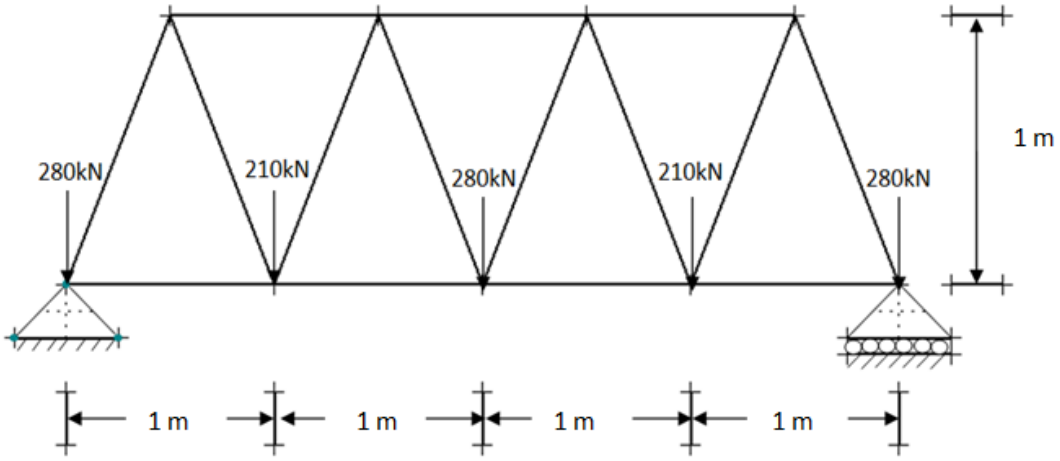
STATIC ANALYSIS OF TRUSS ELEMENTS



Properties

Area = 0.5 m^2

Modulus of Elasticity = 70 GPa

Poisson's Ratio = 0.3

Mass density=2800kg/m^3

PROCEDURE

1. Create geometric coordinates (0,0,0),(1,0,0),(0.5,1,0), (2,0,0),(1.5,1,0), (3,0,0), (2.5,1,0), (4,0,0) ,(3.5,1,0)

Commands : POINT, ADD

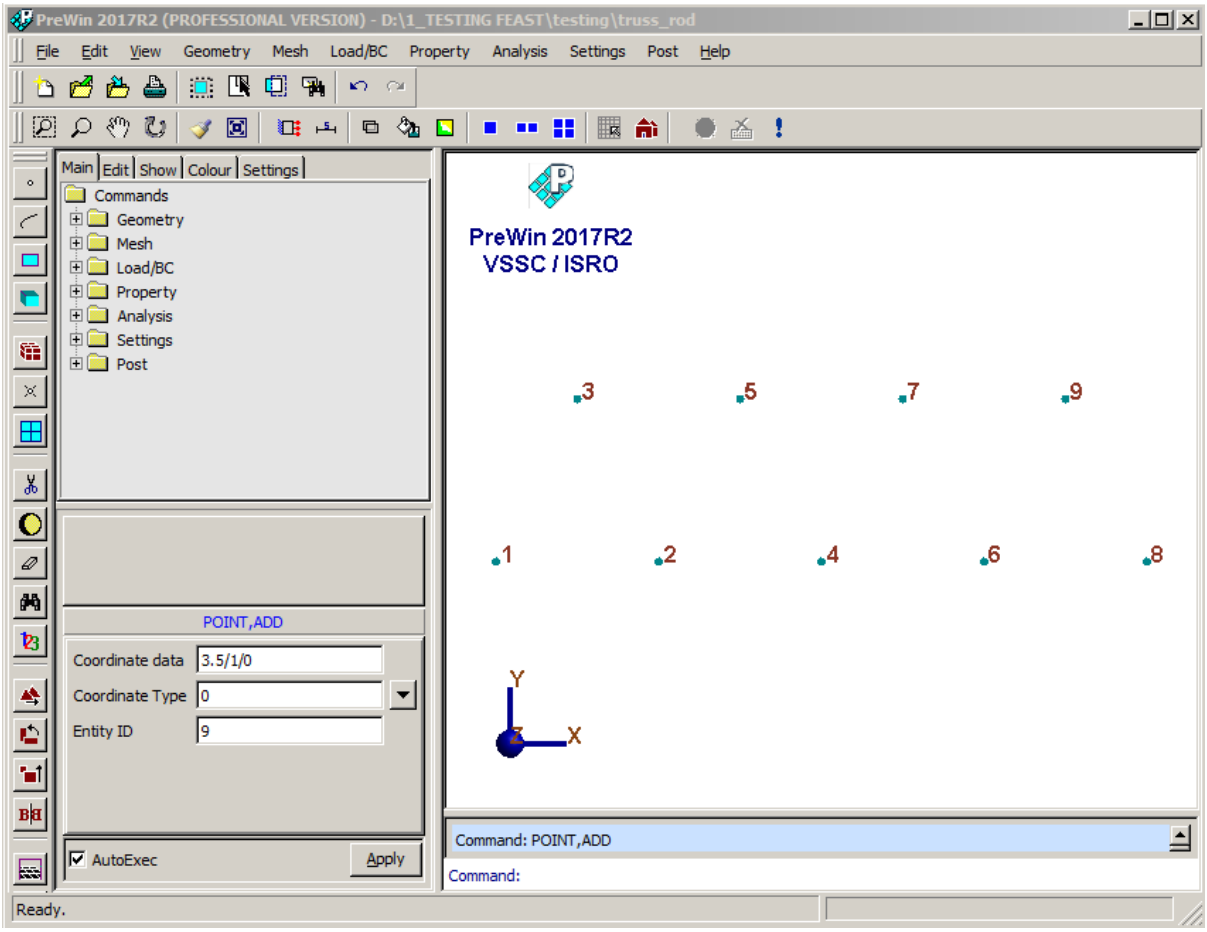
Menu : Geometry → Key point → Create → By X/Y/Z

Parameters : (To be filled by the user)

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

After entering the coordinate data click 'Apply' or press 'Cntrl+Enter'. Likewise each coordinate is created using the above command options.

At the end of the above operations your screen looks like this.



2. Generate nodes for each point ids created

Commands : NODE, SNAP

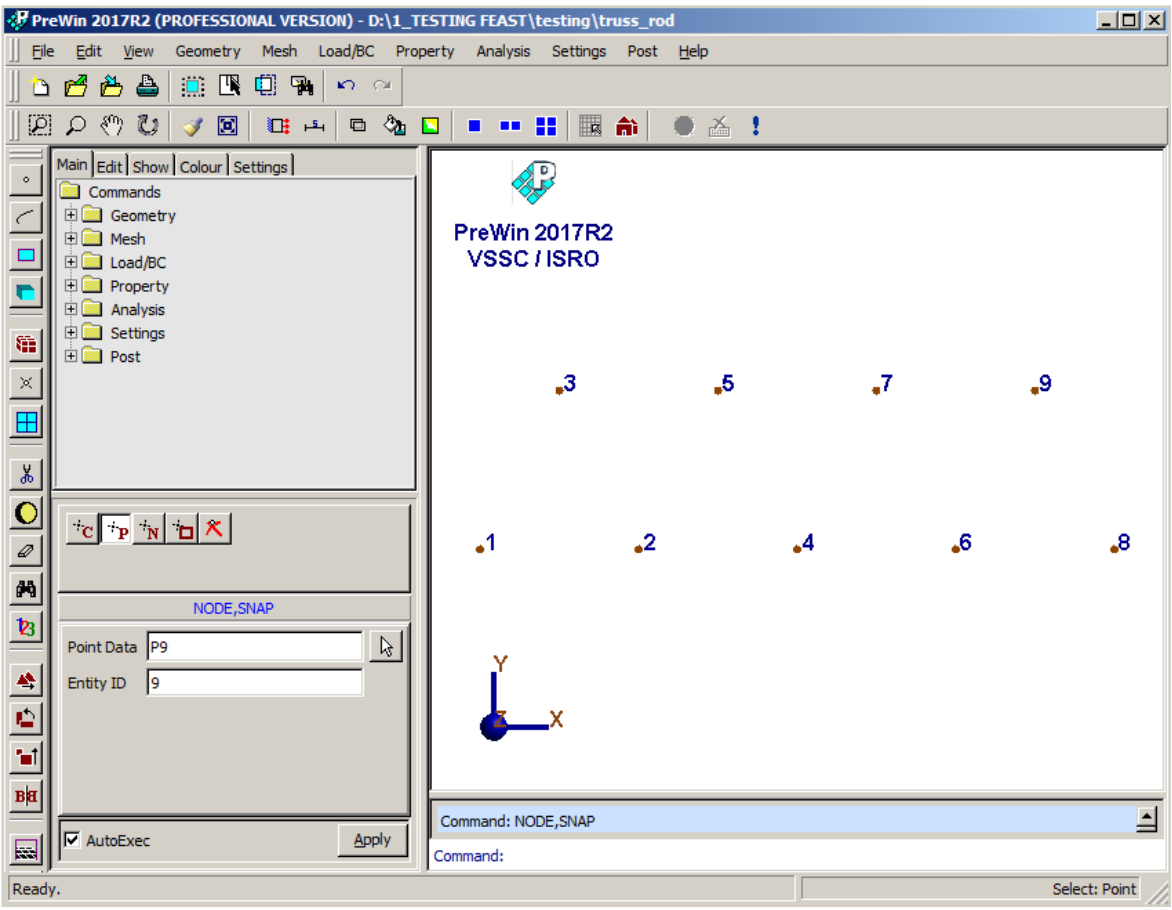
Menu : Mesh → Node → Create → Snap

Parameters : (To be filled by the user)

Click on button above the parameter box in order to select the points created in above step and generate corresponding nodes at locations.

Coordinate Data	P1/ P2/ P3/ P4/ P5/ P6/ P7/ P8/ P9
Coordinate Type	0
Entity ID	

At the end of the above operations your screen looks like this.



Note

*Erase the points created first to visible the nodes snapped by [POINT,ERASE,ALL] command option.

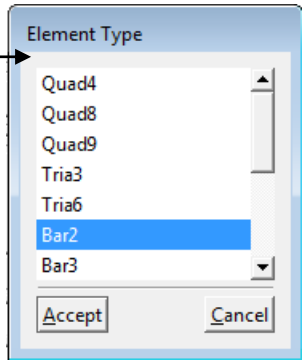
3. Create beam elements

Commands : ELEMENT,BYNODE

Menu : Mesh → Element → Create → ByNode

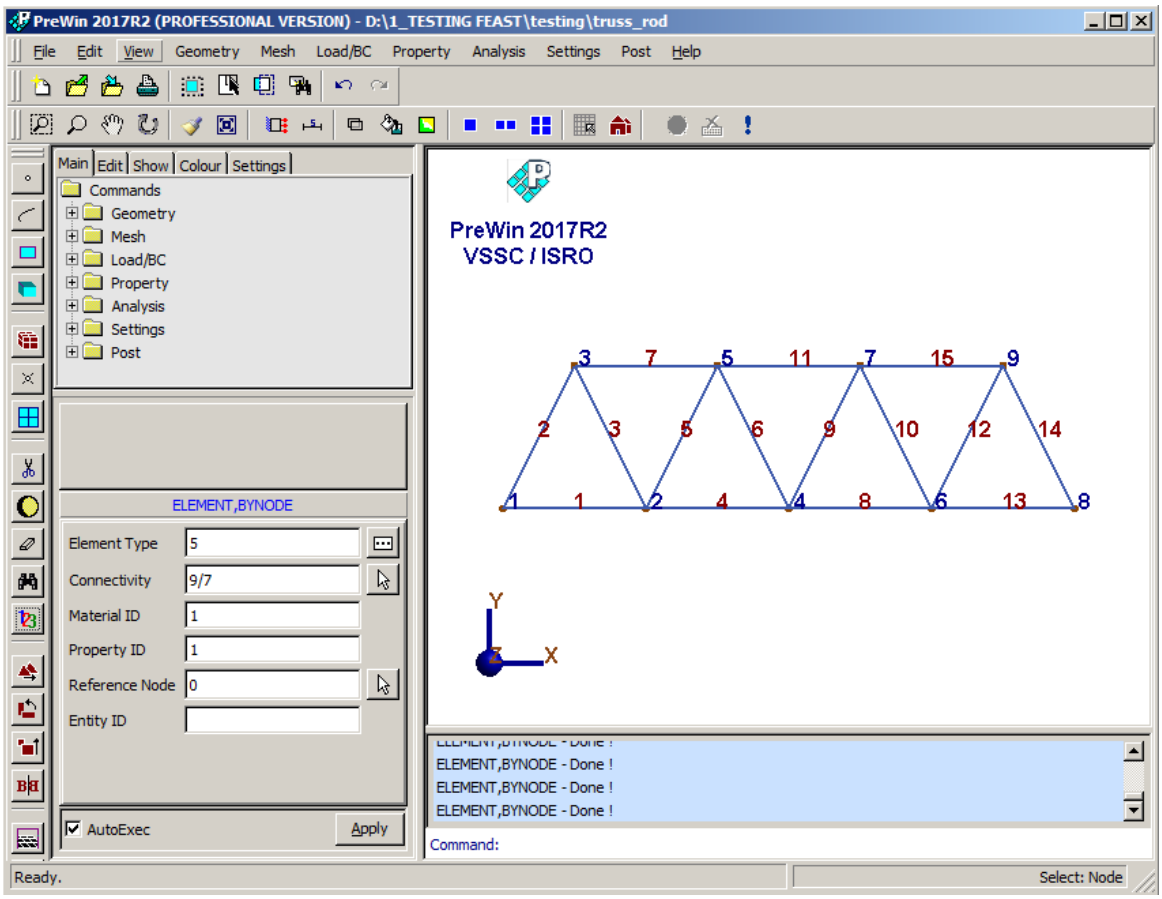
Parameters : (To be filled by the user)

Element Type	5
Connectivity	9/7
Material ID	1
Property ID	1
Reference Node	0
Entity ID	15



Keeping the element type as ‘Bar2’, here 9 beam elements are created by changing the connectivity nodes. After doing the above operations a total number of 9 nodes and 15 elements are displayed.

At the end of the above operations your screen looks like this.



4. Set the element type

Commands : ELEMENT,TYPE

Menu : Mesh → Element → Modify → Type

Parameters : (To be filled by the user)

Element IDs	All
Element Type	Rod

After selecting the element type from the list click ‘Apply’.

5. Define the element properties

Commands : RODPROP,ADD

Menu : Property → Physical → Rod Property → Add

Parameters : (To be filled by the user)

Element IDs	All
Area of cross-section	0.5
Set ID	1

6. Specify Displacement constraints

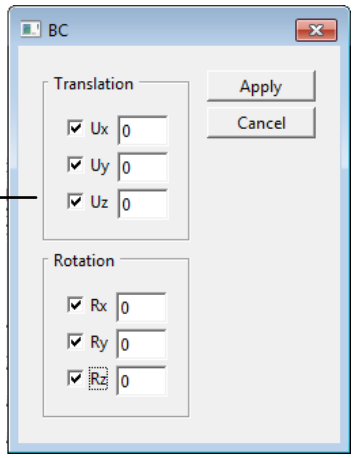
Commands : BC,ADD

Menu : Load/BC → Displacement BC → Add

Parameters : (To be filled by the user)

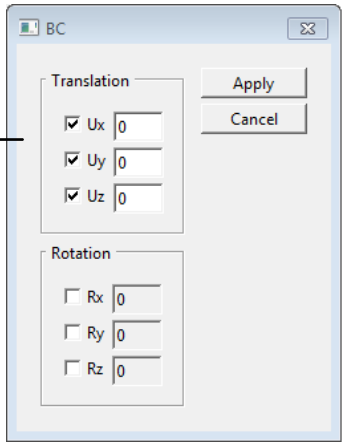
i.) Fixed support

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

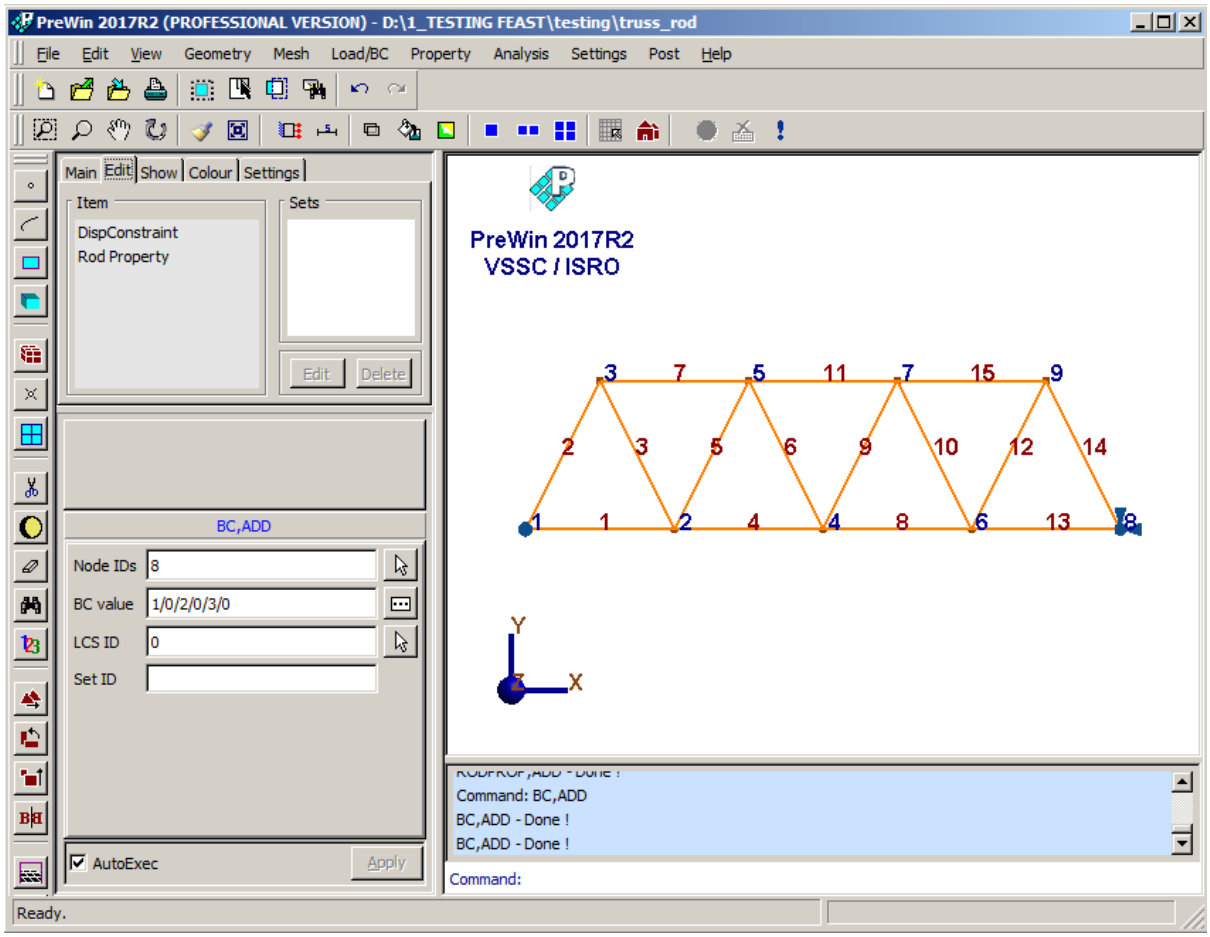


ii.) Roller support

Node IDs	8
BC Value	1/0/2/0/3/0
LCS ID	0
Set ID	2



At the end of the above operations your screen looks like this.



7. Specify loads applied

Commands : FORCE,ADD

Menu : Load/BC → Point Load → Add

Parameters : (To be filled by the user)

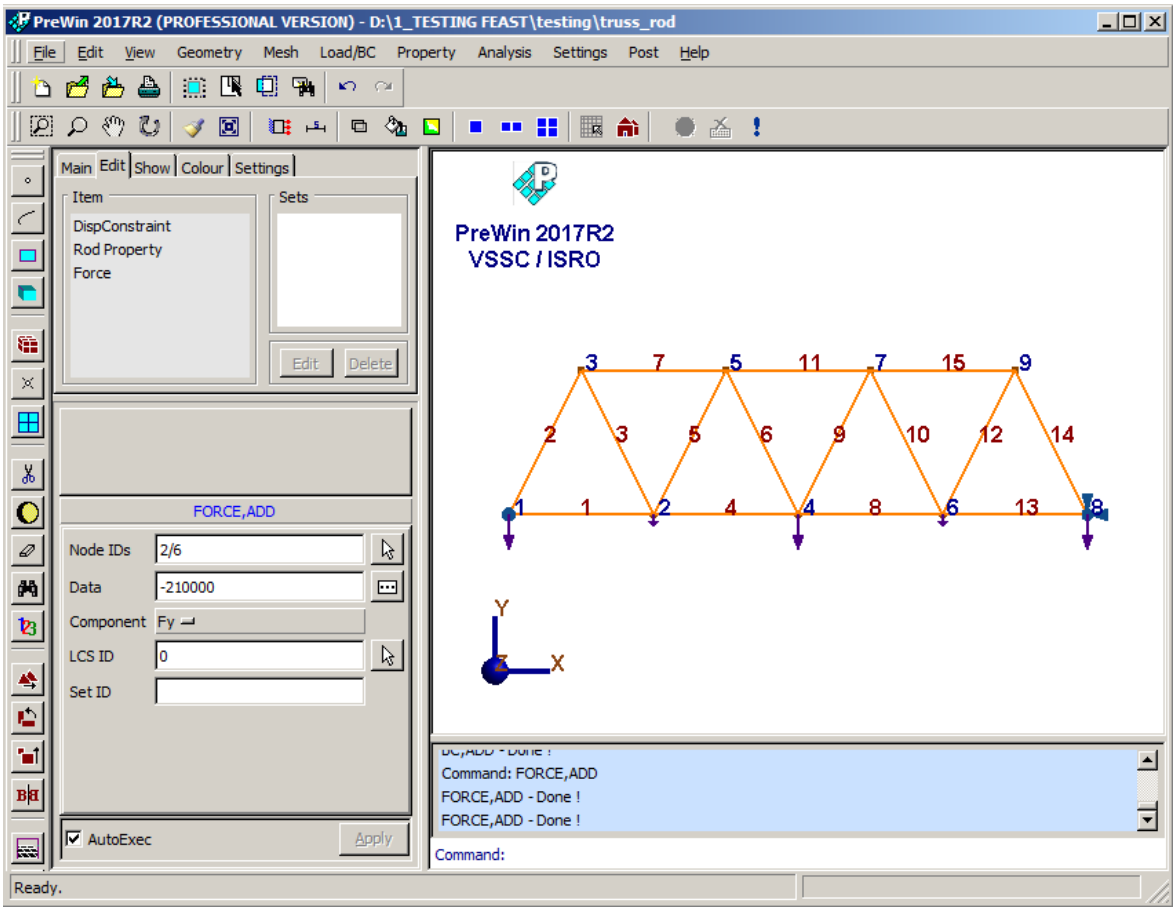
i.) 280kN at node id 1,4&8

Node IDs	1/4/8
Data	-280000
Component	Fy
LCS ID	0
Set ID	1

ii.) 210kN at node id 2&6

Node IDs	2/6
Data	-210000
Component	Fy
LCS ID	0
Set ID	2

At the end of the above operations your screen looks like this.



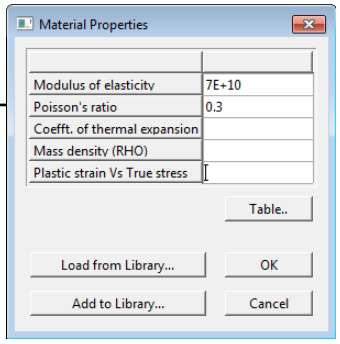
8. Specify material properties

Commands : MATERIAL,ISO

Menu : Property → Material → Isotropic→ Add

Parameters : (To be filled by the user)

Element IDs	All
Material-Data	7E+10/0.3/0/0/0
Material ID	1



Note:

It is possible for the user to add material properties from material library provided.

9. Set analysis type

Commands : ANTYPE,SET

Menu : Analysis → Analysis Type

Parameters : (To be filled by the user)

Analysis Type	Static

10. Set analysis options

Commands : ANOPTION,SET

Menu : Analysis → Analysis Options

Parameters : (To be filled by the user)

Linear Solver	MultiFrontal

11. Set analysis general

Commands : STATGEN,ADD

Menu : Analysis → Static → General → Add

Parameters : (To be filled by the user)

Stress Output	ON

12. Save the job and submit the job into FEAST

Menu : Analysis → Run Solver

Note:

For solving user can either follow the menu path or click icon ! in right end of tool bar.

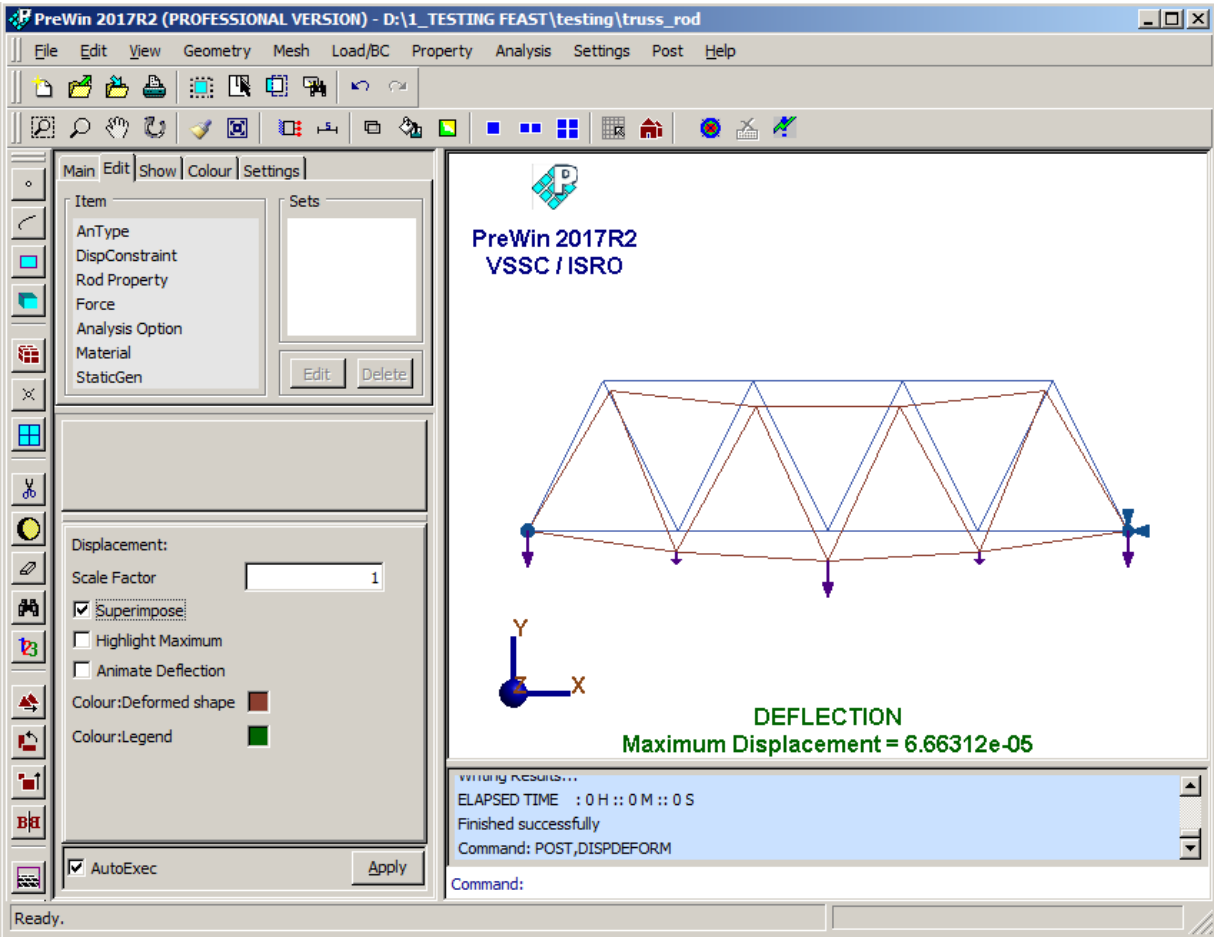
13. Perform post processing

i.) Deformed shape

Command : POST,DISPDEFORM

Menu : Post → Deformed shape

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



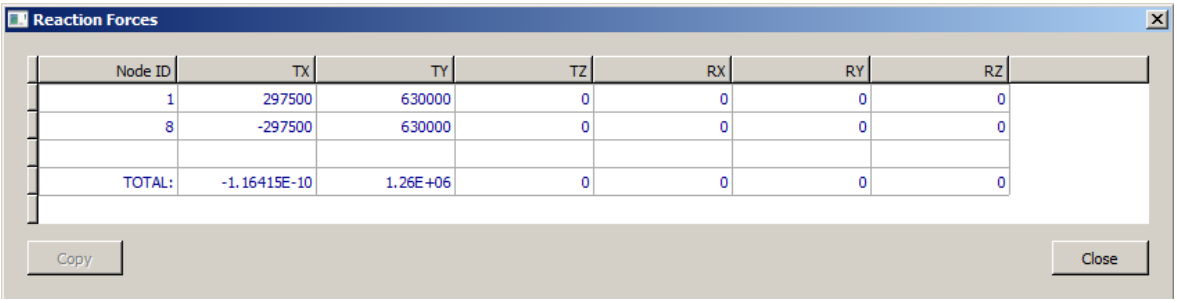
ii.) Reaction force

Command : POST,VIEWRESULTS

Menu : Post → View Results

Parameters : (To be filled by the user)

Item	Reaction Force
Nodes	All
LCS	Global Cartesian


Reaction Forces

Node ID	TX	TY	TZ	RX	RY	RZ
1	297500	630000	0	0	0	0
8	-297500	630000	0	0	0	0
TOTAL:	-1.16415E-10	1.26E+06	0	0	0	0

Copy Close

iii) Output file can be seen in *.OUT