STATIC ANALYSIS OF A SPRING SYSTEM

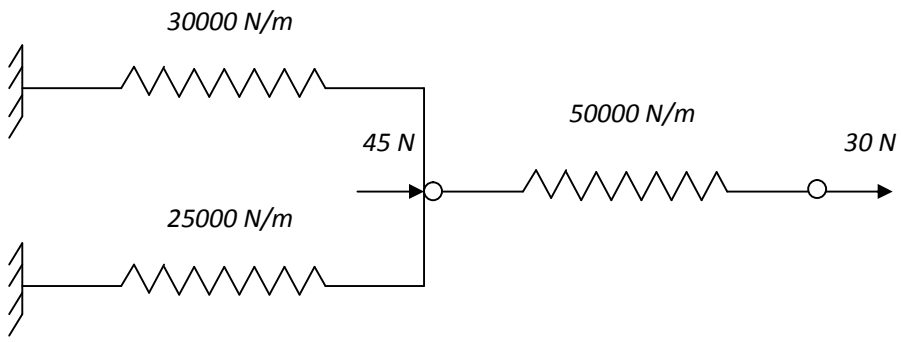


Figure 1

Determine the displacements at nodes of the spring system shown in Figure1

PROCEDURE

STEP

1. Create four points at (0, 1, 0), (0, -1, 0), (2, 0, 0) and (4, 0, 0)

Commands : POINT, ADD

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

Parameters : (To be filled by the user)

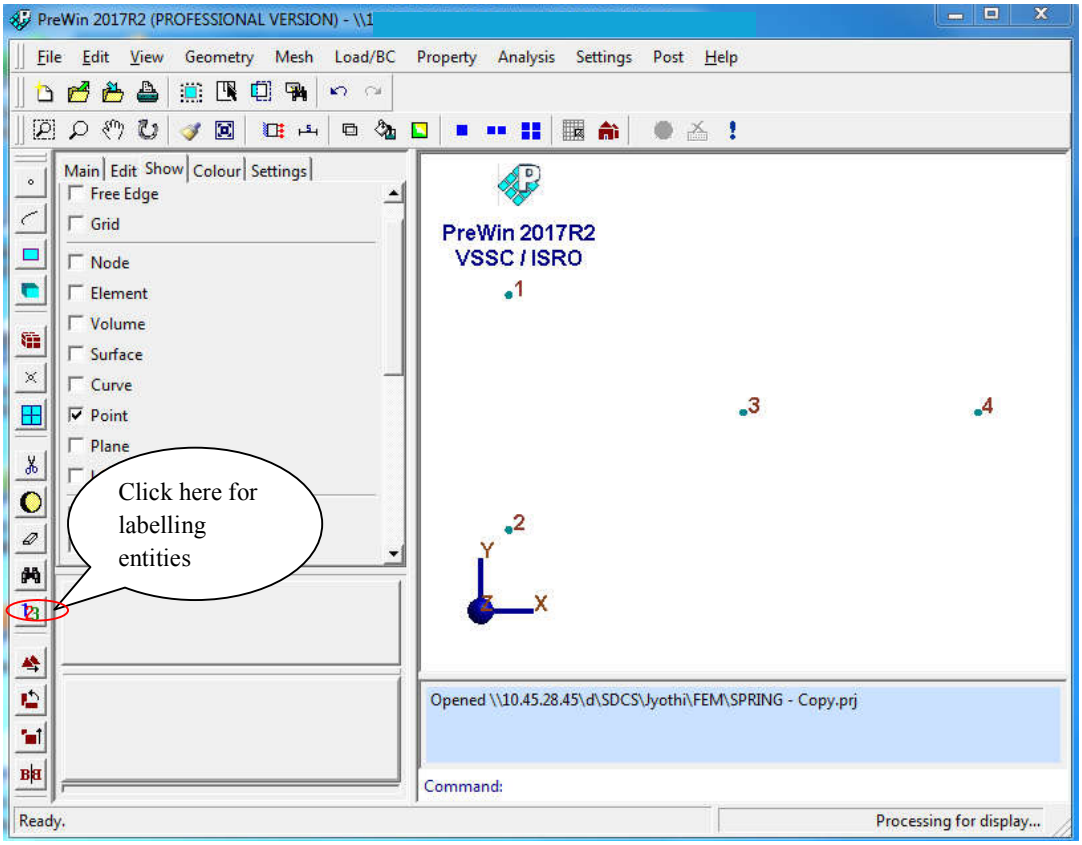
Coordinate Data	0/0/0
Coordinate Type	0
Entity ID	

Similarly create the other points.

Note:

Click "***Apply***" button or press ***cntrl+enter*** key after completing each step. "Done" message appears on message box for every step executed successfully.

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



2. Create nodes

Commands : NODE, SNAP

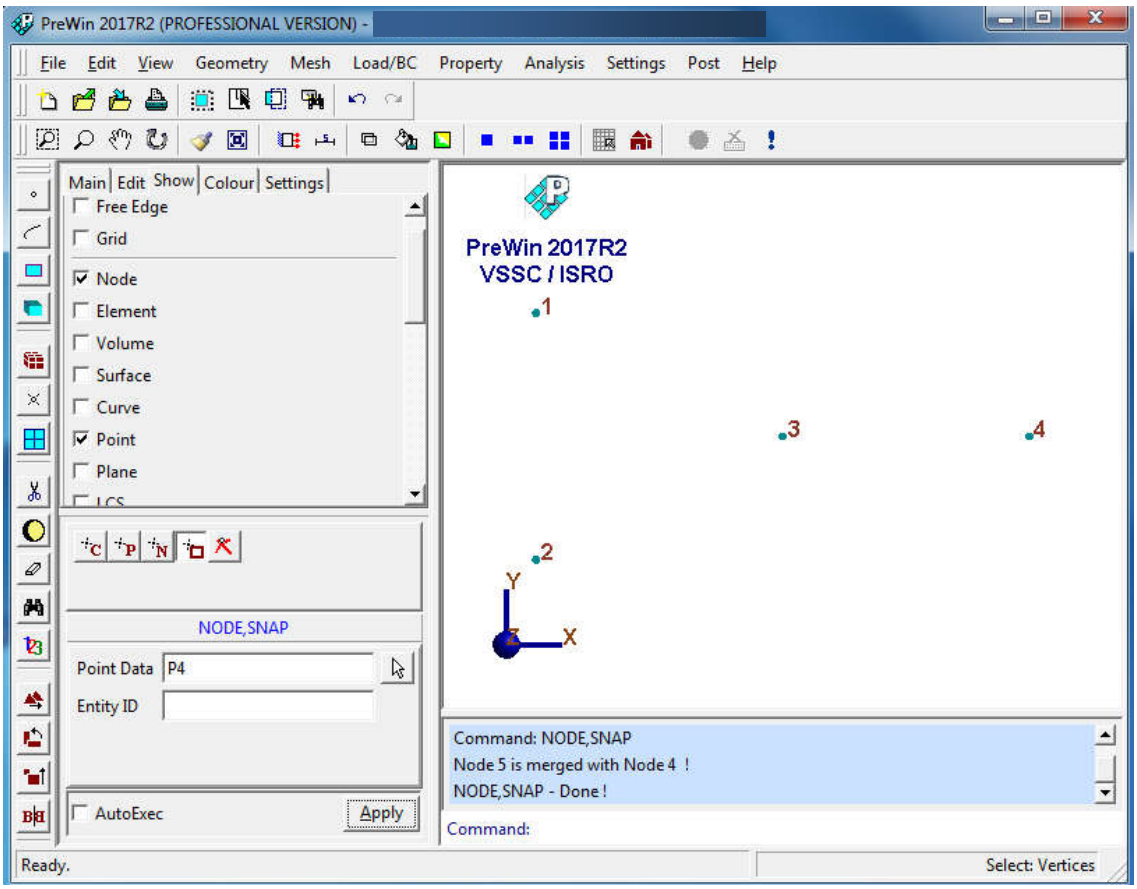
Menu : Mesh → Node → Create →Snap

Parameters :

Point Data	P1
Entity ID	1

Similarly create the other nodes.

At the end of the operation/s your screen should look like this.



3. Create spring elements

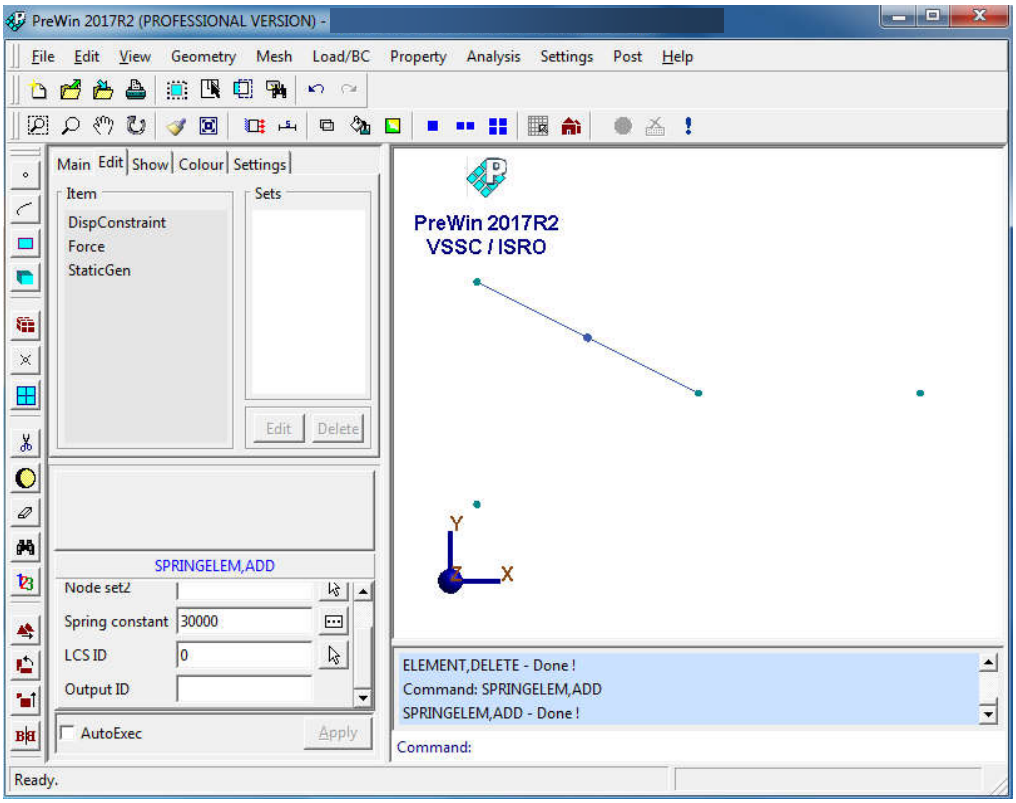
Commands : SPRINGELEM, ADD

Menu : Mesh → Element → Create → Spring →Add

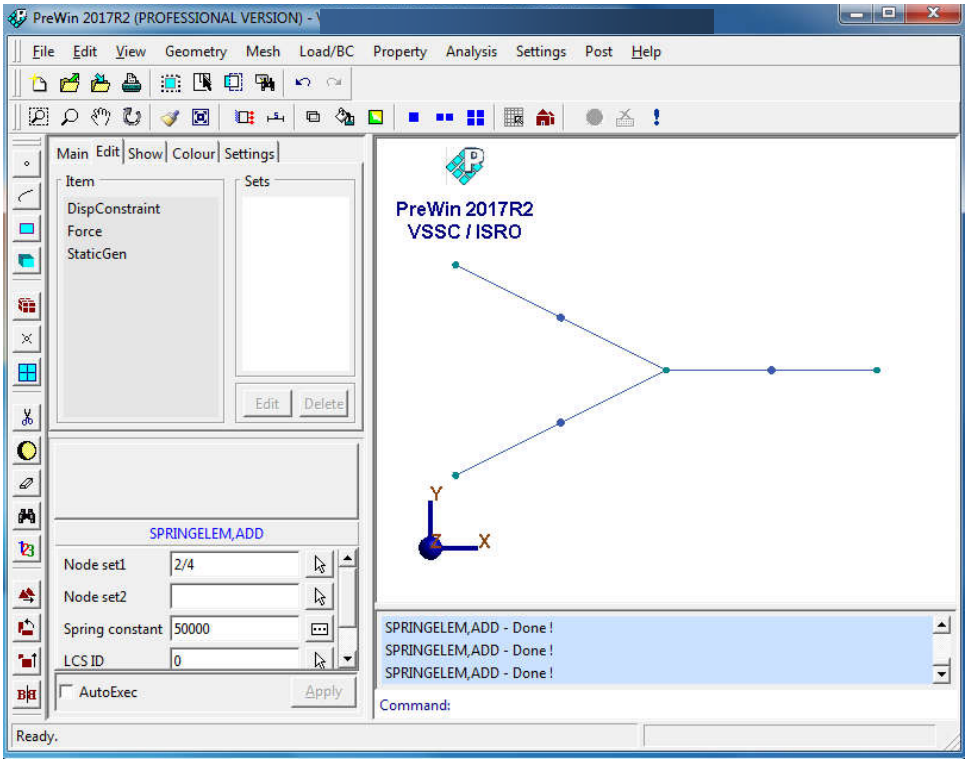
Parameters :

Node Set 1	1/2
Node Set 2	
Spring Constant	30000
LCS ID	0
Output ID	1

At the end of the operation/s your screen should look like this.



Similarly create the other two spring elements.



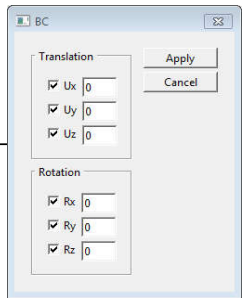
4. Specify displacement boundary conditions

Commands : BC,ADD

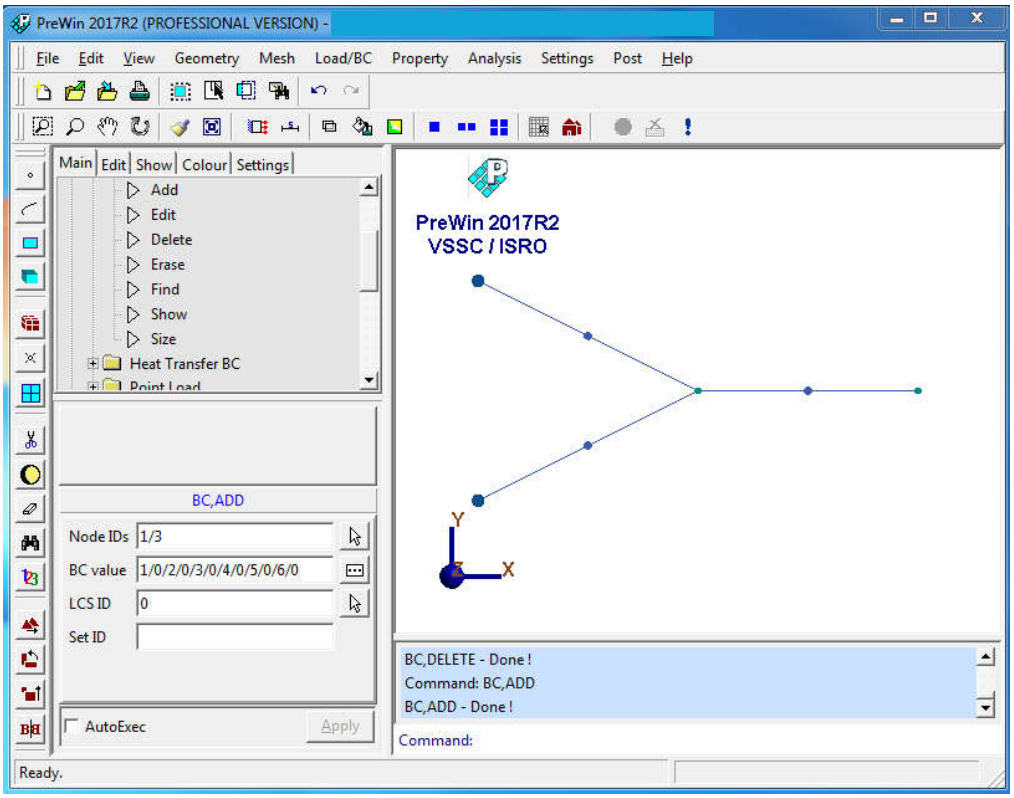
Menu : Load/BC → Displacement BC → Add

Parameters :

Node IDs	1/3
BC Value	1/0/2/0/3/0/4/0/5/0/6/0
LCS ID	0



At the end of the operation/s your screen should look like this.



5. Specify point load

Command : FORCE, ADD

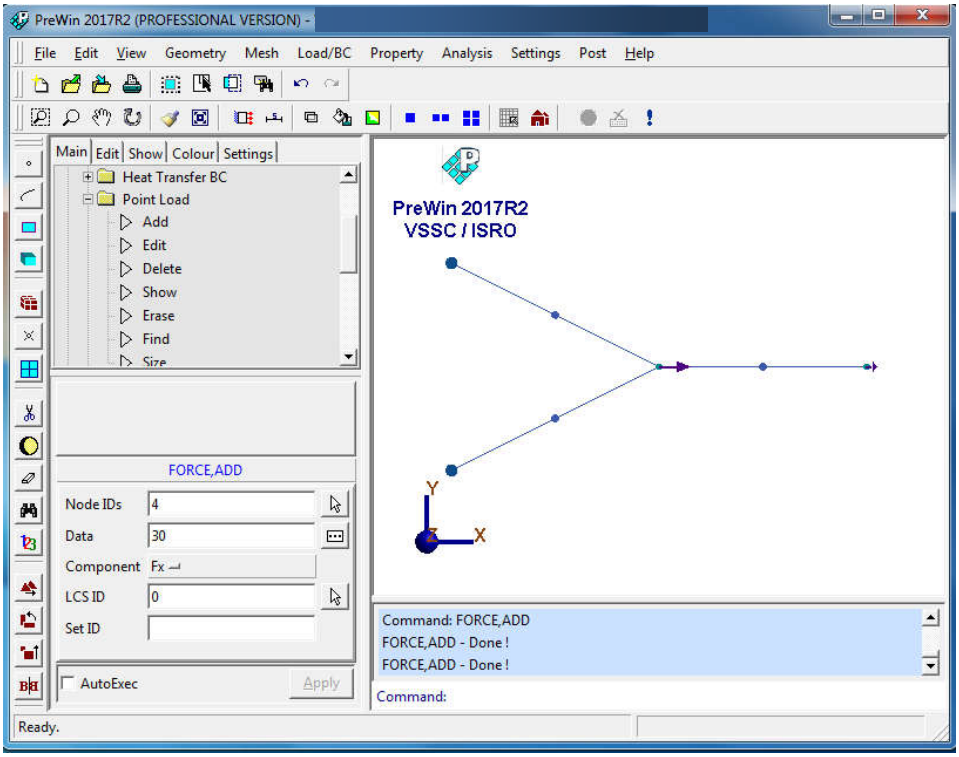
Menu : Load/BC → Point Load → Add

Parameters :

Node IDs	2
Data	45
Component	F_x
LCS ID	0
Set ID	

Similarly create the other load at node 4

At the end of the operation/s your screen should look like this.



6. Set the analysis type

Command : ANTYPE, SET

Menu : Analysis → Analysis Type

Parameters :

Analysis Type	Static

7. Set the analysis options

Command : ANOPTION, SET

Menu : Analysis → Analysis Options

Parameters :

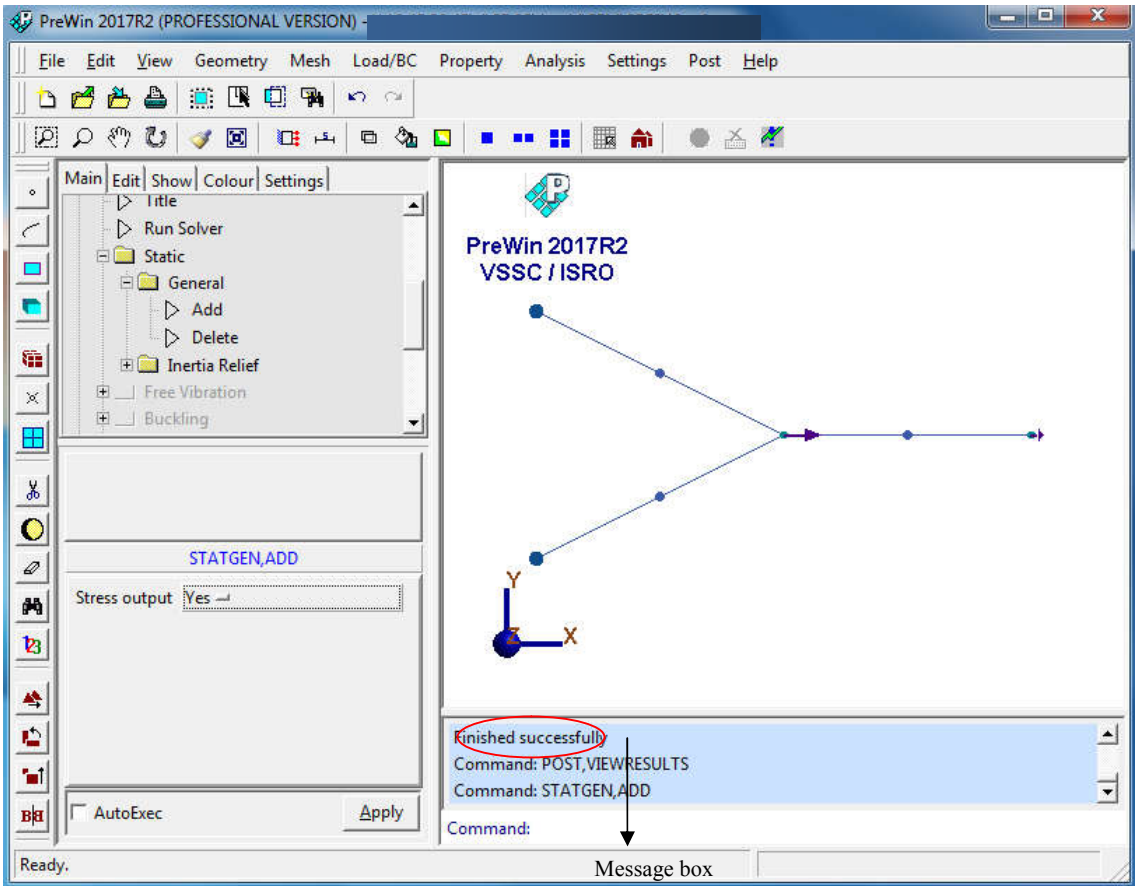
Linear Solver	Multi Frontal

8. Save the project model

Menu : File → Save

9. Submit the job into FEAST

Menu : Analysis →Run solver



Note:

"Finished successfully" message appears on message box after executing is completed.

10. Perform post processing

i) Deformed shape

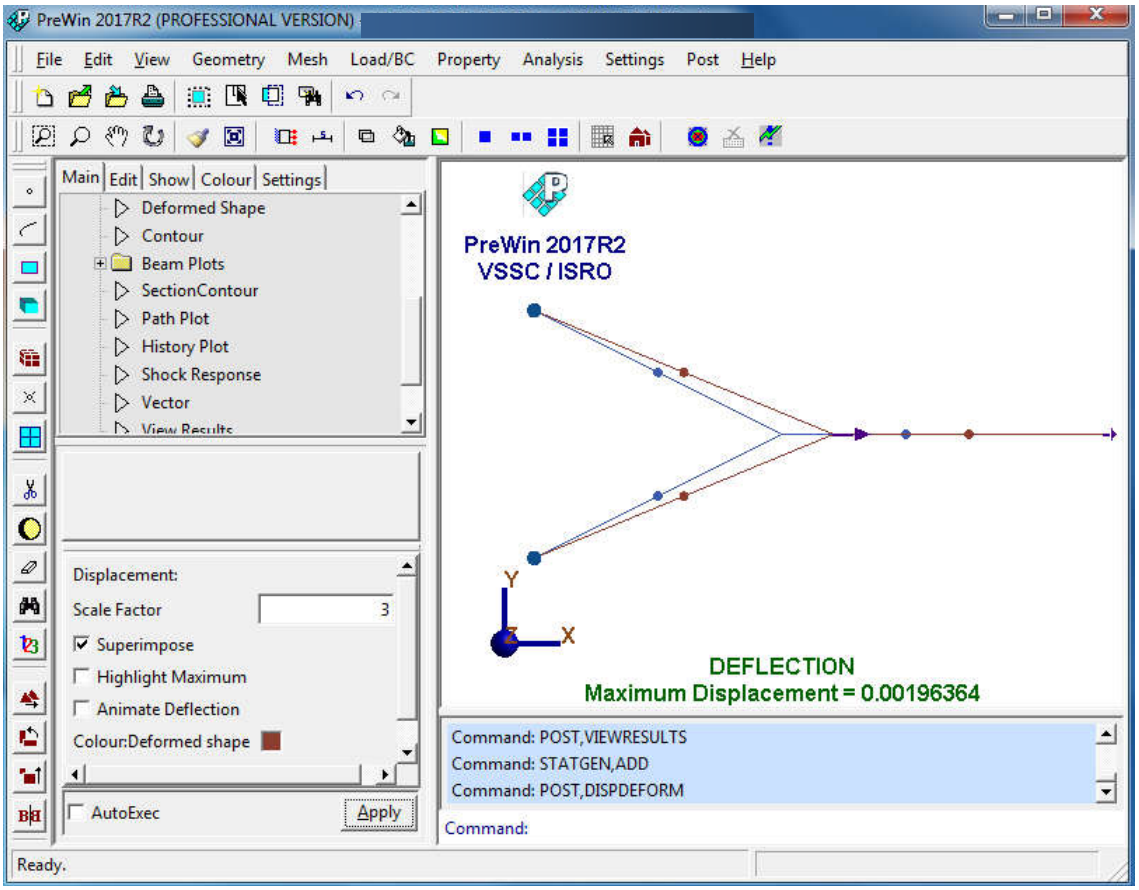
Command : POST, DISPDEFORM

Menu : Post → Deformed shape

Parameters :

Scale factor	1

At the end of the operation/s your screen should look like this.



STATIC ANALYSIS OF A PLANAR TRUSS

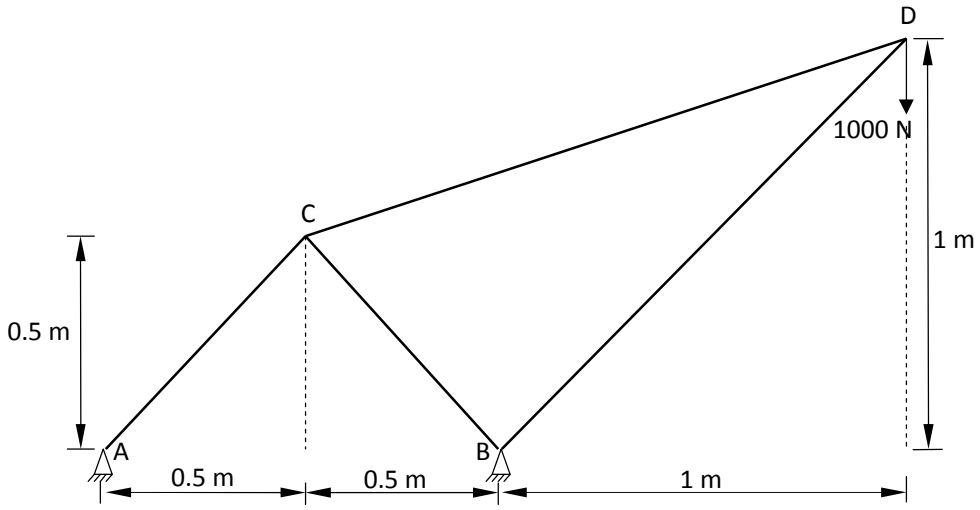


Figure 1

Consider a planar truss loaded as shown in the Figure 1. Find the maximum displacement and deformed shape of the truss.

Material property: ***E*** *= 2.1E+11 Pa, ν = 0.3*

Point load, ***P*** *= -1000 N*

Cross sectional area of AC and BC = $0.0002m^2$

Cross sectional area of CD and BD =$0.0001m^2$

PROCEDURE

STEP

1. Create the points A, B, C and D at (0,0,0), (1,0,0), (0.5,0.5,0), (2,1,0) respectively.

Commands	:	POINT, ADD
Menu	:	Geometry → Key point → Create → By X/Y/Z
Parameters	:	(To be filled by the user)

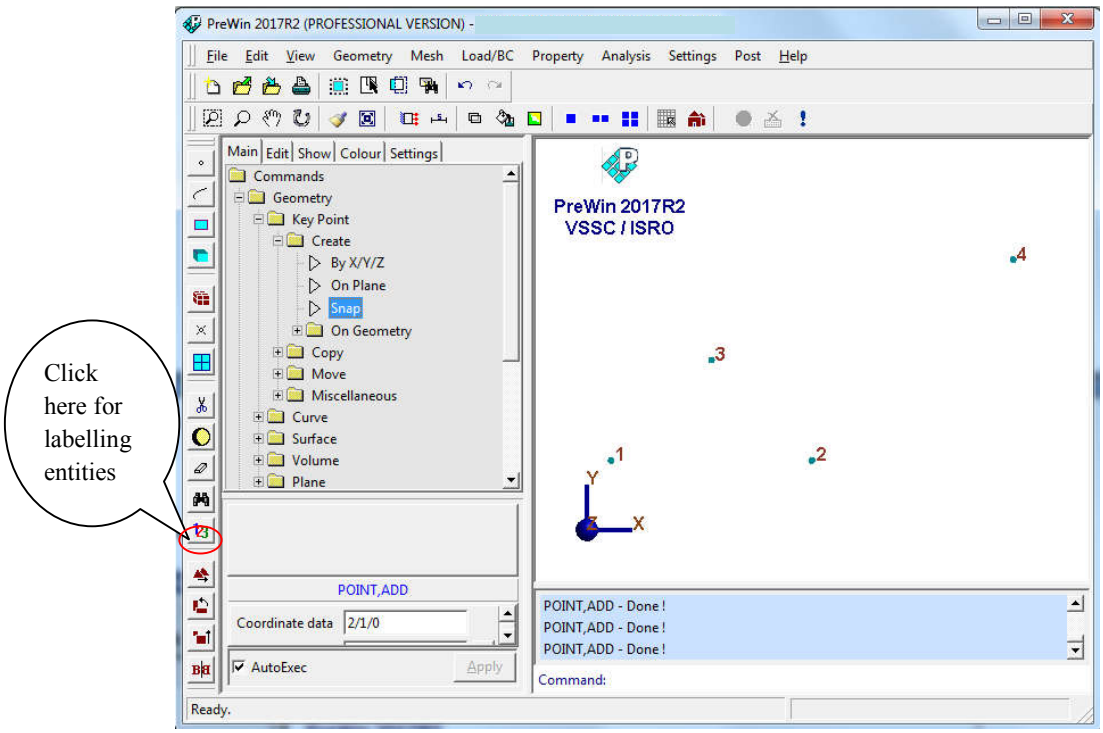
Coordinate Data	0/0/0
Coordinate Type	0
Entity ID	

Similarly create the other points.

Note:

Click "***Apply***" button or press ***cntrl+enter*** key after completing each step. "Done" message appears on message box for every step executed successfully.

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



2. Create lines AC, BC, CD and BD.

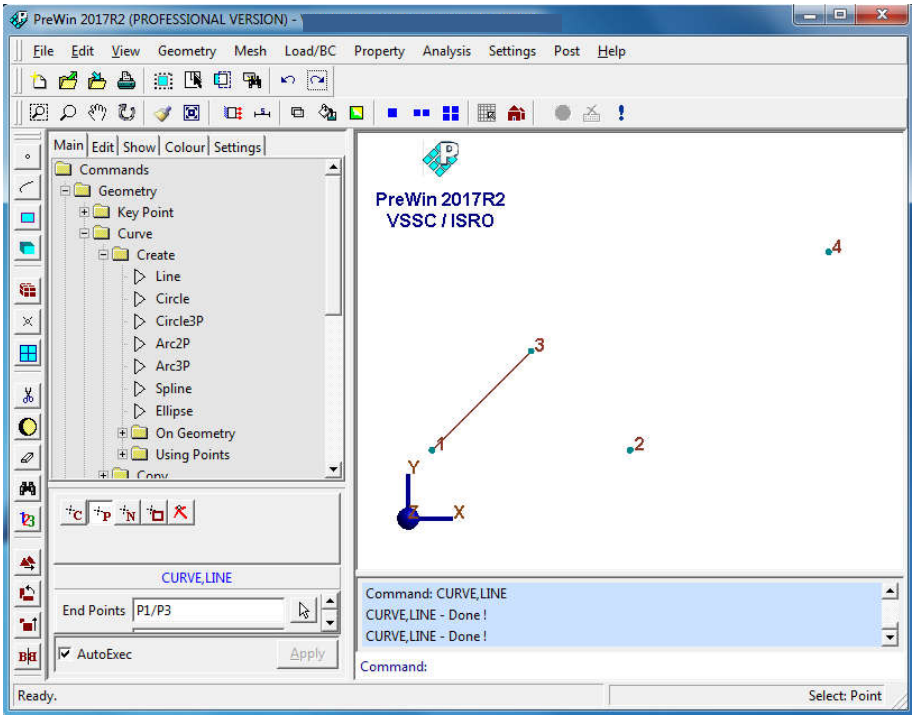
Commands : CURVE, LINE

Menu : Geometry → Curve → Create →Line

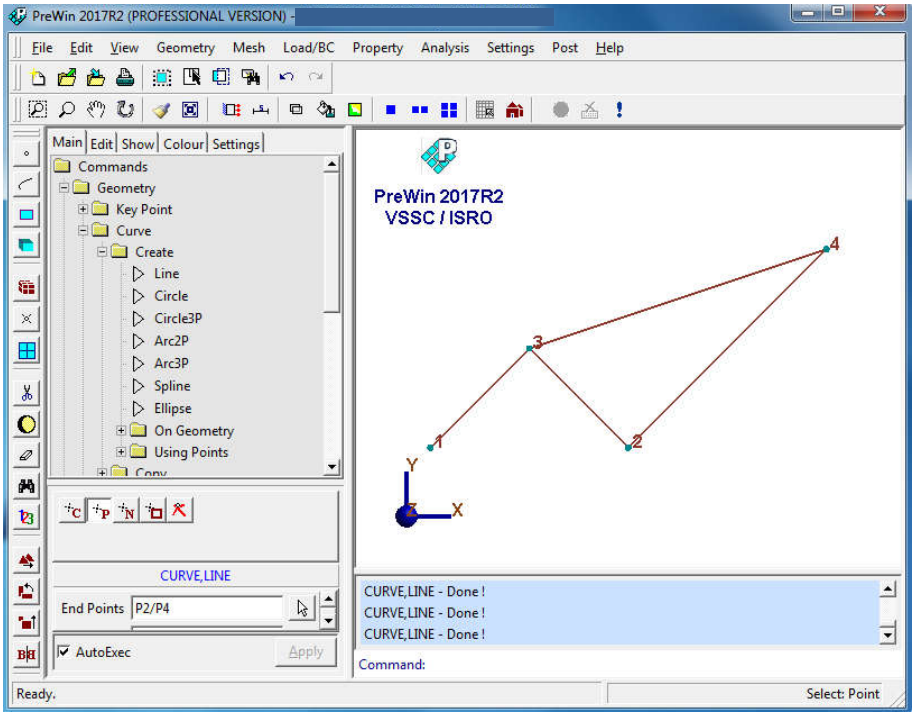
Parameters :

End points	Use Mouse to pick the points 1 & 3
Entity ID	1

This will create the line AC.



Similarly create the lines BC, CD and BD. At the end of the above operations, your screen should look like this.



3. Meshing using beam elements

For creating rod elements first meshing is done using beam elements and then modified as rod elements.

Commands : FEM, BAR

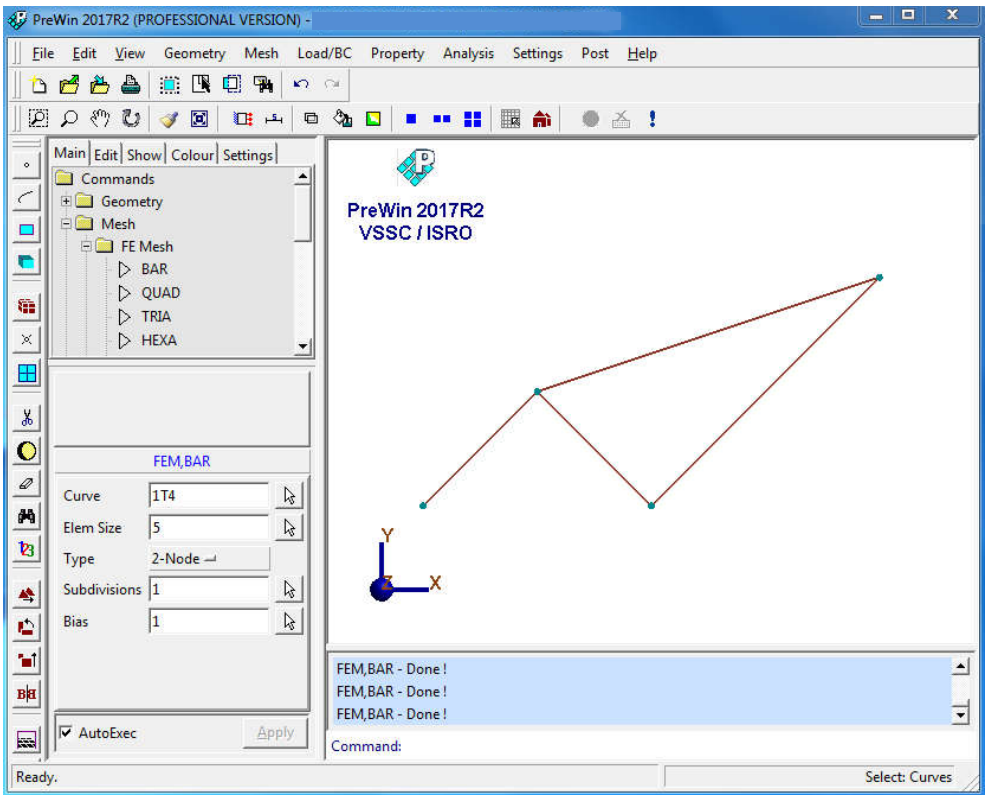
Menu : Mesh → FE Mesh → Bar

Parameters :

Curve	1T4
Element Size	5
Type	2-node
Subdivisions	1
Bias	1.0

Note:

Right click/left click mouse point to alter sub-divisions.



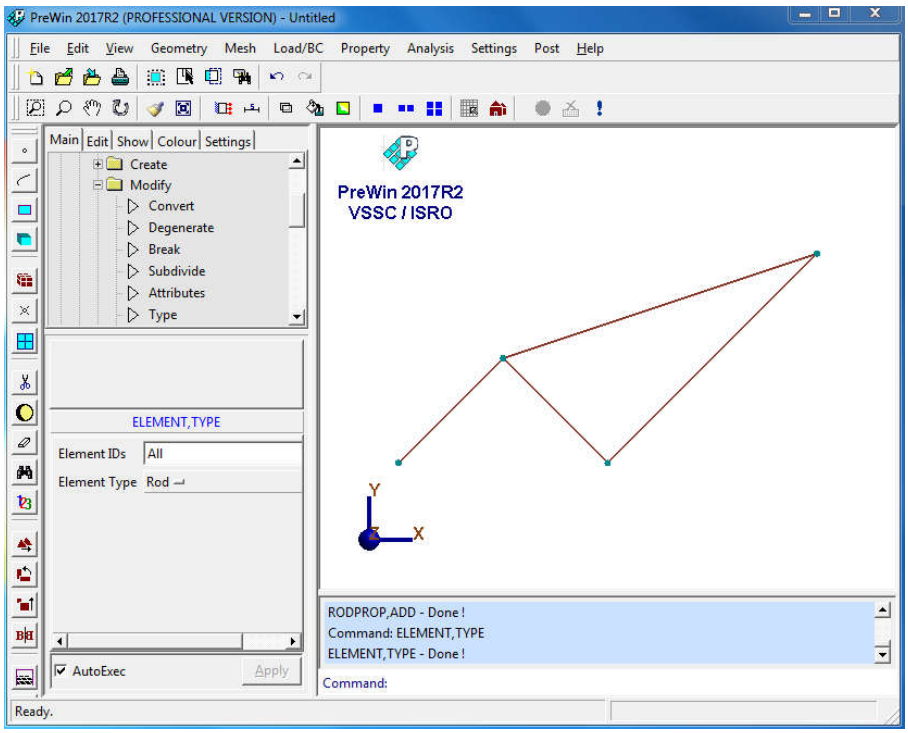
4. Convert beam elements into rod elements

Commands : ELEMENT, TYPE

Menu : Mesh → Element→ Modify→Type

Parameters :

Element IDs	All
Element Type	Rod



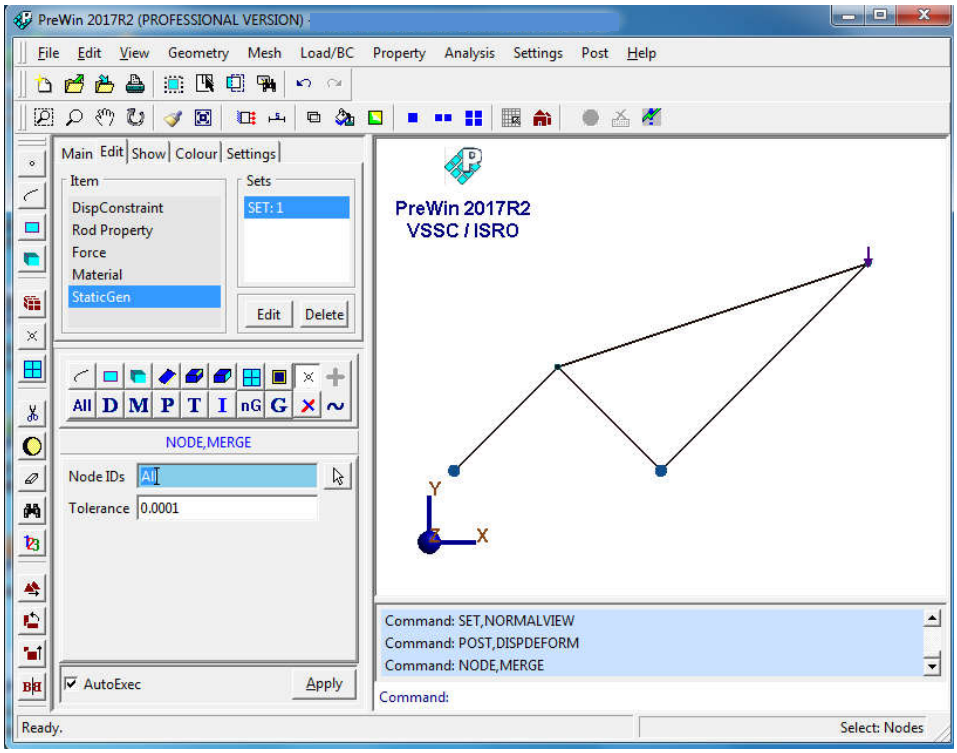
5. Merge duplicate nodes.

Commands : NODE, MERGE

Menu : Mesh → Node→ Modify→Merge

Parameters :

Element IDs	All
Tolerance	0.0001

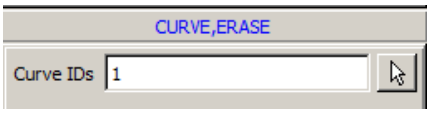


6. Erase Curve (Optional)

Commands : CURVE,ERASE

Menu : Geometry → Curve → Miscellaneous → Erase

Parameters :



Note:

Likewise erase key points by POINT,ERASE,ALL command.

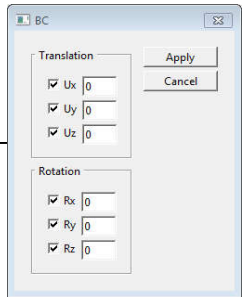
7. Specify displacement boundary conditions

Commands : BC,ADD

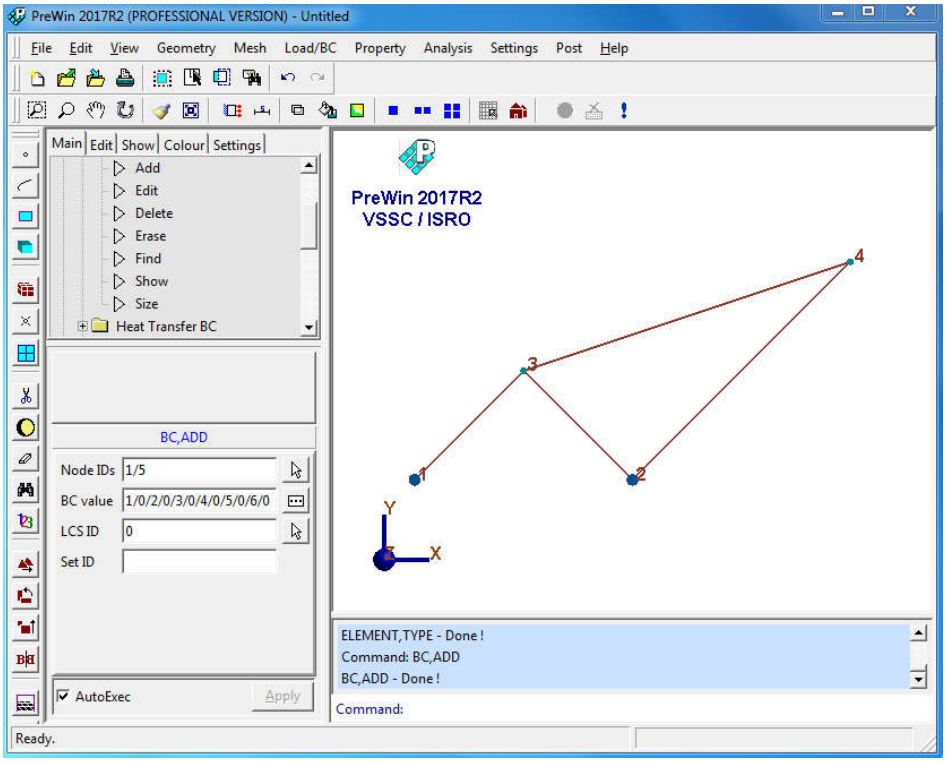
Menu : Load/BC → Displacement BC → Add

Parameters :

Node IDs	Select the node at the points A and B
BC Value	1/0/2/0/3/0/4/0/5/0/6/0
LCS ID	0



At the end of the operation/s your screen should look like this.



8. Specify material properties

Command : MATERIAL, ISO

Menu : Property → Material → Isotropic → Add

Parameters :

Element IDs	All
Material Data	2.1E+11/0.3/0/7850/0
Material ID	1

9. Specify Rod Properties

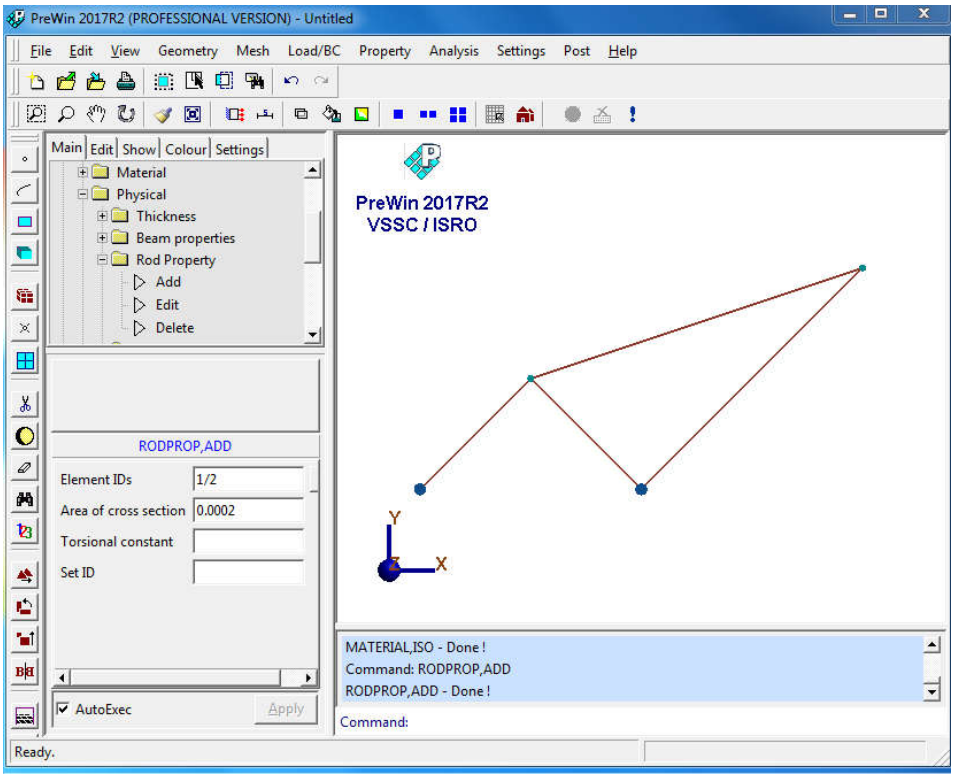
Command : RODPROP, ADD

Menu : Property → Physical → Rod Properties →Add

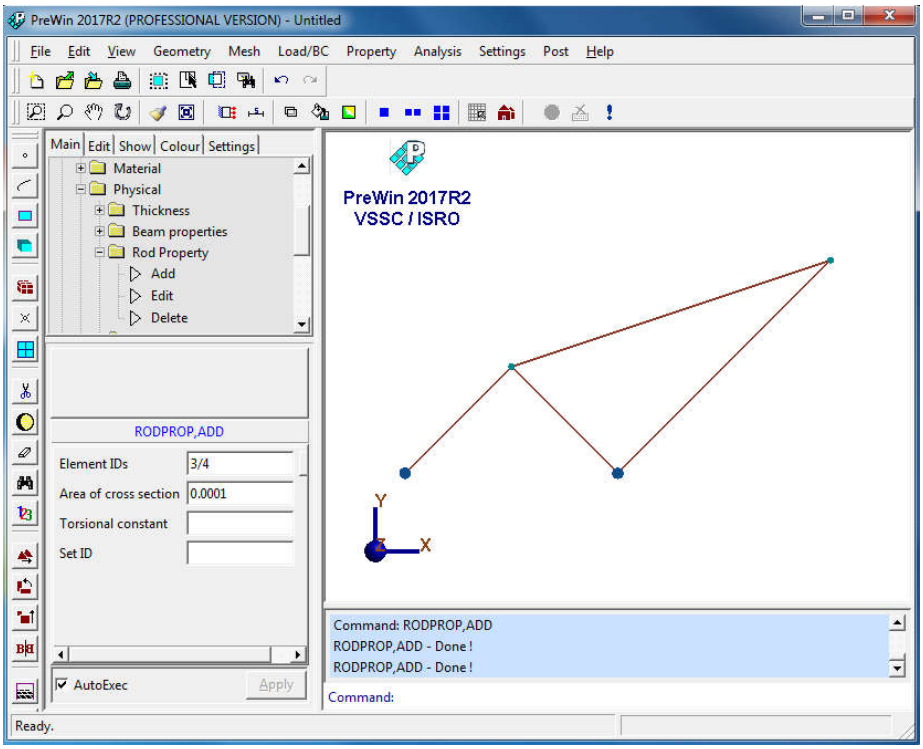
Parameters :

Element IDs	**1/2**
Area of cross section	**0.0002**
Torsional Constant	
Set ID	

At the end of the operation/s your screen should look like this.



similarly specify the rod properties for elements 3 and 4



10. Specify point load

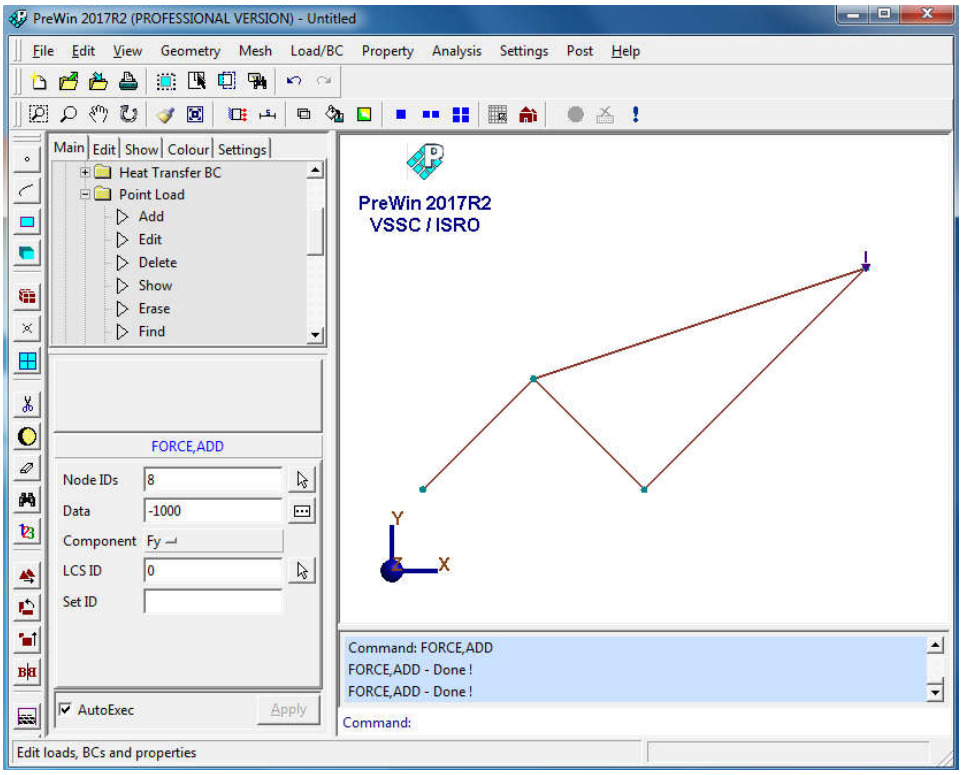
Command : FORCE, ADD

Menu : Load/BC → Point Load → Add

Parameters :

Node IDs	8
Data	-1000
Component	F_y
LCS ID	0
Set ID	1

At the end of the operation/s your screen should look like this.



11. Set the analysis type

Command : ANTYPE, SET

Menu : Analysis → Analysis Type

Parameters :

Analysis Type	Static

12. Set the analysis options

Command : ANOPTION, SET

Menu : Analysis → Analysis Options

Parameters :

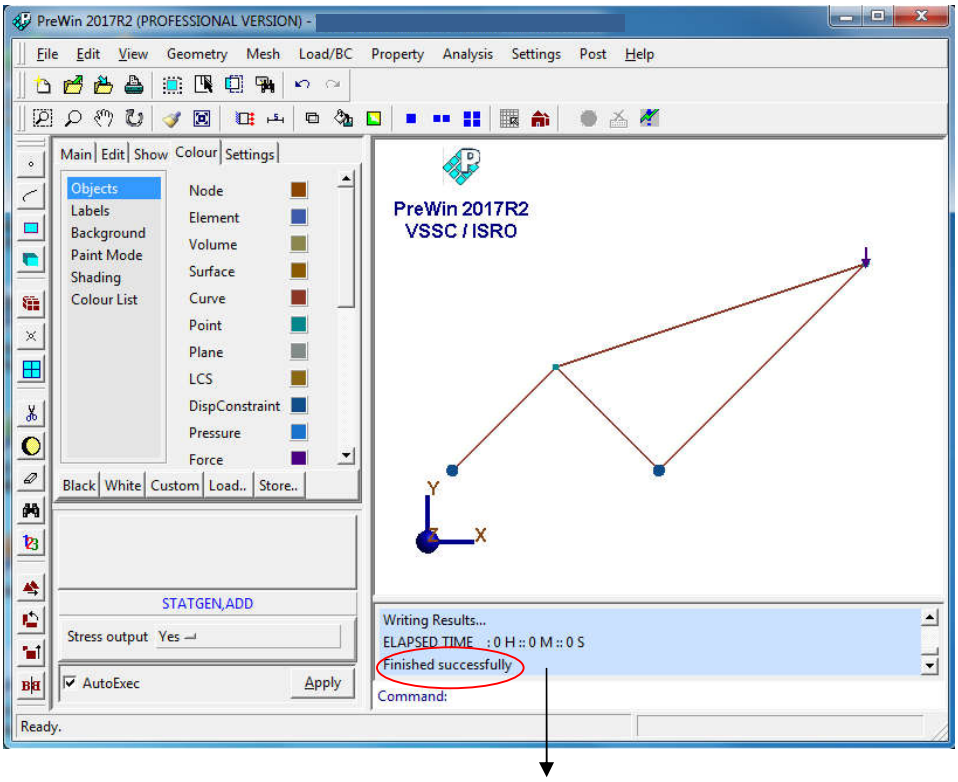
Linear Solver	Multi Frontal

13. Save the project model

Menu : File → Save

14. Submit the job into FEAST

Menu : Analysis →Run solver



Message box

Note:

"Finished successfully" message appears on message box after executing is completed.

15. Perform post processing

i) Deformed shape

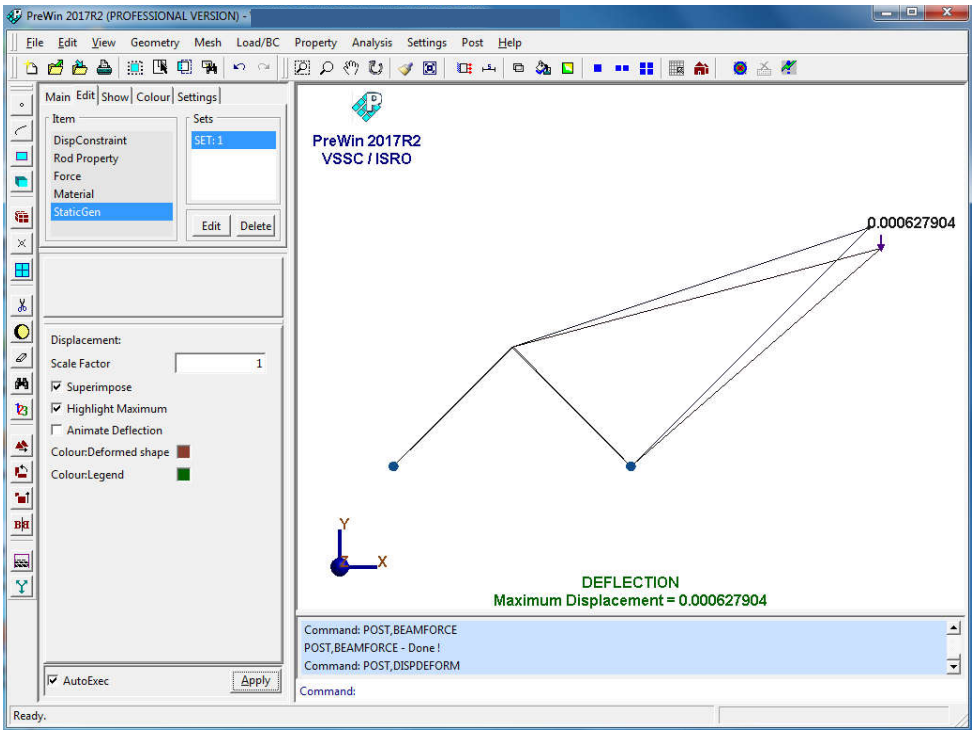
Command : POST, DISPDEFORM

Menu : Post → Deformed shape

Parameters :

Scale factor	1

At the end of the operation/s your screen should look like this.



ii) Beam force

a. Shear force diagram

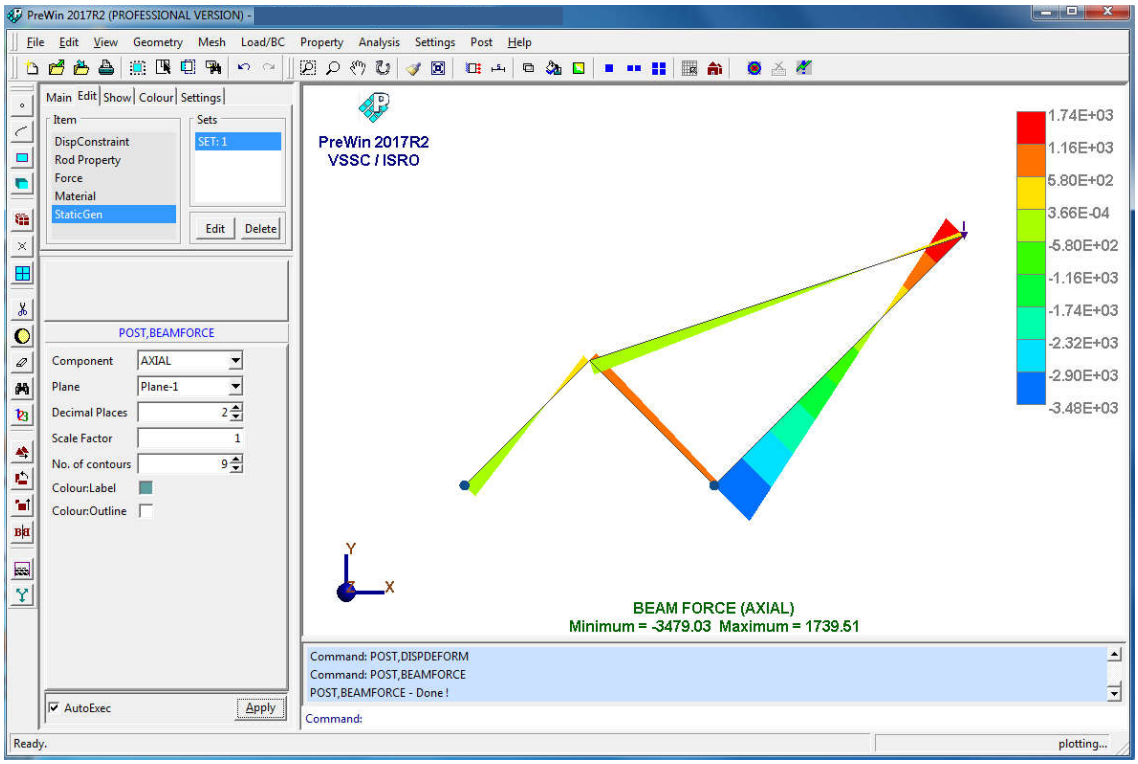
Command : POST, BEAMFORCE

Menu : Post → Beam Plots → Force Diagram

Parameters :

Component	SHEAR 1
Plane	Plane-1
Decimal Places	2
Scale Factor	1
No. of contours	9

At the end of the operation/s your screen should look like this.



b. *.DAT file shows the input data and *.OUT file shows the output file.