
1. Post-Works Tree : Response_X-Dir > RESPONSE SPEC 1(1) > Displacement
2. Double Click "TDtXYZ(V)"
3. Click "Sens." Button
4. Select "Deformed+Undeformed" for Mesh Shape (See Figure)
5. Select "TDtXYZ(V)" for Deformation Data
6. Property Window : Deformation
7. Undeformed Shape Type : Feature Edge
8. Click [Apply] Button



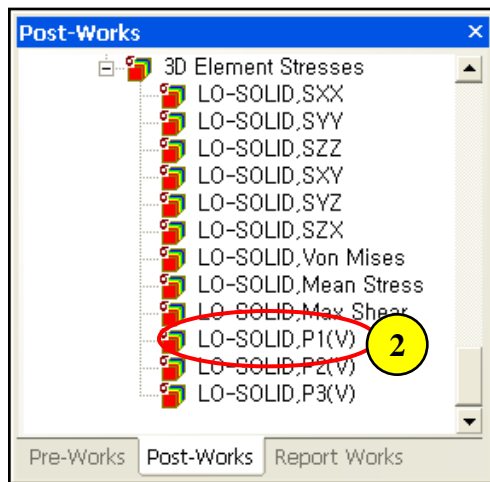
Step 6.



1. Select "Undeformed" for Mesh Shape
2. Post-Works Tree : Response_X-Dir > RESPONSE SPEC 1(1)
> 2D Element Stresses
3. Double Click "LO-PLATE, TOP, P1(V)"
4. Select "Undeformed" for Mesh Shape (See Figure)
5. Property Window : Contour
6. No Result Entity : Feature Edge
7. Click [Apply] Button



Step 7.



1. Post-Works Tree : Response_X-Dir > RESPONSE SPEC 1(1)
> 3D Element Stresses

2. Double Click “LO-SOLID, P1(V)”

