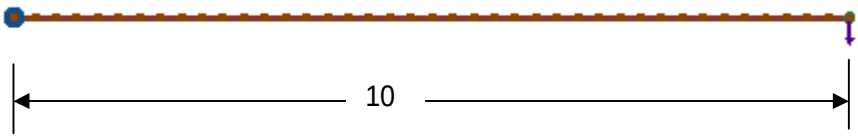
TRANSIENT RESPONSE ANALYSIS OF A CANTILEVER BEAM



Analysis Type : Transient Response

Modulus of elasticity, E =12000 MPa

Poisson's Ratio : 0.2

PROCEDURE

STEP

1. Create two points at (0, 0, 0) and (10, 0,0)

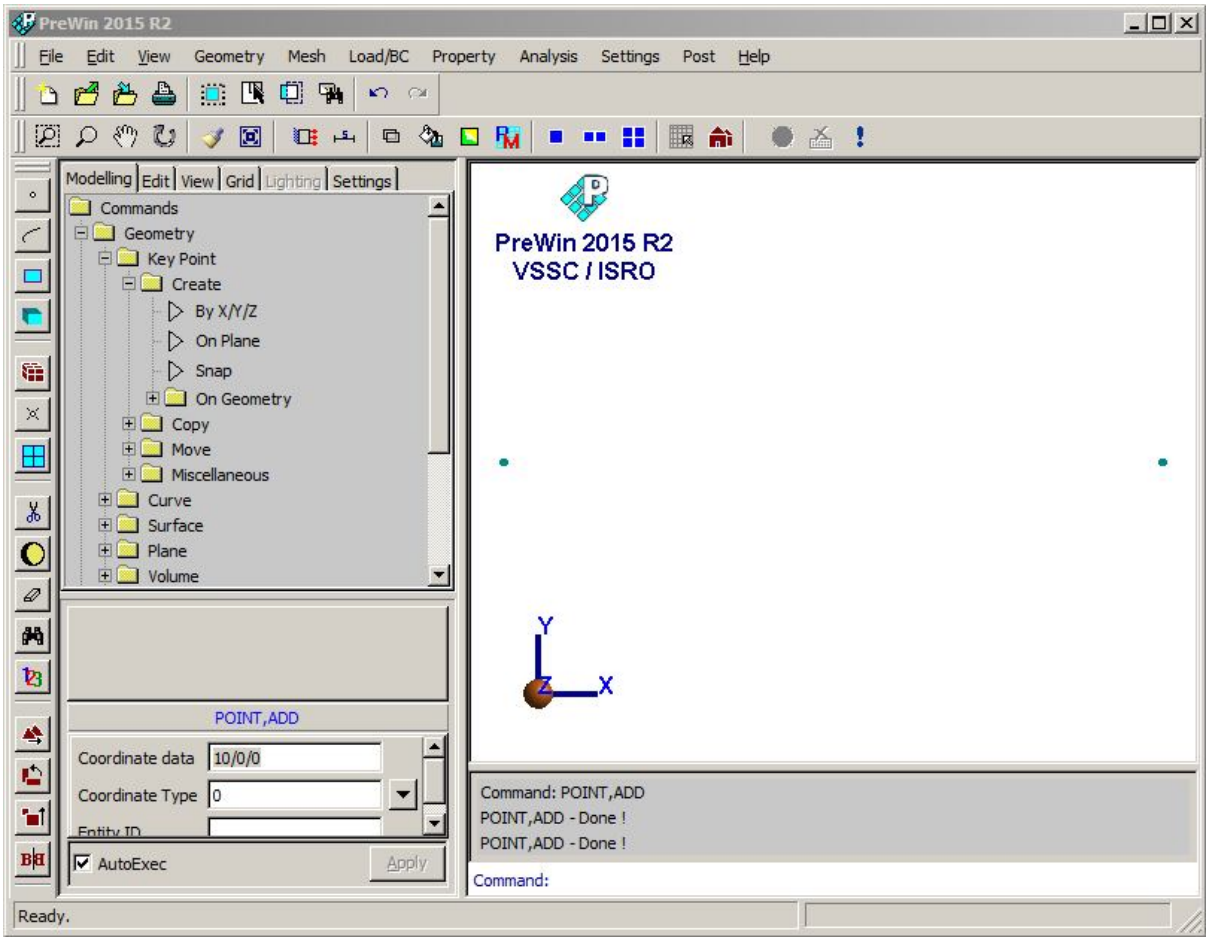
Commands : POINT, ADD

Menu : Geometry → Key point → Create → By X/Y/Z

Parameters : (To be filled by the user)

Coordinate Data	10/0/0
Coordinate Type	0
Entity ID	

At the end of the above operations, your screen should look like this



2. Create a line

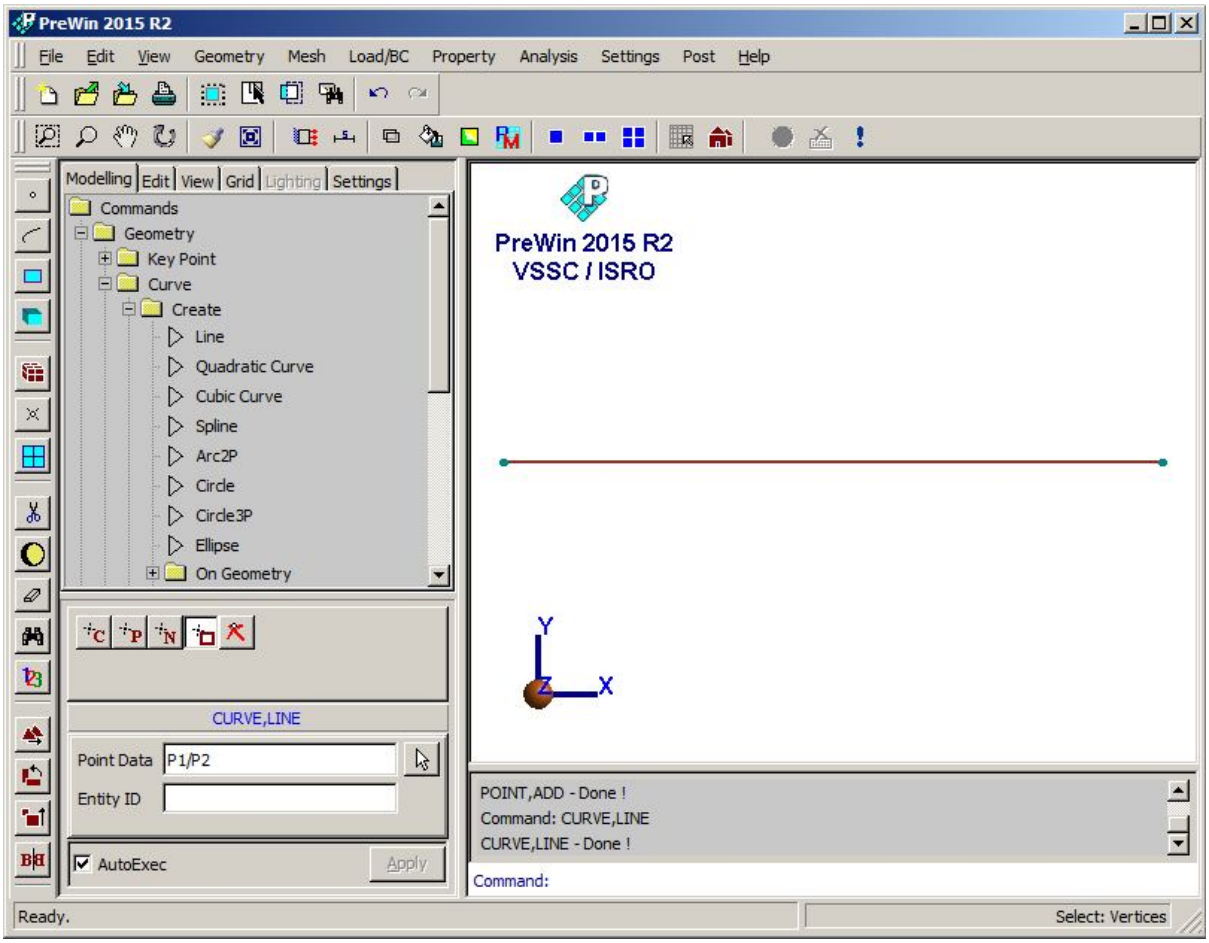
Commands : CURVE, LINE

Menu : Geometry → Curve → Create →Line

Parameters :

Point Data	Use Mouse to pick the points
Entity ID	1

At the end of the above operations, your screen should look like this.



3. Meshing using beam elements

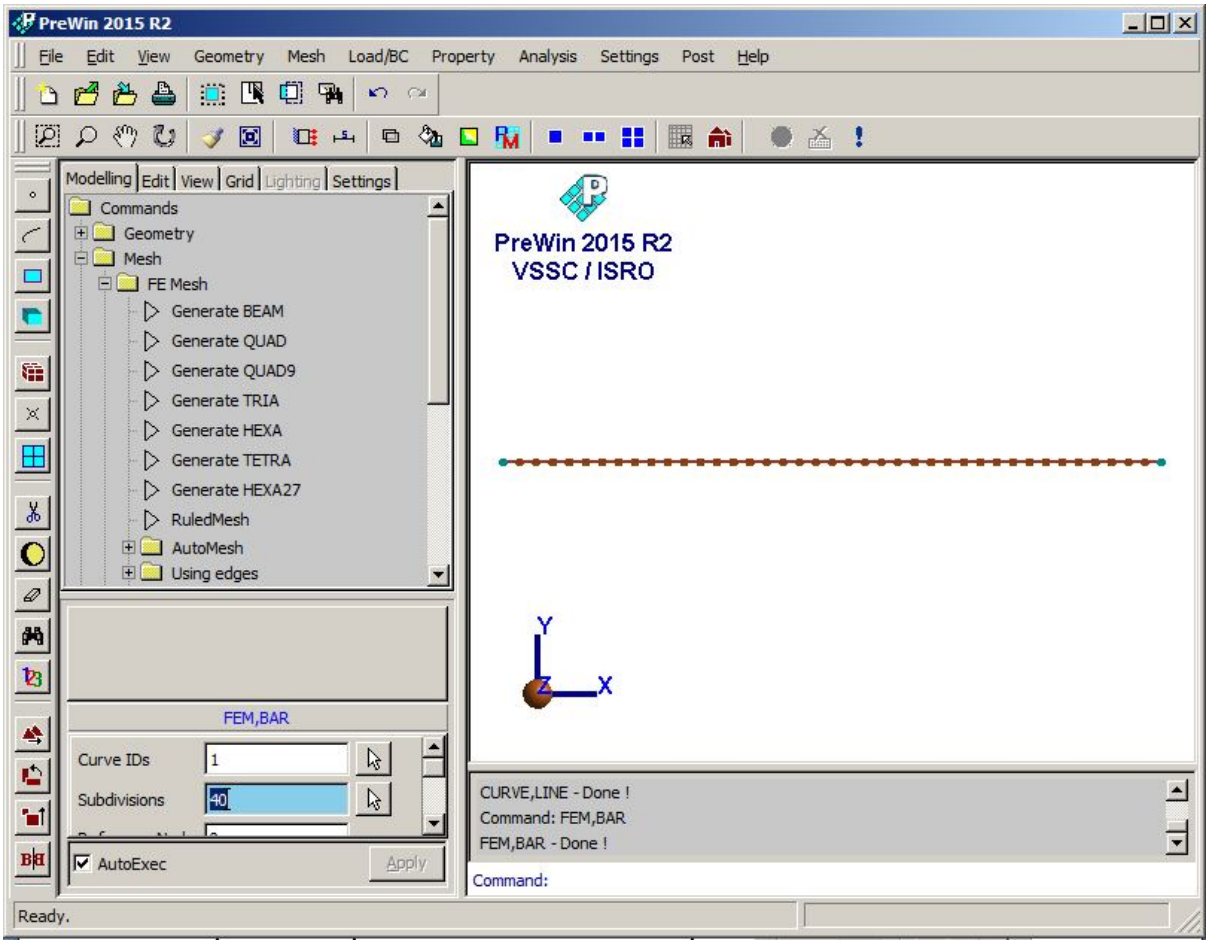
Commands : FEM, BAR

Menu : Mesh → FE Mesh → Generate BEAM

Parameters :

Curve IDs	1
Subdivisions	40
Reference Node	0
Element order	Linear
Bias Factors	1

At the end of the above operations, your screen should look like this.



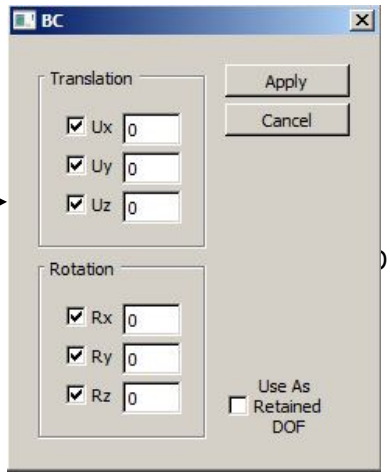
4. Specify displacement boundary conditions

Commands : BC,ADD

Menu : Load/BC → Displacement BC → Add

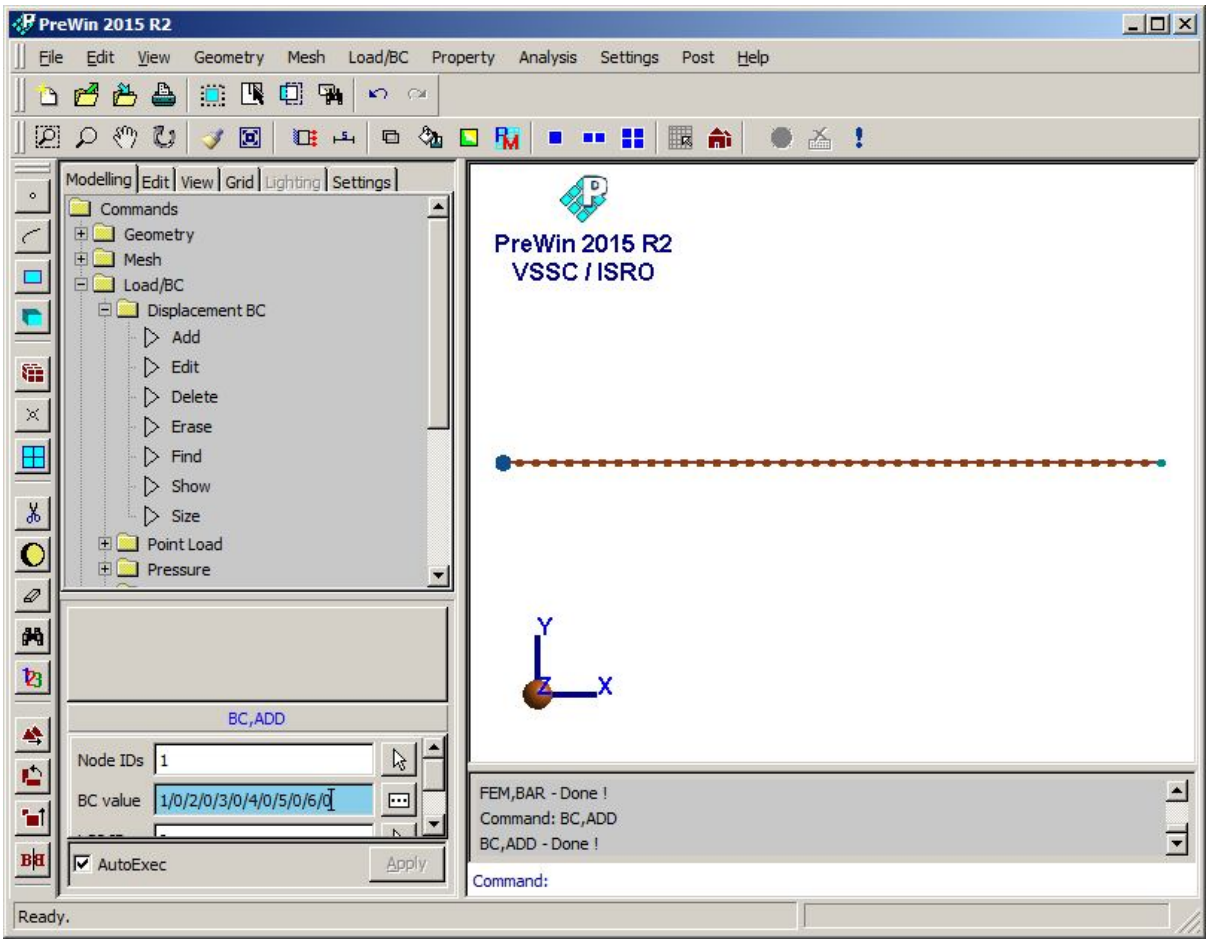
Parameters :

Node IDs	Select the node at the left end
BC Value	1/0/2/0/3/0/4/0/5/0/6/0



LCS ID	0
Set ID	1

At the end of the above operations, your screen should look like this.



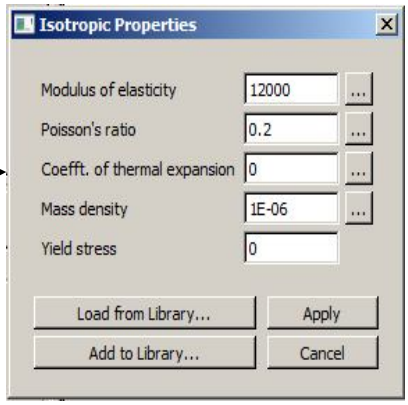
5. Specify material properties

Command : MATERIAL, ISO

Menu : Property → Material → Isotropic → Add

Parameters :

Element IDs	All
Material Data	12000/0.2/0/1E-06/0
Material ID	1



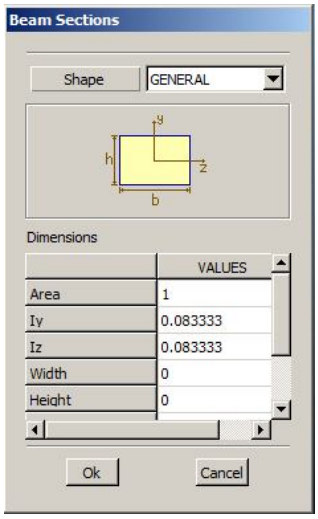
6. Specify Beam Properties

Command : BEAMSECTION, ADD

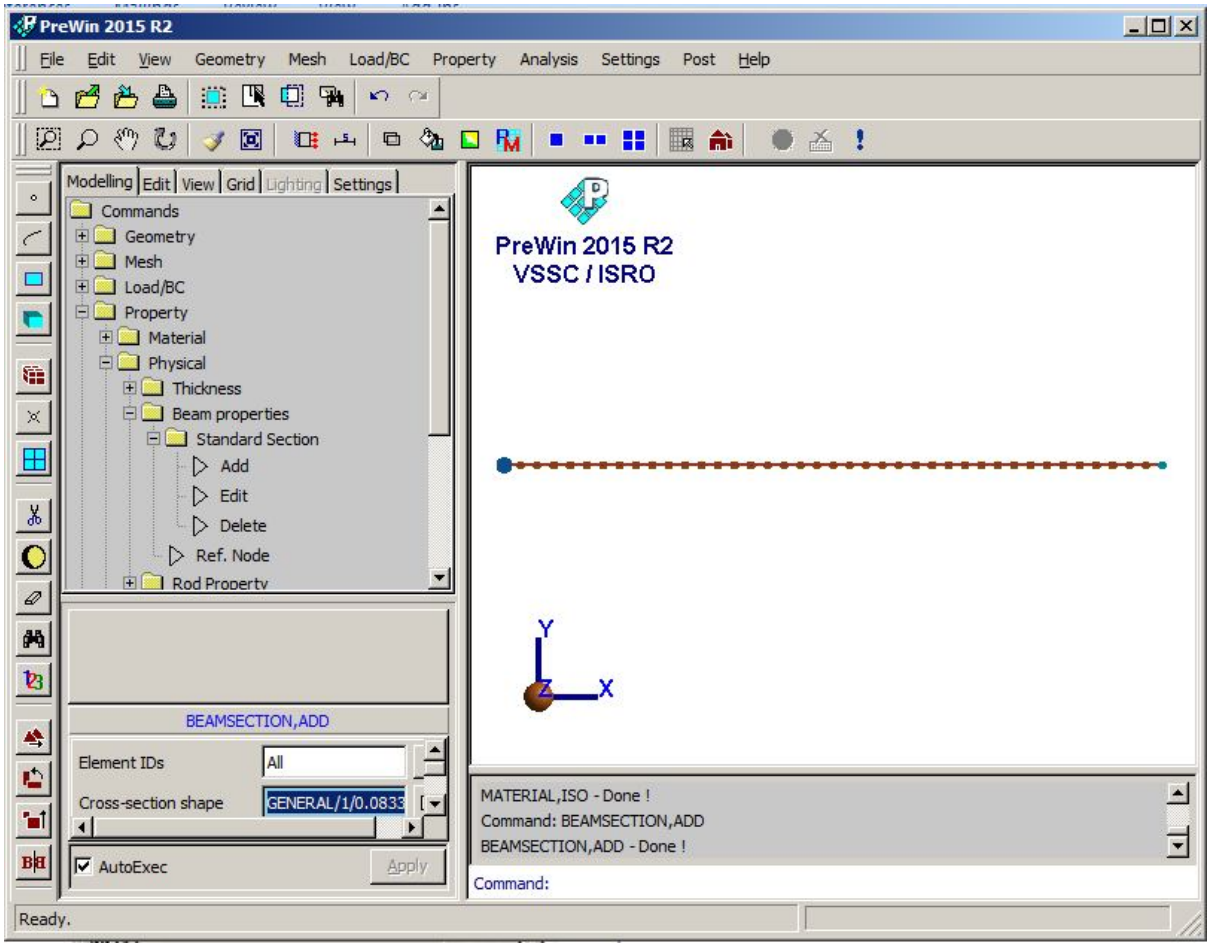
Menu : Property → Physical → Beam Properties →Standard Section →Add

Parameters :

Element IDs	All
Cross section Shape	GENERAL
Angle about axis	0
Node offsets (yoff/zoff)	0/0



At the end of the above operations, your screen should look like this.



7. Specify point load

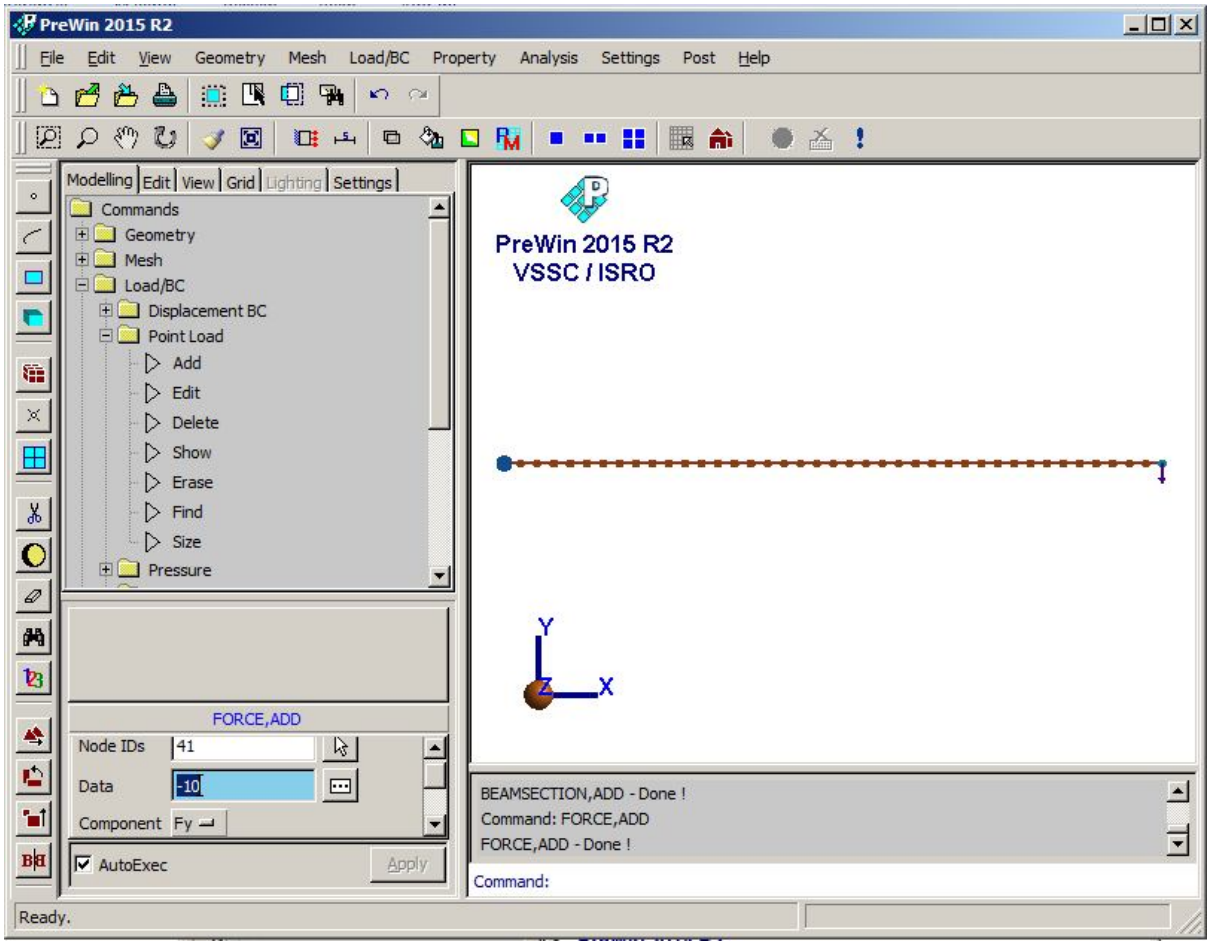
Command : FORCE, ADD

Menu : Load/BC → Point Load → Add

Parameters :

Node IDs	41
Data	-10
Component	Fy
LCS ID	0
Set ID	1

At the end of the above operations, your screen should look like this.



8. Set the analysis type

Command : ANTYPE, SET

Menu : Analysis → Analysis Type

Parameters :

Analysis Type	Transient Response

9. Set the analysis options

Command : ANOPTION, SET

Menu : Analysis → Analysis Options

Parameters :

Linear Solver	Multi Frontal
Eigen Solver	Lanczos

10. Specify control data for transient analysis

Command : TRANSGEN, ADD

Menu : Analysis → Transient Response → General → Add

Parameters :

No. of eigen values	10
No. of time steps	5
Start time for analysis	0
Time step for analysis	0.000135
Mass computation	Consistent

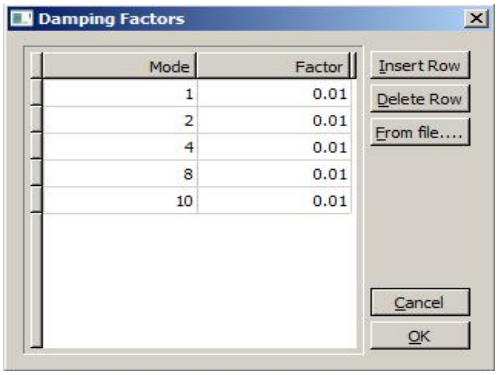
11. Specify data for explicit damping

Command : EXPLICITDAMP, ADD

Menu : Analysis → Transient Response → Damping → Add

Parameters :

Damping factors	



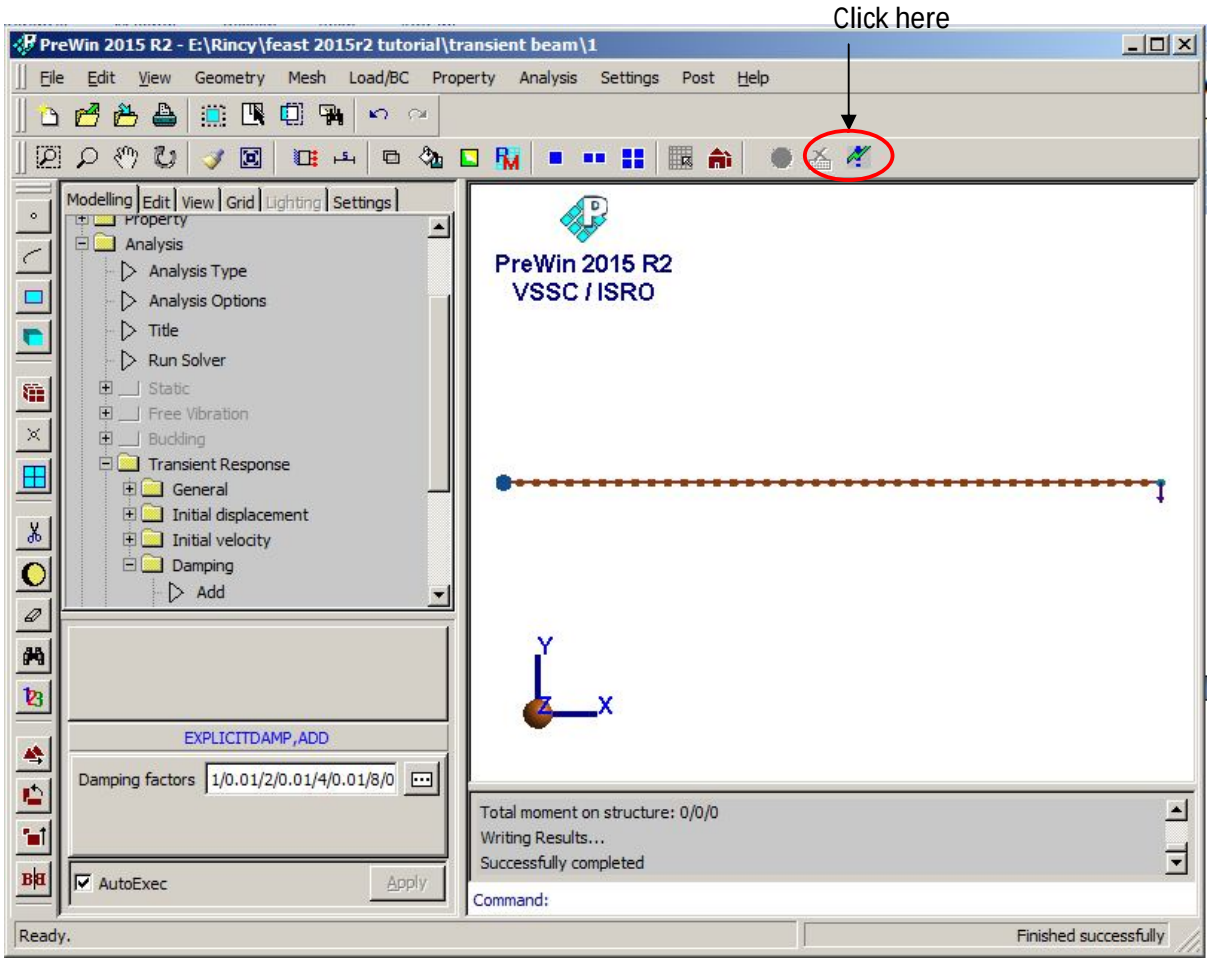
12. Save the project model

Menu : File → Save

13. Submit the job into FEAST

Menu : Analysis →Run solver

At the end of the above operations, your screen should look like this.



14. Perform post processing

i) Graph plots for displacement / velocity / acceleration

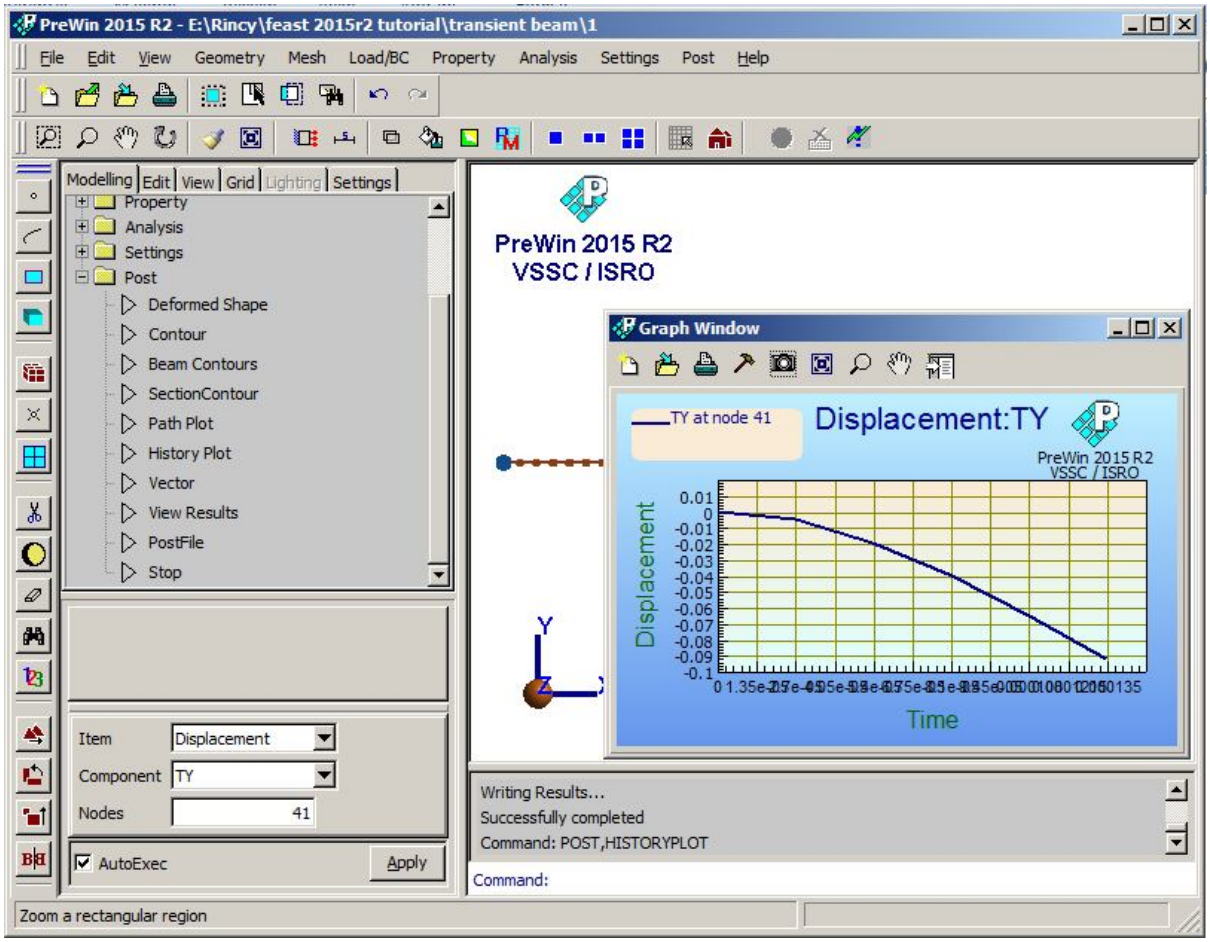
Command : POST, HISTORYPLOT

Menu : Post → History Plot

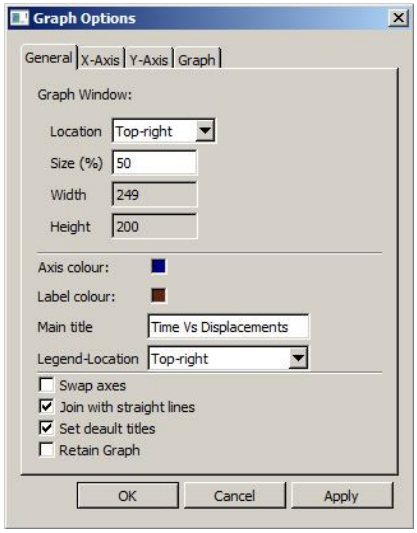
Parameters :

Item	Displacement
Component	TY
Nodes	41

At the end of the above operations a graph as shown appears in the viewport



* If we want to modify graph details then click Graph Options and then the following dialog box appears.



ii) Show resultant displacement / velocity / acceleration

Command : POST, VIEWRESULTS

Menu : Post → View Results

Parameters :

Item	Displacement
Nodes	41
Time steps	2.7E-05
LCS	Global Cartesian