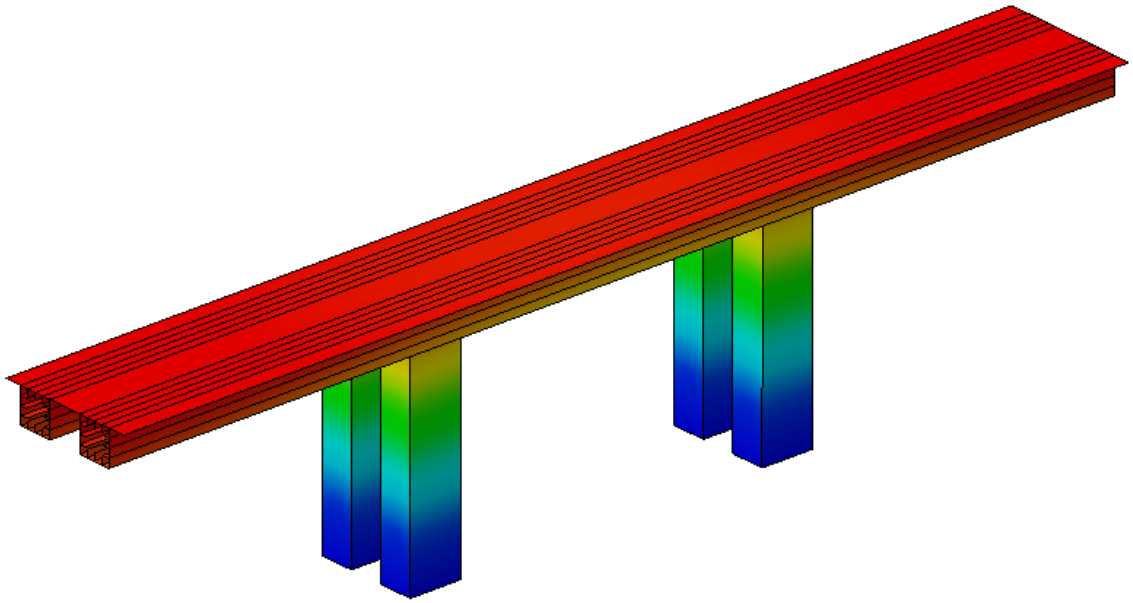
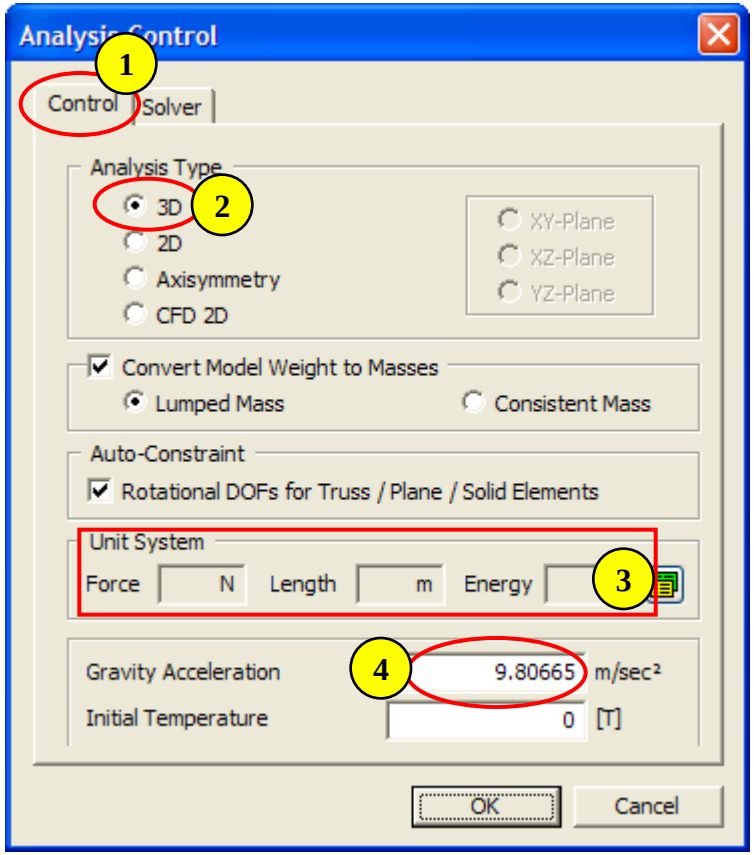
RS-1. Analysis of a Steel Box Bridge



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.

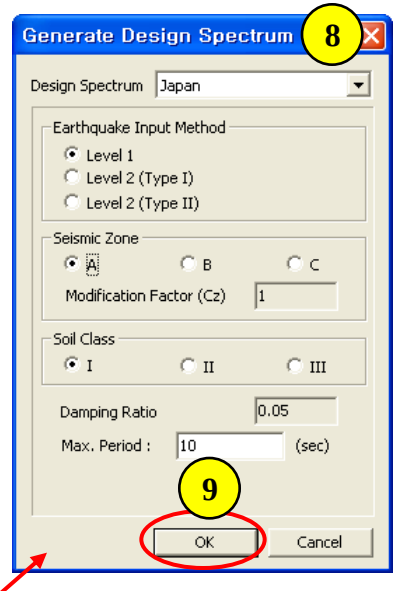


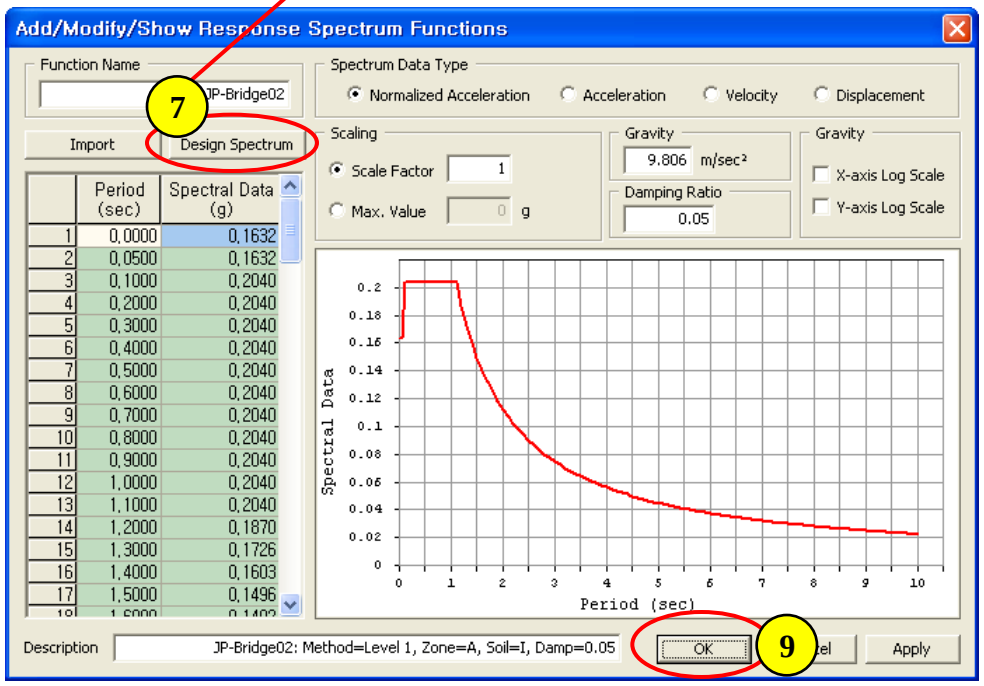
1. *Analysis > Analysis Control – "Control" tab*
2. *Analysis Type : 3D*
3. *Unit : N , m*
4. *Click* **[OK]** *Button*

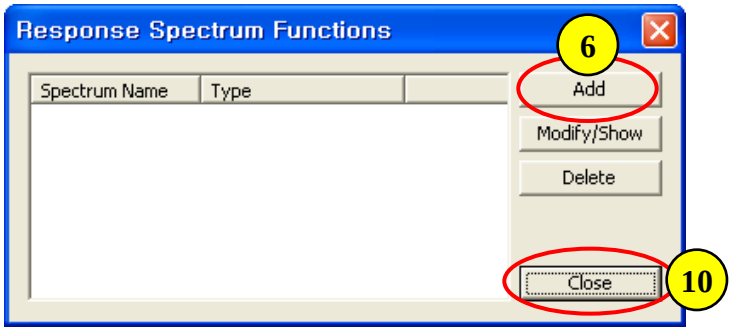
Analysis Control Dialog is automatically activated at startup.

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*

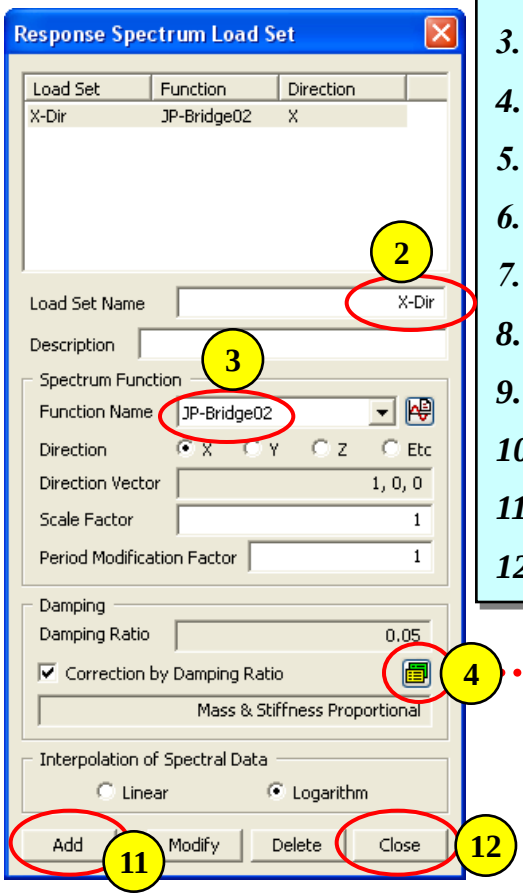


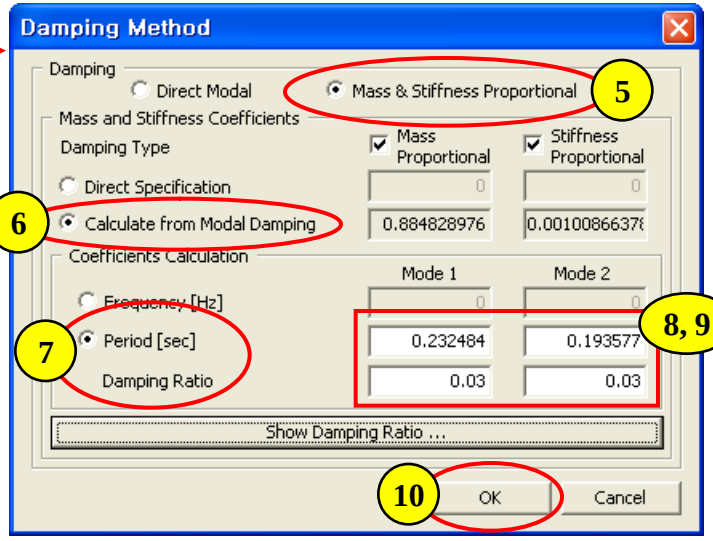




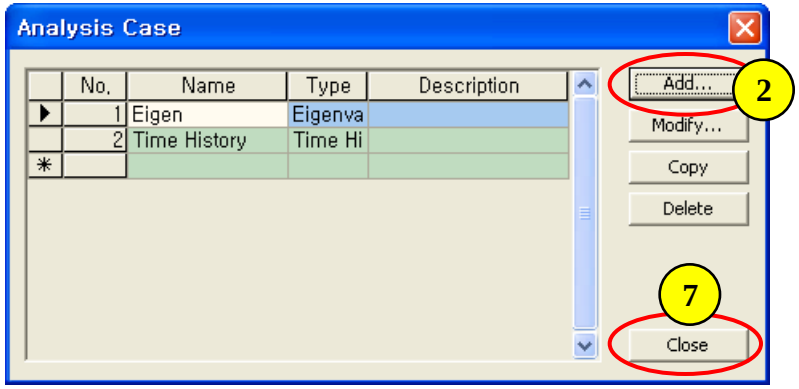
Step 3.

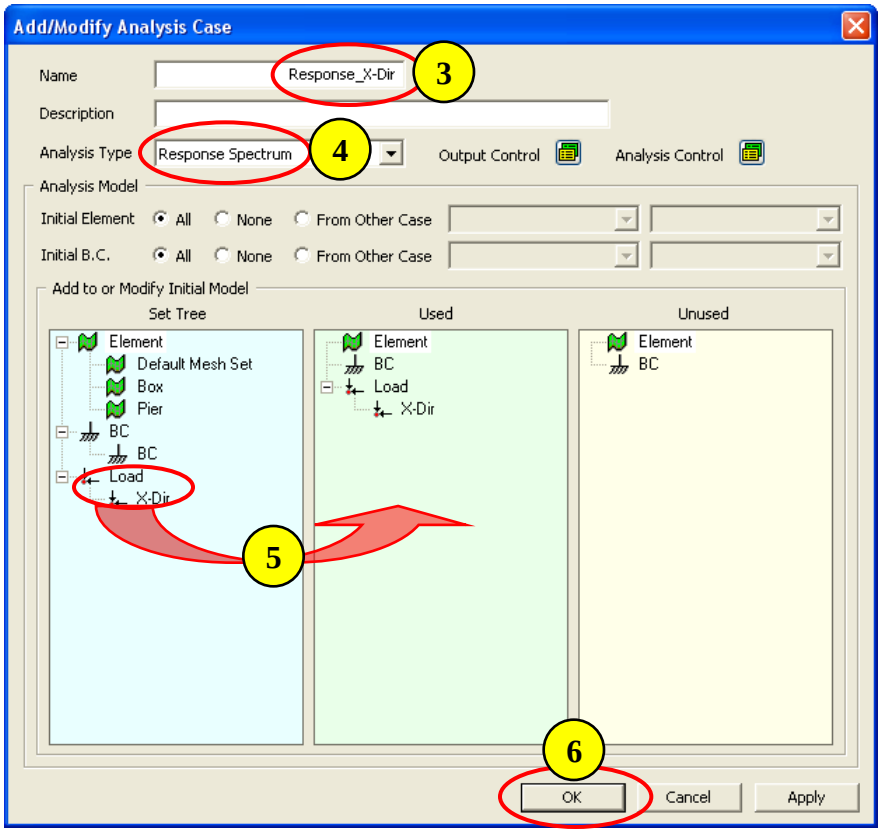
1. *Analysis > Response Spectrum Analysis > Response Spectrum Load Set …*
2. *Load Set Name : X-Dir*
3. *Select "JP-Bridge02" for Function Name*
4. *Click Button of Damping*
5. *Check on "Mass & Stiffness Proportional"*
6. *Check on "Calculate from Modal Damping"*
7. *Check on "Period [sec]" for Coefficients 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*



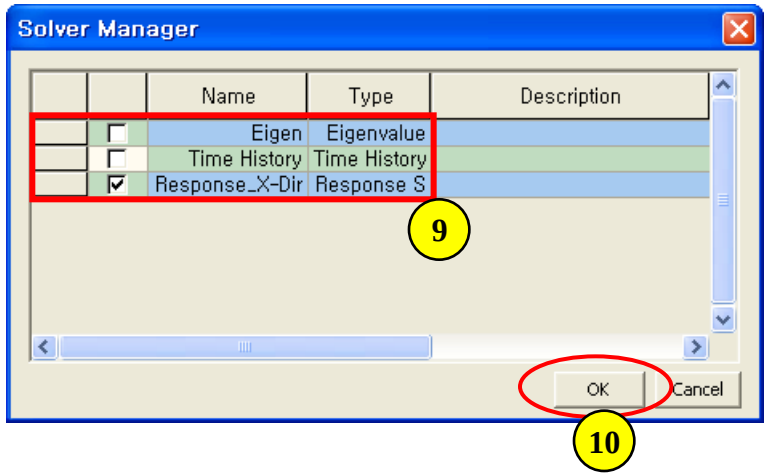


Step 4.



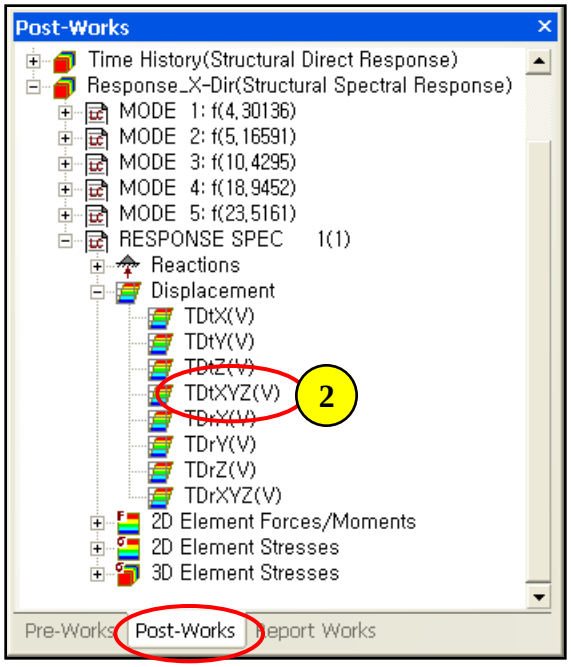


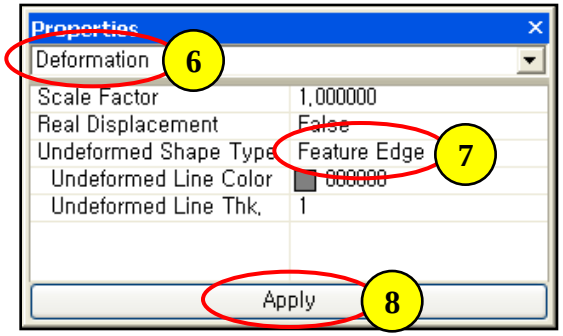
1. *Analysis > Analysis Case …*
2. *Click* **[Add]** *Button*
3. *Name : Response_X-Dir*
4. *Analysis Type : Response Spectrum*
5. *Drag & Drop "Load > X-Dir" to "Used" Window*
6. *Click* **[OK]** *Button*
7. *Click* **[Close]** *Button*
8. *Analysis > Solve ...*
9. *Check off "Eigen" & "Time History"*
10. *Click* **[OK]** *Button*

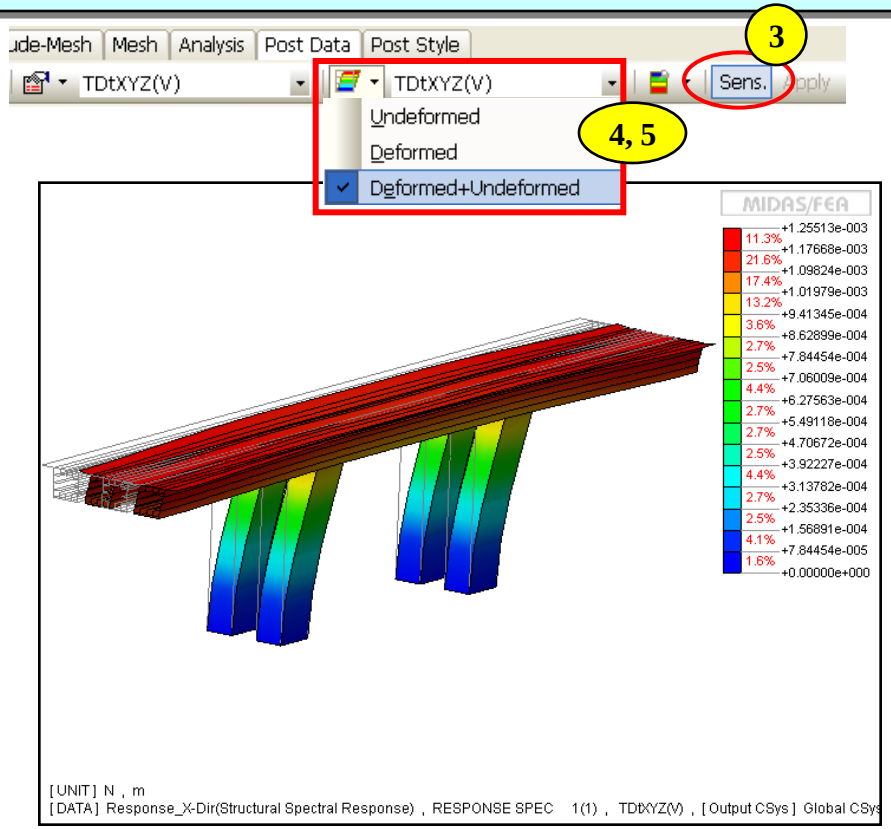


Step 5.

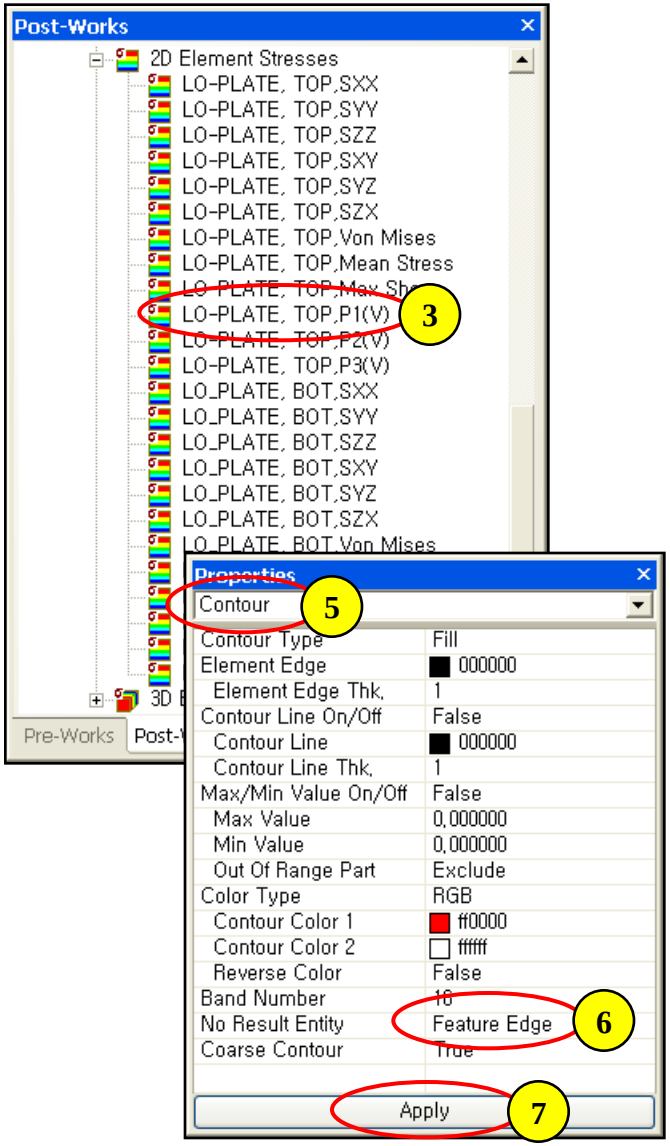
1. Post-Works Tree : Response_X-Dir > RESPONSE SPEC 1(1) > Displacement
2. Double Click "TDtXYZ(V)"
3. Click "Sens." Button
4. Select "Deformed+Undeformed" for Mesh Shape (See Figure)
5. Select "TDtXYZ(V)" for Deformation Data
6. Property Window : Deformation
7. Undeformed Shape Type : Feature Edge
8.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



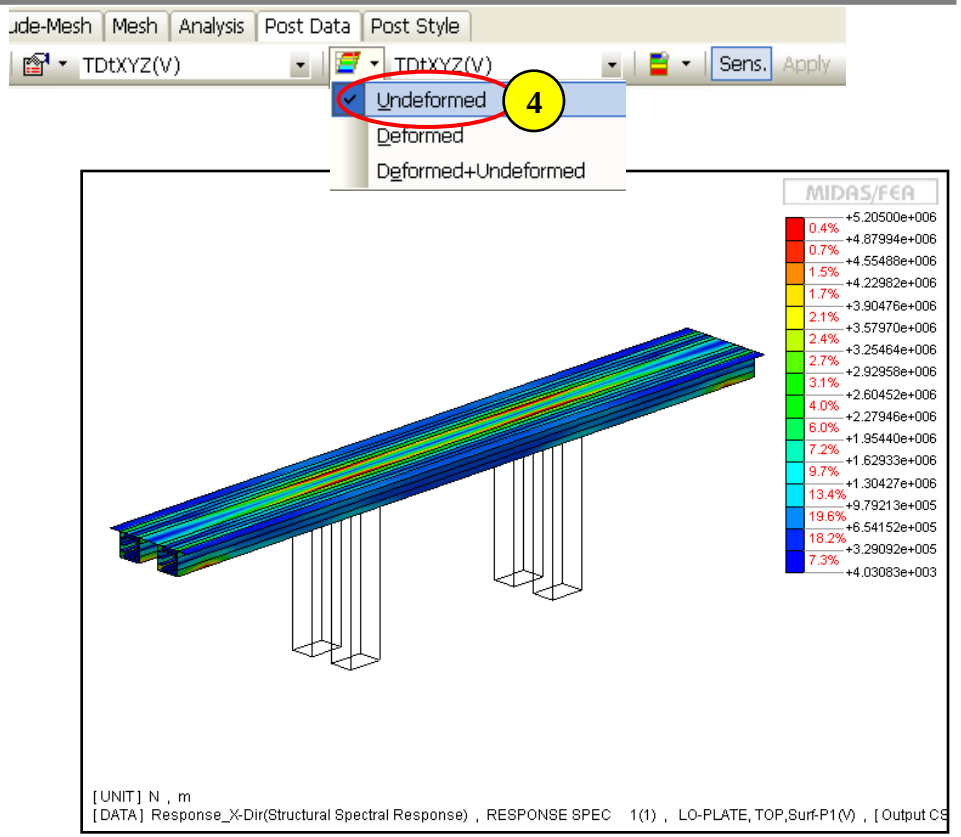




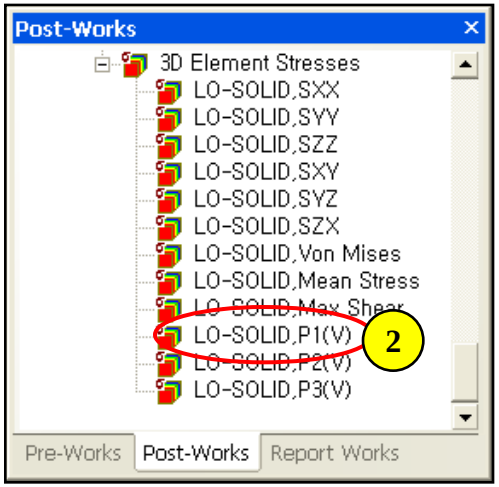
Step 6.



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)"