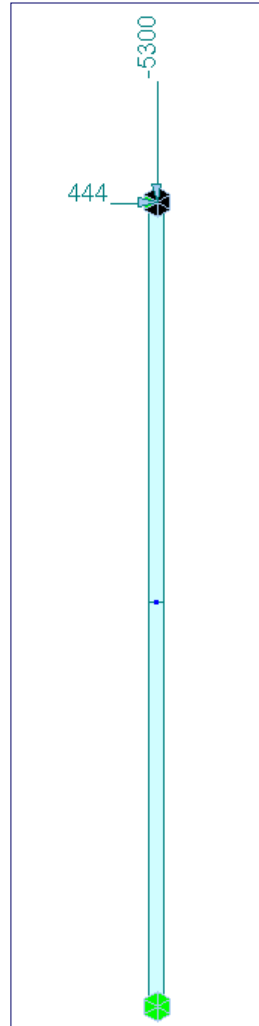


→ P-Delta Analysis

Concept

- P-Delta analysis is required when high vertical and lateral forces act simultaneously on a structure, causing 1st and 2nd order lateral displacement.
- **How P-Delta analysis is performed:**
 1. Linear static analysis is performed first.
 2. Based on the member forces obtained from the first analysis, a new geometric stiffness matrix is formulated.
 3. Static analysis is repeatedly performed and the geometric stiffness matrix is repeatedly modified until the given convergence conditions are met.



Overview

- **P-Delta Effect Analysis**
- **Model**
 - Unit : N, mm
 - Beam Element
- **Load & Boundary Condition**
 - Lateral load
 - Vertical load
 - Supports
- **Analysis**
 - P-Delta Analysis
- **Result Evaluation**
 - Deflection

→ *Step 1. Define material properties.*

Material Data

General
Material ID: 1 Name: MAT-1

Elasticity Data
Type of Design: Steel

Steel
Standard: BS(S)
DB: 43A

Concrete
Standard:
DB:

Type of Material
 Isotropic Orthotropic

Steel
Modulus of Elasticity : 2.0500e+005 N/mm²
Poisson's Ratio : 0.3
Thermal Coefficient : 1.2000e-005 1/°C
Weight Density : 7.698e-005 N/mm³
 Use Mass Density: 7.85e-009 N/mm³/q

Concrete
Modulus of Elasticity : 0.0000e+000 N/mm²
Poisson's Ratio : 0
Thermal Coefficient : 0.0000e+000 1/°C
Weight Density : 0 N/mm³
 Use Mass Density: 0 N/mm³/q

Plasticity Data
Plastic Material Name: NONE

Thermal Transfer
Specific Heat : 0 Btu·q/N·°C
Heat Conduction : 0 Btu/mm·hr·°C

OK Cancel Apply

- 1. Model > Properties > Material
- 2. Name: MAT-1
- 3. Type of Design: Steel
- 4. Standard: BS(S)
- 5. DB: 43A
- 6. Click [OK] Button.

Step 2. Define section properties

Section Data

DB/User Value **2**

Section ID 1 **3** Solid Rectangle

Name Beam **4** Built-Up Section

Size

| | | |
|---|---------|----|
| H | 25.0000 | mm |
| B | 25.0000 | mm |

5

Section Properties **6**

Calc. Section Properties

| | | |
|--------|--------------|-----------------|
| Area | 6.25000e+002 | mm ² |
| Asy | 5.20833e+002 | mm ² |
| Asz | 5.20833e+002 | mm ² |
| Ixx | 5.49316e+004 | mm ⁴ |
| Iyy | 3.25521e+004 | mm ⁴ |
| Izz | 3.25521e+004 | mm ⁴ |
| Cyp | 12.5000 | mm |
| Cym | 12.5000 | mm |
| Czp | 12.5000 | mm |
| Czm | 12.5000 | mm |
| Qyb | 78.1250 | mm ² |
| Qzb | 78.1250 | mm ² |
| Peri:O | 1.00000e+002 | mm |

Consider Shear Deformation.

Offset : Center-Center
Change Offset ...

Show Calculation Results... **7** OK Cancel Apply

1. Model > Properties > Section...

2. Select "Value" tab.

3. Section Type: Solid Rectangle

4. Name: Beam

5. Size – H: 25 mm, B: 25 mm

6. Click [Calc. Section Properties] Button.

7. Click [OK] Button.

Step 3. Create elements (1)



Column Wizard

Input/Edit | Insert

Distance: 1250 mm Repeat: 2

Add
Delete
Delete All

| No. | Z Coord. |
|-----|----------|
| 1 | 0 |
| 2 | 1250 |
| 3 | 2500 |

Boundary Condition
 Pin Fix
 None

Show Element No.

Material: 1: MAT-1 **6**

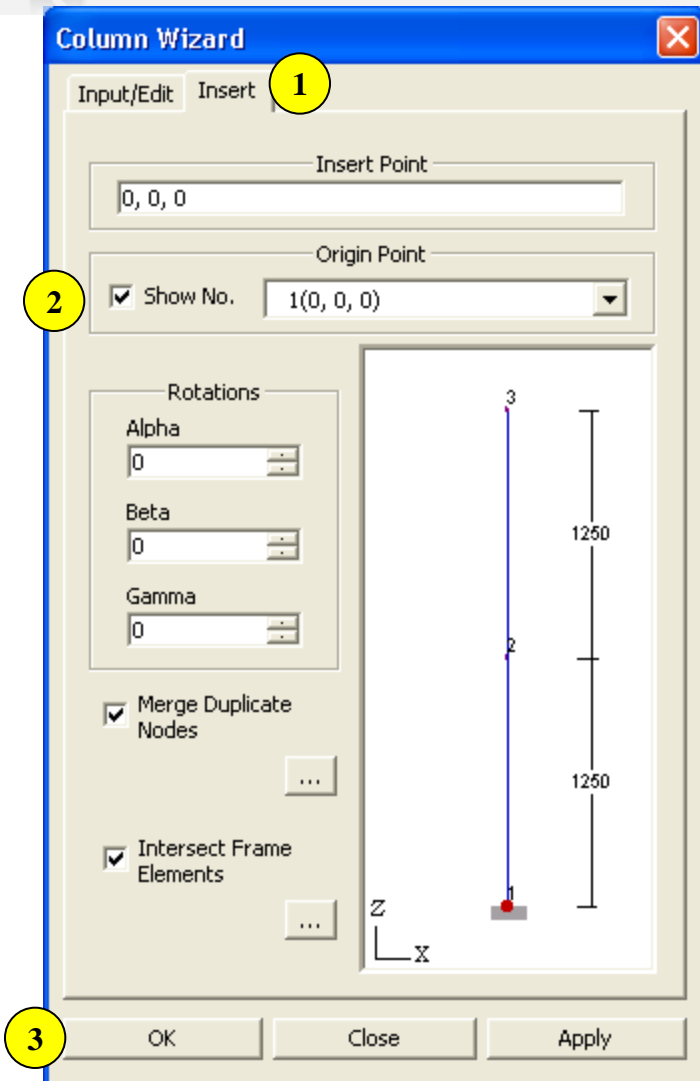
Section: 1: Beam **7**

Redraw & Update Data

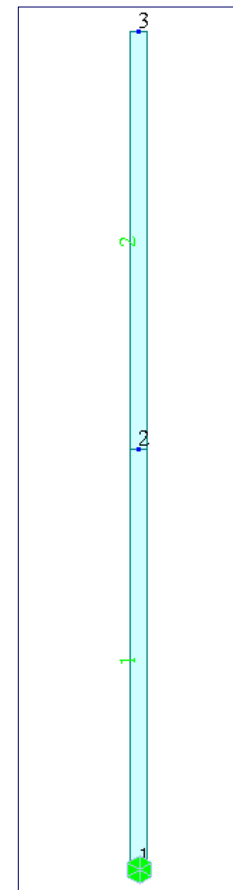
OK Close Apply

1. *Model > Structure Wizard > Column*
2. *Distance: 1250 mm, Repeat: 2*
3. *Click [Add] Button.*
4. *Boundary Condition: Fix*
5. *Check on “Show Element No.”.*
6. *Material: 1: MAT-1*
7. *Section: 1: Beam*

Step 3. Create elements (2)



1. Model > Structure Wizard > Column – “Insert” tab
2. Check on “Show No.”.
3. Click [OK] Button.



Step 4. Define Structure Type



1. Model > Structure Type
2. Structure Type > X-Z Plane
3. Click [OK] Button.

2. 2D analysis is carried out in X-Z plane.

Structure Type

Structure Type

3-D X-Z Plane Y-Z Plane X-Y Plane Constraint RZ

Conversion of Structure Self-weight into Masses

Do not Convert

Lumped Mass

Convert to X, Y, Z Convert to X, Y Convert to Z

Consider Mass Offset

Consistent Mass

Gravity acceleration : 9806 mm/sec²

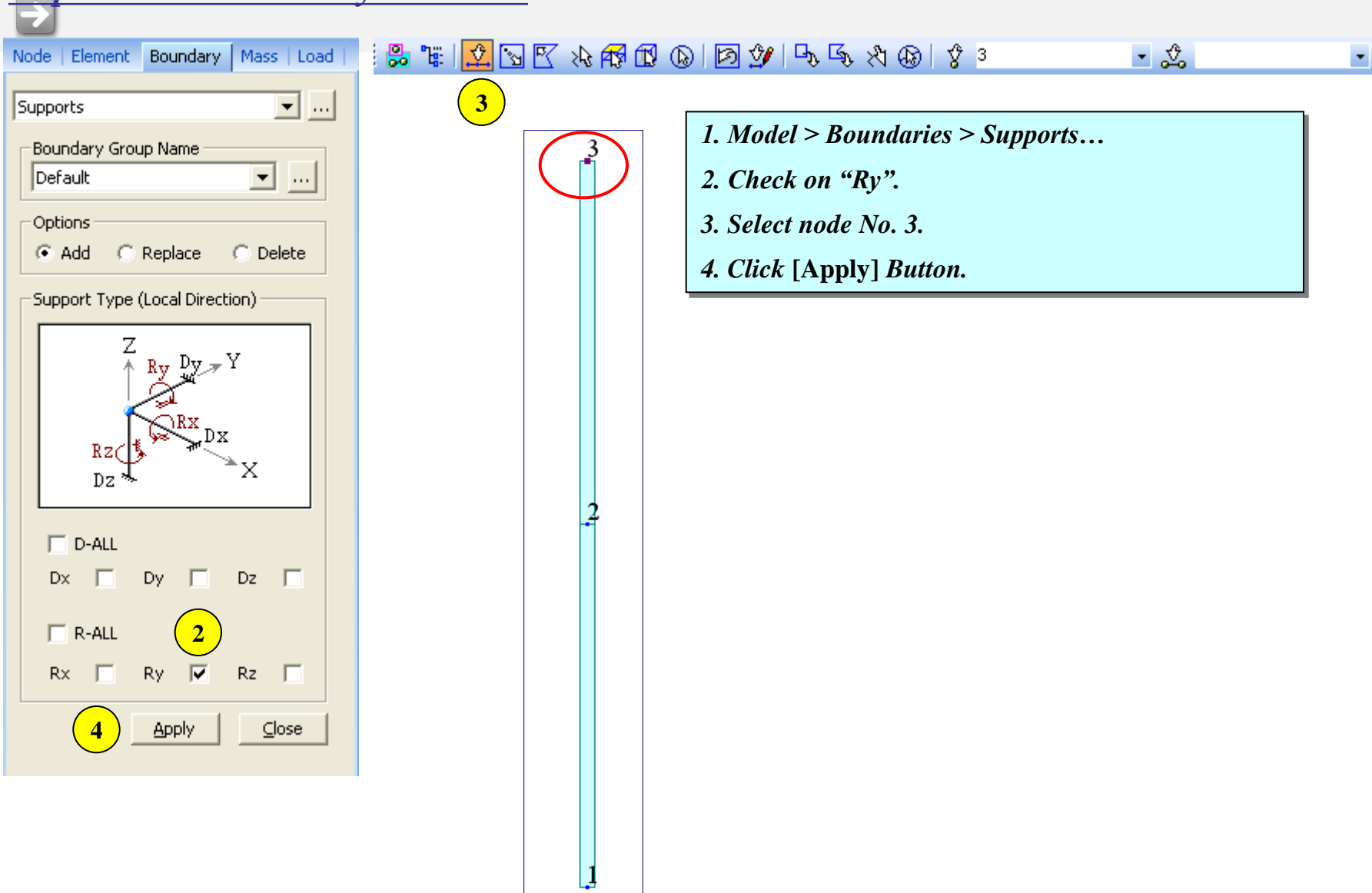
Initial Temperature : 0 [C]

Align Top of Beam Section with Center Line (X-Y Plane) for Display

Align Top of Slab(Plate) Section with Center Line (X-Y Plane) for Display

3 OK Cancel

Step 5. Create boundary condition



The screenshot displays the 'Supports' dialog box in the midas Gen software. The dialog box is titled 'Supports' and has a 'Boundary Group Name' set to 'Default'. Under 'Options', the 'Add' radio button is selected. The 'Support Type (Local Direction)' section shows a 3D coordinate system with axes X, Y, and Z. Below this, the 'R-ALL' section has the 'Ry' checkbox checked, which is circled in yellow with the number '2'. The 'Apply' button is circled in yellow with the number '4'. In the background, a 3D model of a vertical member is shown with nodes 1, 2, and 3. Node 3 is circled in red with the number '3'.

1. *Model > Boundaries > Supports...*
2. *Check on “Ry”.*
3. *Select node No. 3.*
4. *Click [Apply] Button.*

Step 6. Create static load cases.

Static Load Cases

Name : HL

Type : User Defined Load (USER)

Description :

| No | Name | Type | Description |
|----|------|--------------------------|-------------|
| 1 | HL | User Defined Load (USER) | |
| 2 | VL | User Defined Load (USER) | |
| * | | | |

Close

1. Load > Static Load Cases...
2. Name: HL, Type: User Defined Load (User)
3. Click [Add] Button.
4. Name: VL, Type: User Defined Load (User)
5. Click [Add] Button.
6. Click [Close] Button.

Step 7. Create nodal loads (1).

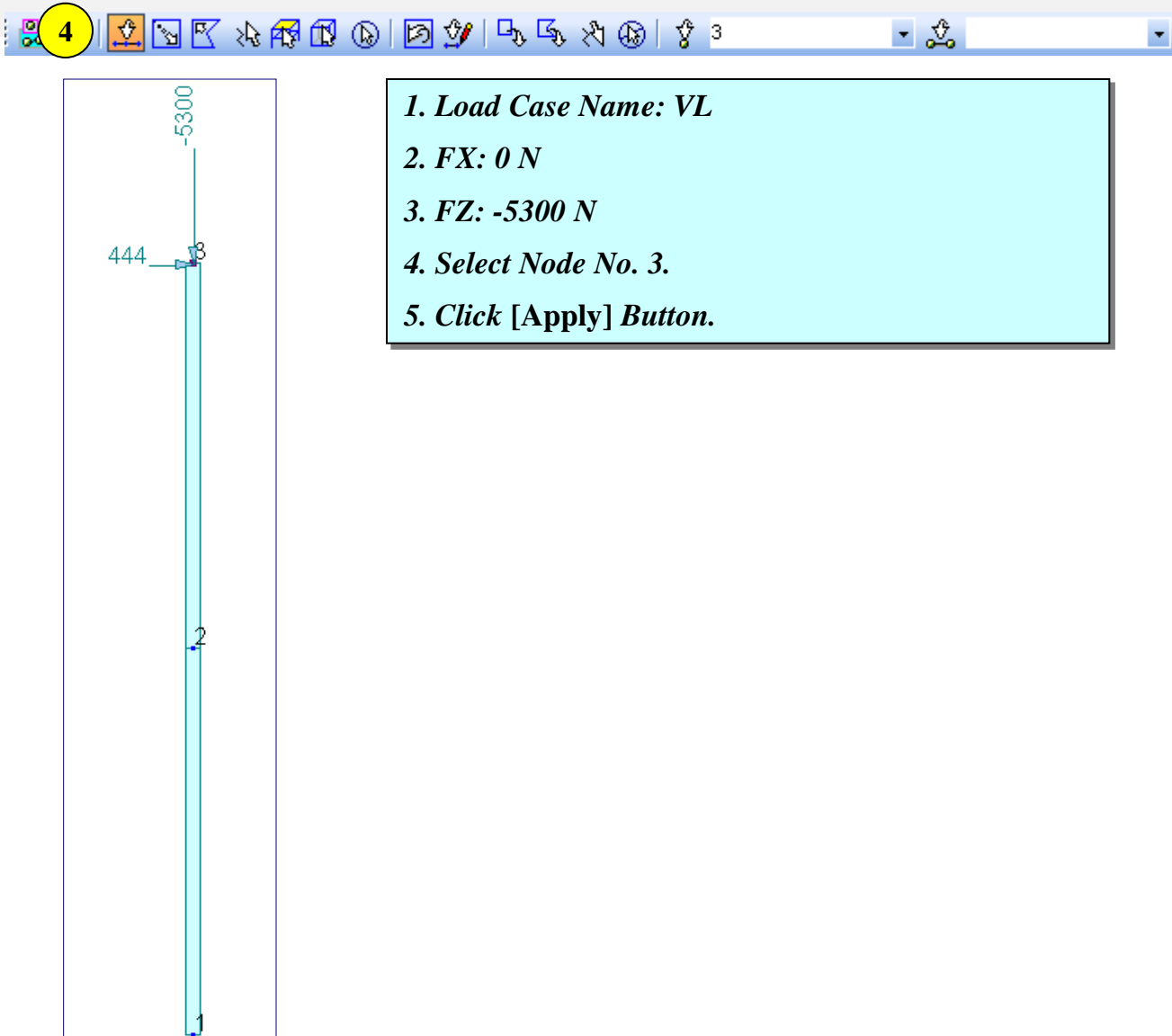
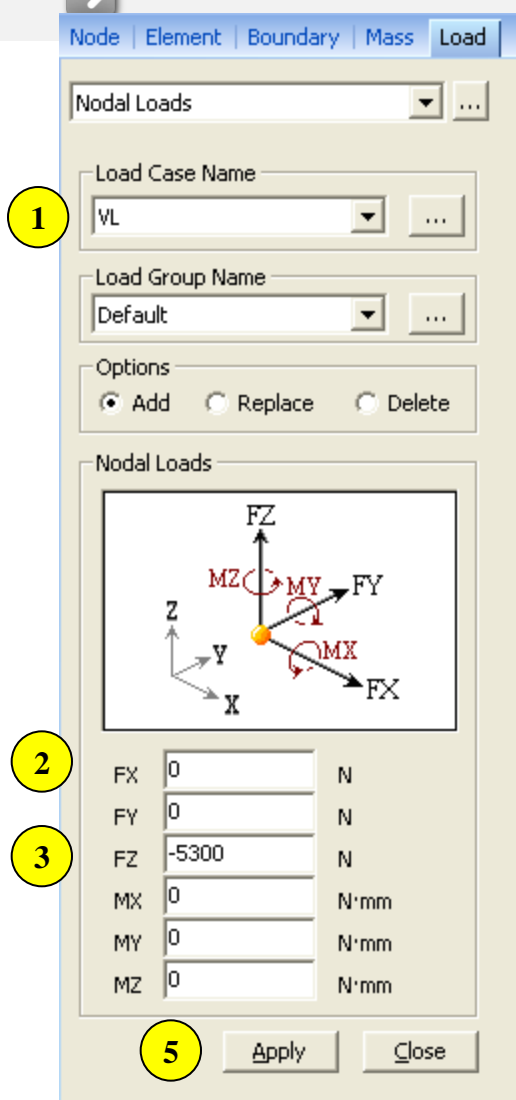
The screenshot shows the 'Nodal Loads' dialog box in the midas Gen software. The 'Load Case Name' is set to 'HL'. The 'Load Group Name' is 'Default'. The 'Options' are set to 'Add'. The 'Nodal Loads' section shows a 3D coordinate system with axes X, Y, and Z. The load values are: FX = 444 N, FY = 0 N, FZ = 0 N, MX = 0 N·mm, MY = 0 N·mm, and MZ = 0 N·mm. The 'Apply' button is highlighted with a yellow circle labeled '5'. In the background, a 3D model of a vertical member is shown with a nodal load of 444 N applied at the top node, which is circled in red and labeled '4'.

1. Load > Nodal Load...
2. Load Case Name: HL
3. FX: 444 N
4. Select Node No. 3
5. Click [Apply] Button.

The screenshot shows the 'Display' dialog box in the midas Gen software. The 'Load' tab is selected. The 'Load Value' checkbox is checked. The 'Display by Member' checkbox is also checked. The 'Load Value' checkbox is highlighted with a yellow circle labeled 'ii'. In the background, a 3D model of a vertical member is shown with a nodal load of 444 N applied at the top node, which is circled in red and labeled '4'.

1. Display nodal load value.
 - i. View > Display > "Load" tab
 - ii. Check on "Load Value".

Step 7. Create nodal loads (2).



1. Load Case Name: VL
2. FX: 0 N
3. FZ: -5300 N
4. Select Node No. 3.
5. Click [Apply] Button.

Step 8. P-Delta analysis control




P-Delta Analysis Control

Control Parameters

Number of Iterations : 5

Convergence Tolerance: 1e-005

P-Delta Combination

Load Case : VL 

Scale Factor : 1

| Load Case | Scale |
|-----------|-------|
| VL | 1 |

Add

Modify

Delete

Remove P-Delta Analysis Data

OK Cancel


2 **3** **4**

1. Analysis > P-Delta Analysis Control...

2. Load Case: VL

3. Click [Add] Button.

4. Click [OK] Button.

 1. When a lateral load acts upon a column member, resulting in moments and shear forces in the member, an additional vertical axial load (compression) increases the member forces. Consider the p-delta effect due to vertical axial load.

Step 9. Check the input data



1. Model > Boundaries > Supports Table...

| Model View | | Supports | | | | | | | |
|------------|------|----------|----|----|----|----|----|---------|--|
| | Node | Dx | Dy | Dz | Rx | Ry | Rz | Group | |
| | 1 | 1 | 1 | 1 | 1 | 1 | 1 | Default | |
| | 3 | 0 | 0 | 0 | 0 | 1 | 0 | Default | |
| * | | | | | | | | | |

2. Load > Load Tables > Nodal Loads...

| Model View | | Supports | | Nodal Loads | | | | | | |
|------------|------|-----------|--------|-------------|----------|-----------|-----------|-----------|---------|--|
| | Node | Load Case | FX (N) | FY (N) | FZ (N) | MX (N·mm) | MY (N·mm) | MZ (N·mm) | Group | |
| | 3 | VL | 0.00 | 0.00 | -5300.00 | 0.00 | 0.00 | 0.00 | Default | |
| | 3 | HL | 444.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | Default | |
| * | | | | | | | | | | |

Step 10. Perform analysis

1. Analysis > Perform Analysis

Step 11. Check displacements

Reactions | **Deformations** | Forces | Stresses

Deformed Shape

Load Cases/Combinations
 ST: HL
 Step

Displacement Velocity
 Acceleration
 Absolute Acceleration

Components
 DX DY DZ
 DXY DYZ DXZ
 DXYZ

Type of Display
 Deform Undeformed
 Values Legend
 Animate Mirrored
 Current Step Displ.
 Stage/Step Real Displ.
 Hinge Status

Apply Close

0.000 43.331 86.661

0.000 86.289 172.579

1. Results > Deformations > Deformed Shape...
2. Load Cases/Combinations: ST: HL
3. Components: DX
4. Type of Display: Undeformed, Values
5. Click [Apply] Button.

1. Check the displacement due to lateral load.
2. Lateral displacement without p-delta effect.
3. Lateral displacement due to p-delta effect. The additional axial force produces positive p-delta moment.