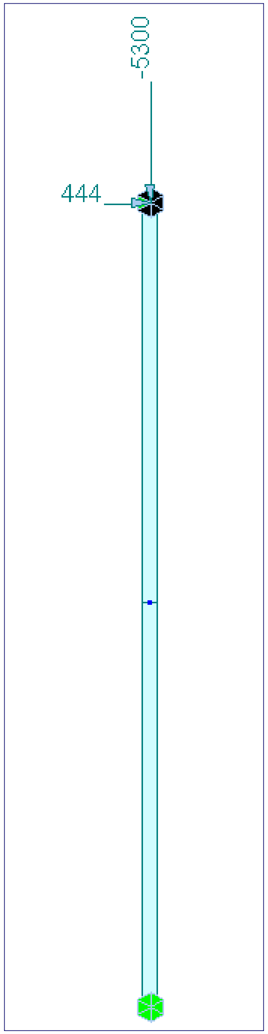
P-Delta Analysis

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:**

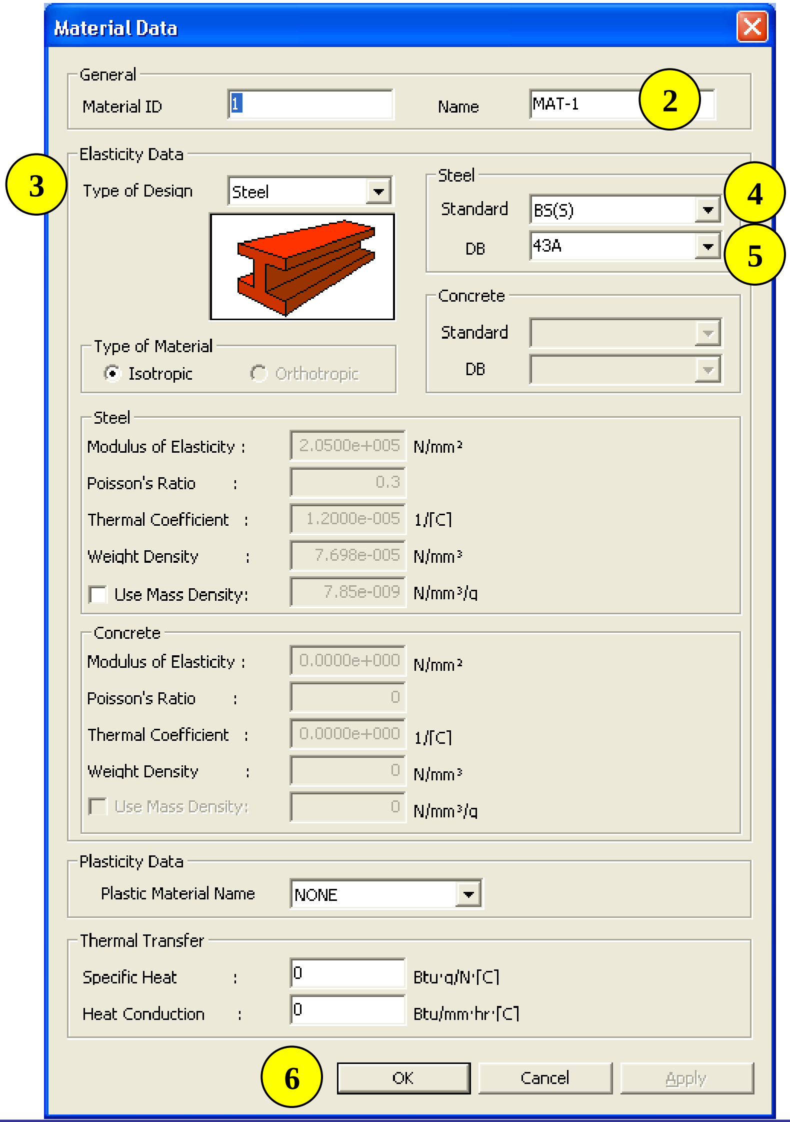
1. Linear static analysis is performed first.
2. Based on the member forces obtained from the first analysis, a new geometric stiffness matrix is formulated.
3. Static analysis is repeatedly performed and the geometric stiffness matrix is repeatedly modified until the given convergence conditions are met.



Overview

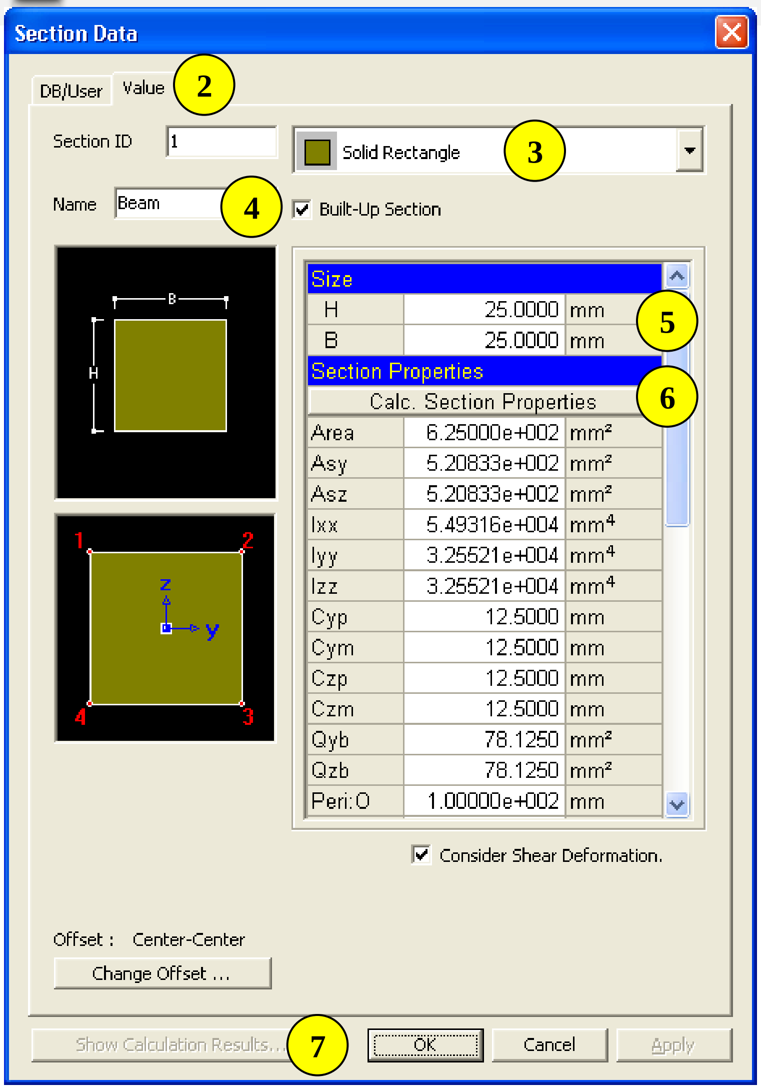
- **P-Delta Effect Analysis**
- **Model**
  - *Unit : N, mm*
  - *Beam Element*
- **Load & Boundary Condition**
  - *Lateral load*
  - *Vertical load*
  - *Supports*
- **Analysis**
  - *P-Delta Analysis*
- **Result Evaluation**
  - *Deflection*

Step 1. Define material properties.



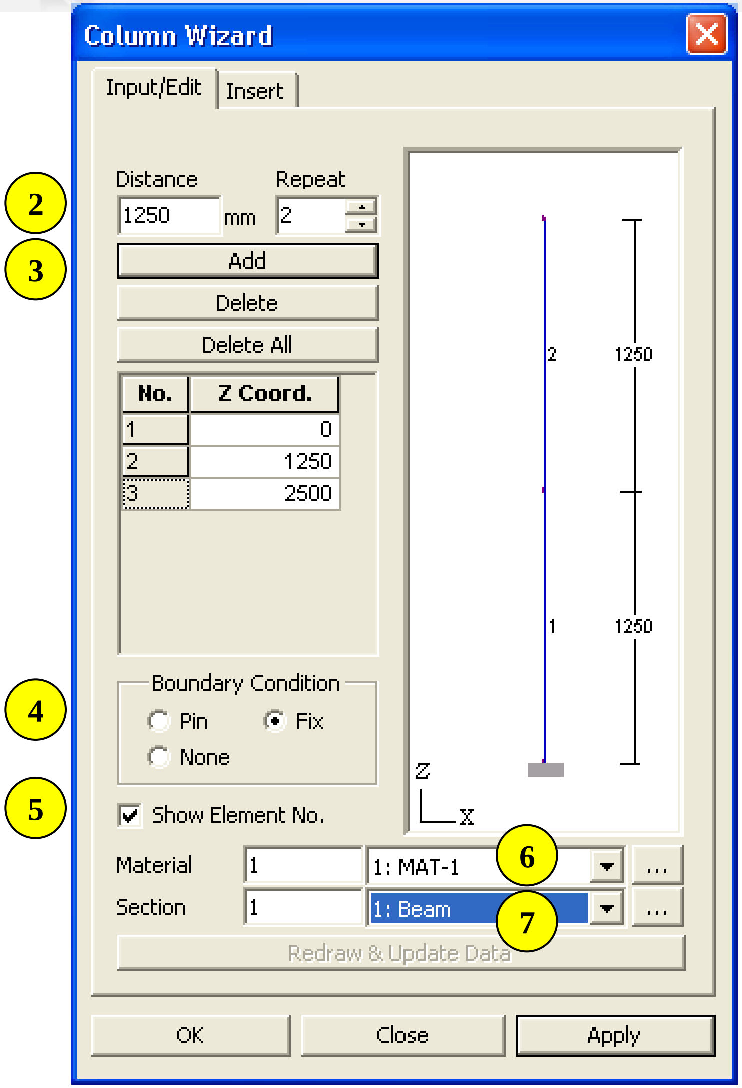
- 1. Model > Properties > Material
- 2. Name: MAT-1
- 3. Type of Design: Steel
- 4. Standard: BS(S)
- 5. DB: 43A
- 6. Click [OK] Button.

Step 2. Define section properties



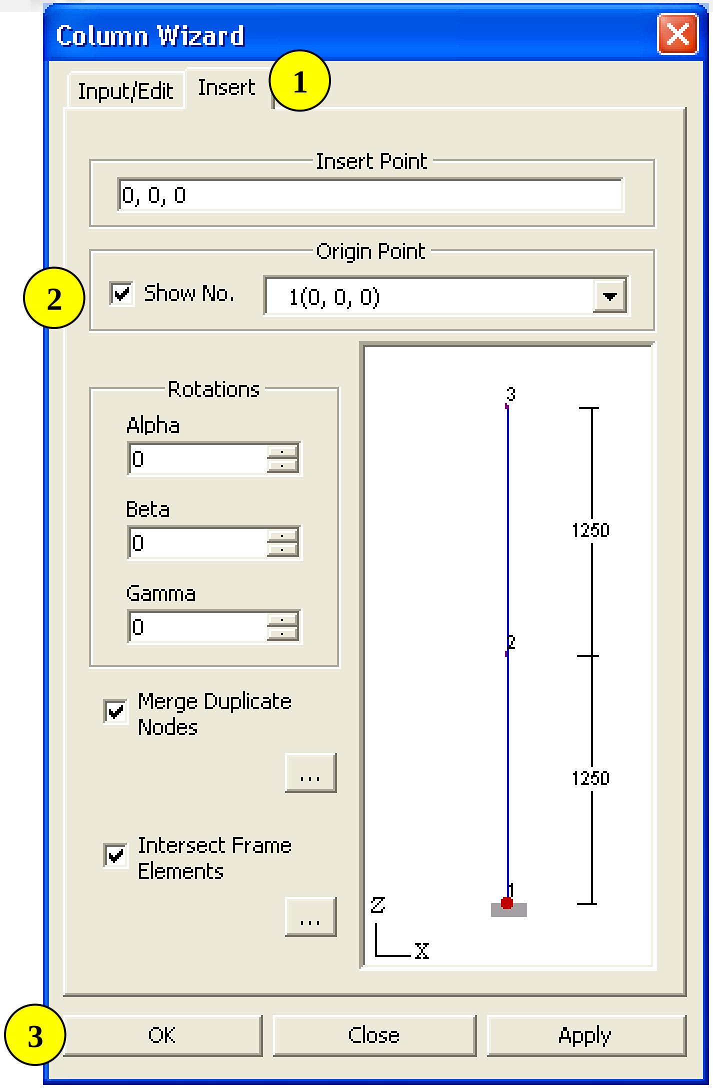
1. *Model > Properties > Section…*
2. *Select* "Value" *tab.*
3. *Section Type: Solid Rectangle*
4. *Name: Beam*
5. *Size – H: 25 mm, B: 25 mm*
6. *Click* [Calc. Section Properties] *Button.*
7. *Click* [OK] *Button.*

Step 3. Create elements (1)

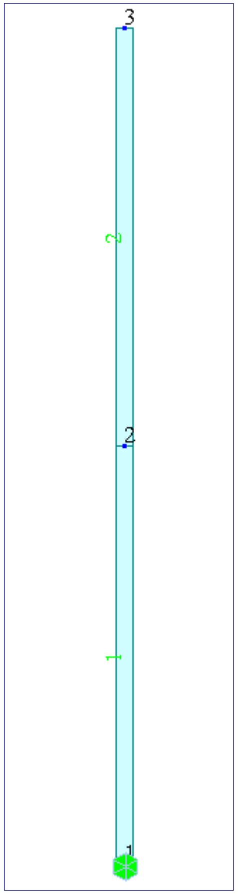


1. *Model > Structure Wizard > Column*
2. *Distance: 1250 mm, Repeat: 2*
3. *Click* [Add] *Button.*
4. *Boundary Condition: Fix*
5. *Check on* "Show Element No.".
6. *Material: 1: MAT-1*
7. *Section: 1: Beam*

Step 3. Create elements (2)



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



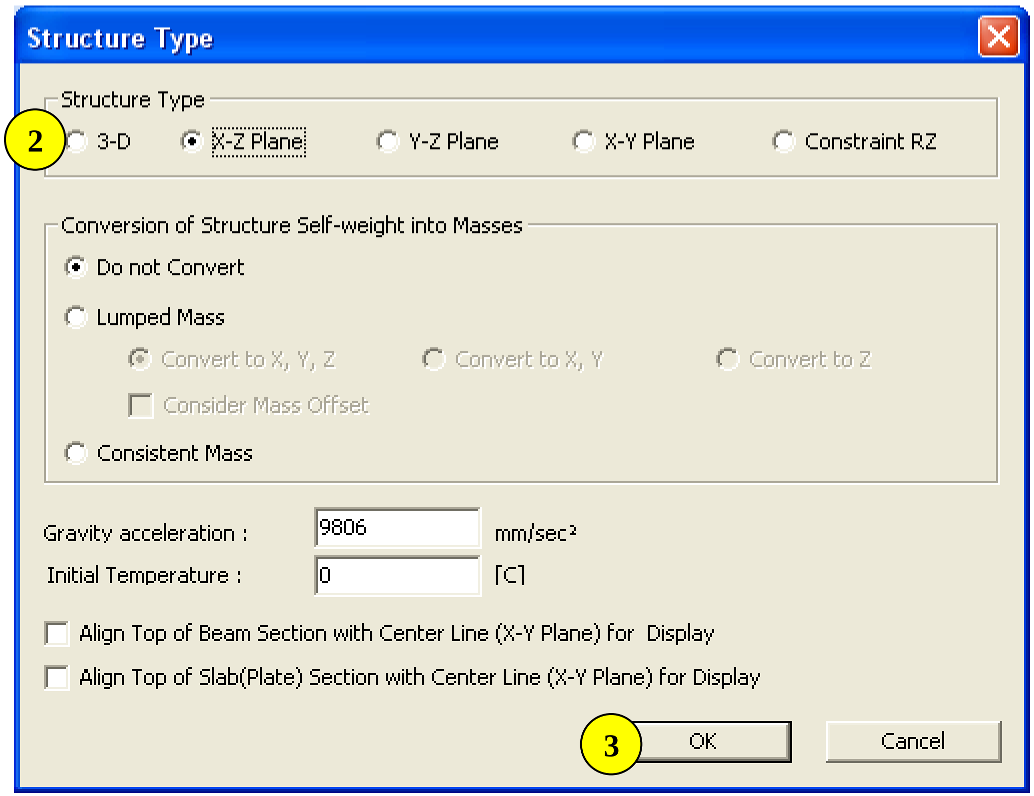
Step 4. Define Structure Type

1. Model > Structure Type

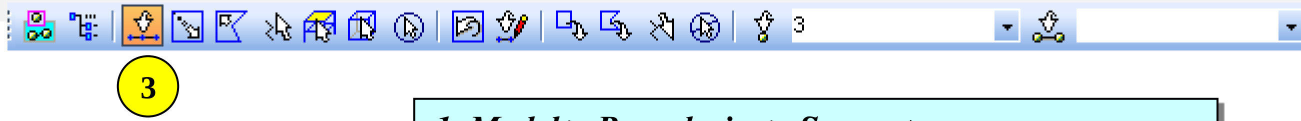
2. Structure Type>X-Z Plane

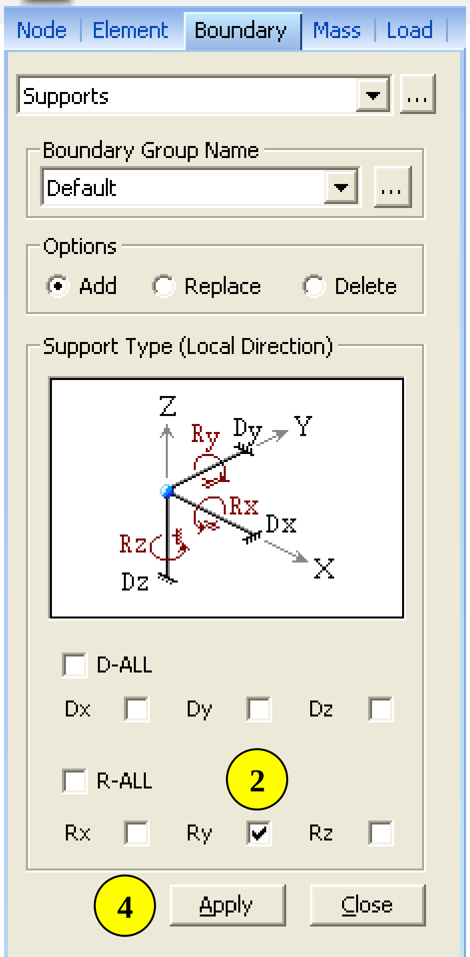
3. Click [OK] Button.

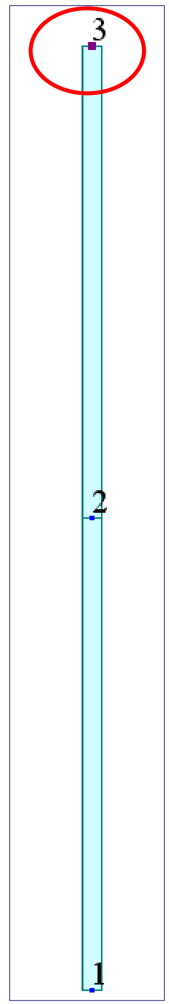
2. 2D analysis is carried out in X-Z plane.



Step 5. Create boundary condition







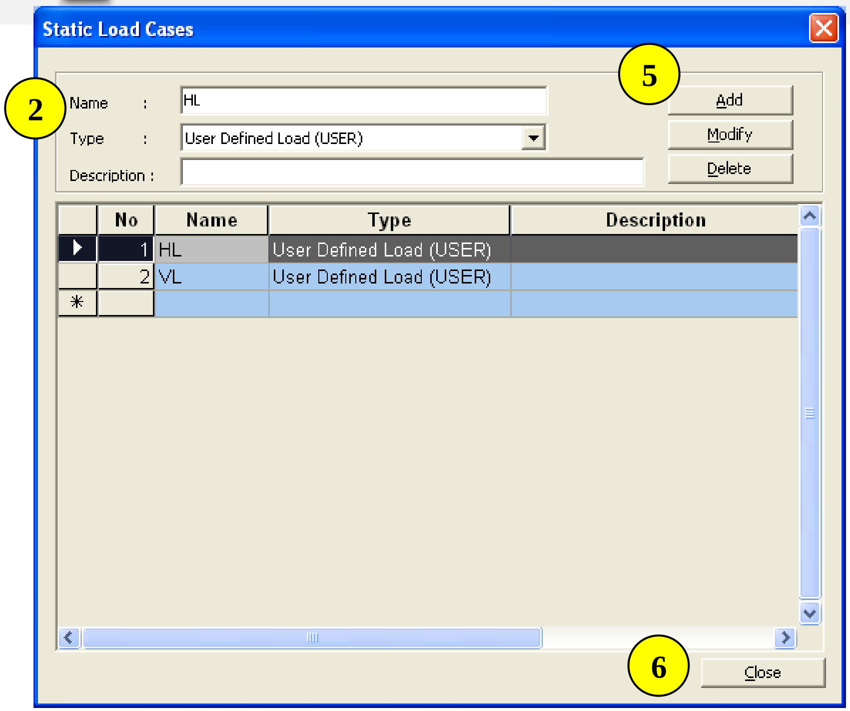
1. Model > Boundaries > Supports…

2. Check on “Ry”.

3. Select node No. 3.

4. Click [Apply] *Button.*

Step 6. Create static load cases.



1. Load > Static Load Cases…

2. Name: HL, Type: User Defined Load (User)

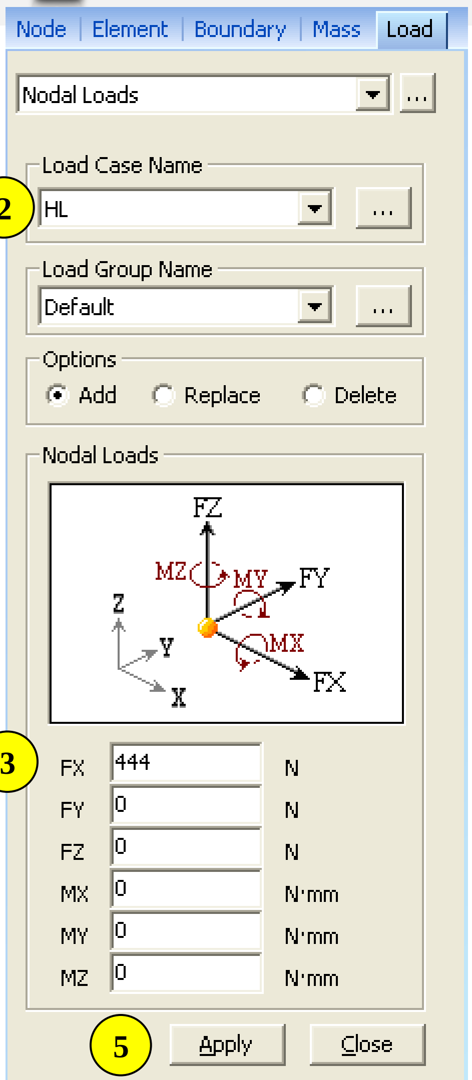
3. Click [Add] *Button.*

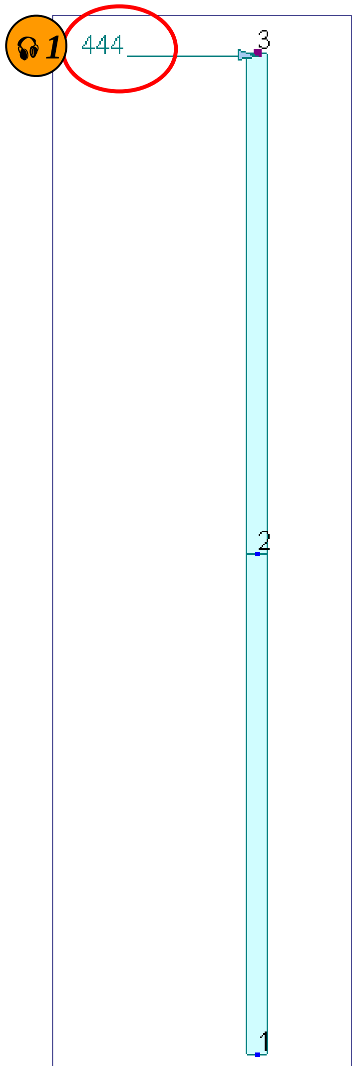
4. Name: VL, Type: User Defined Load (User)

5. Click [Add] *Button.*

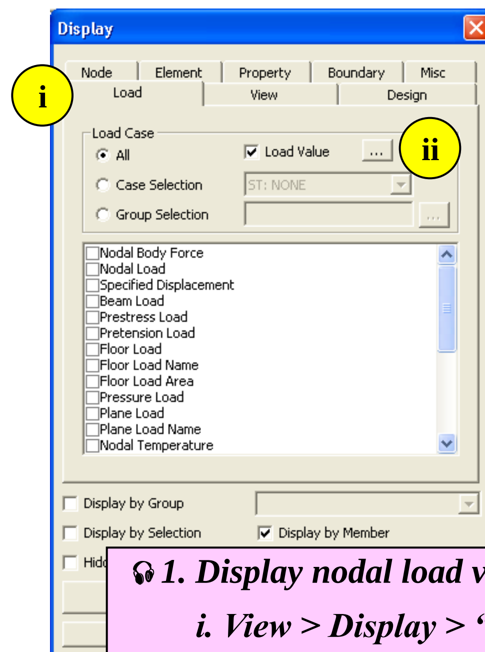
6. Click [Close] *Button.*

Step 7. Create nodal loads (1).





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

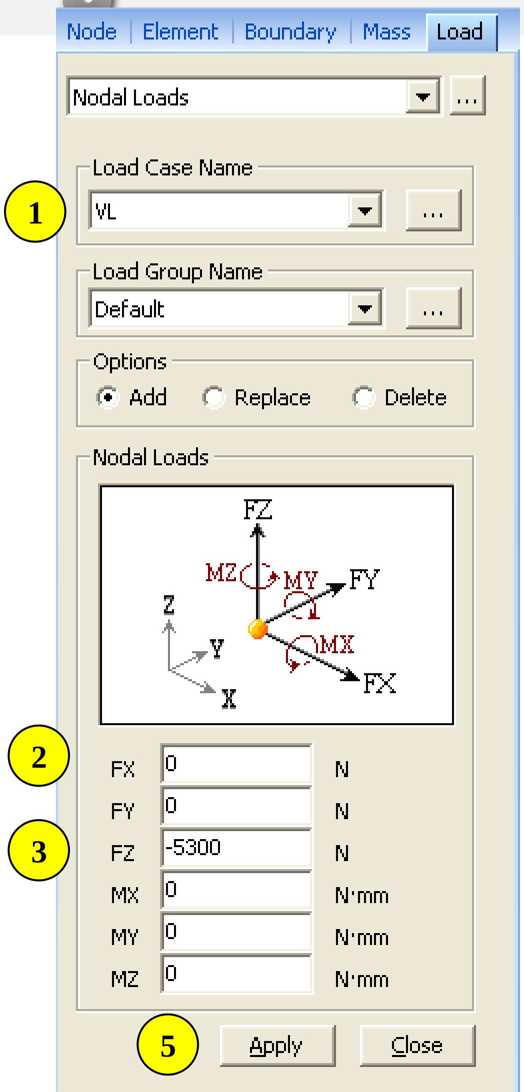


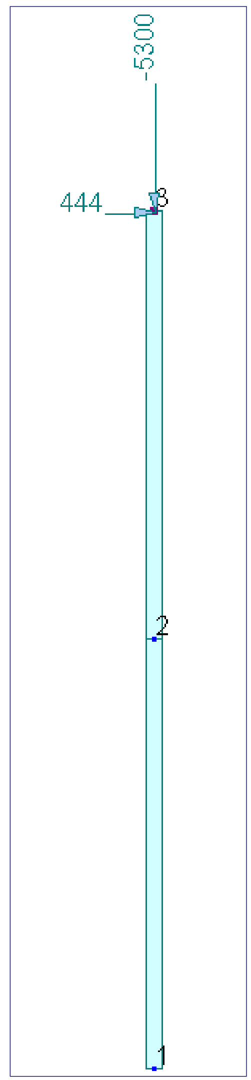
1. Display nodal load value.

i. View > Display > "Load" tab

ii. Check on "Load Value".

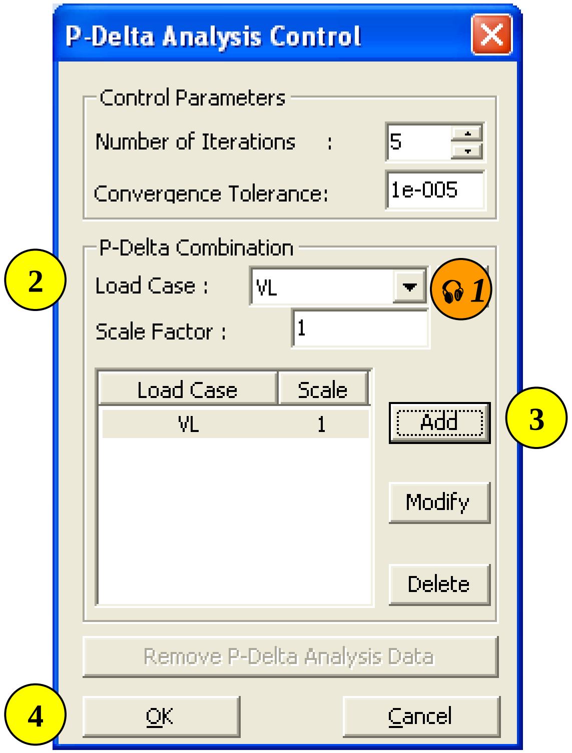
Step 7. Create nodal loads (2).





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



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…

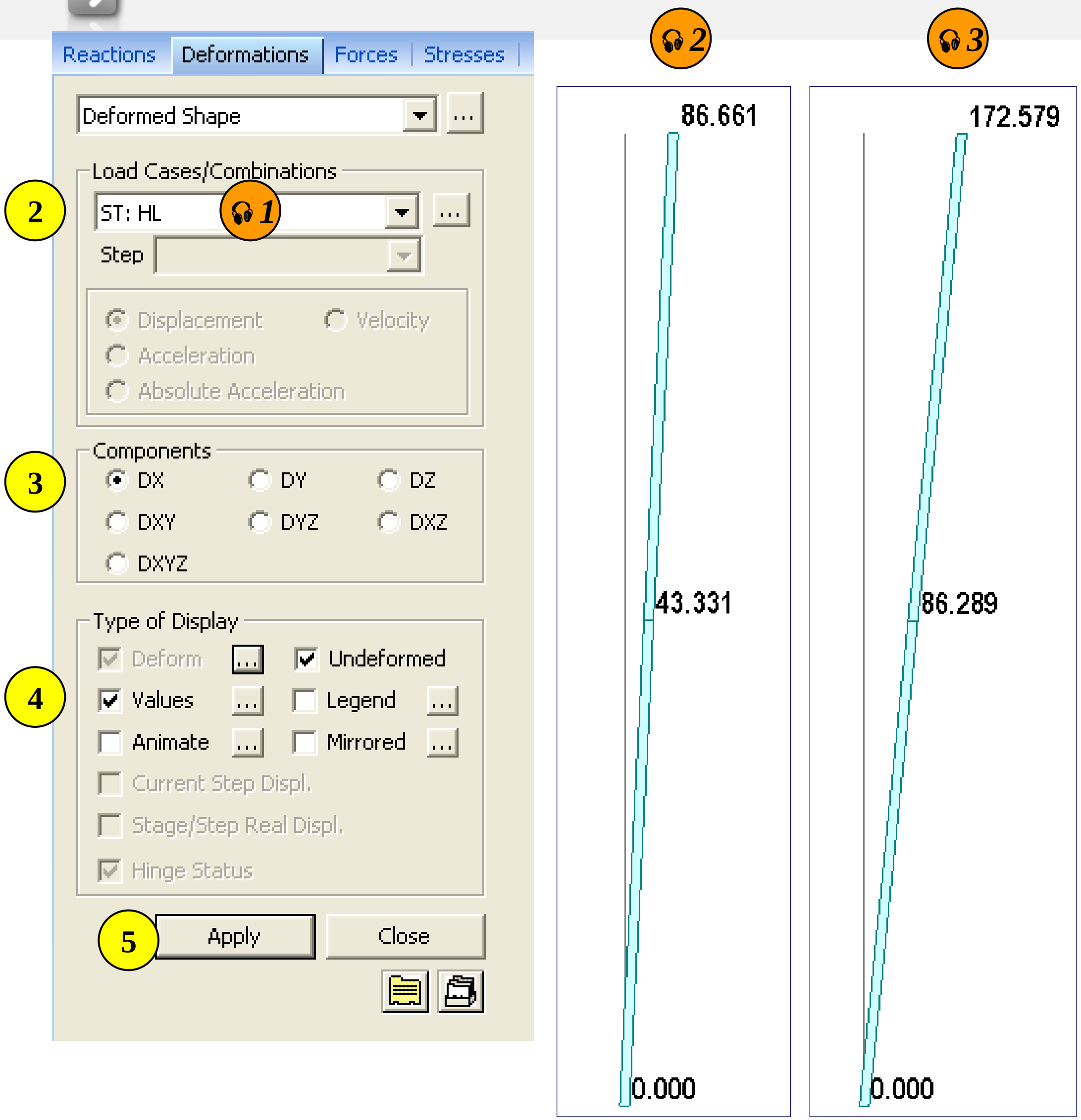
Model View | Supports | Nodal Loads

	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



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.