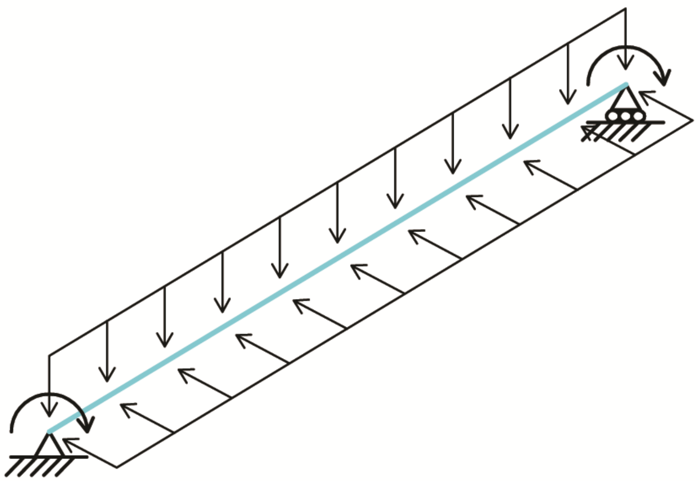
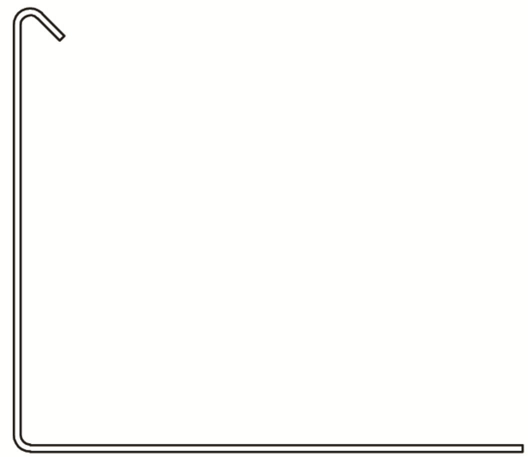
Tutorial for MASTAN2 v5.1 - Pour Stop Beam





Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Bucknell UNIVERSITY

NEW MILLENNIUM BUILDING SYSTEMS

American Iron and Steel Institute

STEEL JOIST SJI INSTITUTE

STEEL DECK INSTITUTE sdi ®

MASTAN2

MASTAN2

Credits

Published 2020

Developed by:

Edward J. Sippel, Ph.D. Student, University of Wisconsin – Madison

Hannah B. Blum, Assistant Professor, University of Wisconsin - Madison

Ronald D. Ziemian, Professor, Bucknell University

Joe Pote, Director of Research & Development, New Millennium Building Systems

Scott Morton, Research and Development Engineer, New Millennium Building Systems

Sponsored by:

American Iron and Steel Institute

New Millennium Building Systems

Steel Joist Institute

Steel Deck Institute

Tutorial Sections

Navigation

- – Skip to Previous Section Title Page
- – Previous Slide
- – Return to Tutorial Sections Page
- – Next Slide
- – Skip to Next Section Title Page

– Open screenshot of MASTAN2 or additional helpful information.

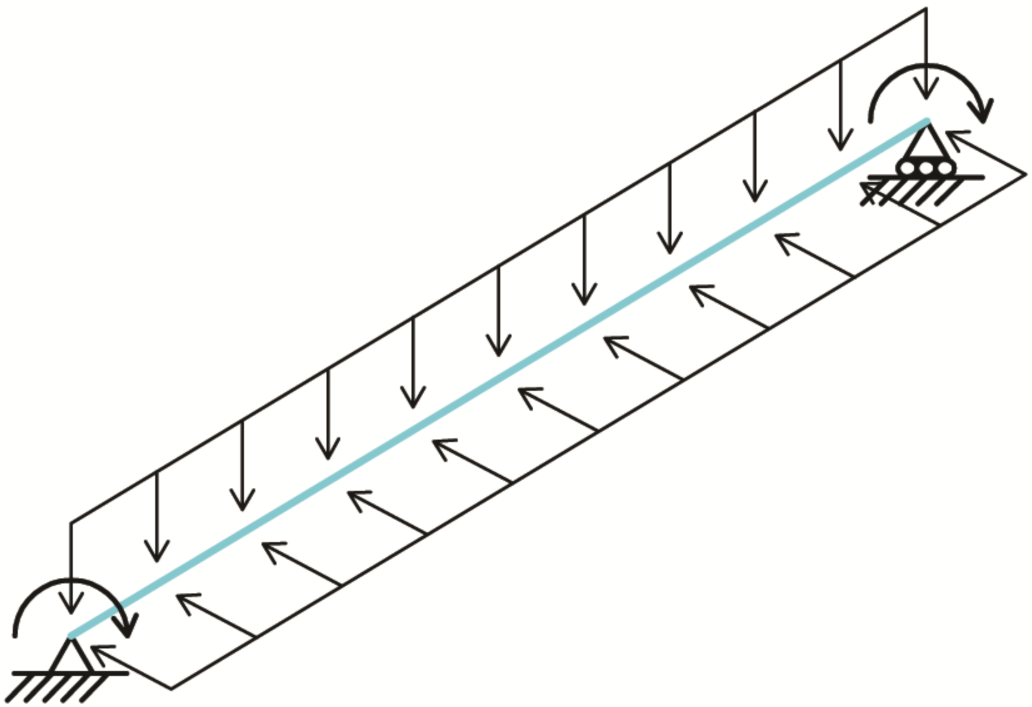
Section 1: Overview

Overview

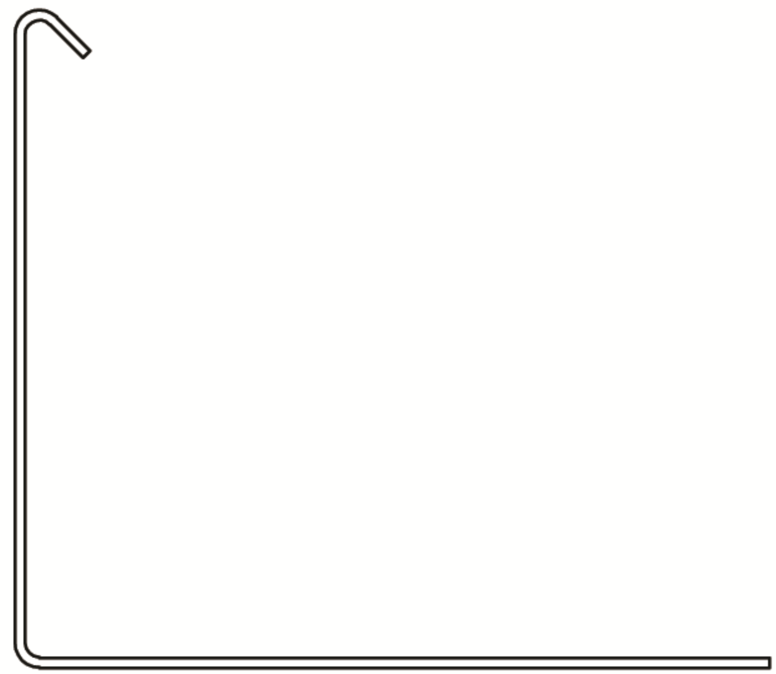
This tutorial provides step-by-step guidance for the sample pour stop beam evaluation. Enough details are provided that the example model with non-doubly symmetric sections can be completed following the instructions here. Not every feature available in MASTAN2 will be mentioned nor utilized in this tutorial. For further information on many of the features within MASTAN2 make use of other tutorials at http://www.mastan2.com/tutorial.html.

Problem Overview

This tutorial completes the analysis of a simply supported non-doubly symmetric section in MASTAN2 and some of the related stress calculations. The section considered is a pour stop that is simply supported. The values used in this tutorial come from the Steel Deck Institute's Technical Note – No. 3: Pour Stops as Beams.



Simple Supported Beam



Cross Section

Section 2: Getting Started

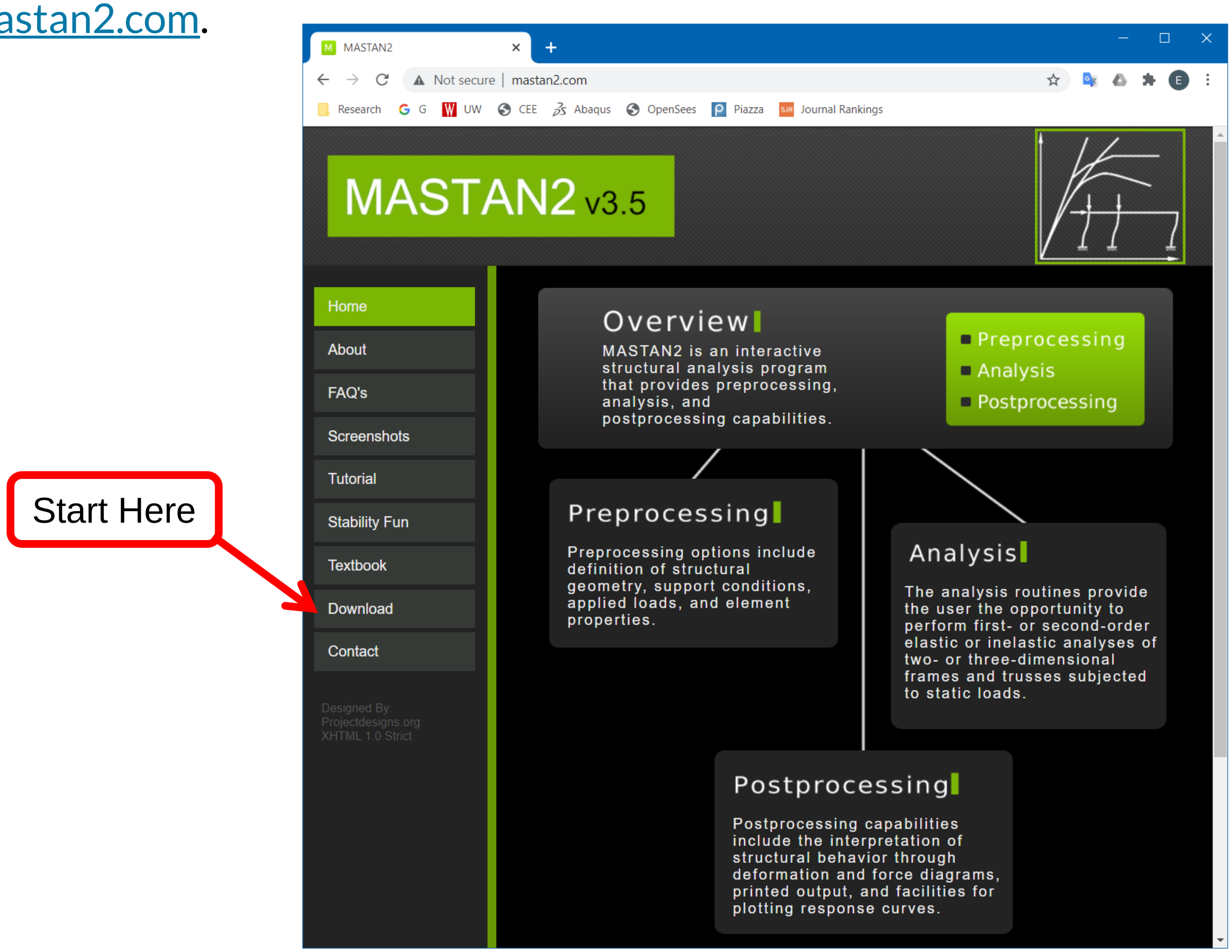
MASTAN2 General Information

MASTAN2 is an interactive graphics program that provides preprocessing, analysis, and postprocessing capabilities. Preprocessing options include definition of structural geometry, support conditions, applied loads, and element properties. The analysis routines provide the user the opportunity to perform first- or second-order elastic or inelastic analyses of two- or three-dimensional frames and trusses subjected to static and dynamic loads. Postprocessing capabilities include the interpretation of structural behavior through deformation and force diagrams, printed output, and facilities for plotting response curves. MASTAN2 is based on MATLAB®, a premier software package for numeric computing and data analysis.

In many ways, MASTAN2 is similar to today's commercially available software in functionality. The number of pre- and post-processing options, however, have been limited in order to minimize the amount of time needed for a user to become proficient at its use. The program's linear and nonlinear analysis routines are based on the theoretical and numerical formulations presented in the text *Matrix Structural Analysis, 2nd Edition*, by McGuire, Gallagher, and Ziemian. In this regard, the reader is strongly encouraged to use this software as a tool for demonstration, reviewing examples, solving problems, and perhaps performing analysis and design studies. Where MASTAN2 has been written in modular format, the reader is also provided the opportunity to develop and implement additional or alternative analysis routines directly within the program.

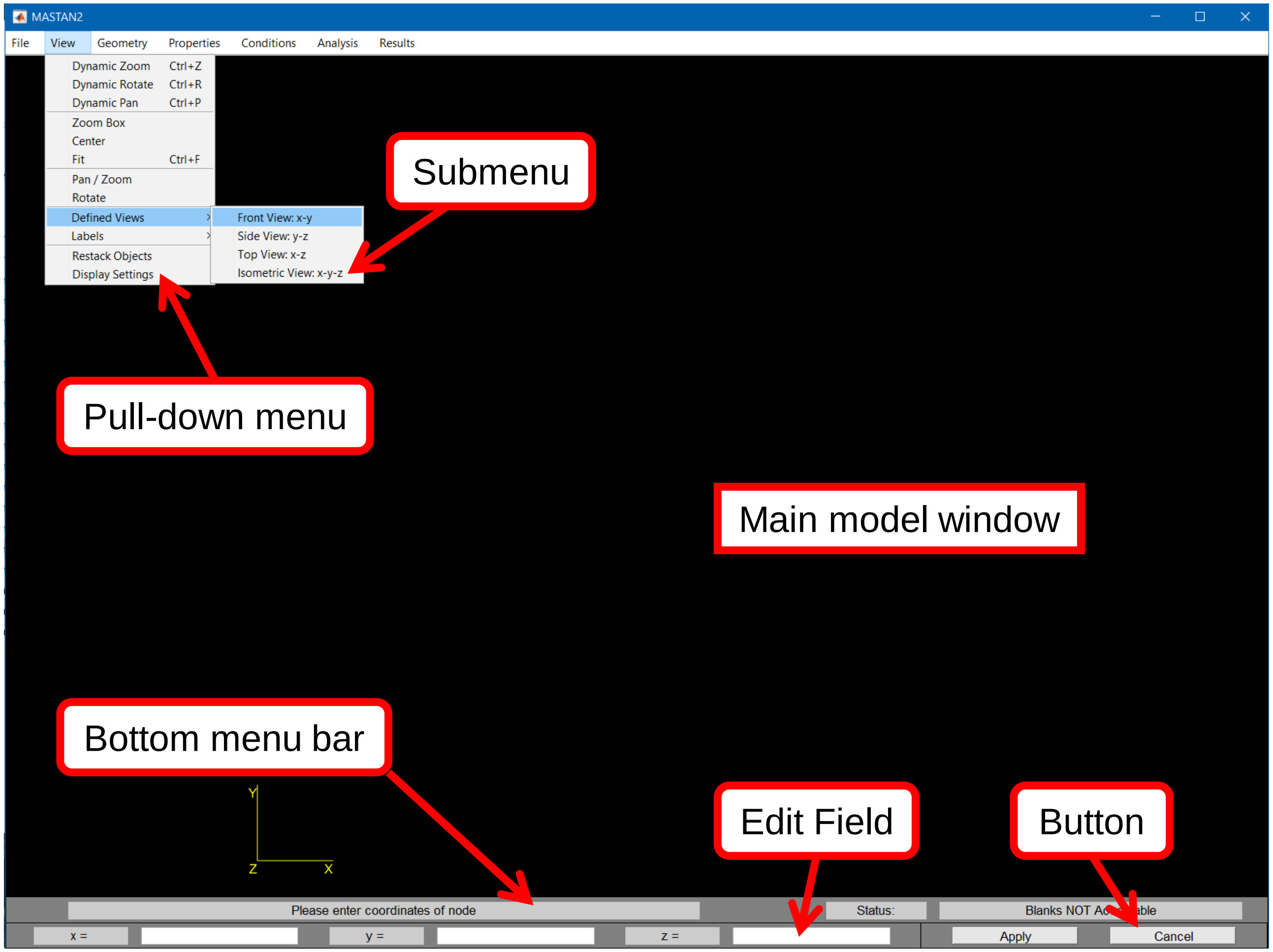
Launching MASTAN2

Two versions of MASTAN2 have been developed and may be installed. One requires you to have access to MATLAB and the other does not. Both versions provide the same functionality, except that the MATLAB version also provides the user an opportunity to develop and implement additional or alternative analysis routines that will directly interact with MASTAN2. Please see the Setup Guides at www.mastan2.com.



Base Layout

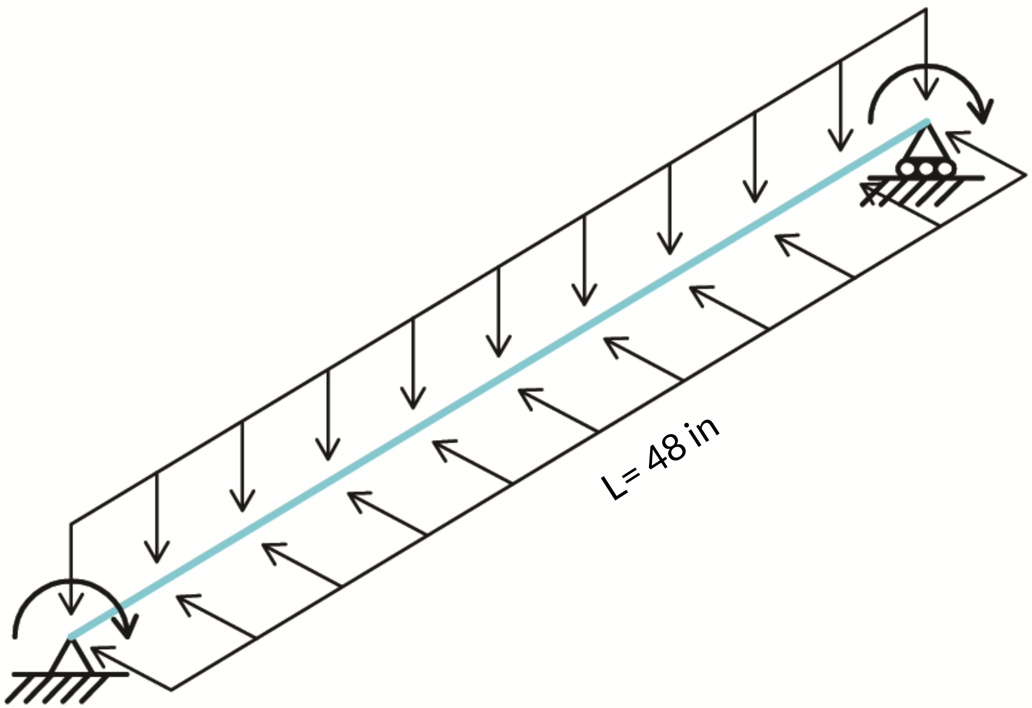
In order to minimize the learning time for MASTAN2, its graphical user interface (GUI) has been designed using a simple and consistent two menu approach. Using a pull-down menu at the top of the GUI, a command is selected. Parameters are then defined in the bottom menu bar and the command is executed by using the Apply button.

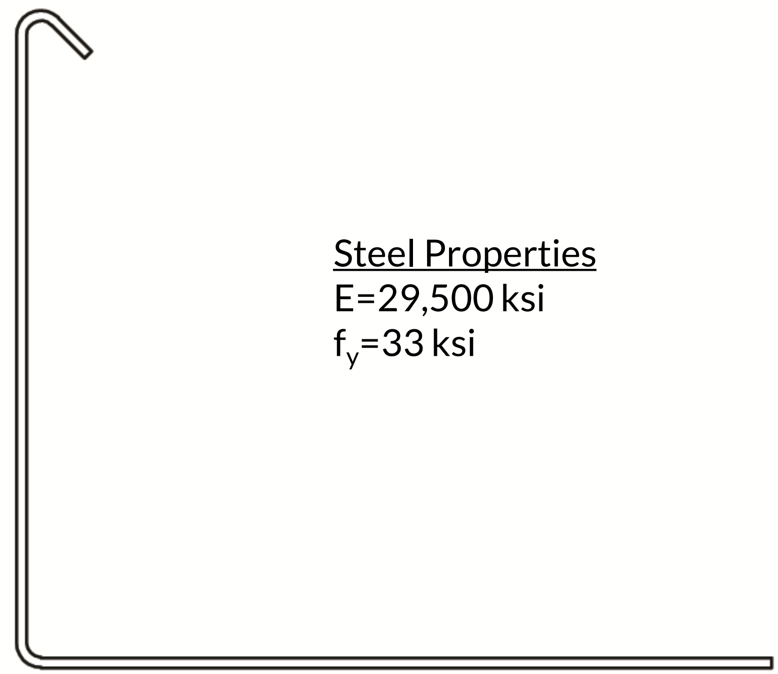


Section 3: Beam Modeling

Problem Description - Figure

The pour stop is a cold-form steel cross-section. The section is subjected to a uniform lateral and uniform vertical load as well as a distributed torsion. The model itself will be a simply supported beam with the ends fixed for torsion, but free to warp. Details on the applied load are on the next page.



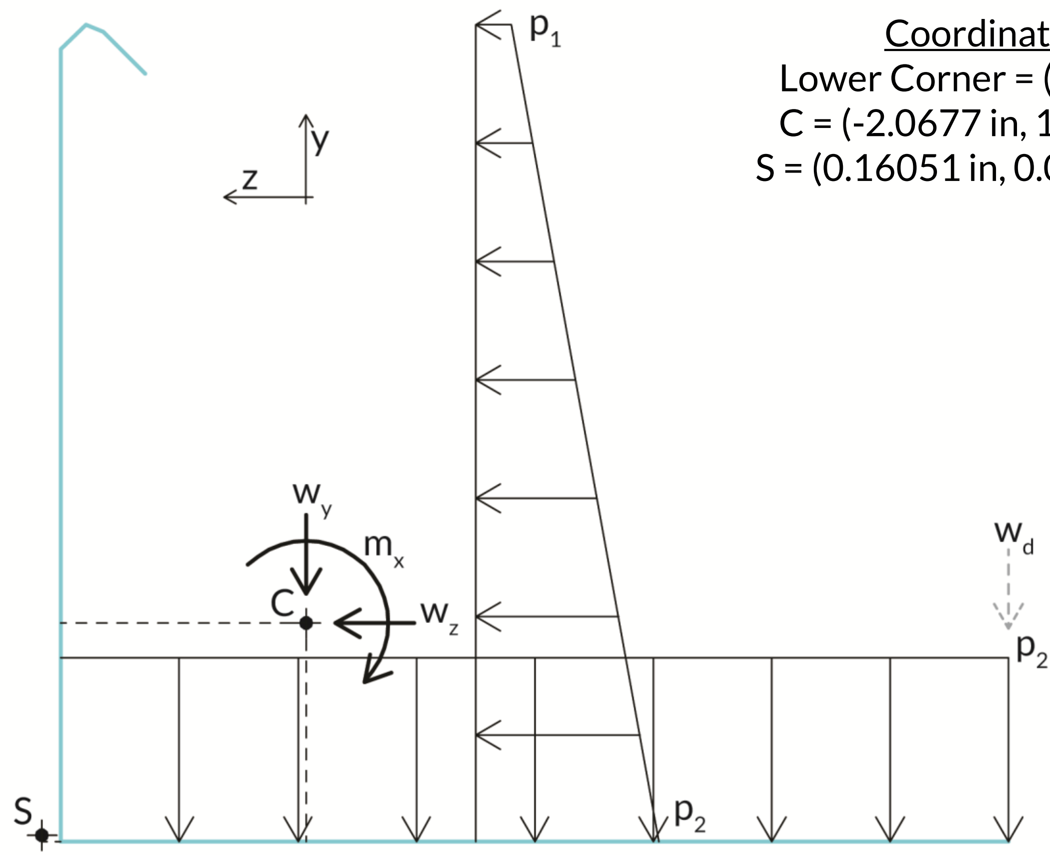


Steel Properties
E=29,500 ksi
f_y=33 ksi

MASTAN2 does not assume any unit system. Models in MASTAN2 require the use of a consistent set of units. This tutorial will use pound and inch for the model. The later section of the tutorial that determines the internal stresses does include a unit conversion to show stresses in ksi.

Problem Description - Loading

The pour stop is subjected to a uniform surcharge load, a pressure from the wet concrete, self-weight, and a linear load from similar sources due to interactions with the deck. Uniform distributed loads are available in MASTAN2; however, distributed torsional moments are not possible. To obtain a similar effect the loading will be applied via concentrated point loads and concentrated moments with many smaller elements along the length of the member



Coordinates

Lower Corner = (0 in, 0 in)
C = (-2.0677 in, 1.8405 in)
S = (0.16051 in, 0.052335 in)

Load Details

See SDI Tech No. 3 for additional information

p_1 = 20 psf
p_2 = 103.3 psf

w_d = 0 plf ↓

Resulting Loads

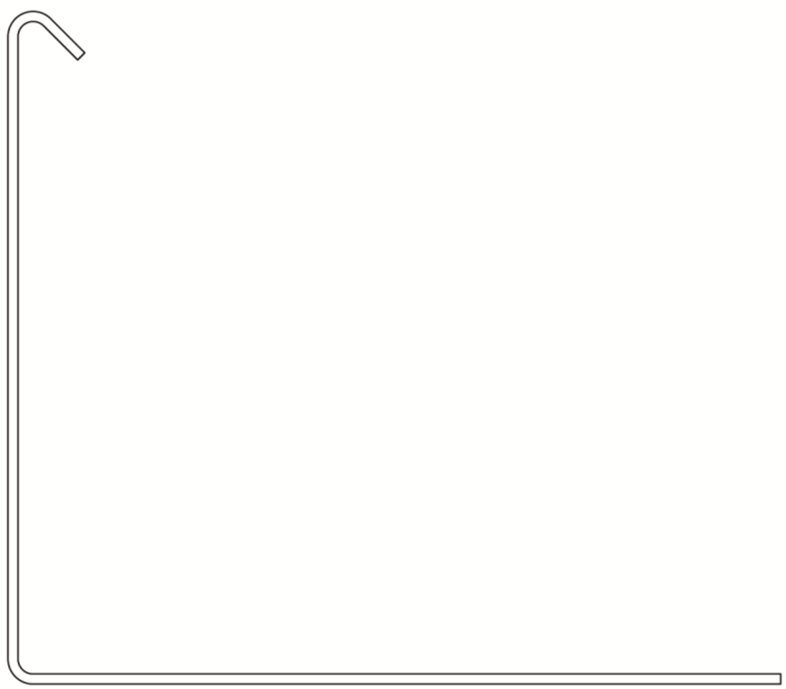
w_y = 74.0 plf ↓
= 6.17 pli ↓

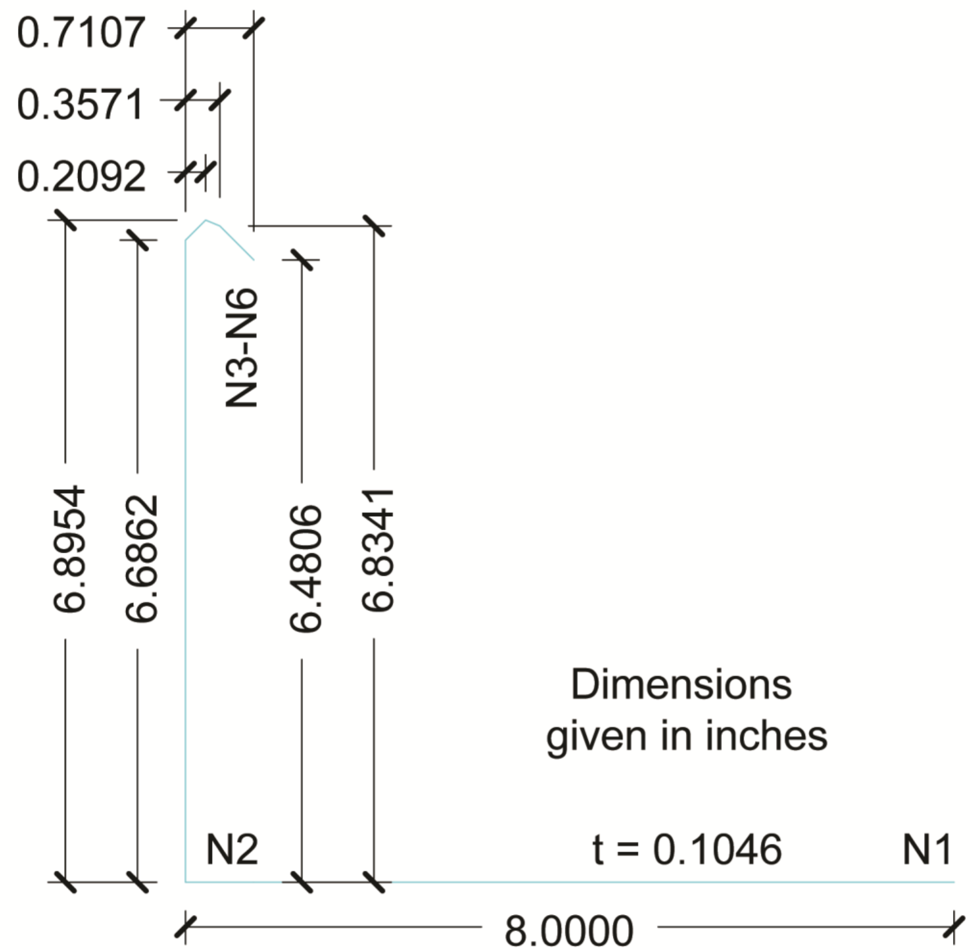
w_z = 35.4 plf ←
= 2.95 pli ←

m_x = 104.5 in-lb/ft ↻
= 8.71 in-lb/in ↻

Problem Description – Cross Section

The real pour stop has the rounded geometry shown on the left. The rounded segments, particularly at the top, could be defined by many closely spaced nodes to account for the full radius. However, this tutorial will use the simplified geometry shown on the right based on SDI Tech No. 3.





Geometry Definition

1) Start with a new, empty model.

2) From the **Geometry** menu select **Define Node**.

3) At the bottom menu bar, click in the edit box to the right of **x =** and enter **0**. Click in the edit box to the right of **y =** and enter **0**. Click in the edit box to the right of **z =** and enter **0**.

4) Click on the **Apply** Button.

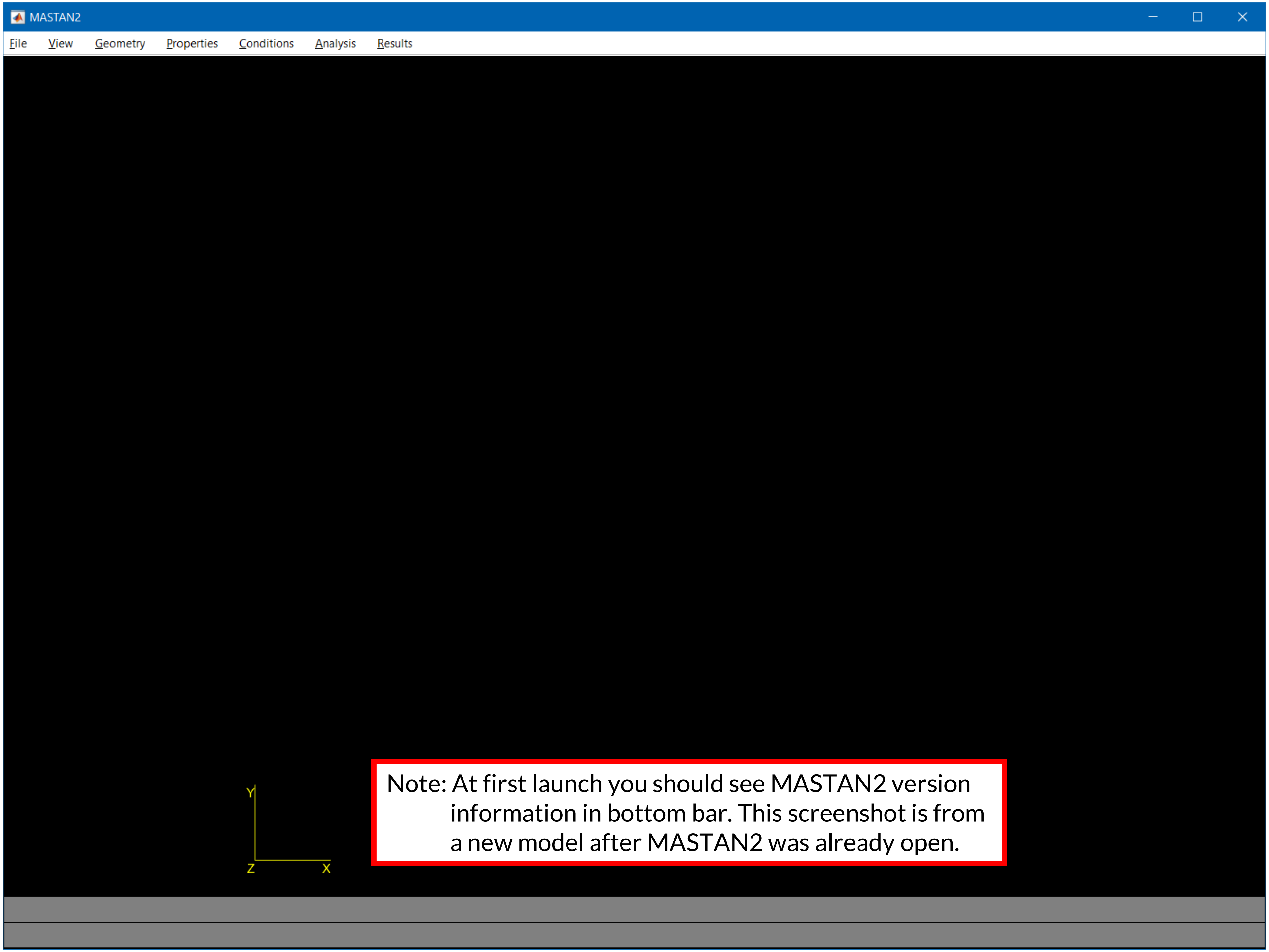
5) From the **Geometry** menu select **Extrude Element**.

6) At the bottom menu bar, click on Node **1** to populate the list of nodes. Click in the edit box to the right of **Delta x =** and change **0** to **2**.

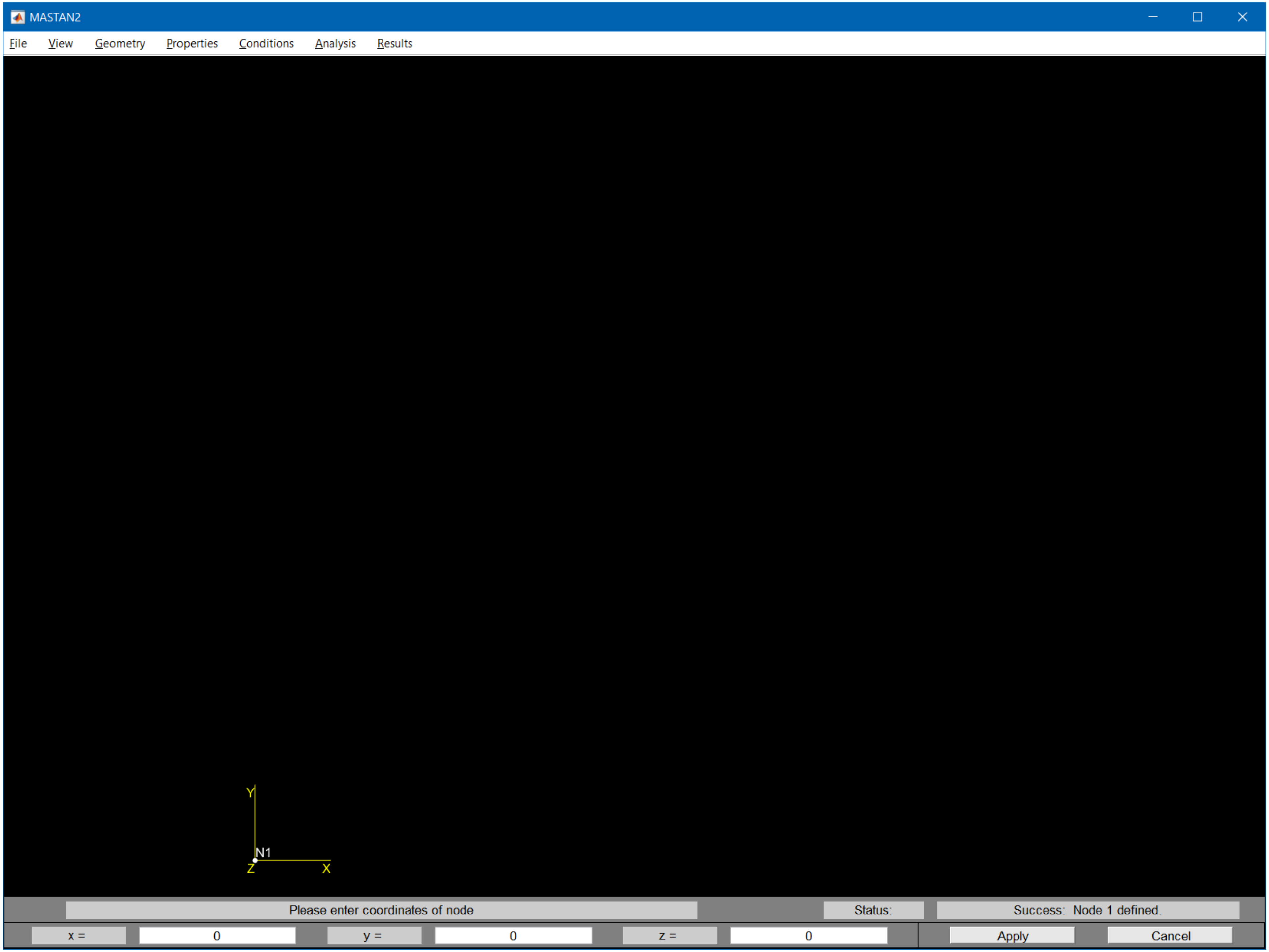
7) Repeatedly click the **>** button to the right of **Times =** to increase **1** to **24**.

8) Click on the **Apply** Button.

Clicking the icon will advance the tutorial to a page that provides an image of the MASTAN2 interface after the corresponding step is executed. Clicking the icon on that page will return you to the step-by-step instructions.



Note: At first launch you should see MASTAN2 version information in bottom bar. This screenshot is from a new model after MASTAN2 was already open.


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
N1
Z
X
Please enter coordinates of node
Status:
Success: Node 1 defined.
x =
0
y =
0
z =
0
Apply
Cancel

MASTAN2


MASTAN2
File View Geometry Properties Conditions Analysis Results
Y
Z
N1 N2 N3 N4 N5 N6 N7 N8 N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N25
E1 E2 E3 E4 E5 E6 E7 E8 E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
Define node(s) and data to extrude elements(s)
Node(s):
All
Clr
Adv
Status:
Success: Element(s) Extruded.
Delta x =
0
Delta y =
0
Delta z =
0
Beta (deg)
0
Times =
<
1
>
Apply
Cancel

Cross Section Visualization

1) From the **View** menu select **Defined Views** and submenu option **Isometric: x-y-z**.

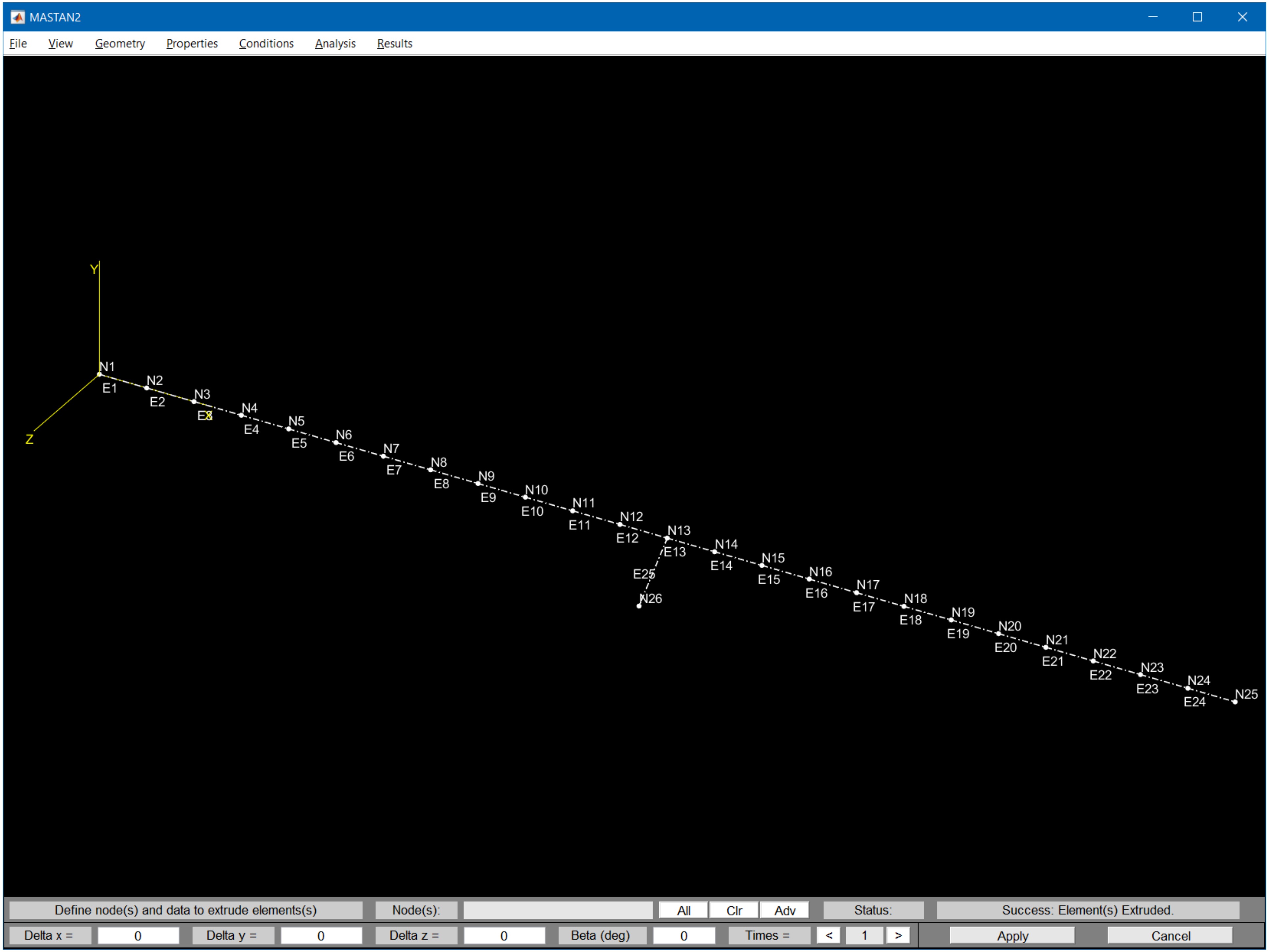
2) Now with the main member defined, continue to use **Extrude Element** to illustrate cross section.

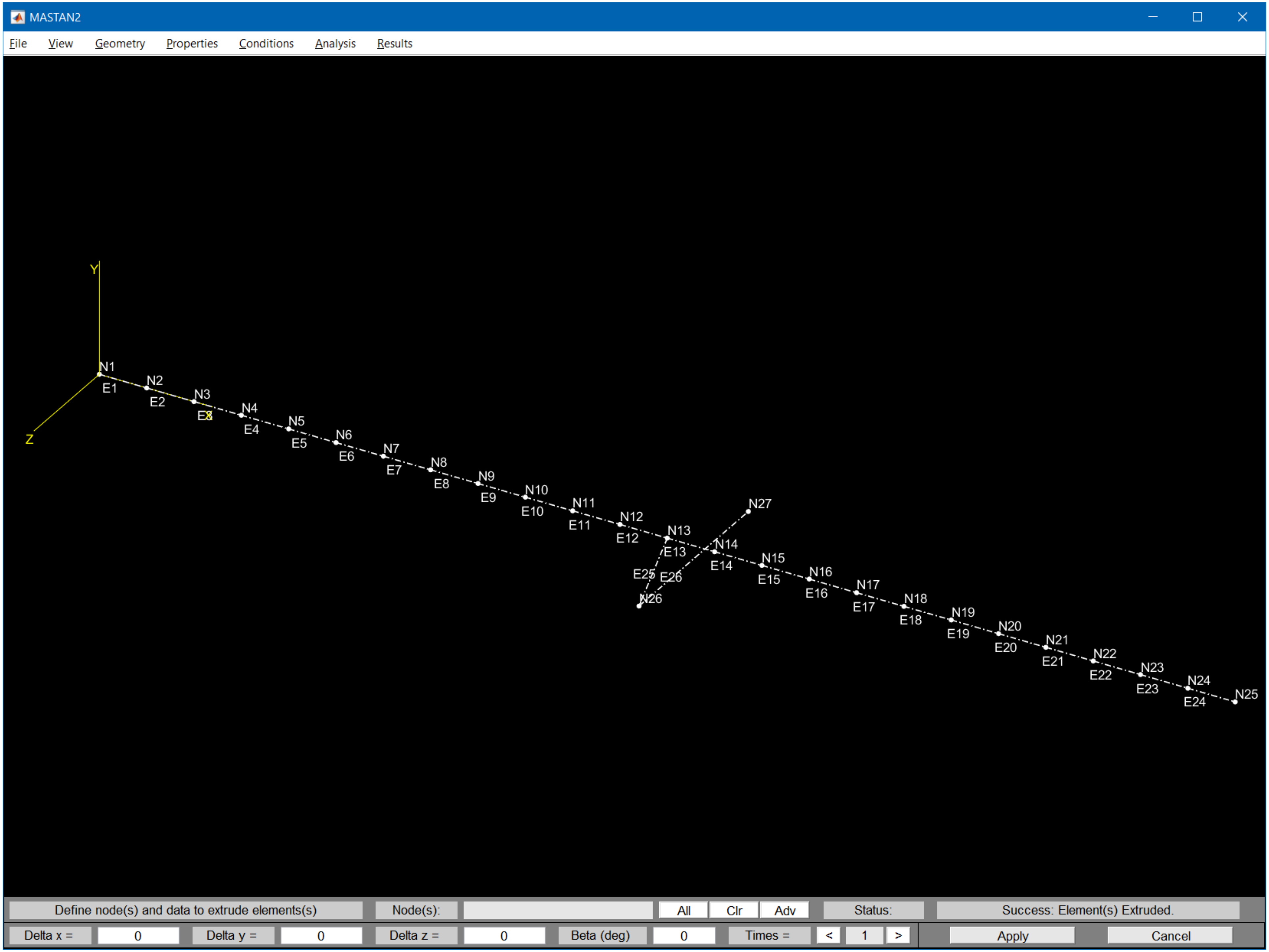
3) At the bottom menu bar, click on node **13** to populate the list of nodes. Click in the edit box to the right of **Delta y =** and change **0** to **-1.8405**. Click in the edit box to the right of **Delta z =** and change **0** to **2.0677**.

4) Click on the **Apply** Button to define an element connecting the centroid to the lower corner of the pour stop cross section which was labeled N2 in the previous cross-section definition.

5) At the bottom menu bar, click on **Node 26** to populate the list of nodes. Click in the edit box to the right of **Delta z =** and change **0** to **-8**. Click on the **Apply** Button.

Note: If there was another position along the bottom flat where the deflection was of interest, 2 elements could be extruded by altering the process to extrude twice in the z direction with the appropriate **Delta z** values or by subdividing the bottom element and then moving the new node to the appropriate location.


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
N1
N2
N3
N4
N5
N6
N7
N8
N9
N10
N11
N12
N13
N14
N15
N16
N17
N18
N19
N20
N21
N22
N23
N24
N25
N26
E1
E2
E4
E5
E6
E7
E8
E9
E10
E11
E12
E13
E14
E15
E16
E17
E18
E19
E20
E21
E22
E23
E24
E25
Define node(s) and data to extrude elements(s)
Node(s):
All
Clr
Adv
Status:
Success: Element(s) Extruded.
Delta x =
0
Delta y =
0
Delta z =
0
Beta (deg)
0
Times =
<
1
>
Apply
Cancel


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
N1
N2
N3
N4
N5
N6
N7
N8
N9
N10
N11
N12
N13
N14
N15
N16
N17
N18
N19
N20
N21
N22
N23
N24
N25
N26
N27
E1
E2
E3
E4
E5
E6
E7
E8
E9
E10
E11
E12
E13
E14
E15
E16
E17
E18
E19
E20
E21
E22
E23
E24
E25
E26
Define node(s) and data to extrude elements(s)
Node(s):
All
Clr
Adv
Status:
Success: Element(s) Extruded.
Delta x =
0
Delta y =
0
Delta z =
0
Beta (deg)
0
Times =
<
1
>
Apply
Cancel

Cross Section Visualization Finish

1) Continue defining the cross section by clicking on **Node 26** to populate the list of nodes. Click in the edit box to the right of **Delta y =** and change **0** to **6.6862**. Click on the **Apply** Button.

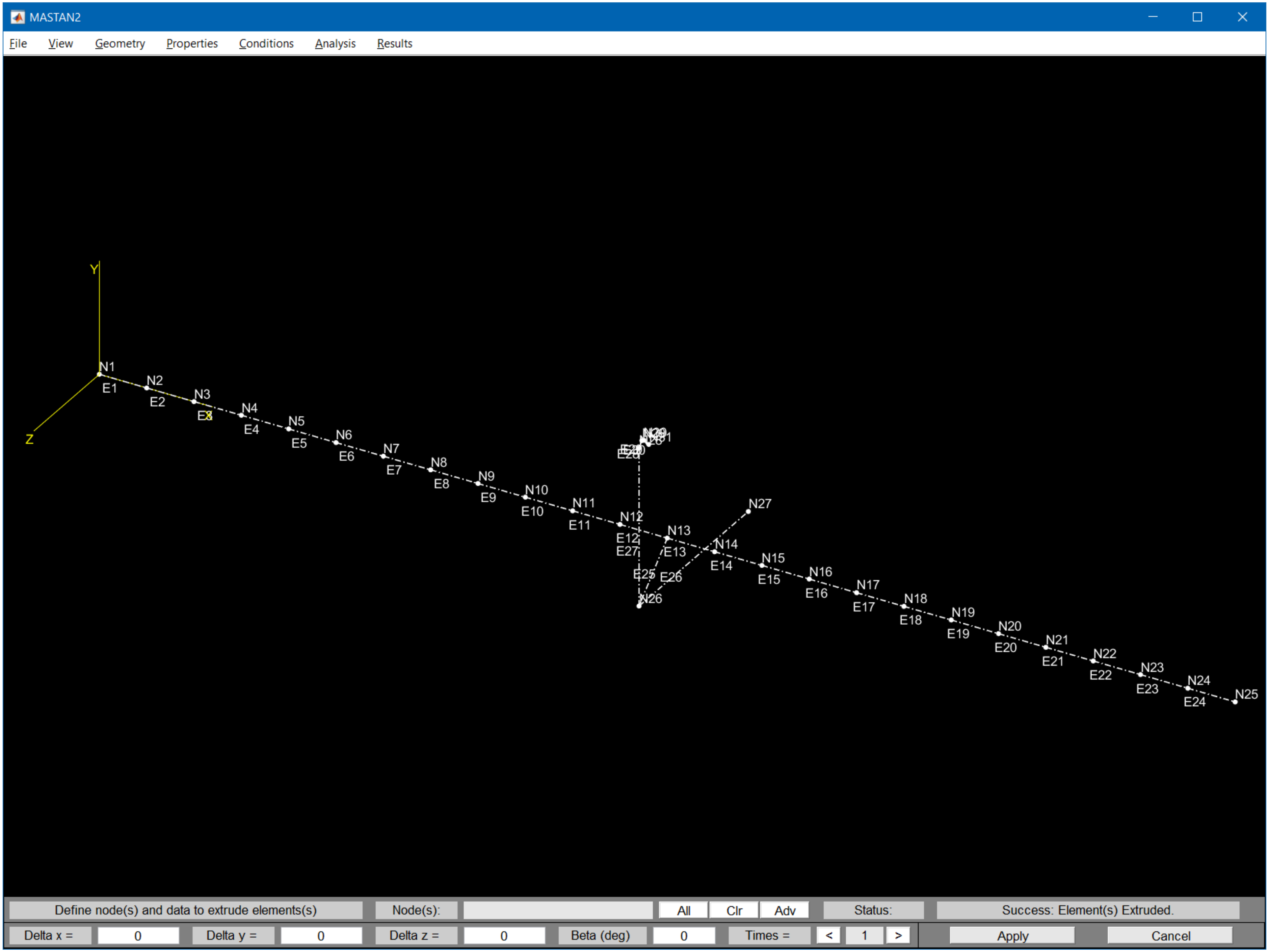
2) Click on **Node 28** to populate the list of nodes. Click in the edit box to the right of **Delta y =** and change **0** to **0.2092**. Click in the edit box to the right of **Delta z =** and change **0** to **-0.2092**. Click on the **Apply** Button.

3) Click on **Node 29** to populate the list of nodes. Click in the edit box to the right of **Delta y =** and change **0** to **-0.0613**. Click in the edit box to the right of **Delta z =** and change **0** to **-0.1479**. Click on the **Apply** Button.

4) Click on **Node 30** to populate the list of nodes. Click in the edit box to the right of **Delta y =** and change **0** to **-0.3535**. Click in the edit box to the right of **Delta z =** and change **0** to **-0.3535**. Click on the **Apply** Button.

Note: For the area where nodes are close together, from the **View** menu select **Zoom Box** to be able to zoom in and easier identify which node you are clicking.

MASTAN2


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Define node(s) and data to extrude elements(s)
Node(s):
All
Clr
Adv
Status:
Success: Element(s) Extruded.
Delta x =
0
Delta y =
0
Delta z =
0
Beta (deg)
0
Times =
<
1
>
Apply
Cancel

Creating Section Properties

1) From the **Properties** menu select **Define Section**.

2) At the bottom menu bar, click on the pop-up menu on the far right that currently displays **Basic**. Click on **Advanced**.

3) Click on **MSASect**.

4) After the interface loads, click on **General** to select the radio button next to it.

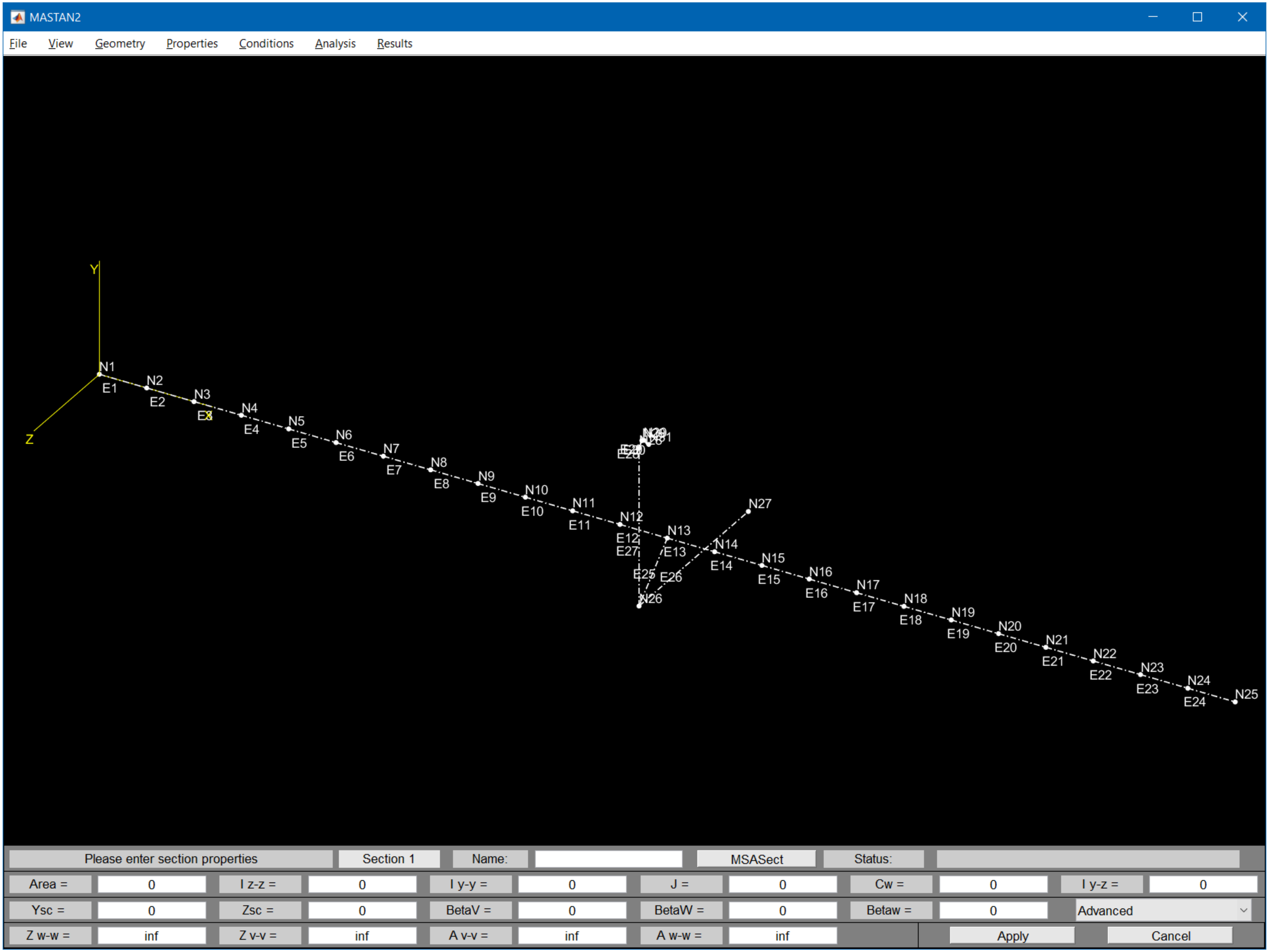
5) Click **Next** to open the editable dialog boxes.

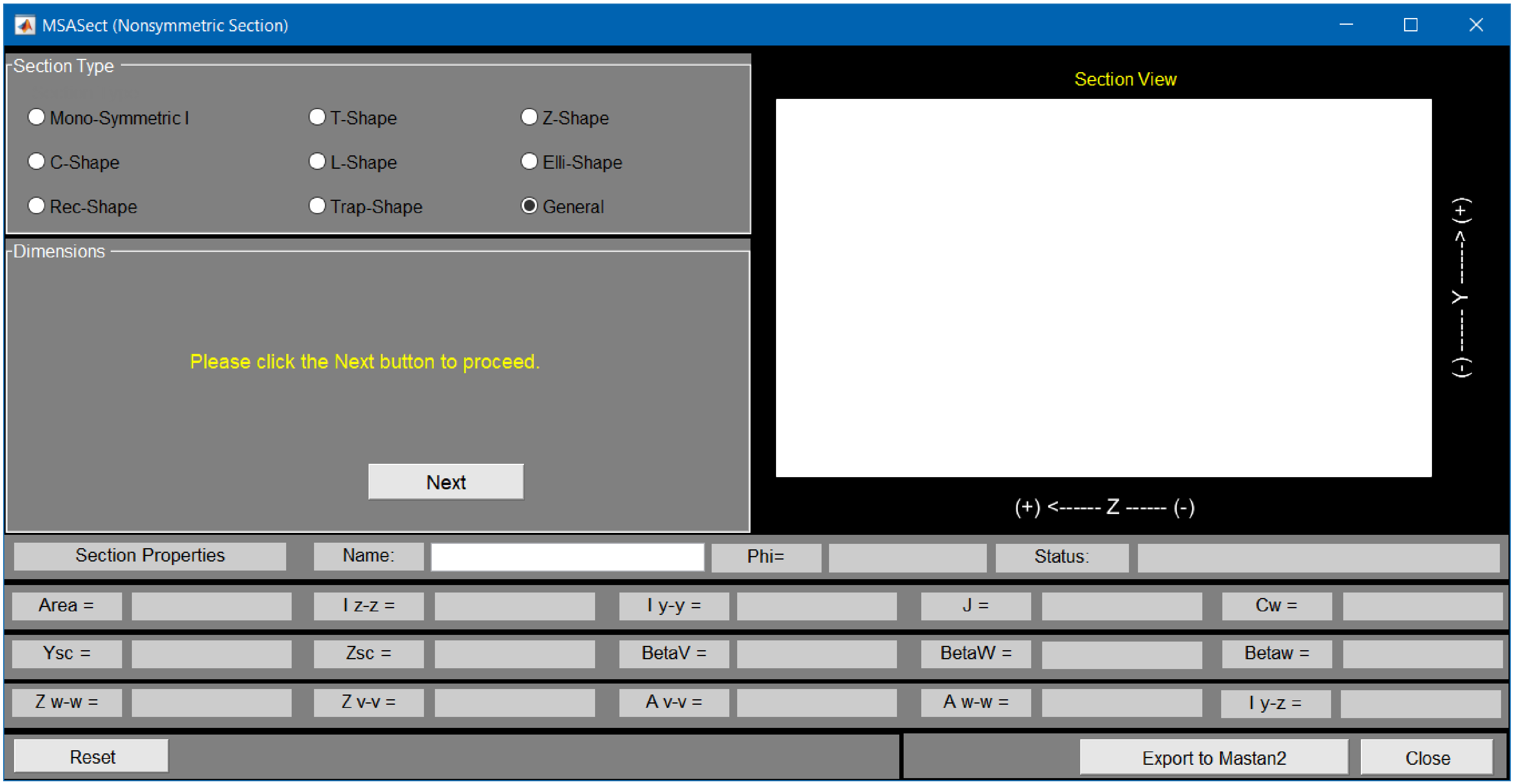
6) Click the edit box to the right of **ID:** and enter **1**. Click the edit box to the right of **Z-Coor.=** and enter **-8**. Click the edit box to the right of **Y-Coor.=** and enter **0**. Click **Add** to save the node.

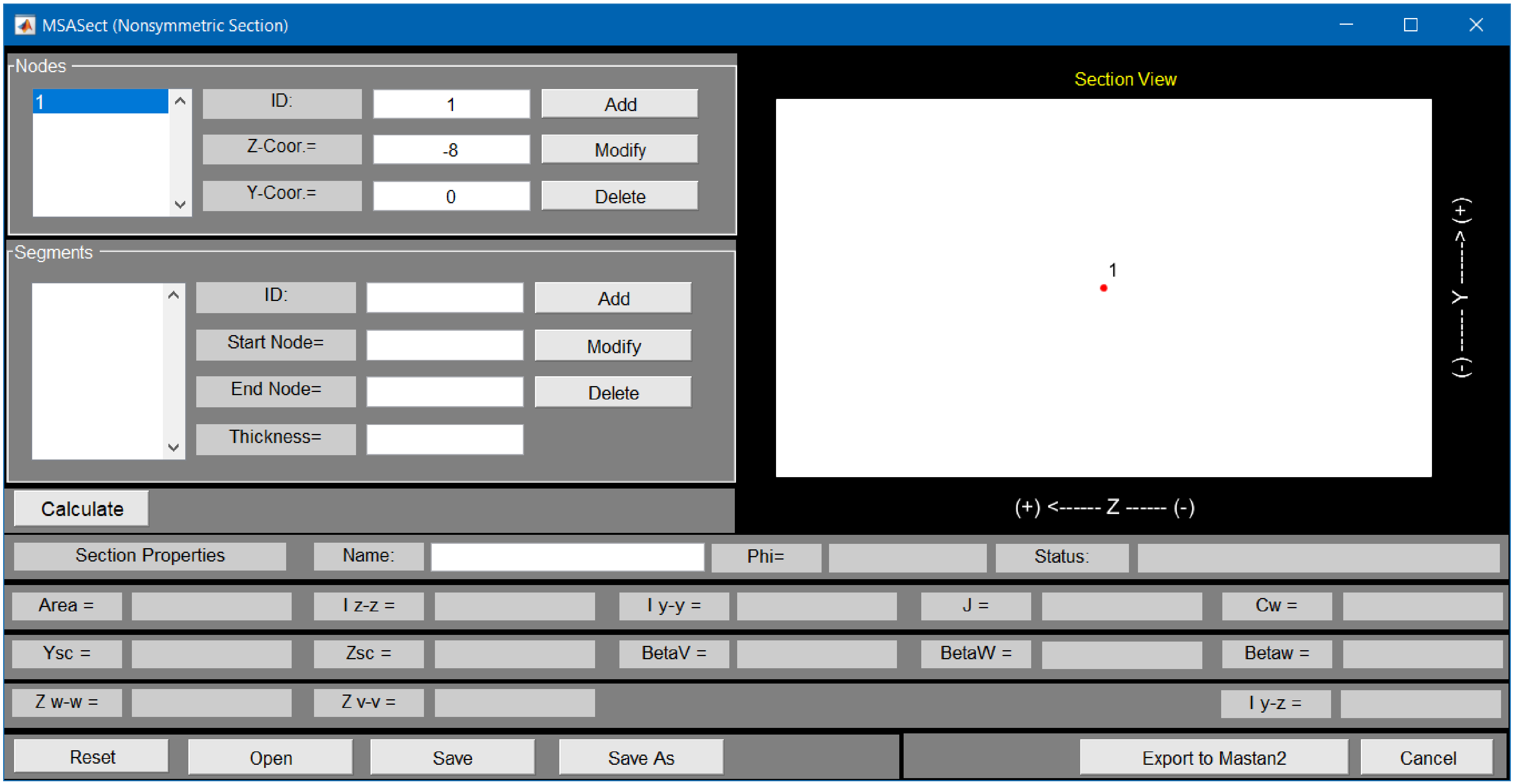
7) Repeat entering the values below for each node clicking **Add** after each one.

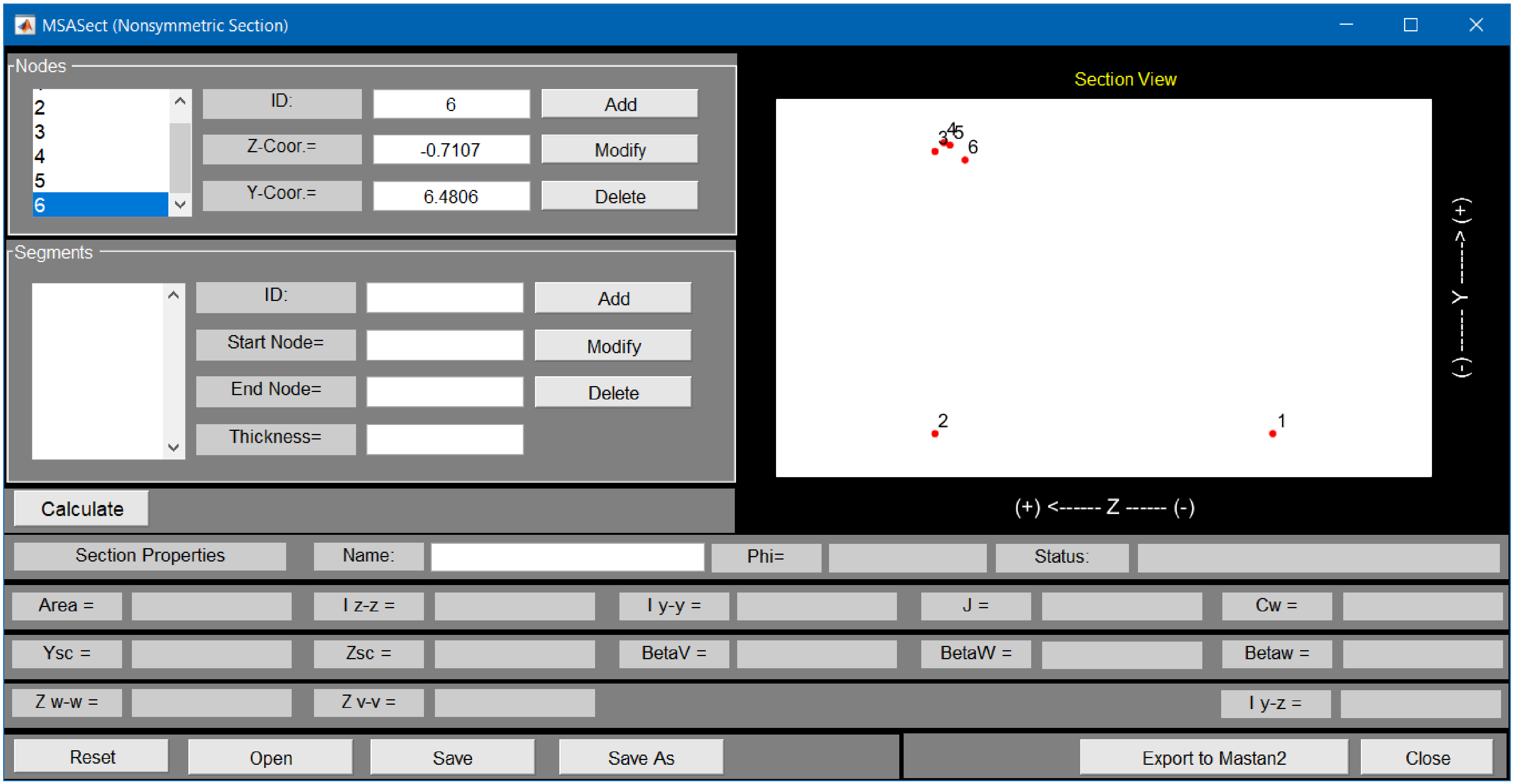
ID:	2	3	4	5	6
Z-Coor.=	0	0	-0.2092	-0.3571	-0.7107
Y-Coor.=	0	6.6862	6.8954	6.8341	6.4806

MASTAN2


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
Please enter section properties
Section 1
Name:
MSASect
Status:
Area = 0
I z-z = 0
I y-y = 0
J = 0
Cw = 0
I y-z = 0
Ysc = 0
Zsc = 0
BetaV = 0
BetaW = 0
Betaw = 0
Advanced
Z w-w = inf
Z v-v = inf
A v-v = inf
A w-w = inf
Apply
Cancel


MSASect (Nonsymmetric Section)
Section Type
Mono-Symmetric I
T-Shape
Z-Shape
C-Shape
L-Shape
Elli-Shape
Rec-Shape
Trap-Shape
General
Dimensions
Please click the Next button to proceed.
Next
Section View
(+) <------ Y ------ (-)
(+) <------ Z ------ (-)
Section Properties
Name:
Phi=
Status:
Area =
I z-z =
I y-y =
J =
Cw =
Ysc =
Zsc =
BetaV =
BetaW =
Betaw =
Z w-w =
Z v-v =
A v-v =
A w-w =
I y-z =
Reset
Export to Mastan2
Close


MSASect (Nonsymmetric Section)
Nodes
ID:
1
Add
Z-Coor.=
-8
Modify
Y-Coor.=
0
Delete
Segments
ID:
Add
Start Node=
Modify
End Node=
Delete
Thickness=
Calculate
Section View
1
(+) <------ Y ------ (-)
(+) <------ Z ------ (-)
Section Properties
Name:
Phi=
Status:
Area =
I z-z =
I y-y =
J =
Cw =
Ysc =
Zsc =
BetaV =
BetaW =
Betaw =
Z w-w =
Z v-v =
I y-z =
Reset
Open
Save
Save As
Export to Mastan2
Cancel


MSASect (Nonsymmetric Section)
Nodes
2
3
4
5
6
ID:
6
Add
Z-Coor.=
-0.7107
Modify
Y-Coor.=
6.4806
Delete
Segments
ID:
Add
Start Node=
Modify
End Node=
Delete
Thickness=
Calculate
Section View
(+) <------ Z ------ (-)
(-) ------ Y ------> (+)
Section Properties
Name:
Phi=
Status:
Area =
I z-z =
I y-y =
J =
Cw =
Ysc =
Zsc =
BetaV =
BetaW =
Betaw =
Z w-w =
Z v-v =
I y-z =
Reset
Open
Save
Save As
Export to Mastan2
Close

Creating Section Properties – Cont.

1) Under the segments section, click the edit box to the right of **ID:** and enter **1**. Click the edit box to the right of **Start Node=** and enter **1**. Click the edit box to the right of **End Node=** and enter **2**. Click the edit box to the right of **Thickness=** and enter **0.1046**. Click **Add** to save the segment.

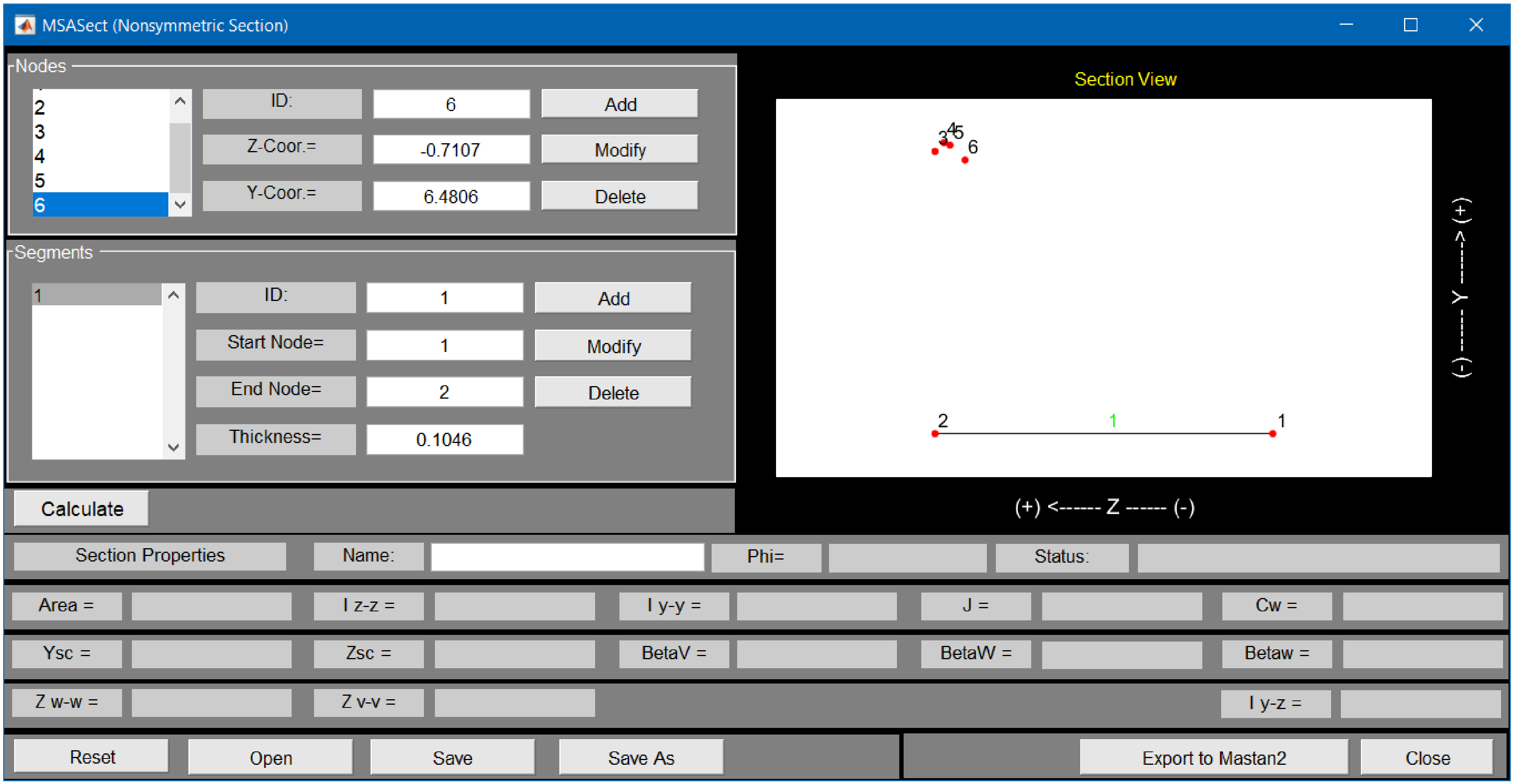
2) Repeat entering the values for each segment updating the **ID:**, **Start Node=**, and **End Node=** values by adding **1** to each number until all 5 segments are entered. Click **Add** after each to save.

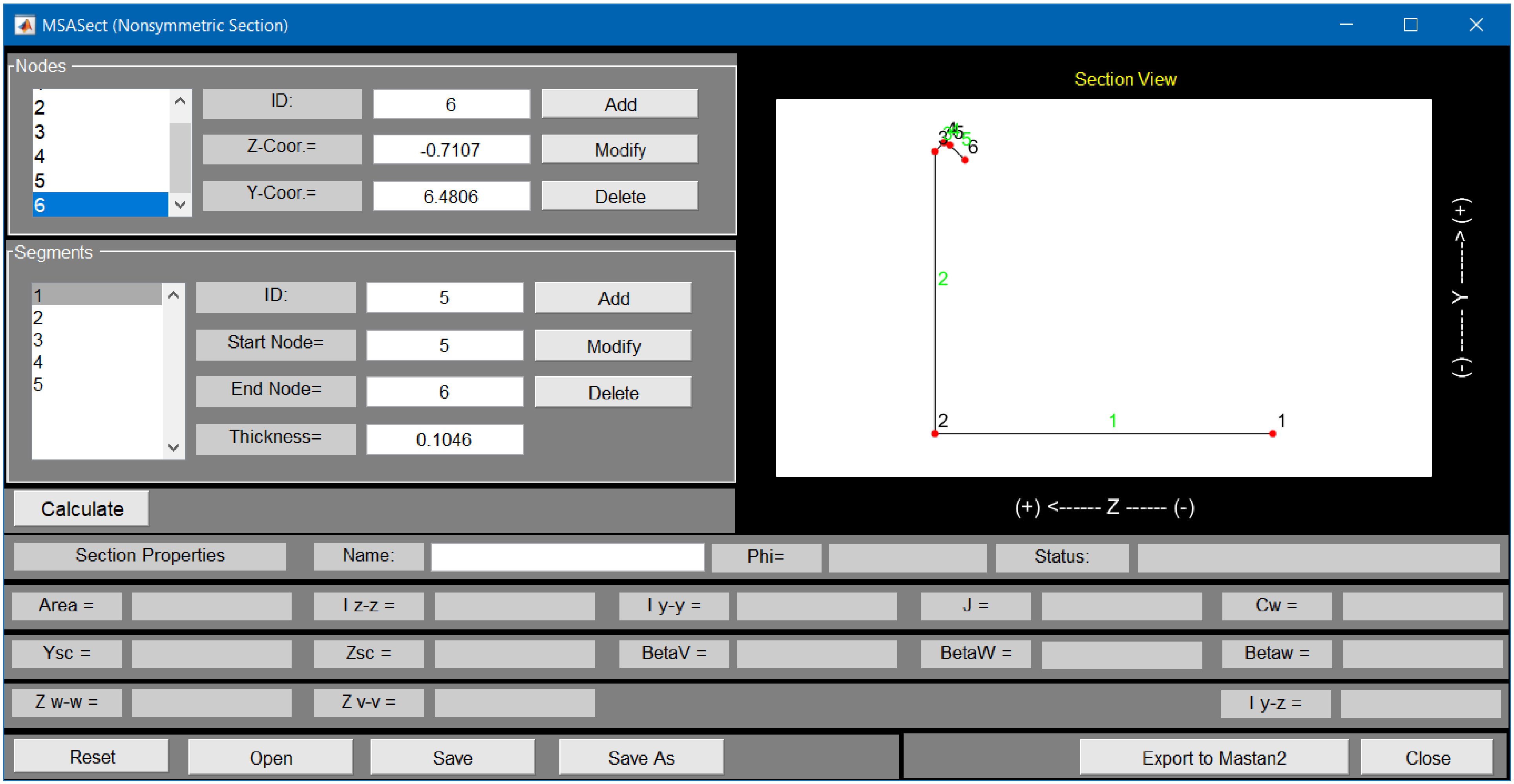
3) Click **Calculate** to determine the properties.

4) Click edit box to right of **Name:** and enter **Pour Stop**.

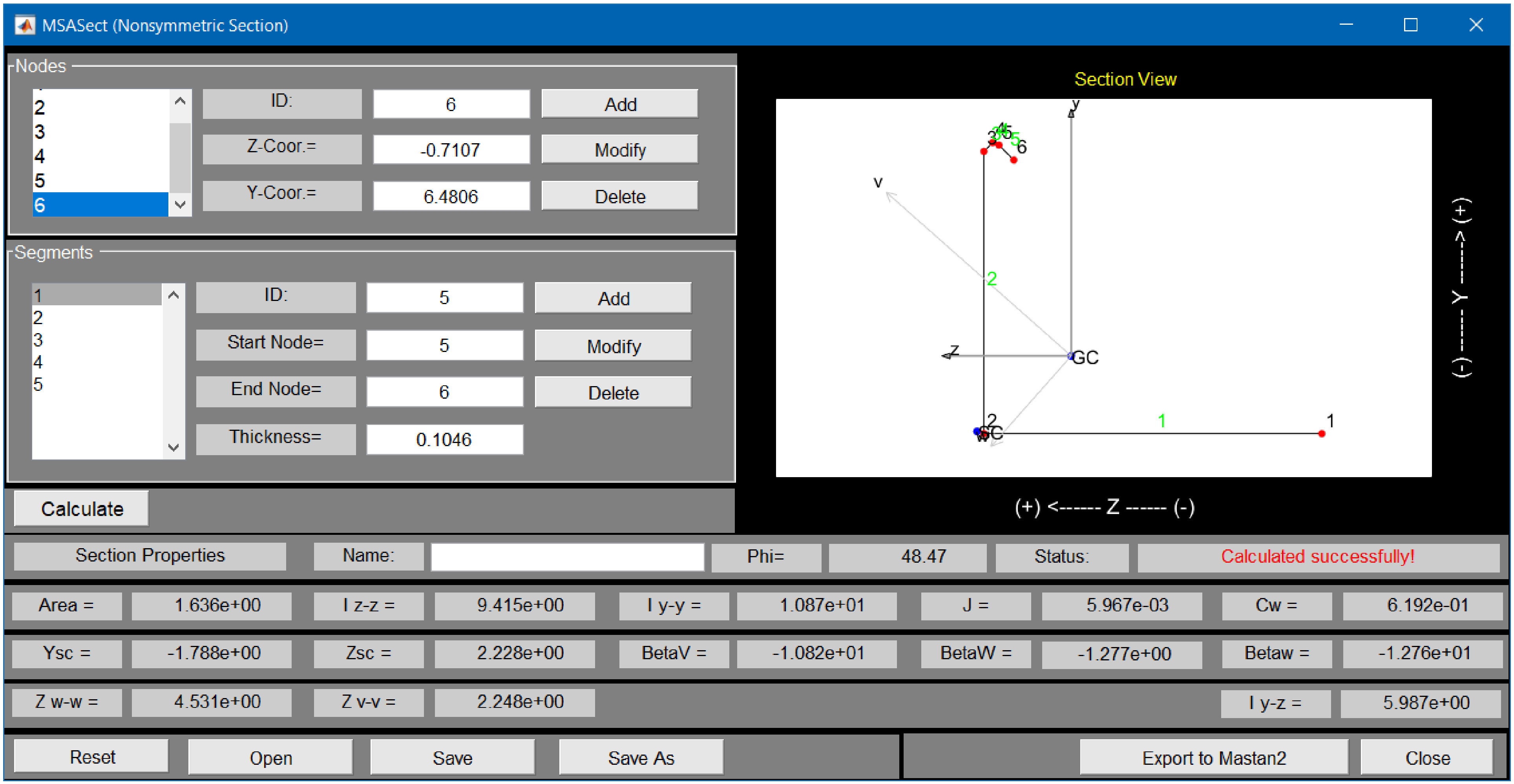
5) Click **Export to MASTAN2** to copy values to main program. Then click **Close** to return.

6) Click **Apply** to save Section 1.

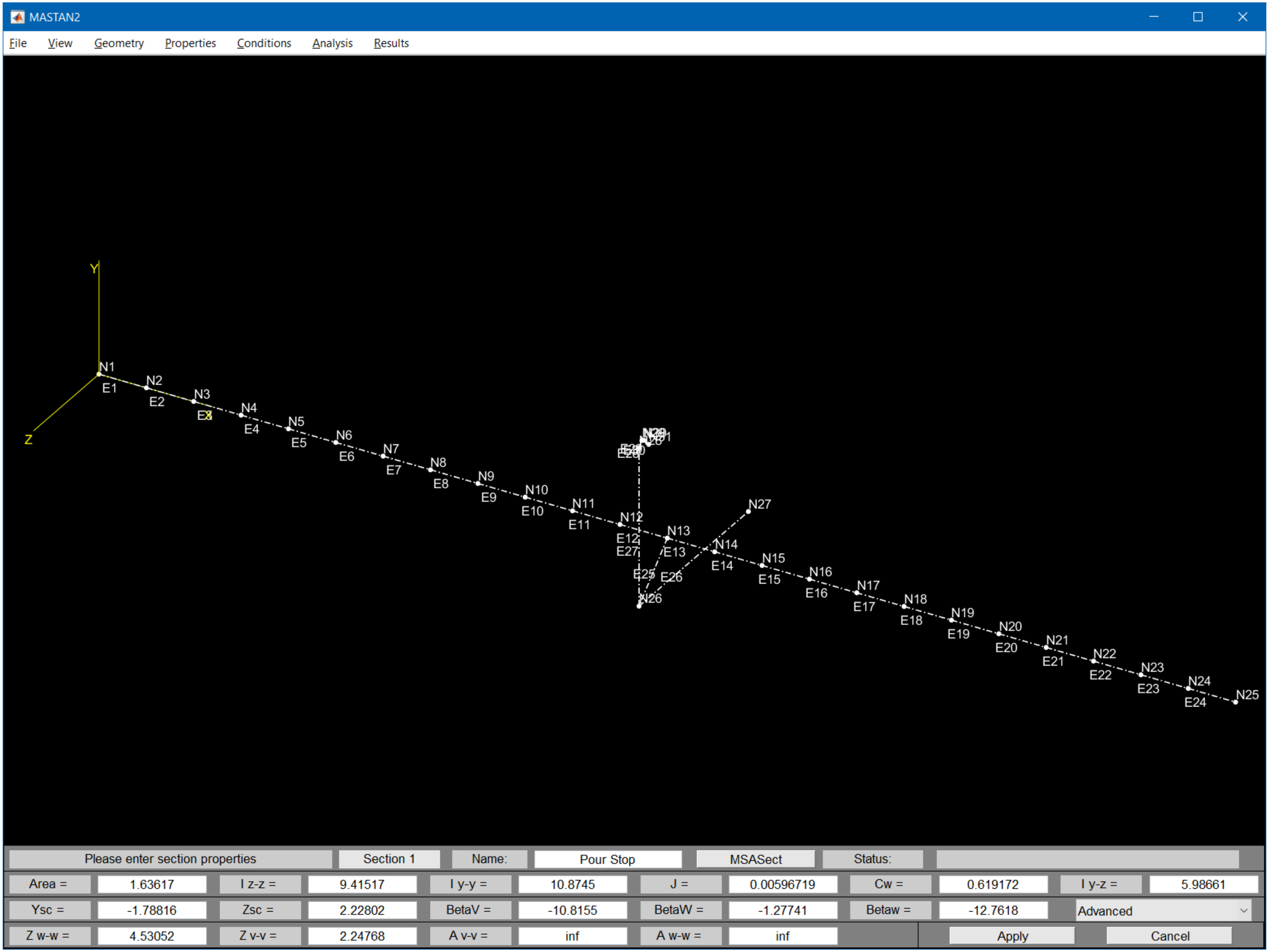

MSASect (Nonsymmetric Section)
Nodes
2
3
4
5
6
ID:
6
Add
Z-Coor.=
-0.7107
Modify
Y-Coor.=
6.4806
Delete
Segments
1
ID:
1
Add
Start Node=
1
Modify
End Node=
2
Delete
Thickness=
0.1046
Calculate
Section View
(+) <------ Z ------ (-)
(-) ------ Y ------> (+)
Section Properties
Name:
Phi=
Status:
Area =
I z-z =
I y-y =
J =
Cw =
Ysc =
Zsc =
BetaV =
BetaW =
Betaw =
Z w-w =
Z v-v =
I y-z =
Reset
Open
Save
Save As
Export to Mastan2
Close


MSASect (Nonsymmetric Section)
Nodes
2
3
4
5
6
ID:
6
Add
Z-Coor.=
-0.7107
Modify
Y-Coor.=
6.4806
Delete
Segments
1
2
3
4
5
ID:
5
Add
Start Node=
5
Modify
End Node=
6
Delete
Thickness=
0.1046
Calculate
Section View
(+) <------ Z ------ (-)
(-) ------ Y ------> (+)
Section Properties
Name:
Phi=
Status:
Area =
I z-z =
I y-y =
J =
Cw =
Ysc =
Zsc =
BetaV =
BetaW =
Betaw =
Z w-w =
Z v-v =
I y-z =
Reset
Open
Save
Save As
Export to Mastan2
Close

MASTAN2


MSASect (Nonsymmetric Section)
Nodes
2
3
4
5
6
ID:
6
Add
Z-Coor.=
-0.7107
Modify
Y-Coor.=
6.4806
Delete
Segments
1
2
3
4
5
ID:
5
Add
Start Node=
5
Modify
End Node=
6
Delete
Thickness=
0.1046
Calculate
Section View
(+) <------ Z ------ (-)
(-) ------- Y ------> (+)
GC
Section Properties
Name:
Phi=
48.47
Status:
Calculated successfully!
Area =
1.636e+00
I z-z =
9.415e+00
I y-y =
1.087e+01
J =
5.967e-03
Cw =
6.192e-01
Ysc =
-1.788e+00
Zsc =
2.228e+00
BetaV =
-1.082e+01
BetaW =
-1.277e+00
Betaw =
-1.276e+01
Z w-w =
4.531e+00
Z v-v =
2.248e+00
I y-z =
5.987e+00
Reset
Open
Save
Save As
Export to Mastan2
Close

MASTAN2


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Please enter section properties
Section 1
Name:
Pour Stop
MSASect
Status:
Area = 1.63617
I z-z = 9.41517
I y-y = 10.8745
J = 0.00596719
Cw = 0.619172
I y-z = 5.98661
Ysc = -1.78816
Zsc = 2.22802
BetaV = -10.8155
BetaW = -1.27741
Betaw = -12.7618
Advanced
Z w-w = 4.53052
Z v-v = 2.24768
A v-v = inf
A w-w = inf
Apply
Cancel

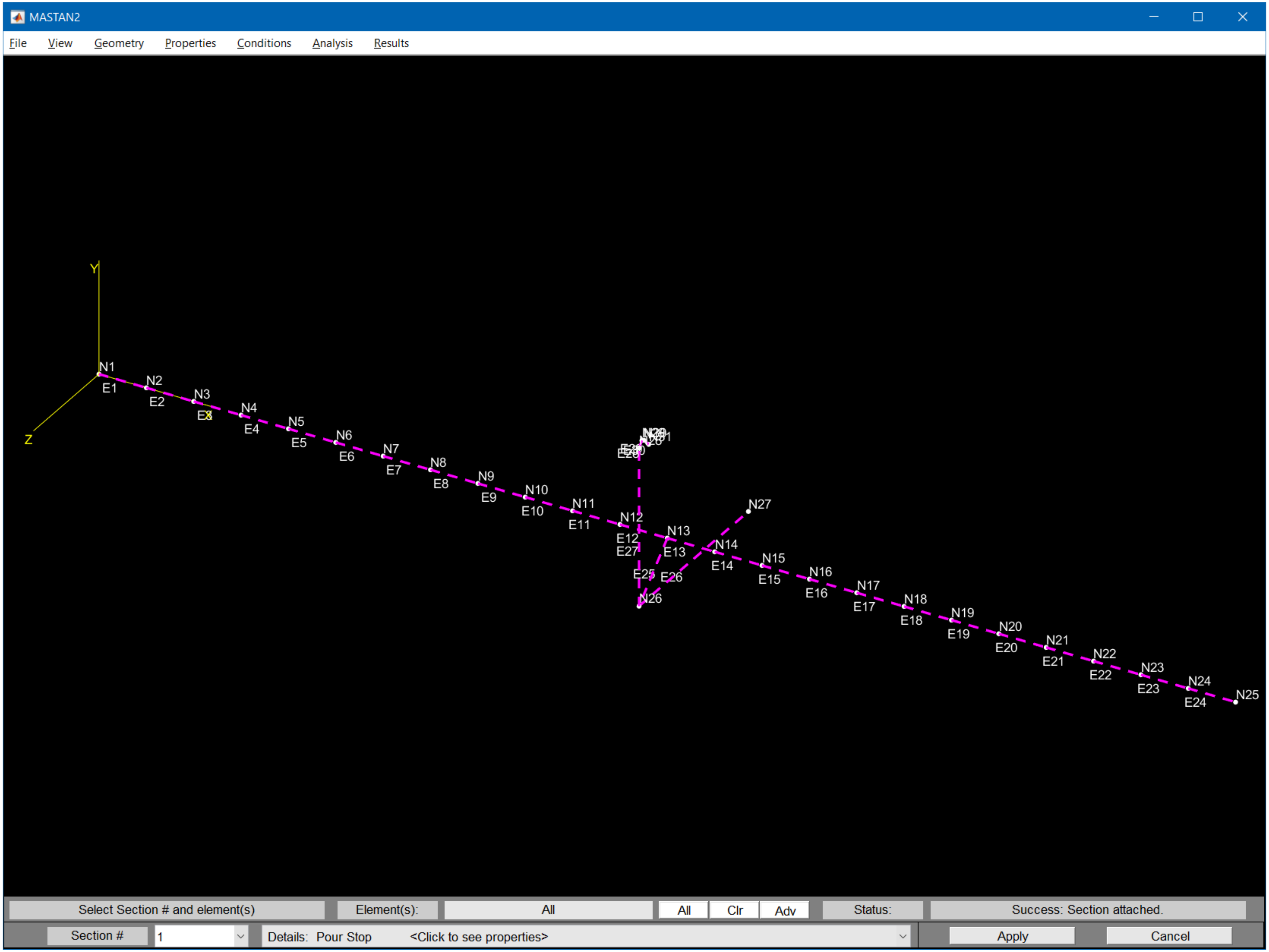
Section Properties - Assigning

1) From the **Properties** menu select **Attach Section**.

2) At the bottom menu bar, use the buttons to the right of **Element(s):** to make the list of elements.

3) Create a list of the elements by clicking the **All** button.

4) Click on the **Apply** button to assign Section 1.


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Select Section # and element(s)
Element(s):
All
All
Clr
Adv
Status:
Success: Section attached.
Section #
1
Details: Pour Stop
<Click to see properties>
Apply
Cancel

Material Properties

1) From the **Properties** menu select **Define Material**.

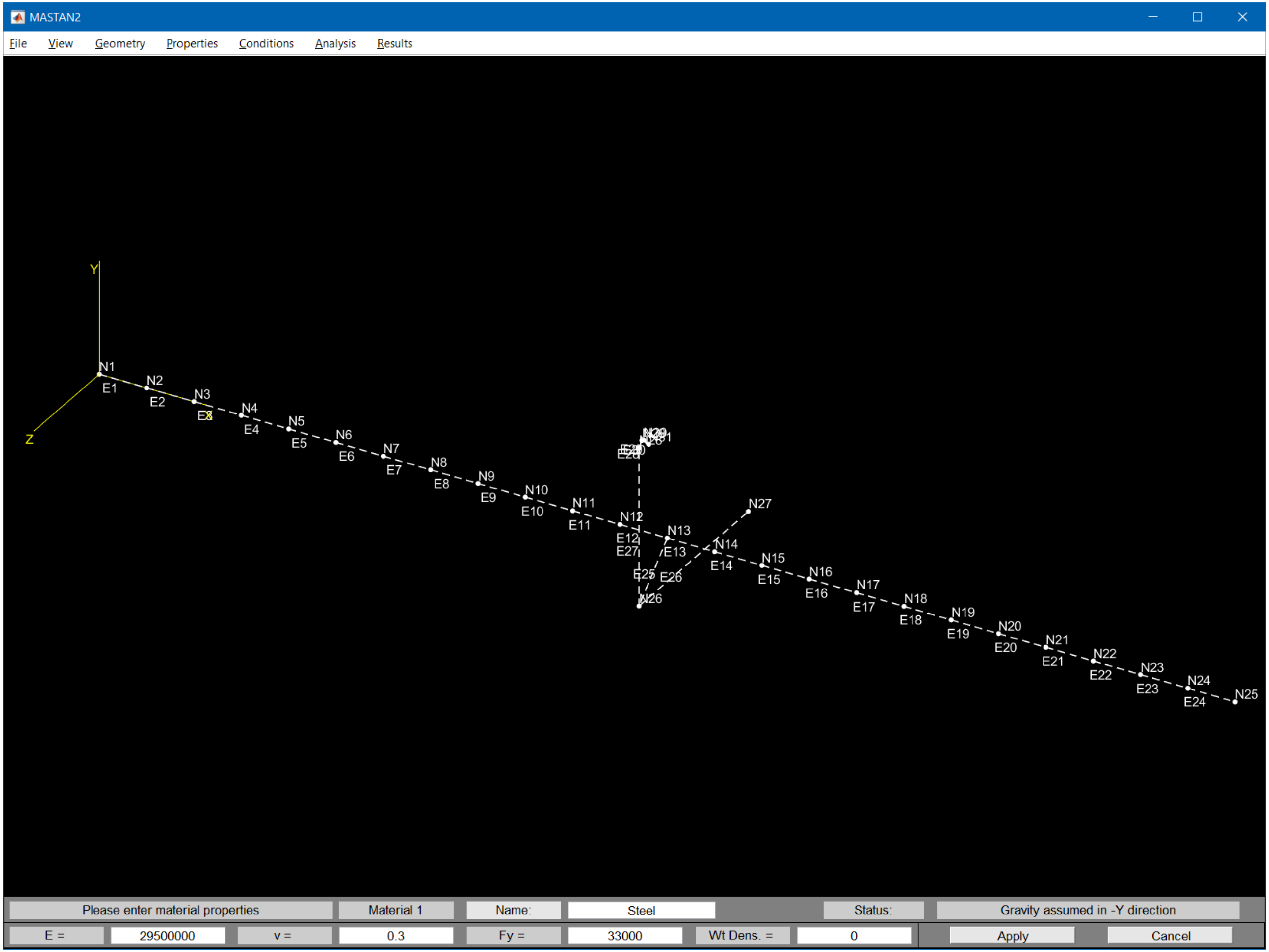
2) At the bottom menu bar, click in the edit box just to the right of **E=** and change the **0** to **29500000** (not 29,500,000). Similarly, click in the edit box just to the right of **Fy=** and change the **inf** to **33000**. Next, click in the edit box to the right of **Name:** and type **Steel**.

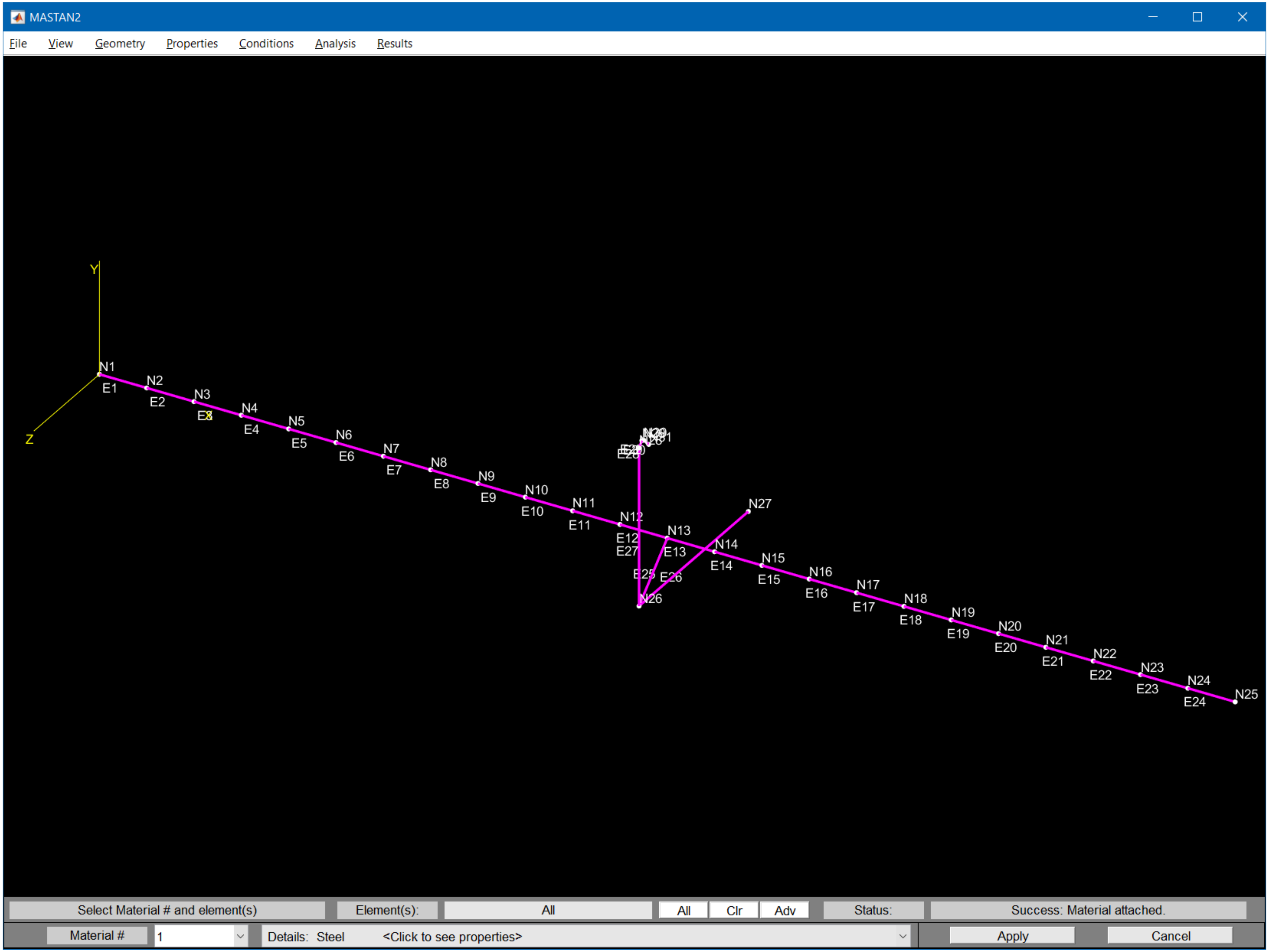
3) Click on the **Apply** button to save Material #1.

4) From the **Properties** menu select **Attach Material**.

5) At the bottom menu bar, create the list of elements to be assigned the properties of Material 1 by clicking on the **All** button to the right of **Elements:**. Click on the **Apply** button.

Since the self-weight is already included in the loading summary, a self-weight was left as zero. If the self-weight was to be included through MASTAN2, a second weightless material would need to be defined and assigned to the members that are being used to visualize the cross section.


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
N1
N2
N3
N4
N5
N6
N7
N8
N9
N10
N11
N12
N13
N14
N15
N16
N17
N18
N19
N20
N21
N22
N23
N24
N25
N26
N27
E1
E2
E4
E5
E6
E7
E8
E9
E10
E11
E12
E13
E14
E15
E16
E17
E18
E19
E20
E21
E22
E23
E24
E25
E26
E27
Please enter material properties
Material 1
Name:
Steel
Status:
Gravity assumed in -Y direction
E =
29500000
v =
0.3
Fy =
33000
Wt Dens. =
0
Apply
Cancel


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
N1 N2 N3 N4 N5 N6 N7 N8 N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N25
E1 E2 E3 E4 E5 E6 E7 E8 E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
N26 N27
E25 E26 E27
Select Material # and element(s)
Element(s):
All
All
Clr
Adv
Status:
Success: Material attached.
Material #
1
Details: Steel
<Click to see properties>
Apply
Cancel

Support Conditions

1) From the **Conditions** menu select **Define Fixities**.

2) At the bottom menu bar, define a pin support with torsion fixed support by clicking in the **check boxes** just to the left of **X-disp**, **Y-disp**, **Z-disp**, and **X-rot**.

3) Create the list of nodes to be assigned this fixity by clicking on node **1**.

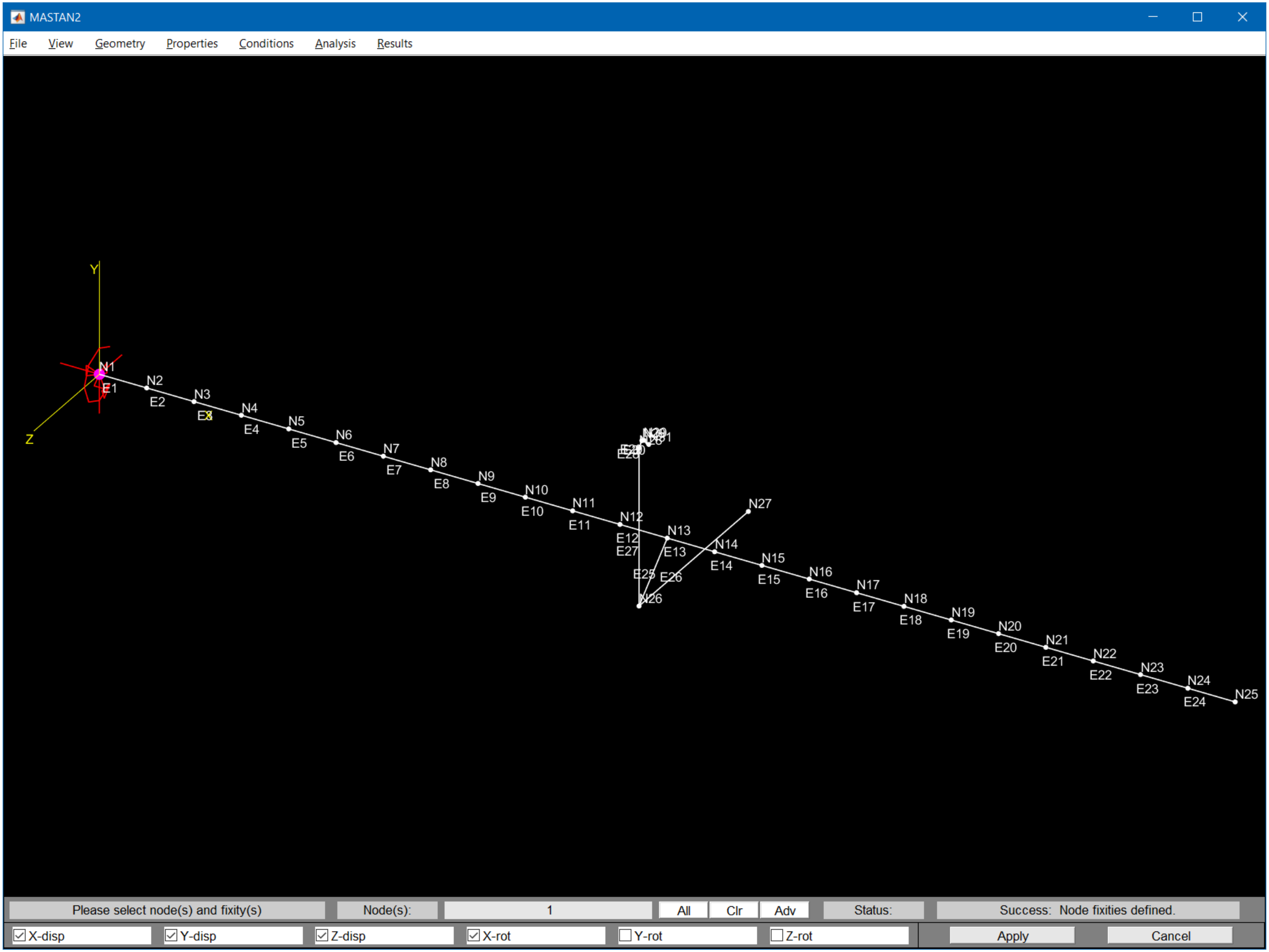
4) Click on the **Apply** button.

5) At the bottom menu bar, define a roller support with torsion fixed support by clicking in the **check boxes** just to the left of **X-disp** to release it and leave **Y-disp**, **Z-disp**, and **X-rot** constrained.

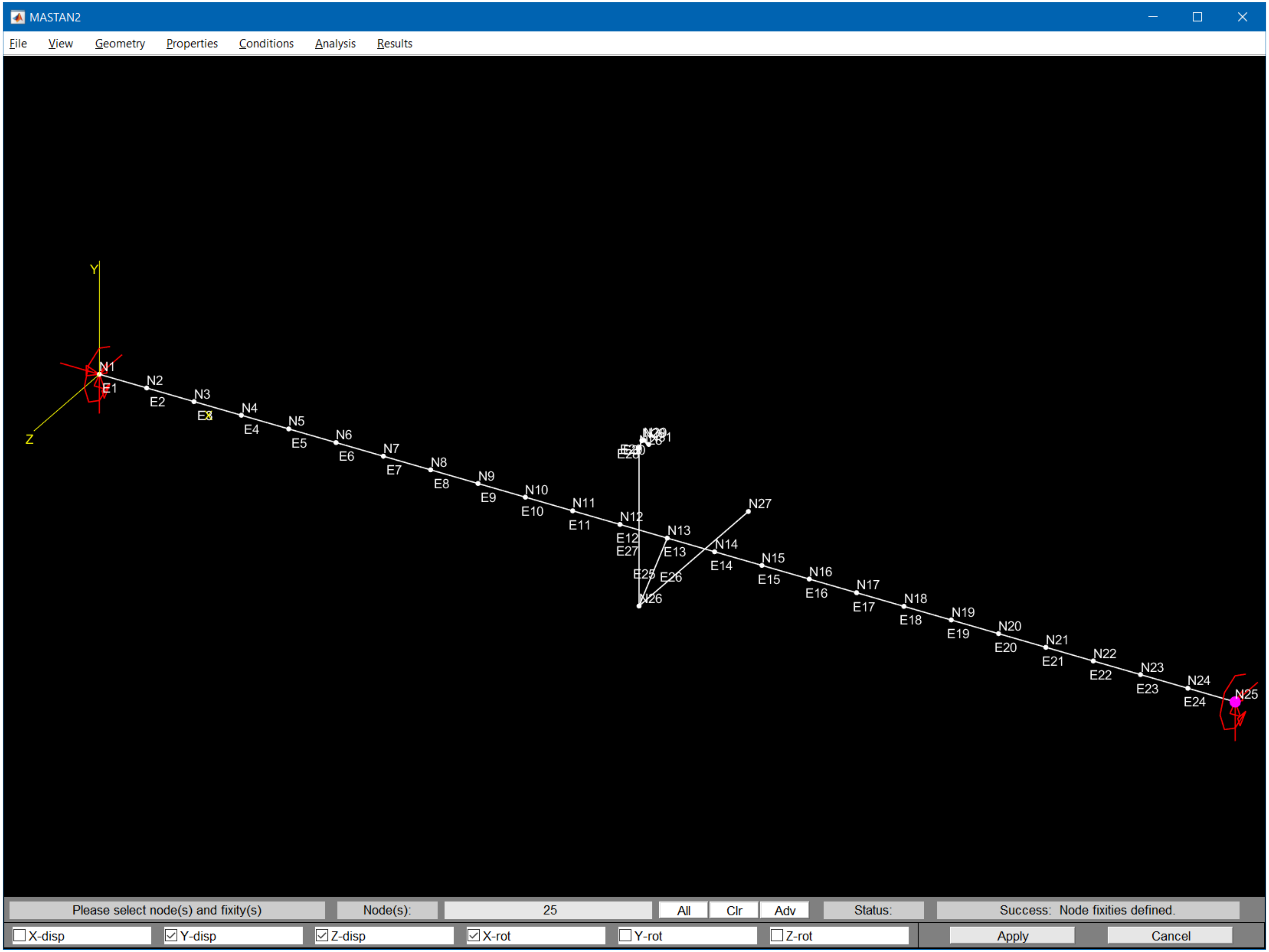
6) Click **Clr** to empty the list of nodes.

7) Create the list of noes to be assigned this fixity by clicking on node **25**.

8) Click on the **Apply** button.


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
N1 N2 N3 N4 N5 N6 N7 N8 N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N25 N26 N27
E1 E2 E4 E5 E6 E7 E8 E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24 E25 E26 E27
Please select node(s) and fixity(s)
Node(s):
1
All
Clr
Adv
Status:
Success: Node fixities defined.
X-disp
Y-disp
Z-disp
X-rot
Y-rot
Z-rot
Apply
Cancel

MASTAN2


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Please select node(s) and fixity(s)
Node(s):
25
All
Clr
Adv
Status:
Success: Node fixities defined.
X-disp
Y-disp
Z-disp
X-rot
Y-rot
Z-rot
Apply
Cancel

Adding Warping Effects

1) From the **Geometry** menu select **Define Connections** and submenu option **Torsion**.

2) At the bottom menu bar, click on the menu to the right of **Warping Restraint for Node i** and set the value to **Continuous**. Repeat this for the **Warping Restraint for Node j**.

3) Click the **Adv** button to open pop-up menu. Click the check box next to the **X-axis** option.

4) Create the list of elements to be assigned continuous warping by clicking on the **Add** button to the advanced menu. Then click on the **Apply** button.

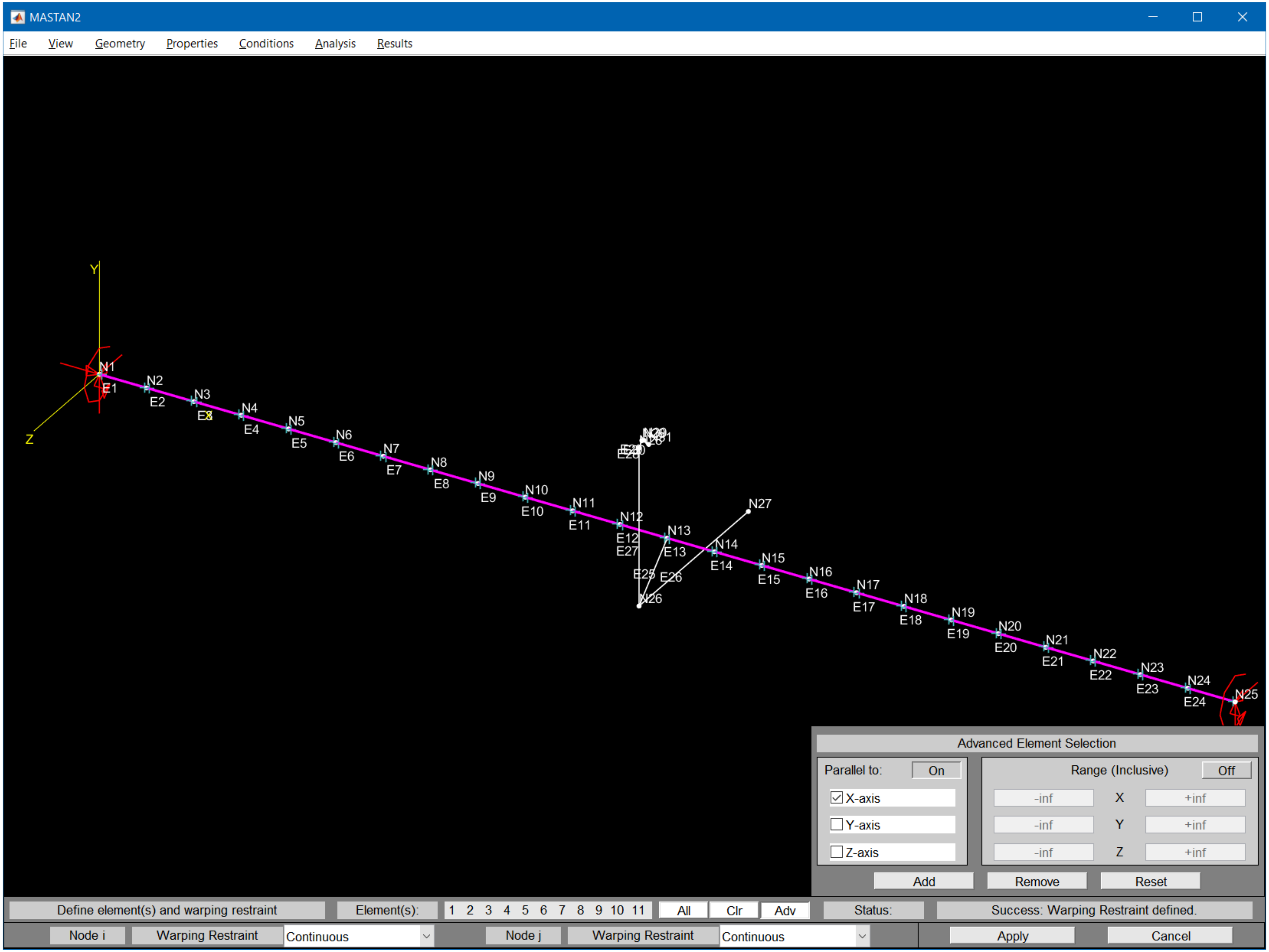
5) Click **Clr** to empty the list of elements. Click on the left most element of the beam.

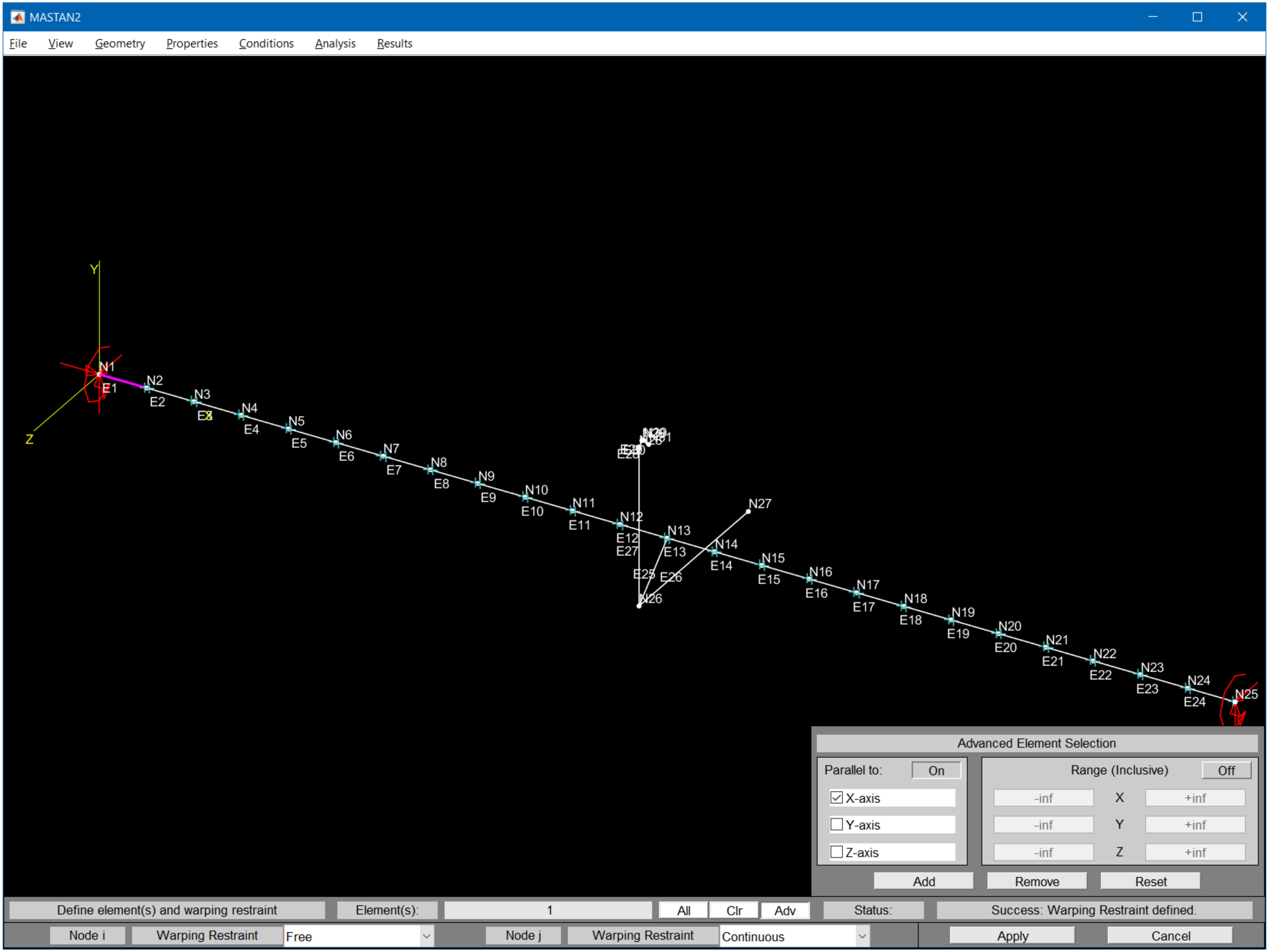
6) Click on the menu to the right of **Warping Restraint for Node i** and set the value to **Free**. Node j is set from the previous step. Click on the **Apply** button.

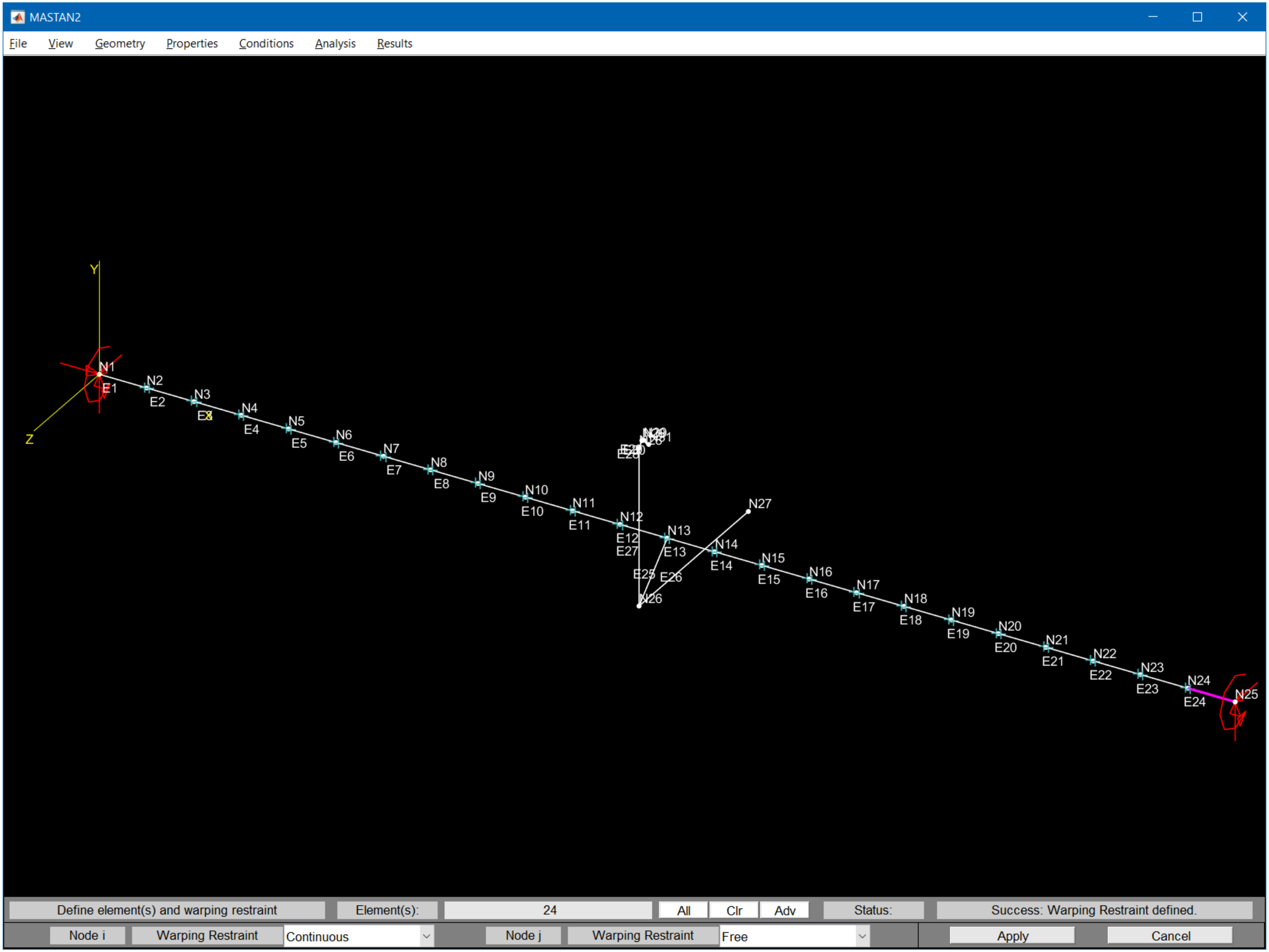
7) Click **Clr** to empty the list of elements. Click on the right most element of the beam. This might require you to click **Adv** to close the pop-up menu.

8) Click on the menu to the right of **Warping Restraint for Node i** and set the value to **Continuous**. Click on the menu to the right of **Warping Restraint for Node j** and set the value to **Free**.

9) Click on the **Apply** button.


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
N1
E1
N2
E2
N3
N4
E4
N5
E5
N6
E6
N7
E7
N8
E8
N9
E9
N10
E10
N11
E11
N12
E12
E27
N13
E13
N14
E14
N15
E15
N16
E16
N17
E17
N18
E18
N19
E19
N20
E20
N21
E21
N22
E22
N23
E23
N24
E24
N25
N26
N27
E25
E26
Advanced Element Selection
Parallel to:
On
X-axis
Y-axis
Z-axis
Range (Inclusive)
Off
-inf
+inf
X
Y
Z
Add
Remove
Reset
Define element(s) and warping restraint
Element(s):
1 2 3 4 5 6 7 8 9 10 11
All
Clr
Adv
Status:
Success: Warping Restraint defined.
Node i
Warping Restraint
Continuous
Node j
Warping Restraint
Continuous
Apply
Cancel


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Advanced Element Selection
Parallel to:
On
Range (Inclusive)
Off
X-axis
Y-axis
Z-axis
-inf
+inf
X
Y
Z
Add
Remove
Reset
Define element(s) and warping restraint
Element(s):
1
All
Clr
Adv
Status:
Success: Warping Restraint defined.
Node i
Warping Restraint
Free
Node j
Warping Restraint
Continuous
Apply
Cancel


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
Define element(s) and warping restraint
Element(s):
24
All
Clr
Adv
Status:
Success: Warping Restraint defined.
Node i
Warping Restraint
Continuous
Node j
Warping Restraint
Free
Apply
Cancel

Loading

1) From the **Conditions** menu select **Define Forces**.

2) At the bottom menu bar, click in the edit box just to the right of **PY =** and change **0** to **-12.34**. Click in the edit box just to the right of **PZ =** and change **0** to **5.9**.

3) Click the **Adv** button to open the pop-up menu. To select the main beam nodes, change the edit box to the left of **Z** to **-1**. Change the edit box to the right of **Z** to **1**.

4) Click **Add** to add all main beam nodes.

5) Click on the **Apply** button.

6) From the **Conditions** menu select **Define Moments**.

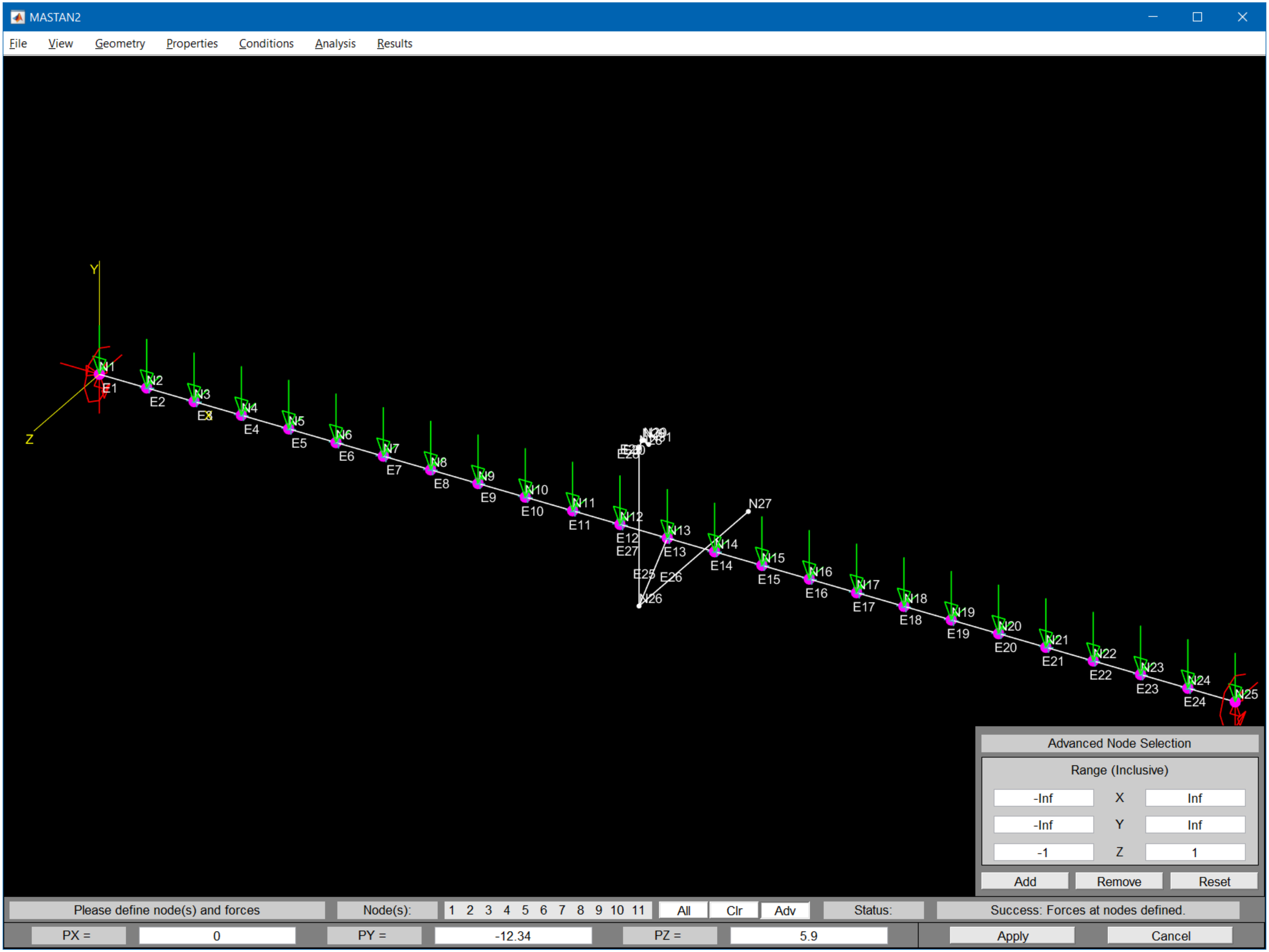
7) At the bottom menu bar, click in the edit box just to the right of **Mx =** and change **0** to **-17.42**.

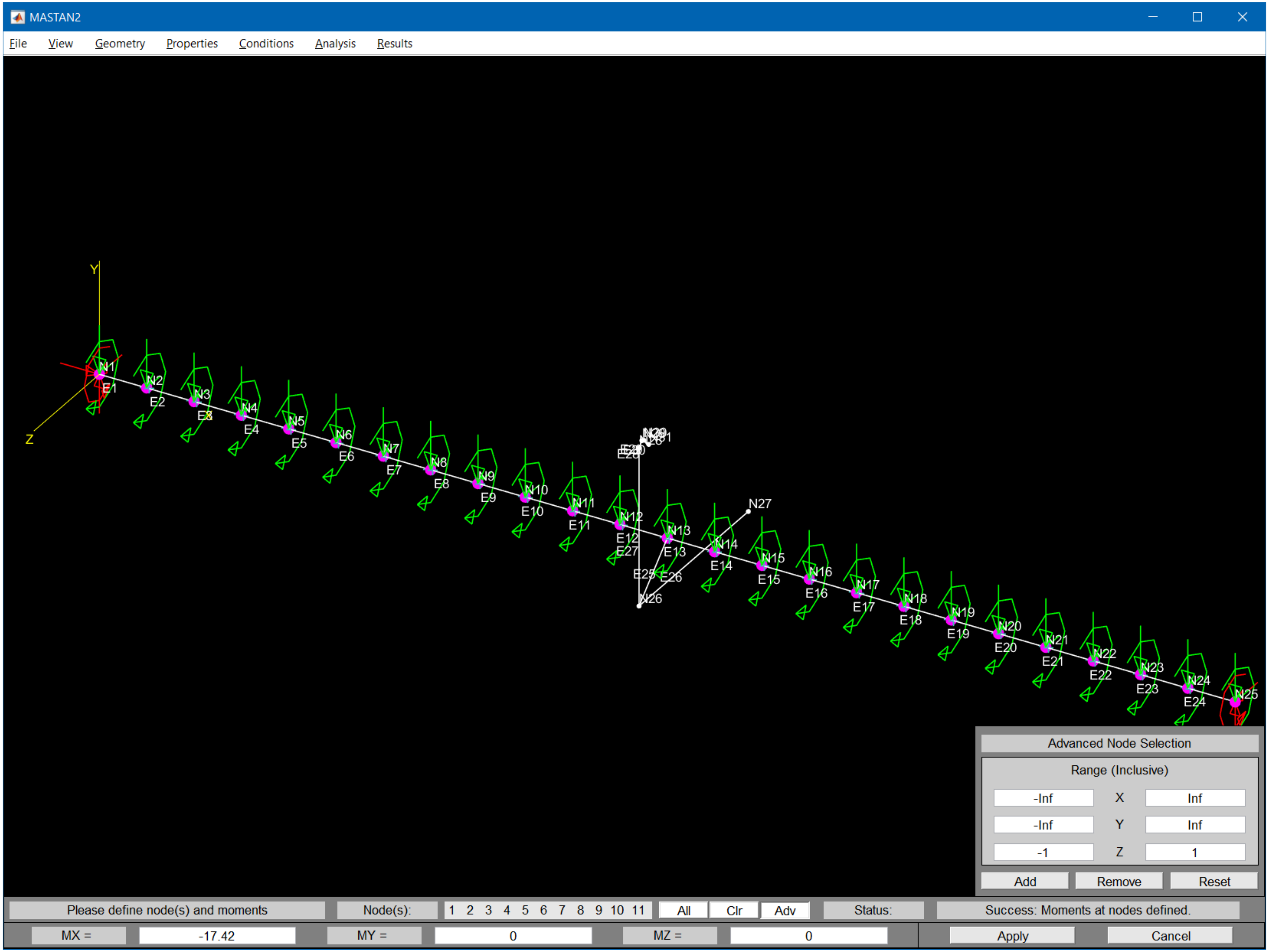
8) Click the **Adv** button to open the pop-up menu. To select the main beam nodes, change the edit box to the left of **Z** to **-1**. Change the edit box to the right of **Z** to **1**.

9) Click **Add** to add all main beam nodes.

10)Click on the **Apply** button.

MASTAN2


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Advanced Node Selection
Range (Inclusive)
-Inf X Inf
-Inf Y Inf
-1 Z 1
Add
Remove
Reset
Please define node(s) and forces
Node(s):
1 2 3 4 5 6 7 8 9 10 11
All
Clr
Adv
Status:
Success: Forces at nodes defined.
PX = 0
PY = -12.34
PZ = 5.9
Apply
Cancel

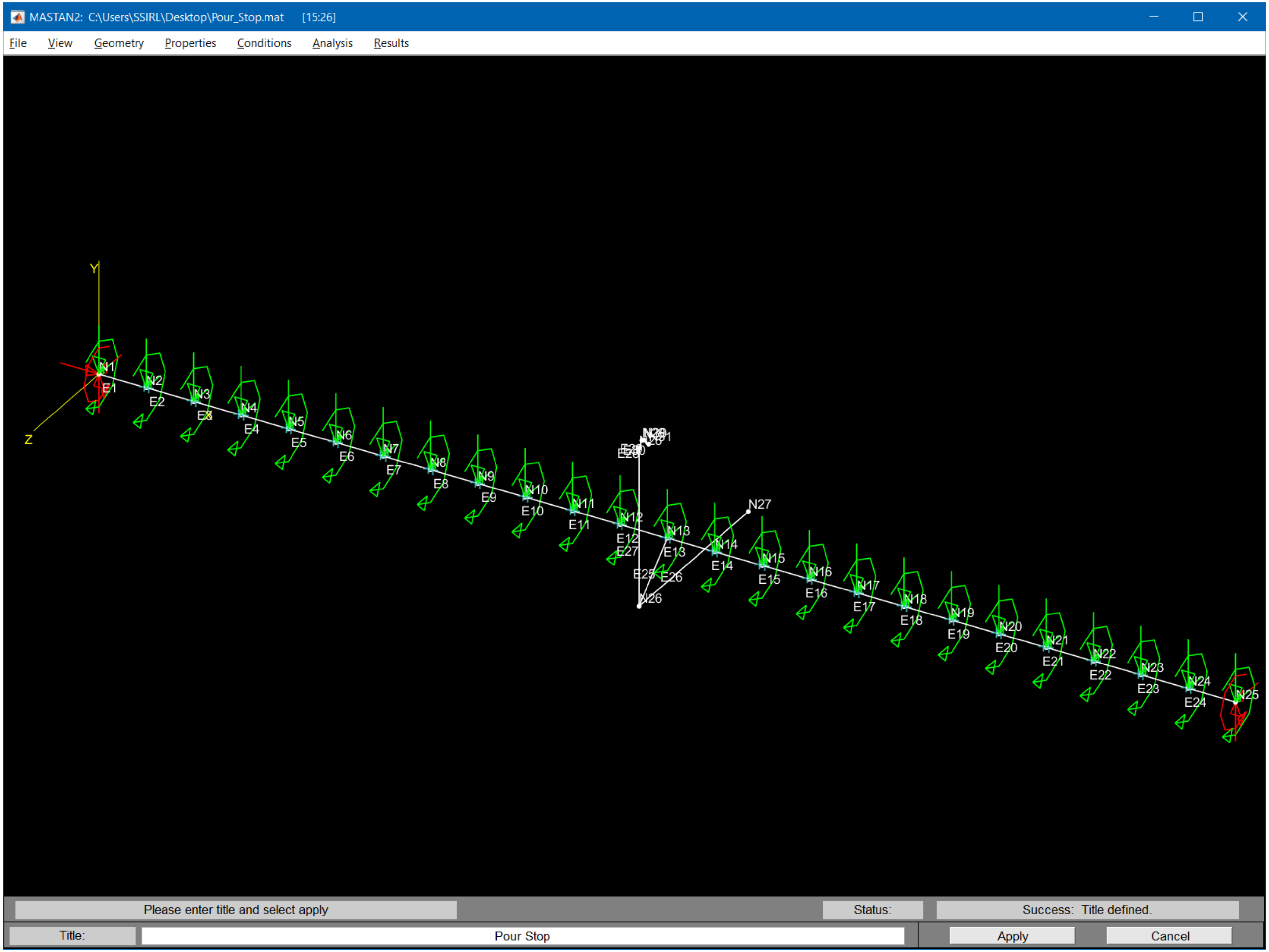

MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
N27
N26
Advanced Node Selection
Range (Inclusive)
-Inf
X
Inf
-Inf
Y
Inf
-1
Z
1
Add
Remove
Reset
Please define node(s) and moments
Node(s):
1 2 3 4 5 6 7 8 9 10 11
All
Clr
Adv
Status:
Success: Moments at nodes defined.
MX =
-17.42
MY =
0
MZ =
0
Apply
Cancel

Naming and Saving

These steps are technically optional as you can complete analysis without saving or applying a title; however, this is a good time to complete this.

1) From the **File** menu select **Define title**. At the bottom menu bar, click in the edit box to the right of **Title:** and type in a brief description of this effort. This text might include the model title, your name, and/or the assignment number. Click on the **Apply** button.
2) From the **File** menu select **Save As …** . After selecting your destination folder, type in the filename **Pour_Stop** and click **Save**. Note that the top of the window has now changed to include the file name and directory as well as the time the file was last saved.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [15:26]
File
View
Geometry
Properties
Conditions
Analysis
Results
Please enter title and select apply
Status:
Success: Title defined.
Title:
Pour Stop
Apply
Cancel

First-Order Elastic Analysis

1) From the **Analysis** menu select **Static** and submenu option **1st-Order Elastic**.

2) At the bottom menu bar, the **Analysis Type:** should already be set to **Space Frame** as desired.

3) Click on the **Apply** button to perform the analysis.

4) From the **Results** menu select **Diagrams** and submenu option **Deflected Shape**.

5) At the bottom menu bar, click on the **Apply** button.

6) From the **Results** menu select **Node Displacements**.

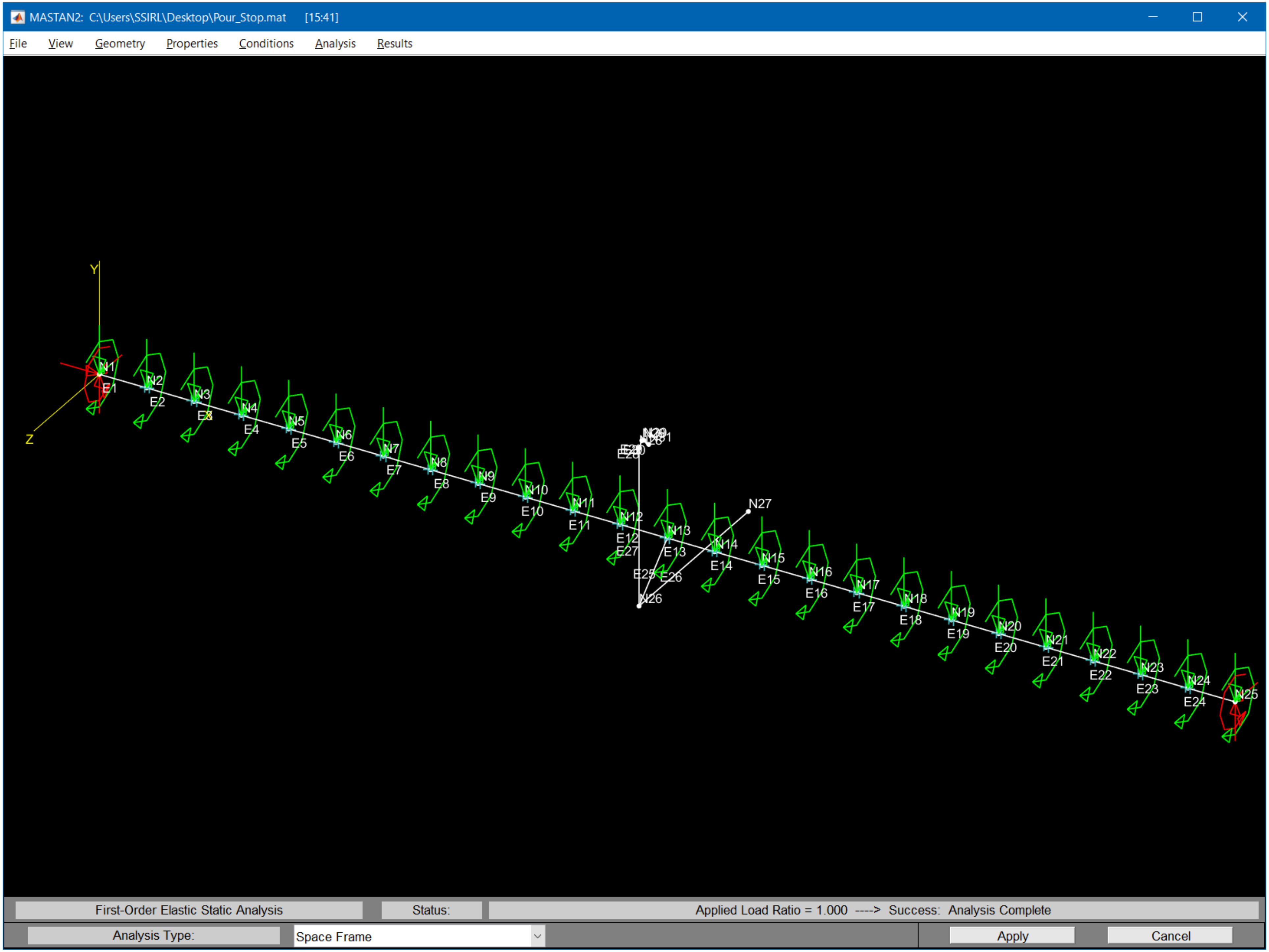
7) On the undeflected shape, click on the midspan node of interest, node **13**, and the displacements for base 6 degree of freedoms are provided in the bottom menu bar.

Results:

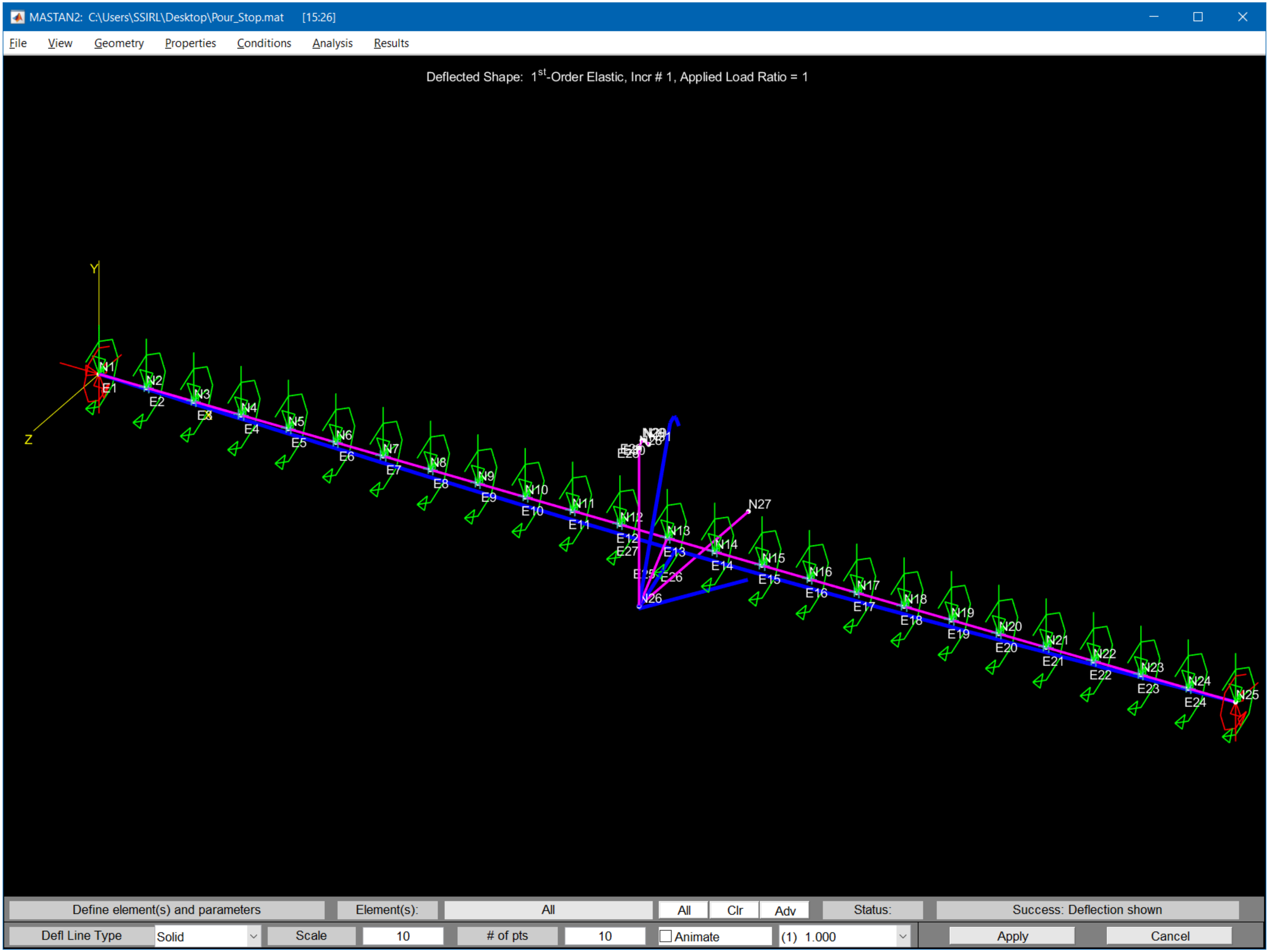
Disp X	Disp Y	Disp Z	Rot X	Rot Y	Rot Z
~0	-0.08033	-0.0598	-0.03472	~0	~0

8) Clicking on any of the additional nodes representing the cross section will cause the displacements at that location due the combined effects of rotation and translation to be displayed. These deflections do not account for any local axial displacement due to warping though.

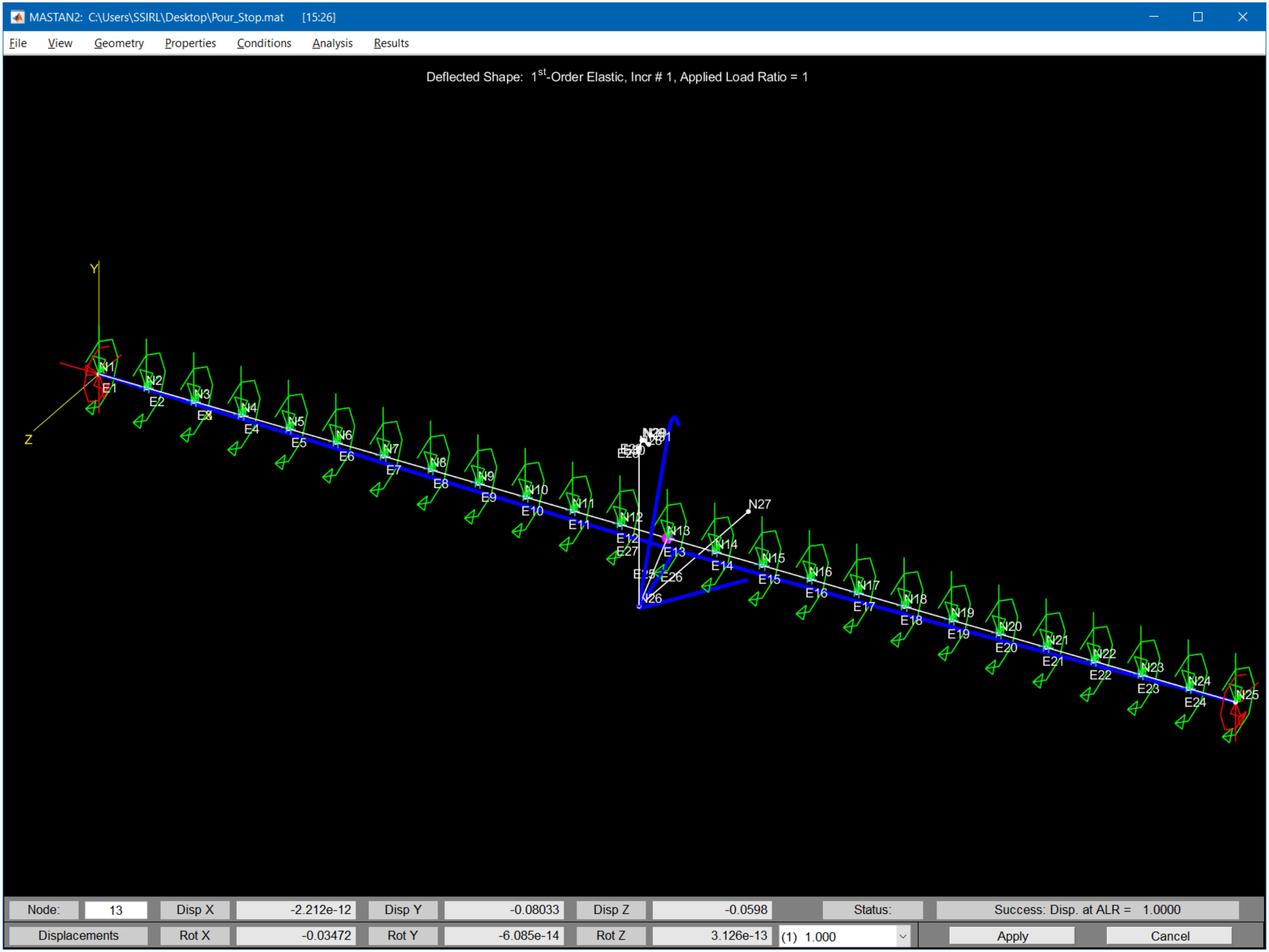
MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [15:41]
File View Geometry Properties Conditions Analysis Results
First-Order Elastic Static Analysis
Status:
Applied Load Ratio = 1.000 ----> Success: Analysis Complete
Analysis Type:
Space Frame
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [15:26]
File
View
Geometry
Properties
Conditions
Analysis
Results
Deflected Shape: 1st-Order Elastic, Incr # 1, Applied Load Ratio = 1
Define element(s) and parameters
Element(s):
All
All
Clr
Adv
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
10
# of pts
10
Animate
(1) 1.000
Apply
Cancel

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [15:26]
File View Geometry Properties Conditions Analysis Results
Deflected Shape: 1st-Order Elastic, Incr # 1, Applied Load Ratio = 1
Y
Z
N27
N26
Node: 13
Disp X -2.212e-12
Disp Y -0.08033
Disp Z -0.0598
Status:
Success: Disp. at ALR = 1.0000
Displacements
Rot X -0.03472
Rot Y -6.085e-14
Rot Z 3.126e-13
(1) 1.000
Apply
Cancel

Second-Order Elastic Analysis

1) From the **Analysis** menu select **Static** and submenu option **2nd-Order Elastic**.

2) At the bottom menu bar, the **Analysis Type:** should already be set to **Space Frame** as desired.

3) Click on the **Apply** button to perform the analysis.

4) From the **Results** menu select **Diagrams** and submenu option **Deflected Shape**.

5) At the bottom menu bar, click on the **Apply** button.

6) From the **Results** menu select **Node Displacements**.

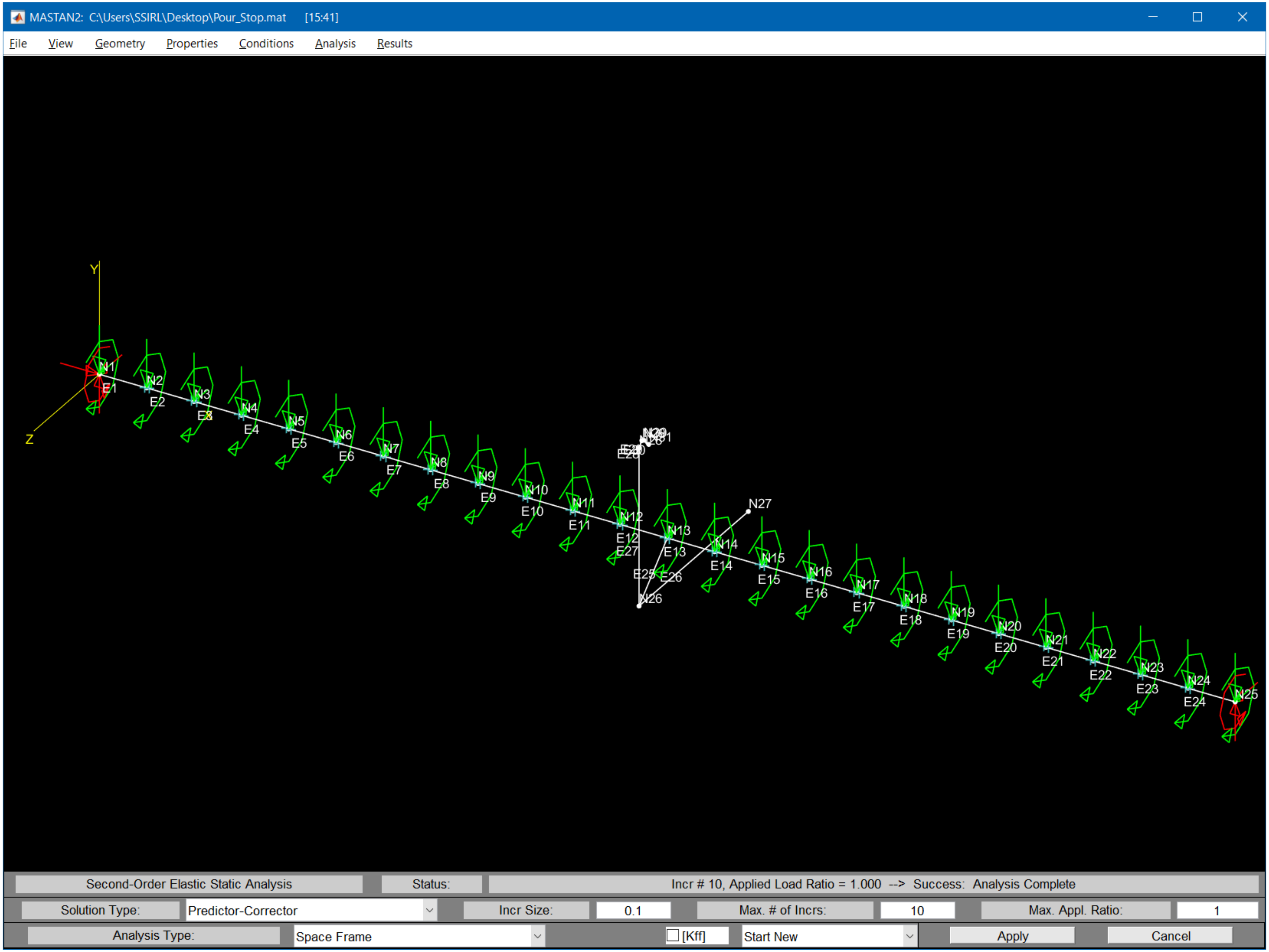
7) On the undeflected shape, click on the midspan node of interest, node **13**, and the displacements for base 6 degree of freedoms are provided in the bottom menu bar.

Results:

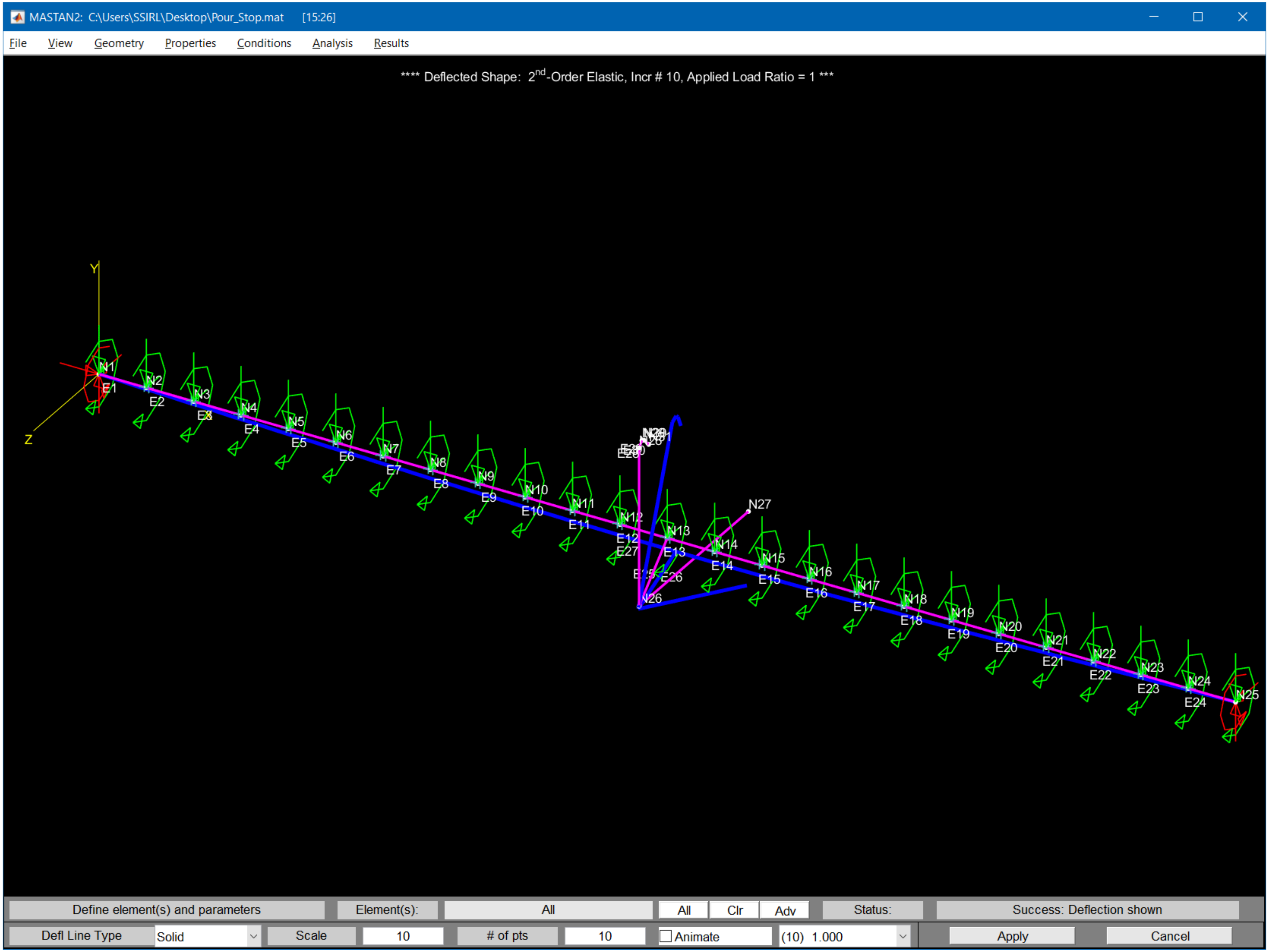
Disp X	Disp Y	Disp Z	Rot X	Rot Y	Rot Z
-2.992e-4	-0.08758	-0.06218	-0.03726	~0	~0

The values are similar to the 1st-order results but capture the change in the applied torsion that exists in the deformed condition. The analysis does not account for any change in the direction the load would be applied. The applied loads will remain in the initial orientation.

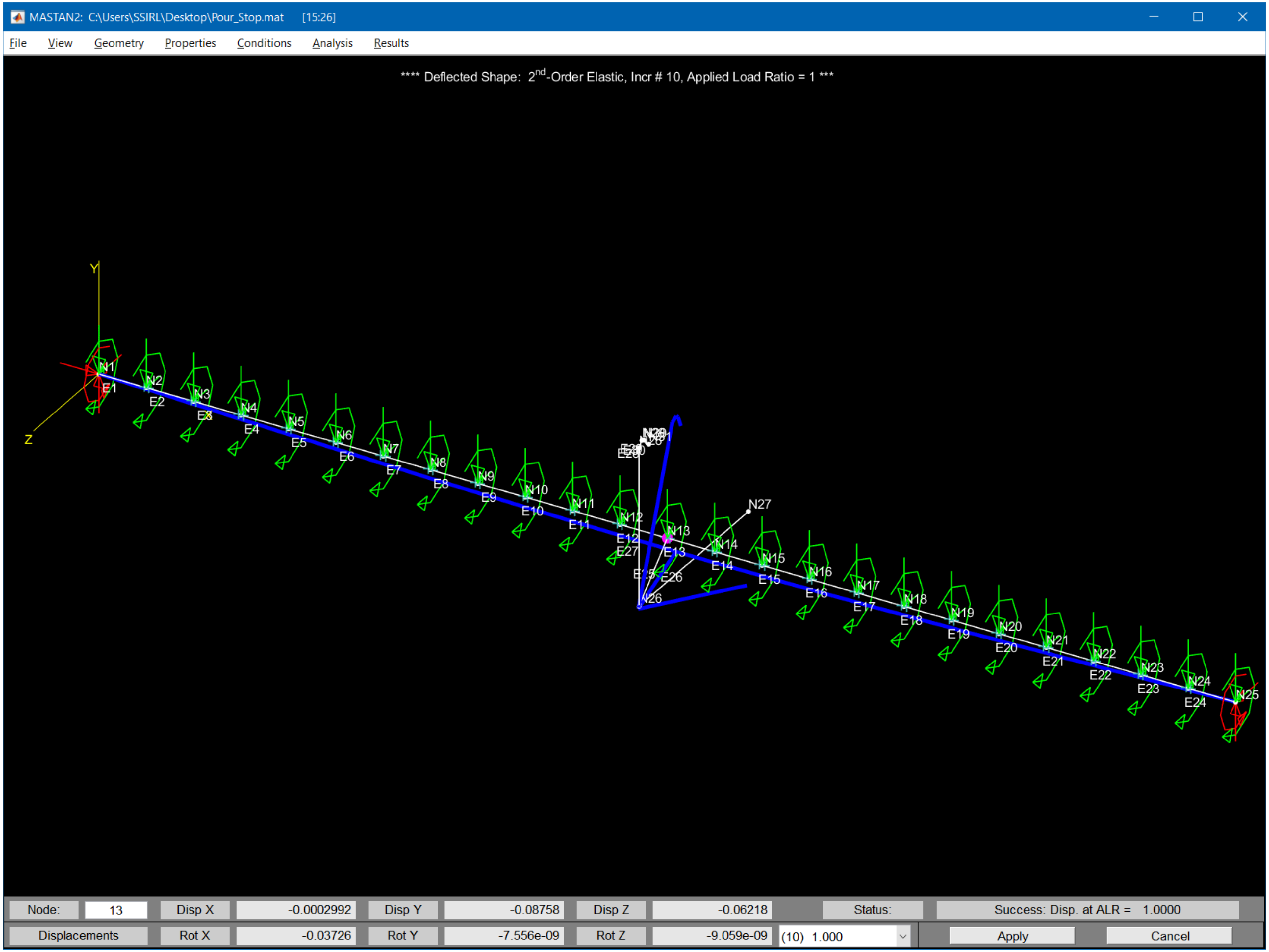
MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [15:41]
File
View
Geometry
Properties
Conditions
Analysis
Results
Second-Order Elastic Static Analysis
Status:
Incr # 10, Applied Load Ratio = 1.000 --> Success: Analysis Complete
Solution Type:
Predictor-Corrector
Incr Size:
0.1
Max. # of Incrs:
10
Max. Appl. Ratio:
1
Analysis Type:
Space Frame
[Kff]
Start New
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [15:26]
File
View
Geometry
Properties
Conditions
Analysis
Results
**** Deflected Shape: 2nd-Order Elastic, Incr # 10, Applied Load Ratio = 1 ***
Y
Z
N1
E1
N2
E2
N25
N26
N27
Define element(s) and parameters
Element(s):
All
All
Clr
Adv
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
10
# of pts
10
Animate
(10) 1.000
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [15:26]
File View Geometry Properties Conditions Analysis Results
**** Deflected Shape: 2nd-Order Elastic, Incr # 10, Applied Load Ratio = 1 ***
Node: 13
Disp X -0.0002992
Disp Y -0.08758
Disp Z -0.06218
Status:
Success: Disp. at ALR = 1.0000
Displacements
Rot X -0.03726
Rot Y -7.556e-09
Rot Z -9.059e-09
(10) 1.000
Apply
Cancel

Section 4: Results and Stresses

Using Internal Forces

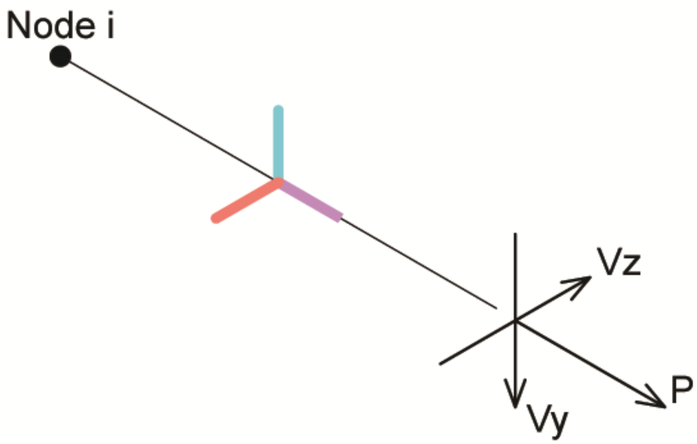
The next section of this tutorial will provide instructions on how it is possible to calculate the internal stresses from the MASTAN2 output information. After showing how to locate the necessary information from within MASTAN2, the next few pages will provide background and the sign conventions that are used with MASTAN2 to calculate the internal stresses. Some information is provided on how this information can be calculated. Finally, the resulting stresses at midspan are identified using the 2nd-order analysis internal forces.

Notes:

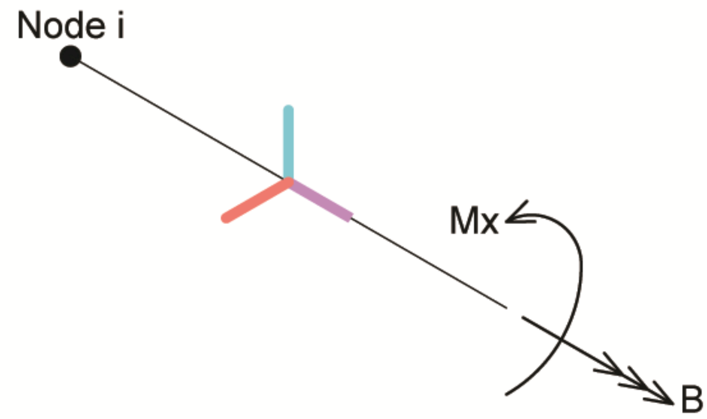
The internal forces taken from MASTAN2 are presented with the sign convention shown on the following page. Also included are the required transformations to get to a principal orientation.

The stress calculations will account for this information.

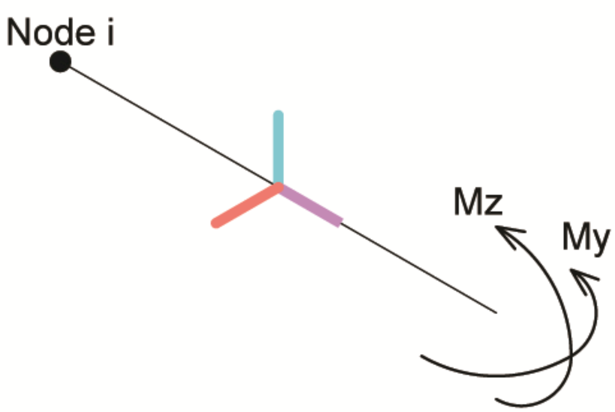
Sign Conventions



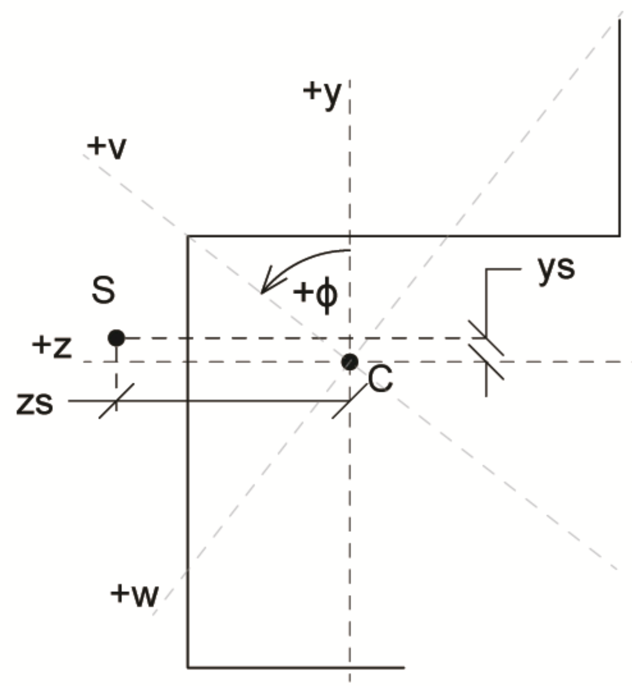
$$\begin{bmatrix}V_v\\ V_w\end{bmatrix}=\begin{bmatrix}cos\phi & sin\phi\\ -sin\phi & cos\phi\end{bmatrix}\cdot\begin{bmatrix}V_y\\ V_z\end{bmatrix}$$



$$\begin{bmatrix}v_s\\ w_s\end{bmatrix}=\begin{bmatrix}cos\phi & sin\phi\\ -sin\phi & cos\phi\end{bmatrix}\cdot\begin{bmatrix}y_s\\ z_s\end{bmatrix}$$



$$\begin{bmatrix}M_v\\ M_w\end{bmatrix}=\begin{bmatrix}cos\phi & -sin\phi\\ sin\phi & cos\phi\end{bmatrix}\cdot\begin{bmatrix}M_y\\ M_z\end{bmatrix}$$



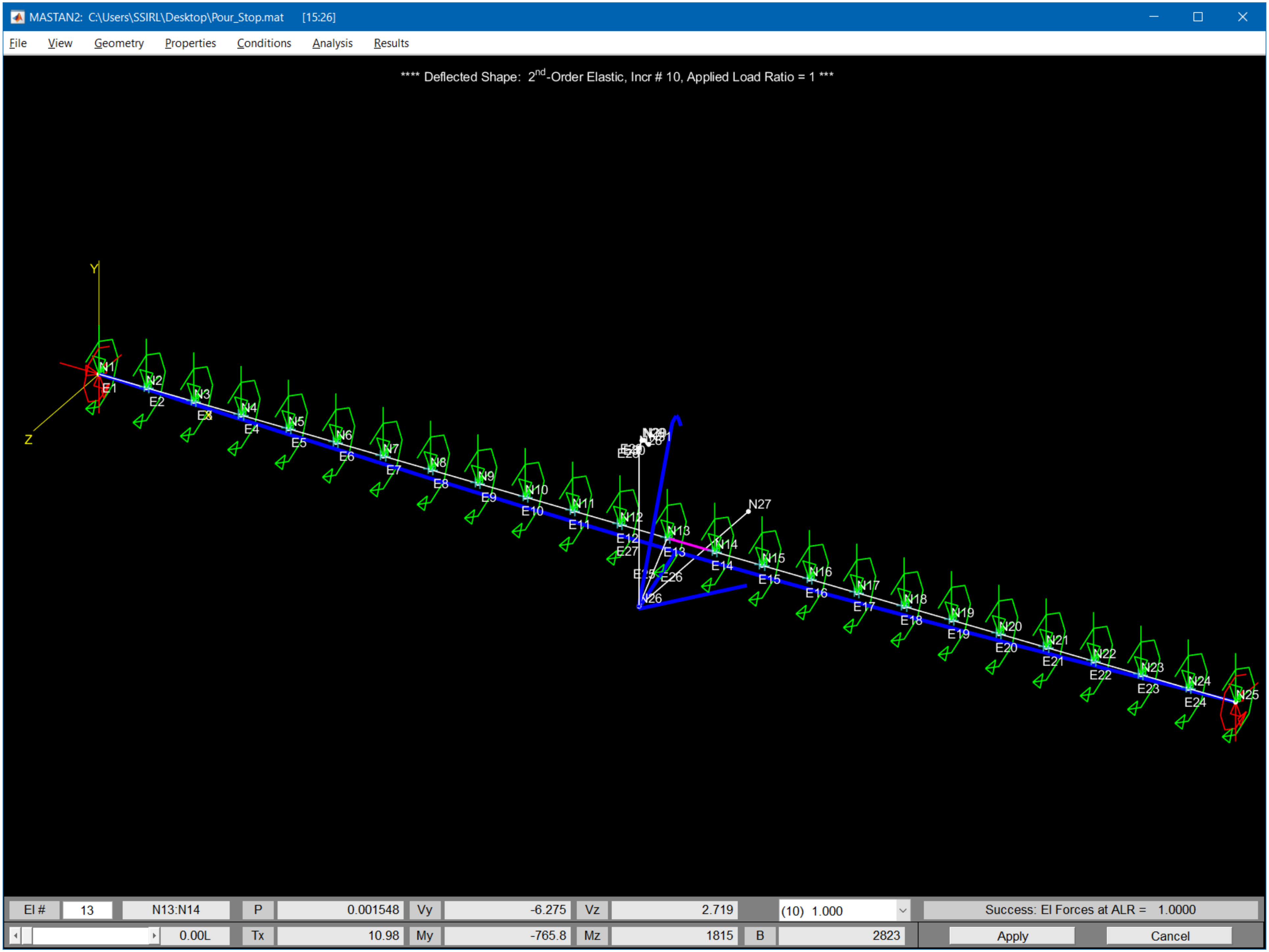
Getting Internal Forces

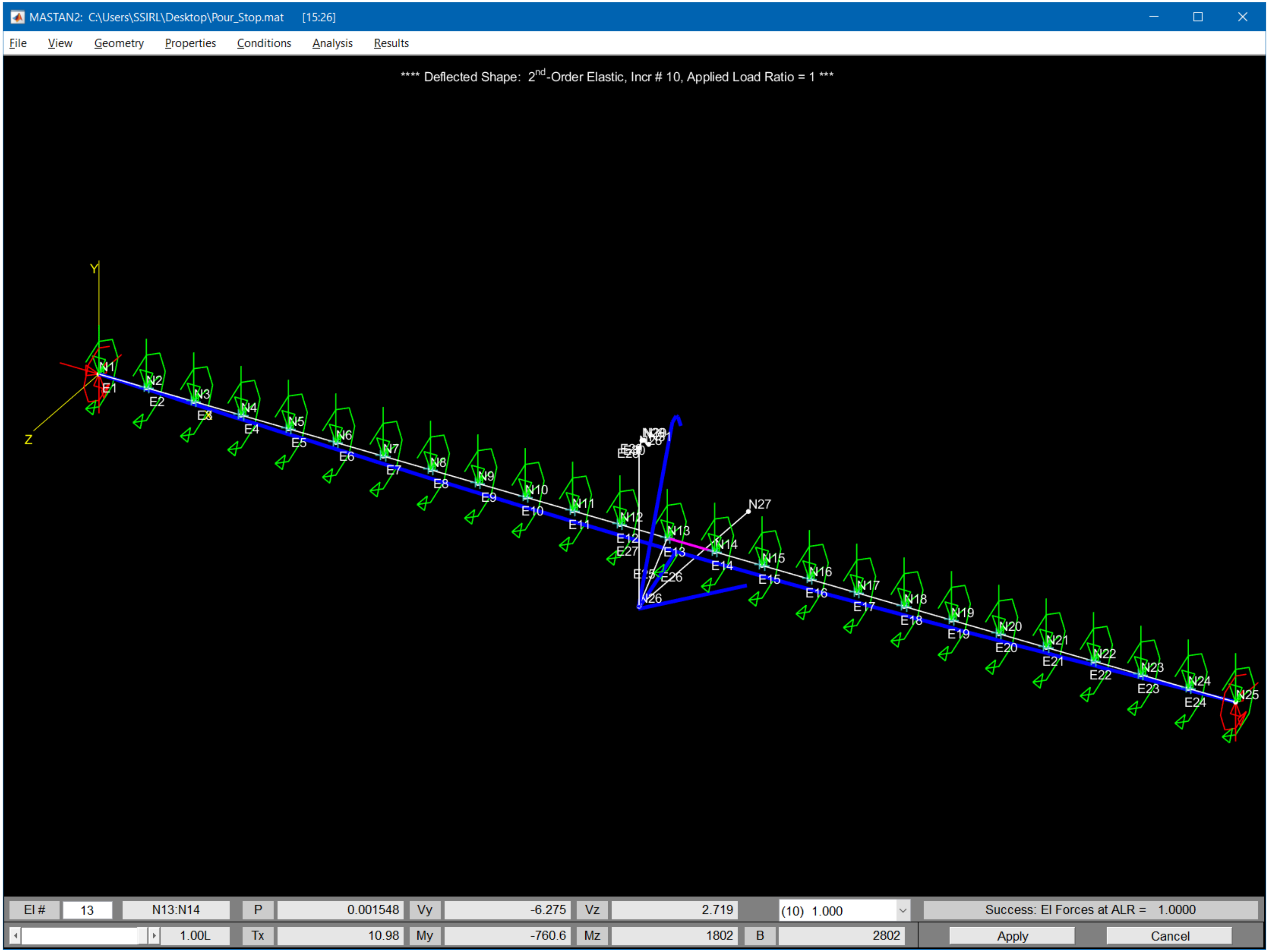
1) From the **Results** menu select **Element Forces**.
2) On the undeflected shape, click on the element of interest, element **13**, and the internal forces are provided in the bottom menu bar. These are the forces at the start of the member and the middle of the beam.
3) These forces will be used to calculate the stresses at the middle of the beam.
4) At the bottom menu bar, drag the slider in the left-hand corner until the position indicator located to the right displays **1.00L**.
5) Click on the **Apply** button. These are the forces at the end of the member.
6) From this position, the bimoment is required to appropriately divide the longitudinal moment into the standard twisting and warping components for stress calculations.
7) Repeat steps 2 through 5 for element **1** to obtain the end forces to calculate the controlling shear location.

Forces at the start of element 1

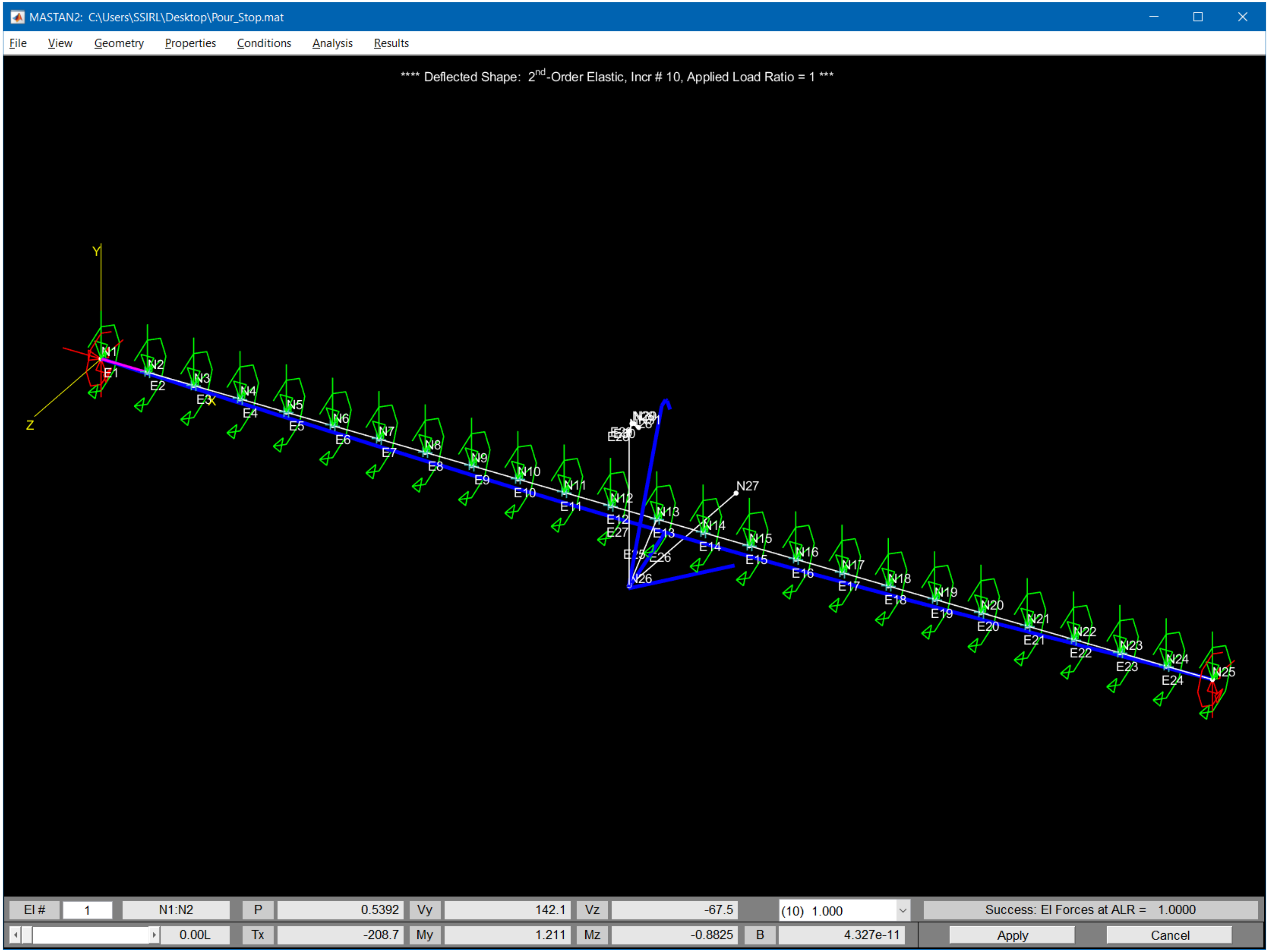
Forces at the end of element 1

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [15:26]
File View Geometry Properties Conditions Analysis Results
**** Deflected Shape: 2nd-Order Elastic, Incr # 10, Applied Load Ratio = 1 ***
El # 13 N13:N14 P 0.001548 Vy -6.275 Vz 2.719
(10) 1.000
Success: El Forces at ALR = 1.0000
0.00L Tx 10.98 My -765.8 Mz 1815 B 2823
Apply
Cancel

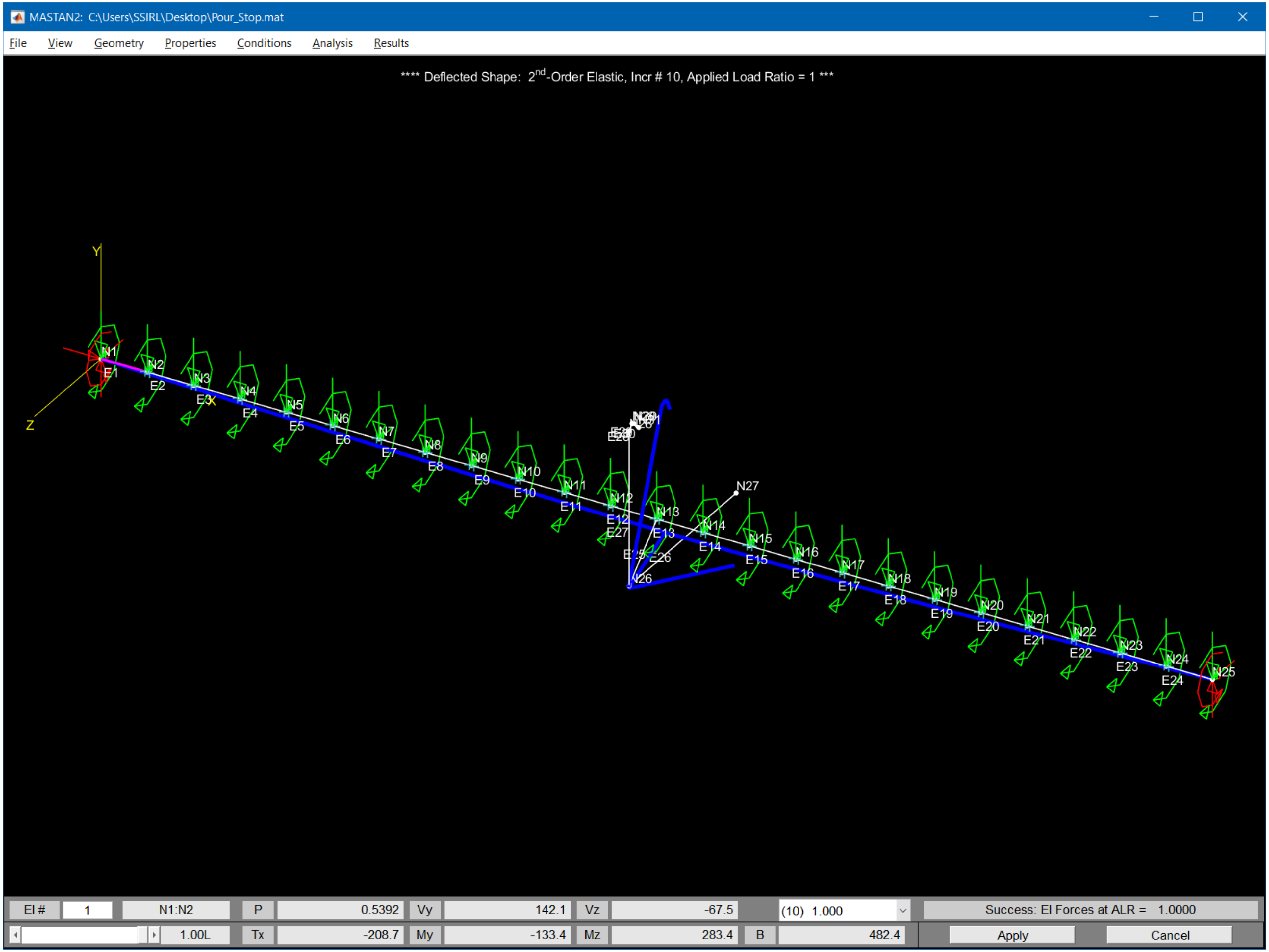

MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [15:26]
File View Geometry Properties Conditions Analysis Results
**** Deflected Shape: 2nd-Order Elastic, Incr # 10, Applied Load Ratio = 1 ***
El # 13 N13:N14 P 0.001548 Vy -6.275 Vz 2.719
(10) 1.000
Success: El Forces at ALR = 1.0000
1.00L Tx 10.98 My -760.6 Mz 1802 B 2802
Apply
Cancel

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
**** Deflected Shape: 2nd-Order Elastic, Incr # 10, Applied Load Ratio = 1 ***
El #
1
N1:N2
P
0.5392
Vy
142.1
Vz
-67.5
(10) 1.000
Success: El Forces at ALR = 1.0000
0.00L
Tx
-208.7
My
1.211
Mz
-0.8825
B
4.327e-11
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
**** Deflected Shape: 2nd-Order Elastic, Incr # 10, Applied Load Ratio = 1 ***
Y
Z
X
N1
N25
N26
N27
El #
1
N1:N2
P
0.5392
Vy
142.1
Vz
-67.5
(10) 1.000
Success: El Forces at ALR = 1.0000
1.00L
Tx
-208.7
My
-133.4
Mz
283.4
B
482.4
Apply
Cancel

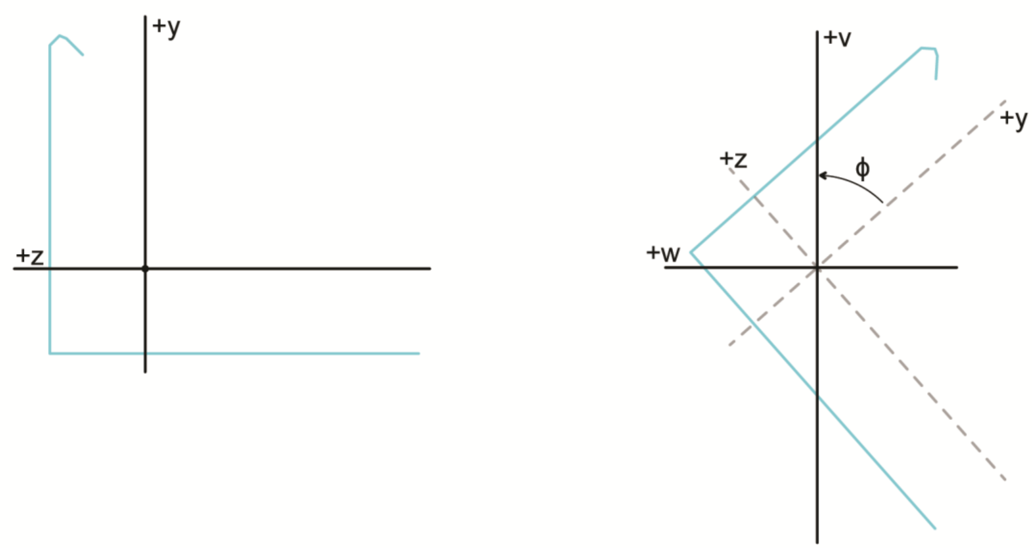
Normal Stresses

The internal normal stresses can be calculated in the geometric orientation by

$$\sigma_x = \frac{P}{A} - \frac{\left(M_y I_{yz} + M_z I_{yy}\right)y + \left(M_y I_{zz} + M_z I_{yz}\right)z}{I_{yy} I_{zz} - I_{yz}^2} + \frac{B\omega_n}{C_\omega}$$

This relationship simplifies when working in the principal orientation to

$$\sigma_x = \frac{P}{A} - \frac{M_w v}{I_{ww}} - \frac{M_v w}{I_{vv}} + \frac{B\omega_n}{C_\omega}$$



Shear Stresses

As part of calculating the shear stresses, the torsional moment must be separated into the component from twisting and the component of warping.

$$M_x - z_s V_y + y_s V_z = M_x - w_s V_v + v_s V_w = T_T - T_\omega$$

$$T_\omega = \frac{d}{dx} B \approx \frac{\Delta B}{\Delta L}$$

The maximum shear stress is the combination of the stress from transverse shear (τ_V), the change in bimoment (τ_B), and torsion (τ_T). The transverse and warping terms are approximately uniform across the thickness of the element, whereas torsion linearly varies across the thickness with the extreme value at the surface.

$$\tau_V = \frac{\left(-V_z I_{yz} + V_y I_{yy}\right)\bar{y} + (V_z I_{zz} - V_y I_{yz})\bar{z}}{\left(I_{yy} I_{zz} - I_{yz}^2\right)t} A_s$$

$$\tau_V = \frac{V_v \bar{v}}{I_{ww} t} A_s + \frac{V_w \bar{w}}{I_{vv} t} A_s$$

$$\tau_B = -\frac{T_\omega S_\omega}{I_\omega} \qquad \tau_T = \frac{T_T t}{J}$$

Calculating Warping Constants

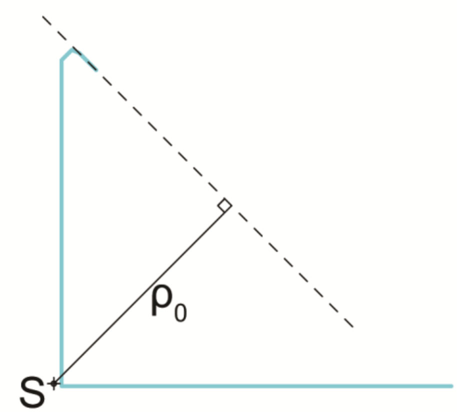
Determining the stresses from warping, both shear stresses and normal stresses, requires the use of warping coefficients. The values can be calculated based on the equations show below with additional details in the following documents. These documents also provide additional information on the stresses calculated as well.

(1) AISC Design Guide 9 (Paul A. Seaburg and Charles J. Carter, 2003)

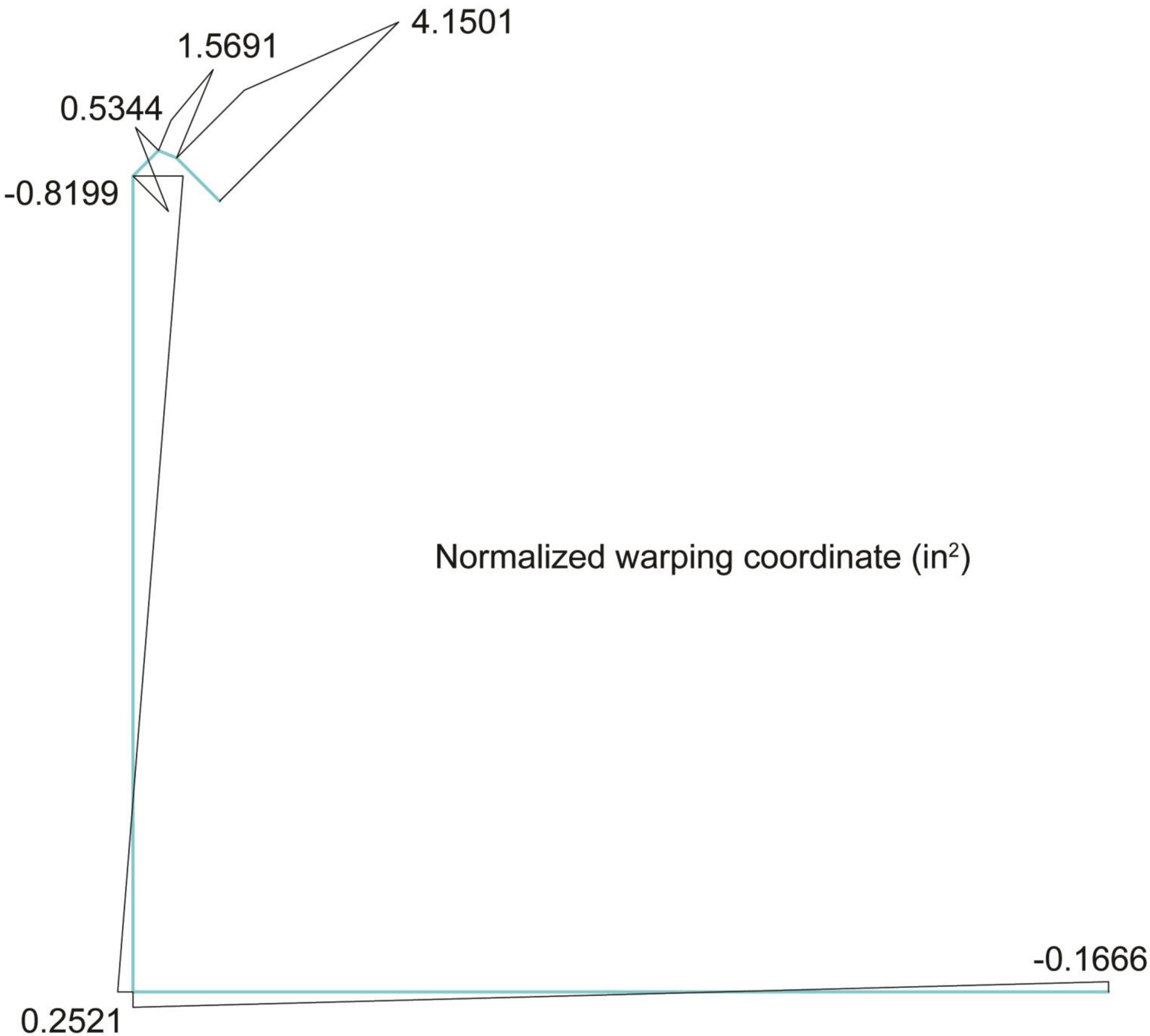
(2) SDI Technical Note 3. (Steel Deck Institute, 2018)

$$\omega_{ns} = \frac{1}{A}\int_0^b \omega_{os}\, t\, ds - \omega_{os}$$

$$\omega_{os} = \int_0^s \rho_0\, ds$$



Different programs exist that can assist in this calculation. One alternative is to make use of CUFSM to calculate these values. The values for the current section are shown here.


4.1501
1.5691
0.5344
-0.8199
Normalized warping coordinate (in²)
-0.1666
0.2521

Resulting Internal Stresses

Each of the items below will link to a stress distribution for the centerline. Note moments and shears below are in the principal orientation.

1) Axial stress from bending moment: **My & Mz** & Axial stress from warping: **B**

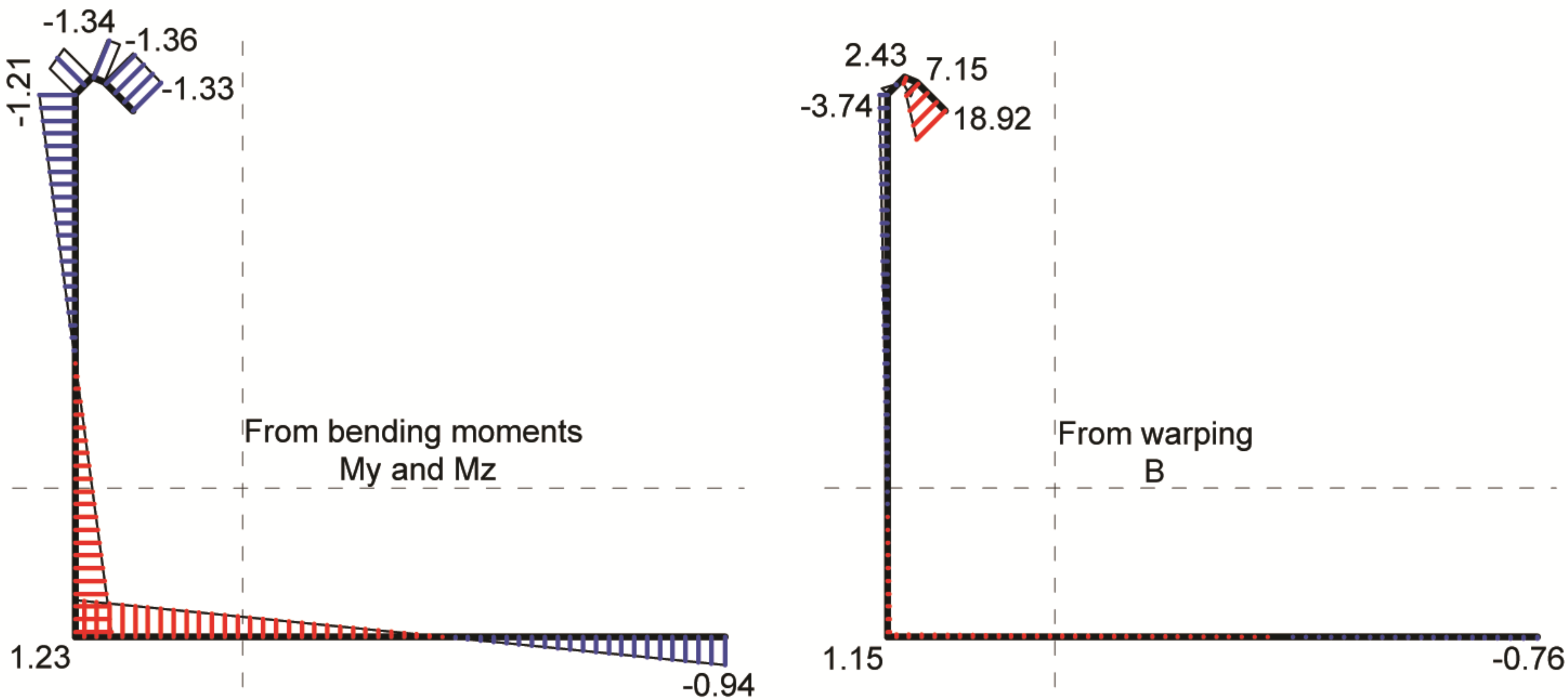
2) Total axial stress from **P, My, Mz, & B**

3) Shear stress from transverse shear: **Vy & Vz** & Shear stress from warping: $\mathbf{T_{\omega}}$

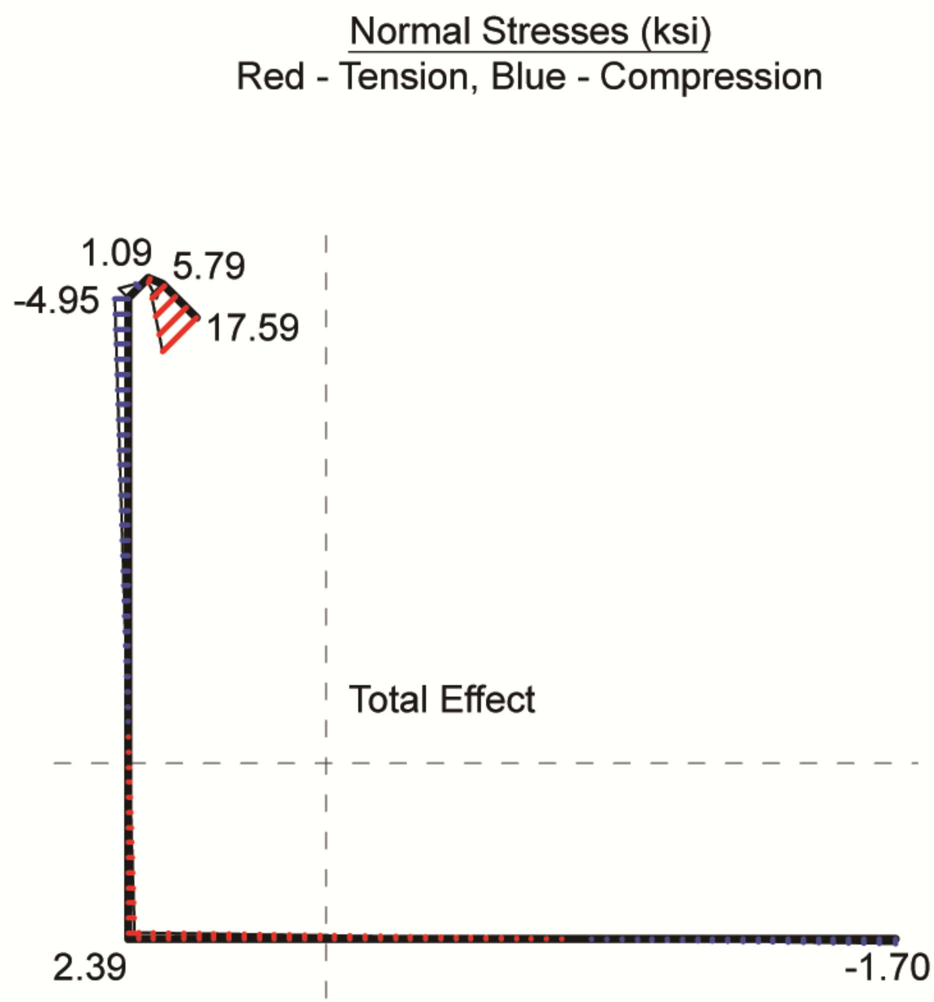
4) Combined shear stress from **Vy, Vz, &** $\mathbf{T_{\omega}}$ & Shear stress from twisting: $\mathbf{T_T}$

No total combined shear stress is shown as the warping and transverse shear stress components are the same across the thickness of the section while the torsional component varies linearly across the thickness of the section with 0 stress at the centerline to ± the value shown at the extremes.

Normal Stresses (ksi)
Red - Tension, Blue - Compression

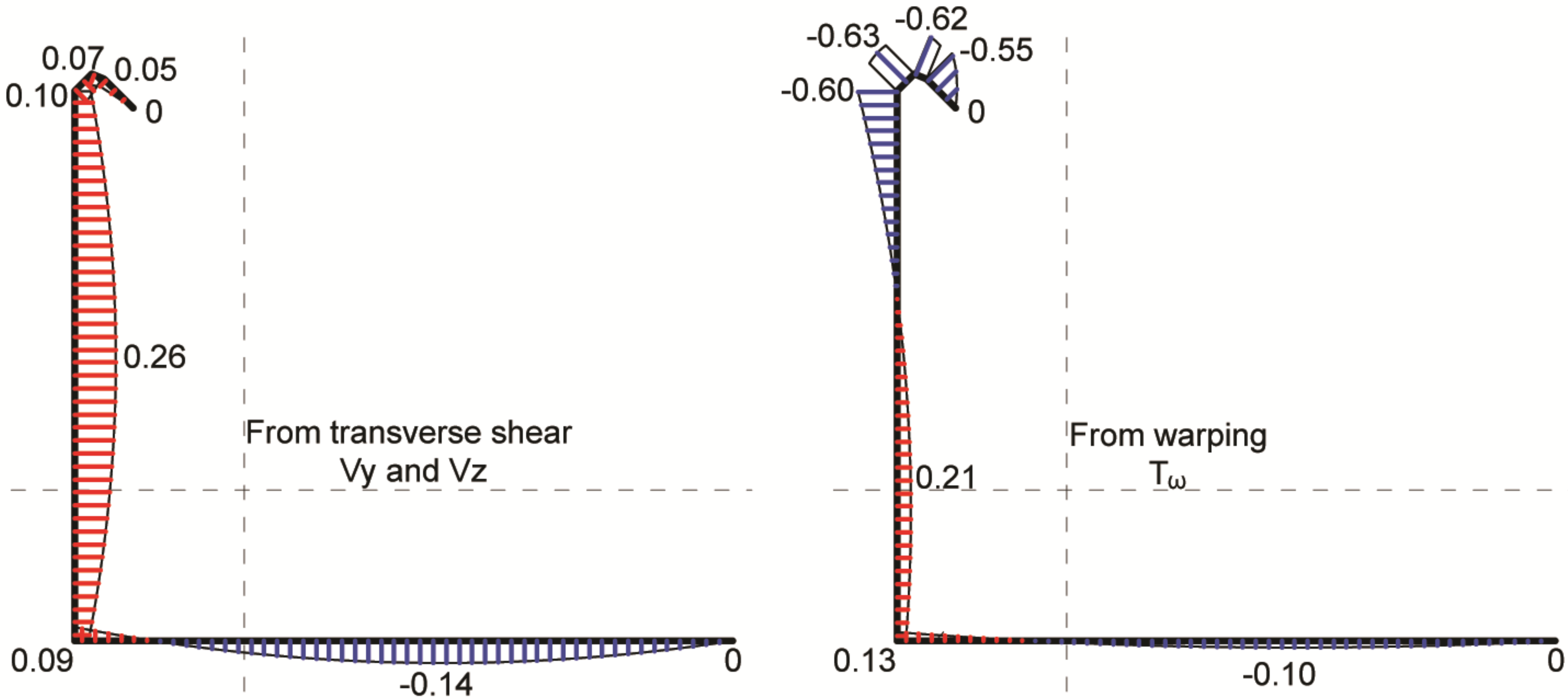


MASTAN2


Normal Stresses (ksi)
Red - Tension, Blue - Compression
1.09
5.79
-4.95
17.59
Total Effect
2.39
-1.70

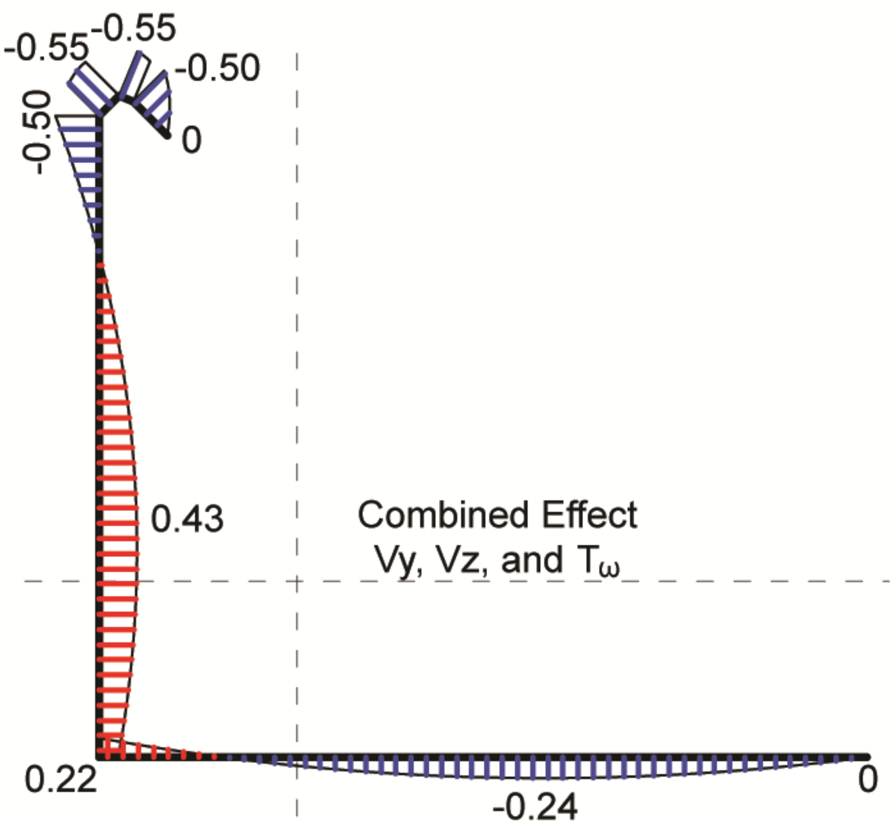
MASTAN2

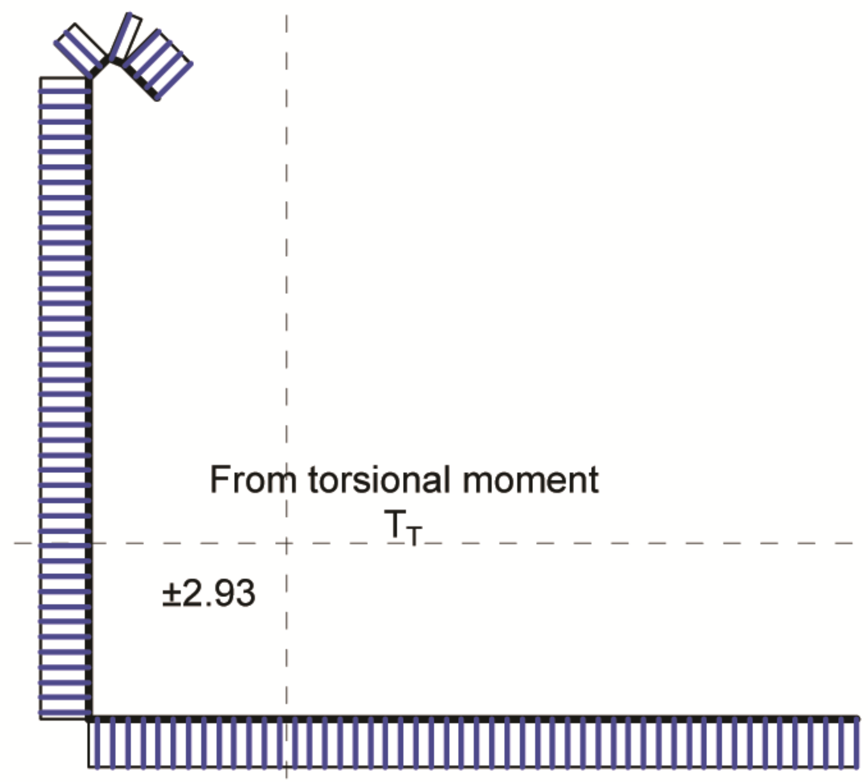
Shear Stresses (ksi)
Red - CCW, Blue - CW



Results do not account for increase that could be considered in shear forces and torsion since part of the end point loads would be on this member.

Shear Stresses (ksi)
Red - CCW, Blue - CW





Results do not account for increase that could be considered in shear forces and torsion since part of the end point loads would be on this member.

Section 5: Additional Options

Other Possibilities

MASTAN2 has many options of what can be done. For this tutorial, three ideas are presented. Each possibility is completed starting with the initial model completed to this point. To compare to each of the following, it is recommended to make sure you have a base version of this model saved to be able to start over.

1) Applying uniform loading instead of point loads for the lateral loading.
2) Adjusting the boundary conditions to consider the effects of fixed warping.
3) Updating the model to calculate the results for other lengths or check other critical geometry.

1) Uniform Loading

While a distributed moment loading is not currently available in MASTAN2, it would be possible to use a distributed uniform load instead of point loads using the steps below.

1) From the **Conditions** menu select **Define Forces**.

2) At the bottom menu bar, all loads should be **0**.

3) Click the **All** button to populate the list of nodes.

4) Click on the **Apply** button to remove all point loads.

5) From the **Conditions** menu select **Define Uniform Loads**.

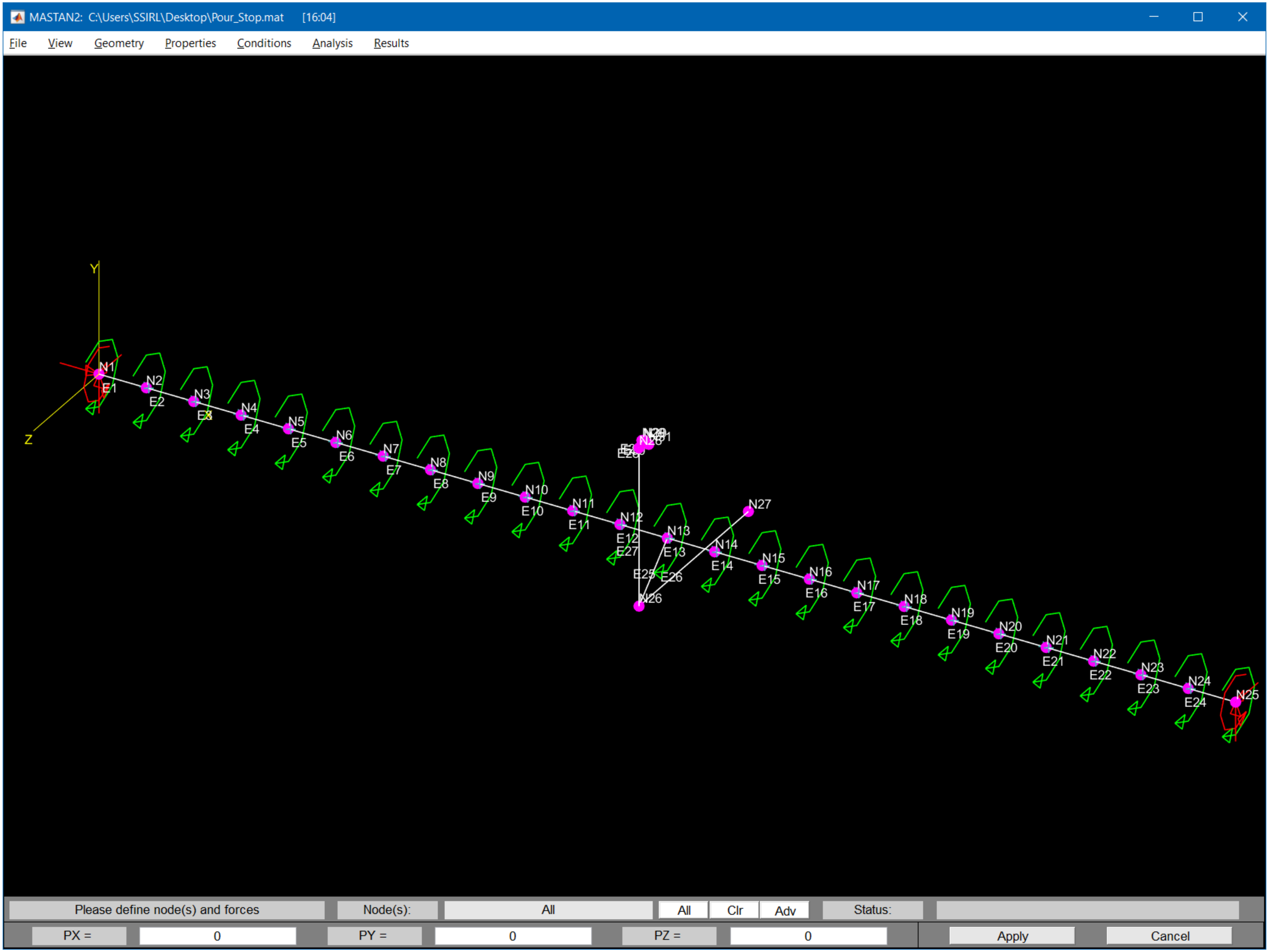
6) At the bottom menu bar, click on **Element(s) local x'-y'-z'** to open the drop-down menu. Select **System global X-Y-Z**.

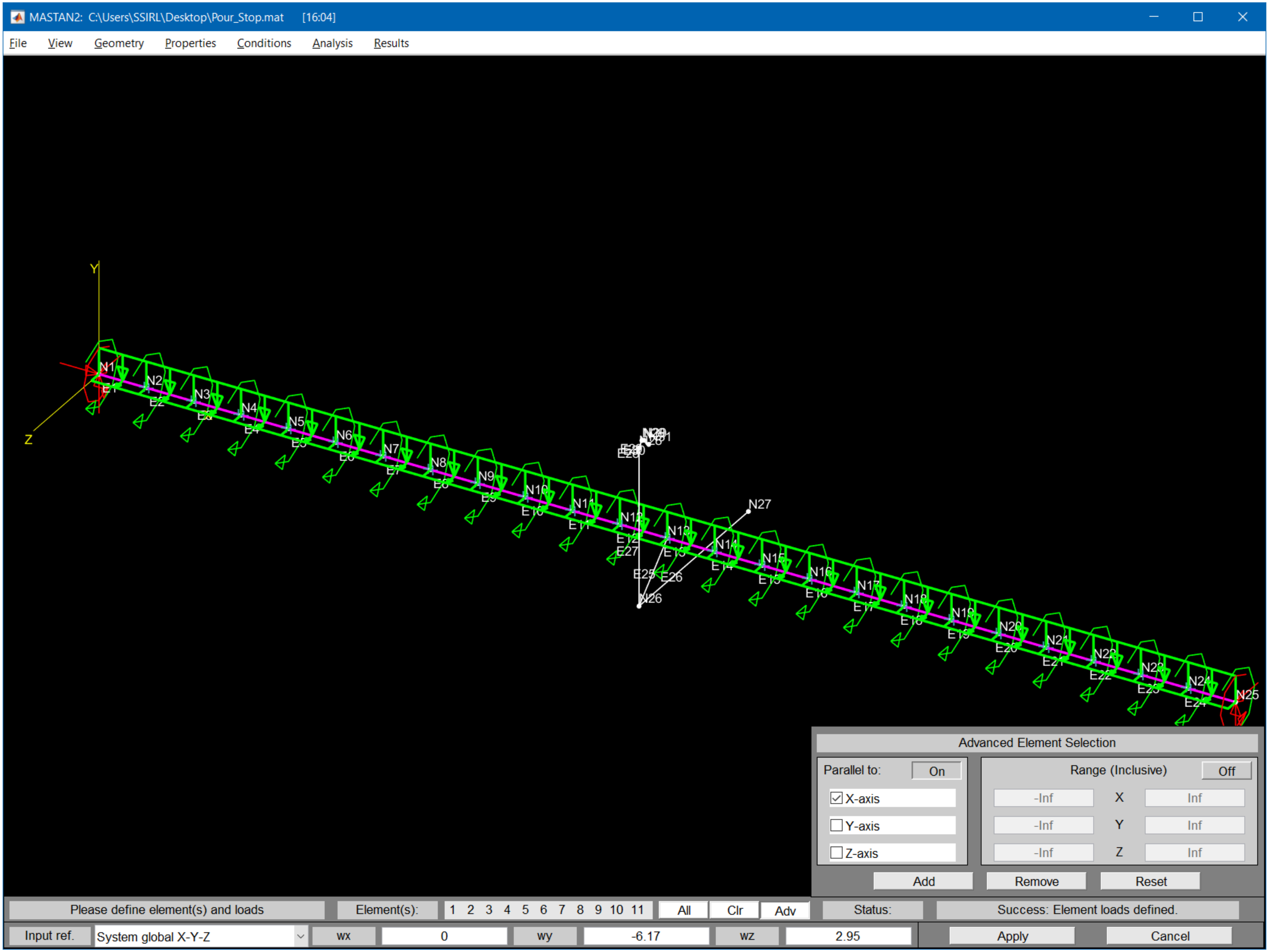
7) Click in the edit box just to the right of **Wy =** and change **0** to **-6.17**. Click in the edit box just to the right of **Wz =** and change **0** to **-2.95**. Click the **Adv** button to open pop-up menu. Ensure the check box next to the **X-axis** option is selected.

8) Click **Add** to add the main beam elements to the element list.

9) Click on the **Apply** button.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [16:04]
File
View
Geometry
Properties
Conditions
Analysis
Results
Please define node(s) and forces
Node(s):
All
All
Clr
Adv
Status:
PX =
0
PY =
0
PZ =
0
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [16:04]
File
View
Geometry
Properties
Conditions
Analysis
Results
N27
N26
Advanced Element Selection
Parallel to:
On
Range (Inclusive)
Off
X-axis
Y-axis
Z-axis
-Inf
X
Inf
-Inf
Y
Inf
-Inf
Z
Inf
Add
Remove
Reset
Please define element(s) and loads
Element(s):
1 2 3 4 5 6 7 8 9 10 11
All
Clr
Adv
Status:
Success: Element loads defined.
Input ref.
System global X-Y-Z
wx
0
wy
-6.17
wz
2.95
Apply
Cancel

1) Uniform Loading -- Second-Order Elastic Analysis

1) From the **Analysis** menu select **Static** and submenu option **2nd-Order Elastic**.

2) At the bottom menu bar, the **Analysis Type:** should already be set to **Space Frame** as desired.

3) Click on the **Apply** button to perform the analysis.

4) From the **Results** menu select **Node Displacements**.

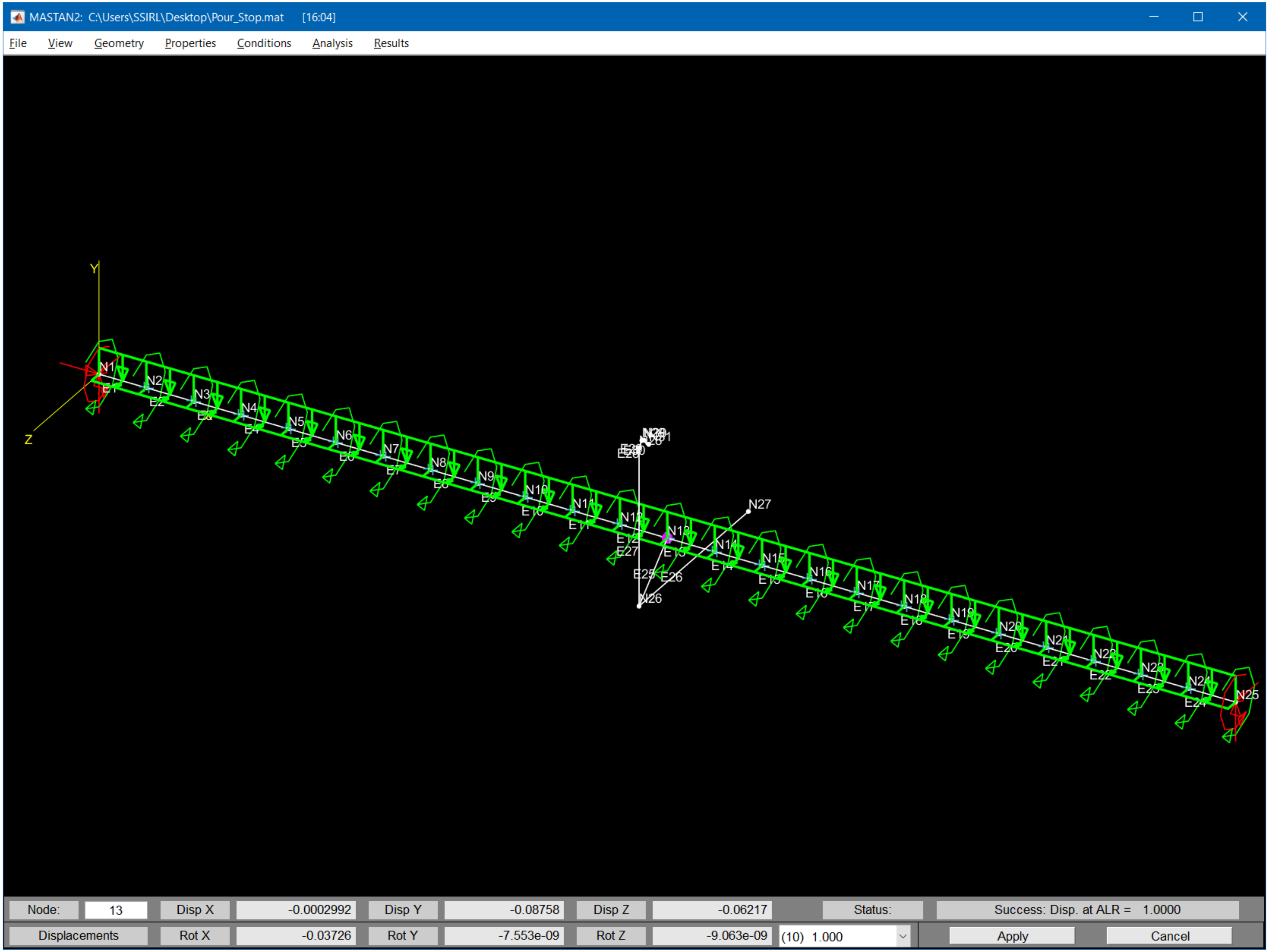
5) On the undeflected shape, click on the midspan node of interest, node **13**, and the displacements for base 6 degree of freedoms are provided in the bottom menu bar.

Results:

Disp X	Disp Y	Disp Z	Rot X	Rot Y	Rot Z
-2.992e-4	-0.08758	-0.06217	-0.03726	~0	~0

The values are similar to using point loads as the model was meshed adequately. If one would be working with less elements with a longer length, a greater variation would be expected with bending moment. There is not an effect of distributed torsion included.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [16:04]
File View Geometry Properties Conditions Analysis Results
Y
Z
N27
N26
Node: 13
Disp X -0.0002992
Disp Y -0.08758
Disp Z -0.06217
Status: Success: Disp. at ALR = 1.0000
Displacements
Rot X -0.03726
Rot Y -7.553e-09
Rot Z -9.063e-09
(10) 1.000
Apply
Cancel

2) Alternate Boundary Conditions

The use of MASTAN2 allows for different boundary conditions. While the model does not need to have symmetric supports, for a reference solution the ends were left as pin supports with torsion fixed while the warping constraint was changed to fixed.

1) From the **Geometry** menu select **Define Connections** and submenu option **Torsion**.

2) Click on the left most element of the beam.

3) At the bottom menu bar, click on the menu to the right of **Warping Restraint for Node i** and set the value to **Fixed**. Click on the menu to the right of **Warping Restraint for Node j** and set the value to **Continuous**.

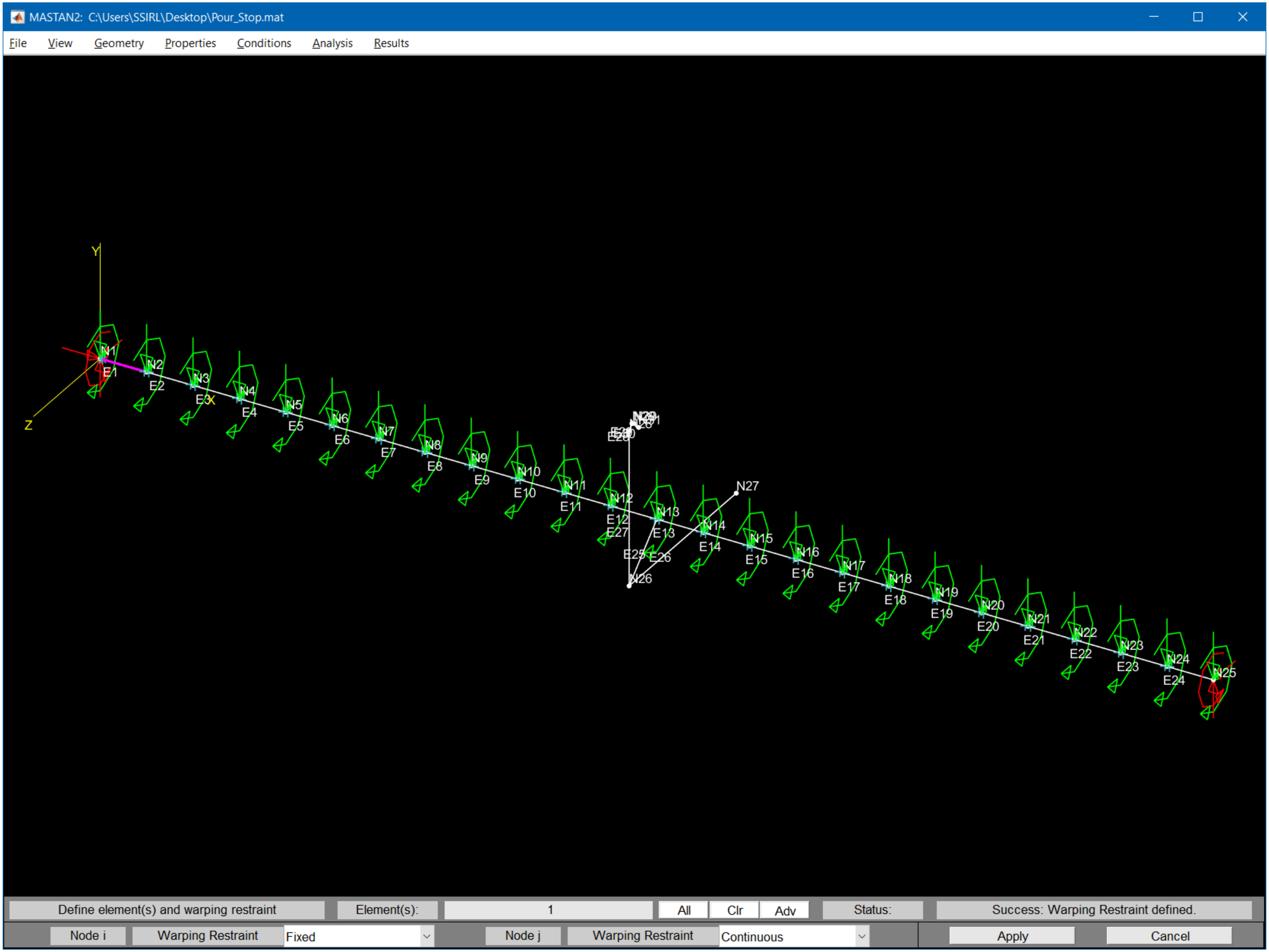
4) Click on the **Apply** button.

5) Click **Clr** to empty the list of elements. Click on the right most element of the beam. This might require you to click **Adv** to close the pop-up menu.

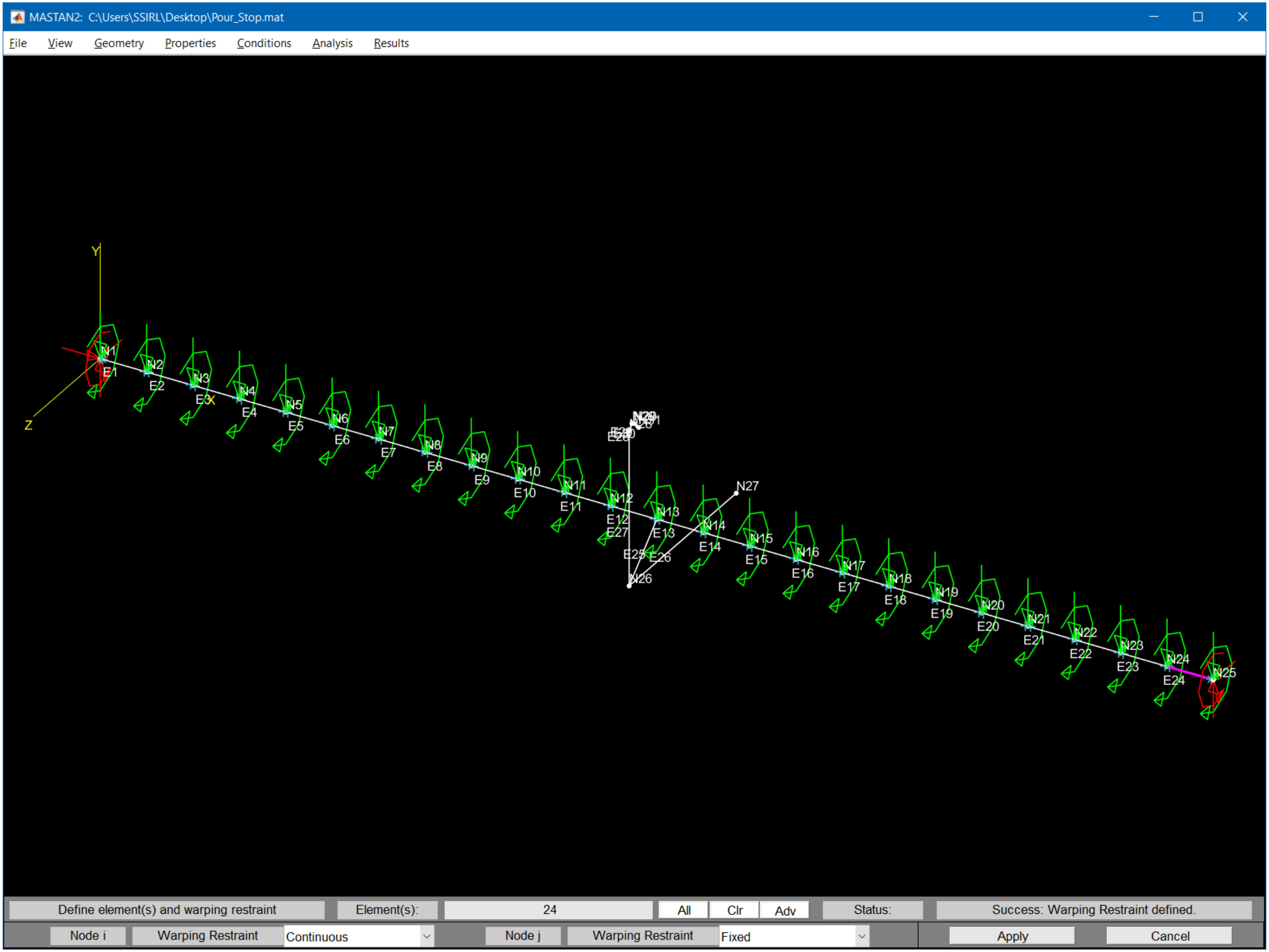
6) Click on the menu to the right of **Warping Restraint for Node i** and set the value to **Continuous**. Click on the menu to the right of **Warping Restraint for Node j** and set the value to **Fixed**.

7) Click on the **Apply** button.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Define element(s) and warping restraint
Element(s):
1
All
Clr
Adv
Status:
Success: Warping Restraint defined.
Node i
Warping Restraint
Fixed
Node j
Warping Restraint
Continuous
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
Define element(s) and warping restraint
Element(s):
24
All
Clr
Adv
Status:
Success: Warping Restraint defined.
Node i
Warping Restraint
Continuous
Node j
Warping Restraint
Fixed
Apply
Cancel

2) Alternate Boundary Conditions -- 2nd-Order Elastic Analysis

1) From the **Analysis** menu select **Static** and submenu option **2nd-Order Elastic**.

2) At the bottom menu bar, the **Analysis Type:** should already be set to **Space Frame** as desired.

3) Click on the **Apply** button to perform the analysis.

4) From the **Results** menu select **Node Displacements**.

5) Click on the midspan node of interest, node **13**, and the displacements for base 6 degree of freedoms are provided in the bottom menu bar.

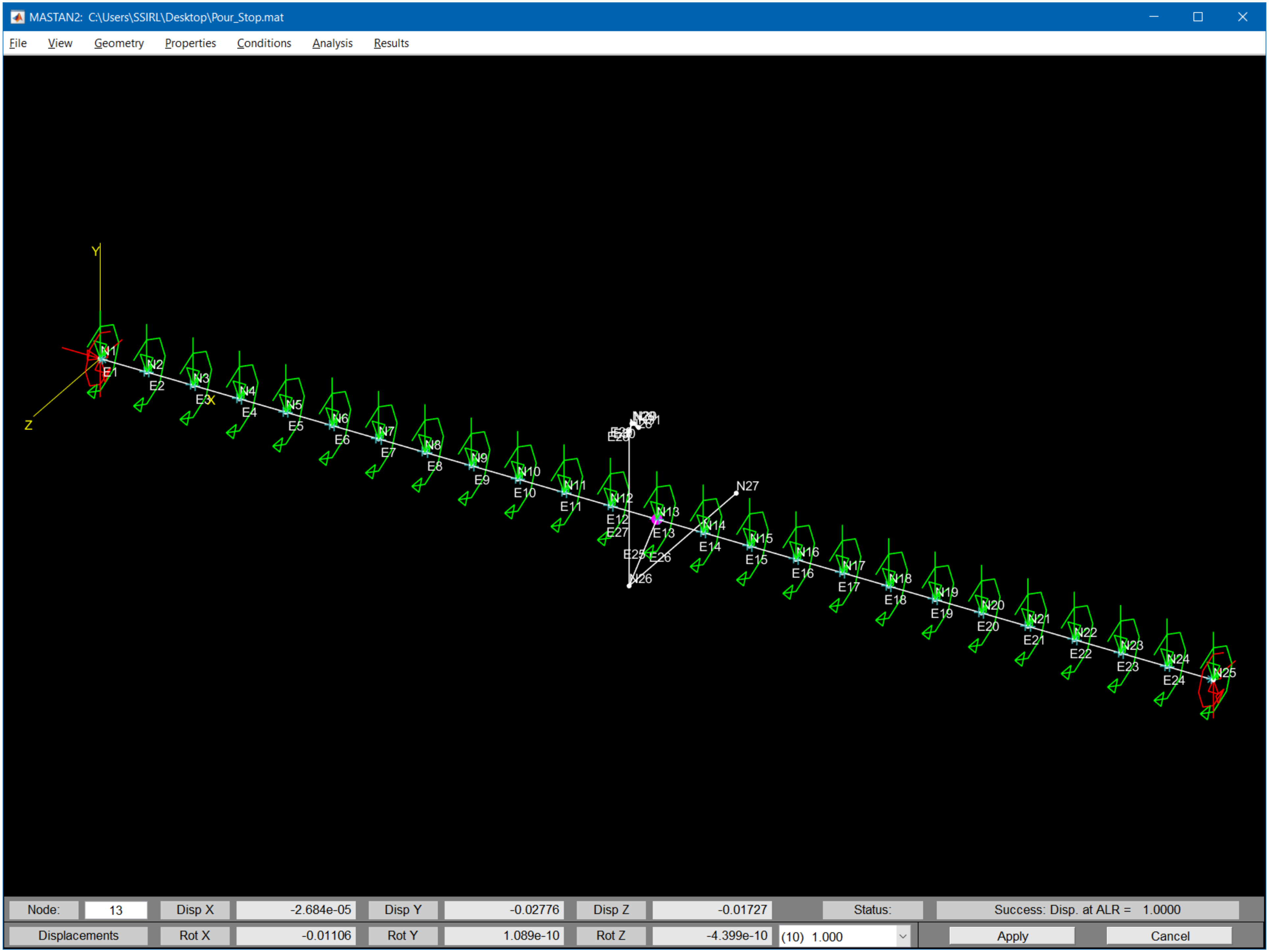
Results:

Disp X	Disp Y	Disp Z	Rot X	Rot Y	Rot Z
-2.684e-5	-0.02776	-0.01727	-0.01106	~0	~0

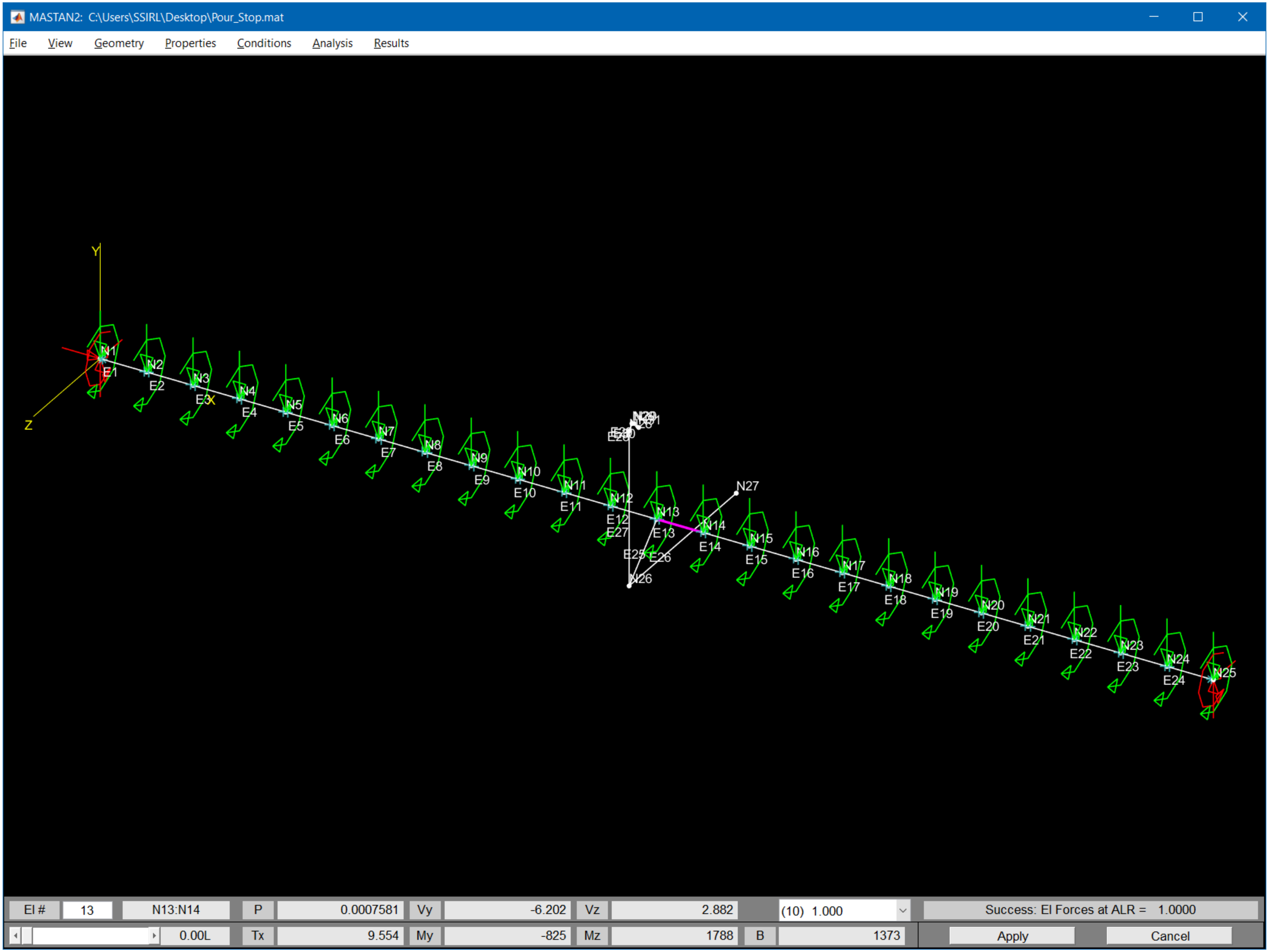
6) From the **Results** menu select **Element Forces**.

7) On the undeflected shape, click on the element of interest, element **13**, and the internal forces are provided in the bottom menu bar. These are the forces at the start of the member and the middle of the beam.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Node: 13
Disp X -2.684e-05
Disp Y -0.02776
Disp Z -0.01727
Status:
Success: Disp. at ALR = 1.0000
Displacements
Rot X -0.01106
Rot Y 1.089e-10
Rot Z -4.399e-10
(10) 1.000
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
El #
13
N13:N14
P
0.0007581
Vy
-6.202
Vz
2.882
(10) 1.000
Success: El Forces at ALR = 1.0000
0.00L
Tx
9.554
My
-825
Mz
1788
B
1373
Apply
Cancel

3) Updating the Geometry -- Scaling

A newer feature of MASTAN2 allows for the length of the model to quickly updated.

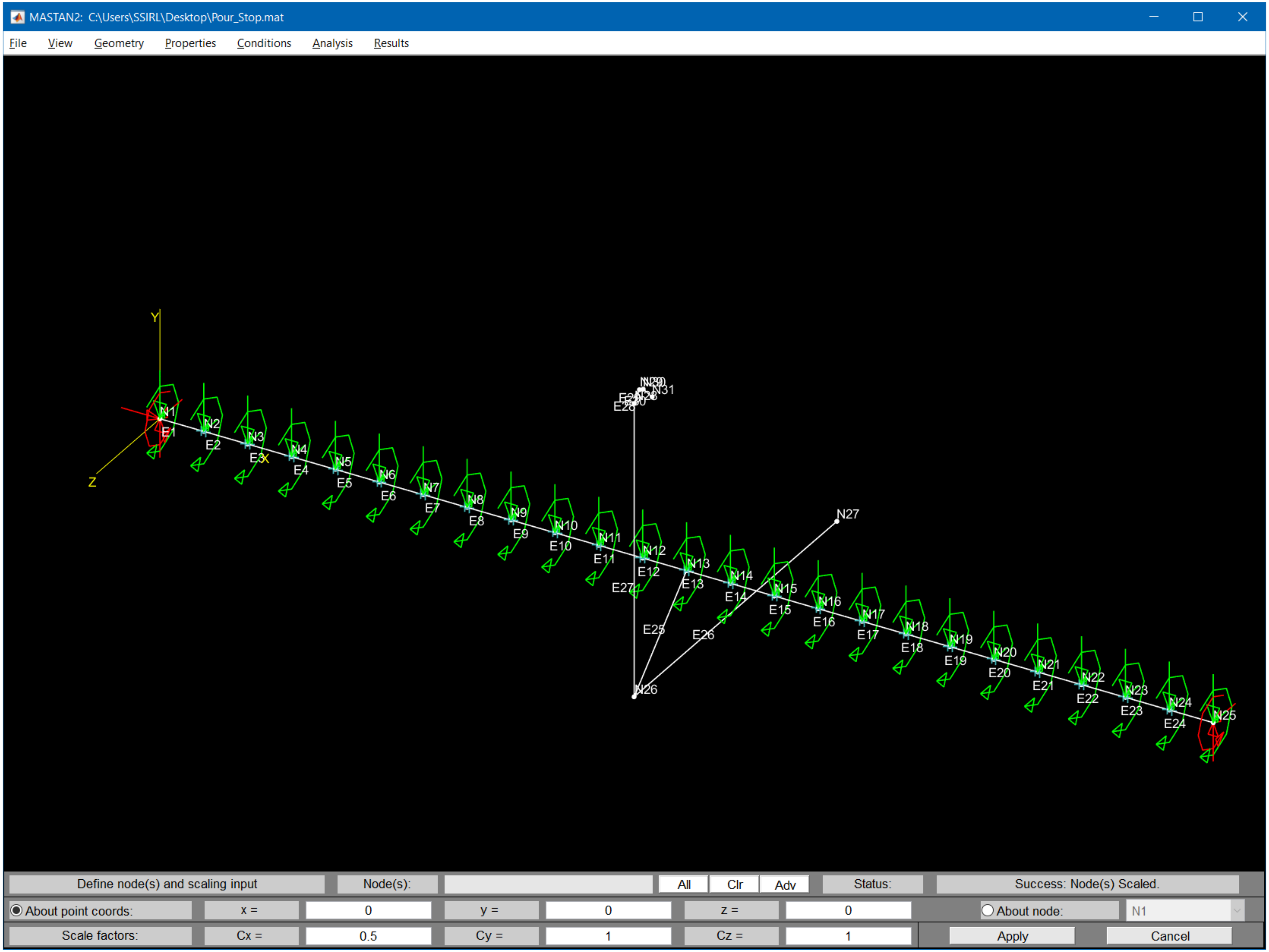
1) From the **Geometry** menu select **Scale Node(s)**.

2) At the bottom menu bar, click in the edit box to the right of **Cx =** and change **1** to **0.5**. This will scale the model only in the x-direction making the overall length of the beam 24 inches.

3) Click the **All** button to populate the list of nodes.

4) Click on the **Apply** button.

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Define node(s) and scaling input
Node(s):
All
Clr
Adv
Status:
Success: Node(s) Scaled.
About point coords:
x =
0
y =
0
z =
0
About node:
N1
Scale factors:
Cx =
0.5
Cy =
1
Cz =
1
Apply
Cancel

3) Updating the Geometry – Update Loading

1) From the **Conditions** menu select **Define Forces**.

2) At the bottom menu bar, click in the edit box just to the right of **PY =** and change **0** to **-6.17**. Click in the edit box just to the right of **PZ =** and change **0** to **2.95**.

3) Click the **Adv** button to open the pop-up menu. To select the main beam nodes, change the edit box to the left of **Z** to **-1**. Change the edit box to the right of **Z** to **1**.

4) Click **Add** to add all main beam nodes.

5) Click on the **Apply** button.

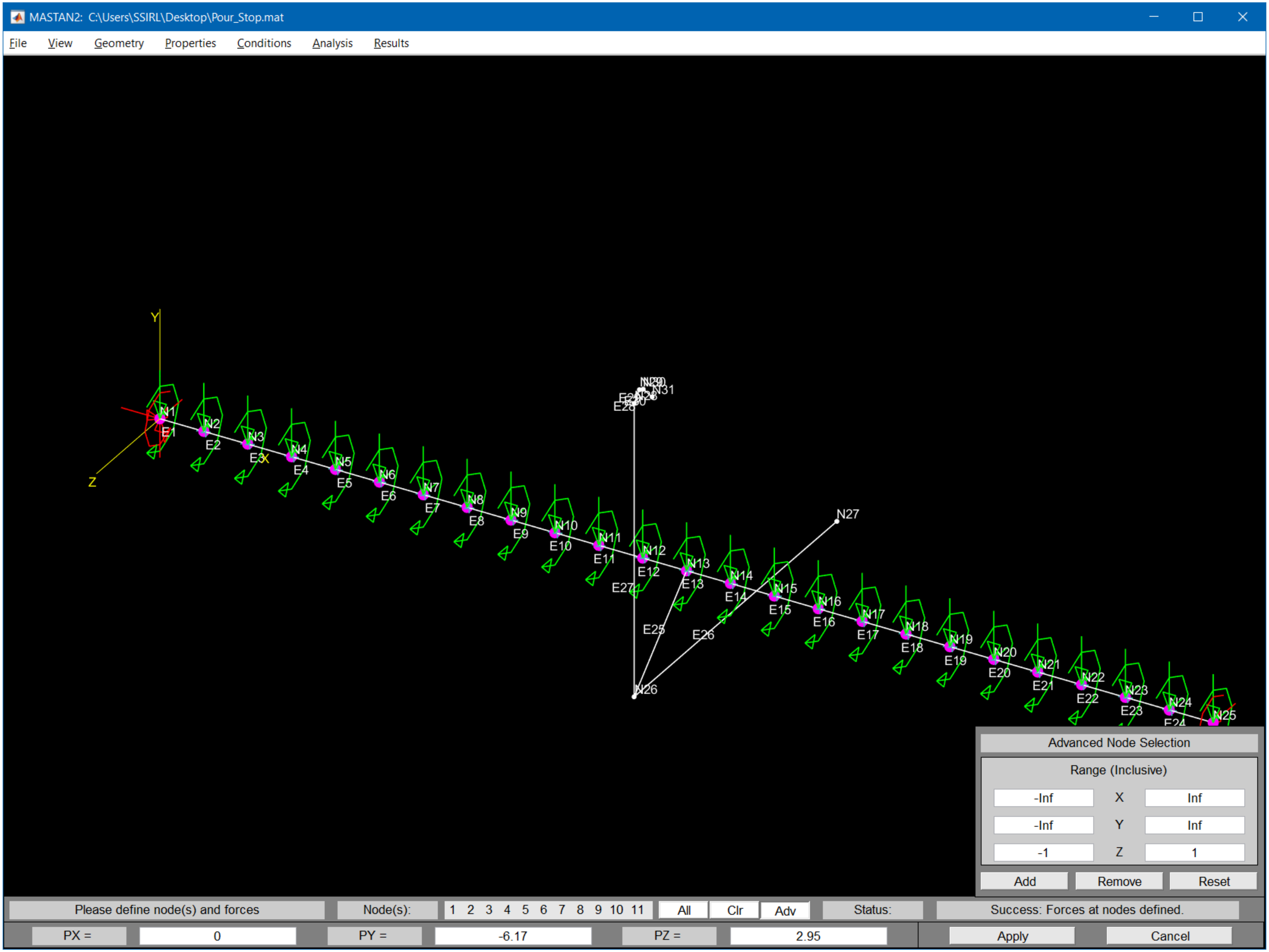
6) From the **Conditions** menu select **Define Moments**.

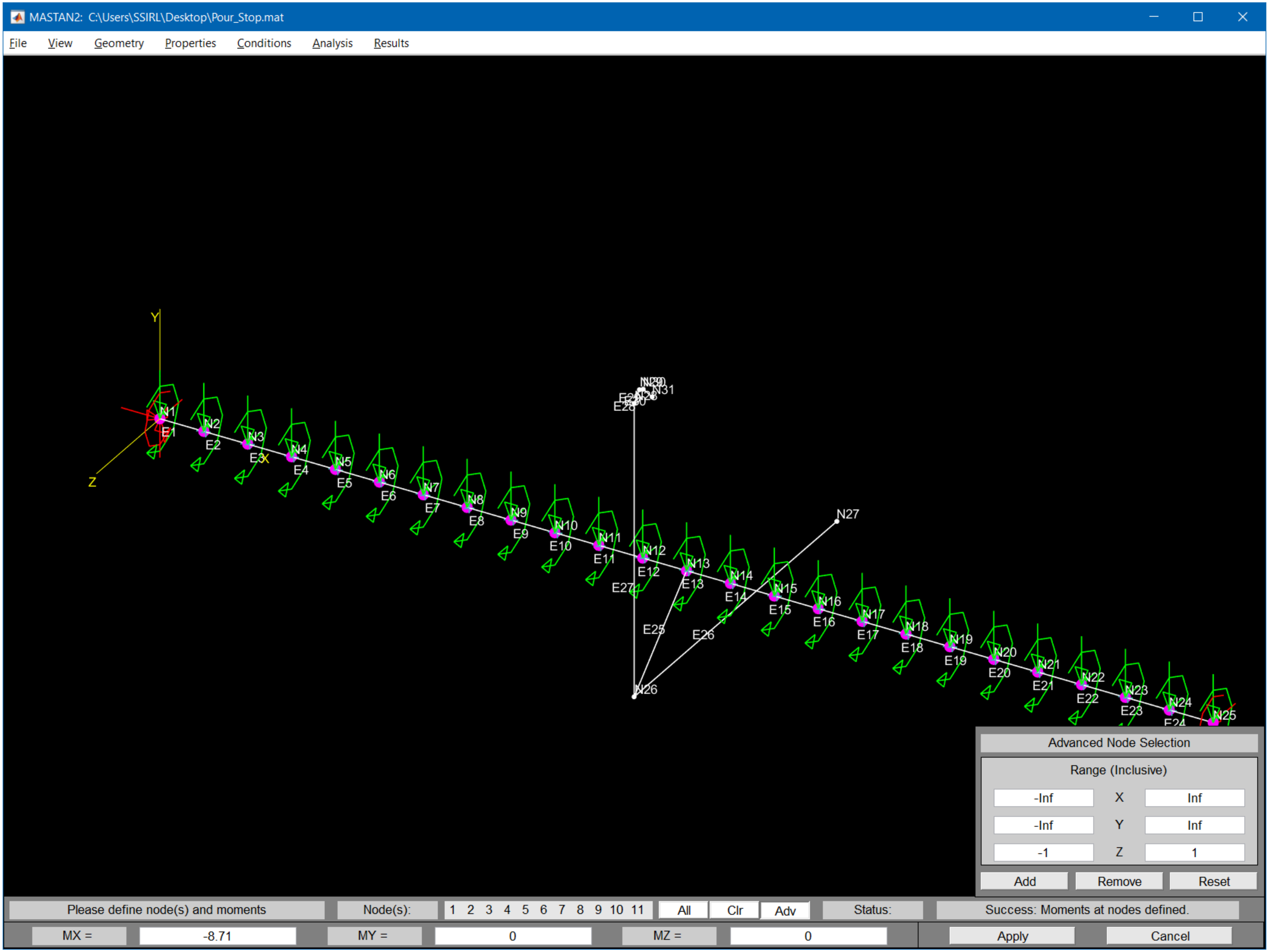
7) At the bottom menu bar, click in the edit box just to the right of **Mx =** and change **0** to **-8.71**.

8) Click the **Adv** button to open the pop-up menu. To select the main beam nodes, change the edit box to the left of **Z** to **-1**. Change the edit box to the right of **Z** to **1**.

9) Click **Add** to add all main beam nodes.

10)Click on the **Apply** button.


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Advanced Node Selection
Range (Inclusive)
-Inf X Inf
-Inf Y Inf
-1 Z 1
Add
Remove
Reset
Please define node(s) and forces
Node(s):
1 2 3 4 5 6 7 8 9 10 11
All
Clr
Adv
Status:
Success: Forces at nodes defined.
PX = 0
PY = -6.17
PZ = 2.95
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Advanced Node Selection
Range (Inclusive)
-Inf X Inf
-Inf Y Inf
-1 Z 1
Add
Remove
Reset
Please define node(s) and moments
Node(s):
1 2 3 4 5 6 7 8 9 10 11
All
Clr
Adv
Status:
Success: Moments at nodes defined.
MX = -8.71
MY = 0
MZ = 0
Apply
Cancel

3) Updating the Geometry -- 2nd-Order Elastic Analysis

1) From the **Analysis** menu select **Static** and submenu option **2nd-Order Elastic**.

2) At the bottom menu bar, the **Analysis Type:** should already be set to **Space Frame** as desired.

3) Click on the **Apply** button to perform the analysis.

4) From the **Results** menu select **Node Displacements**.

5) Click on the midspan node of interest, node **13**, and the displacements for base 6 degree of freedoms are provided in the bottom menu bar.

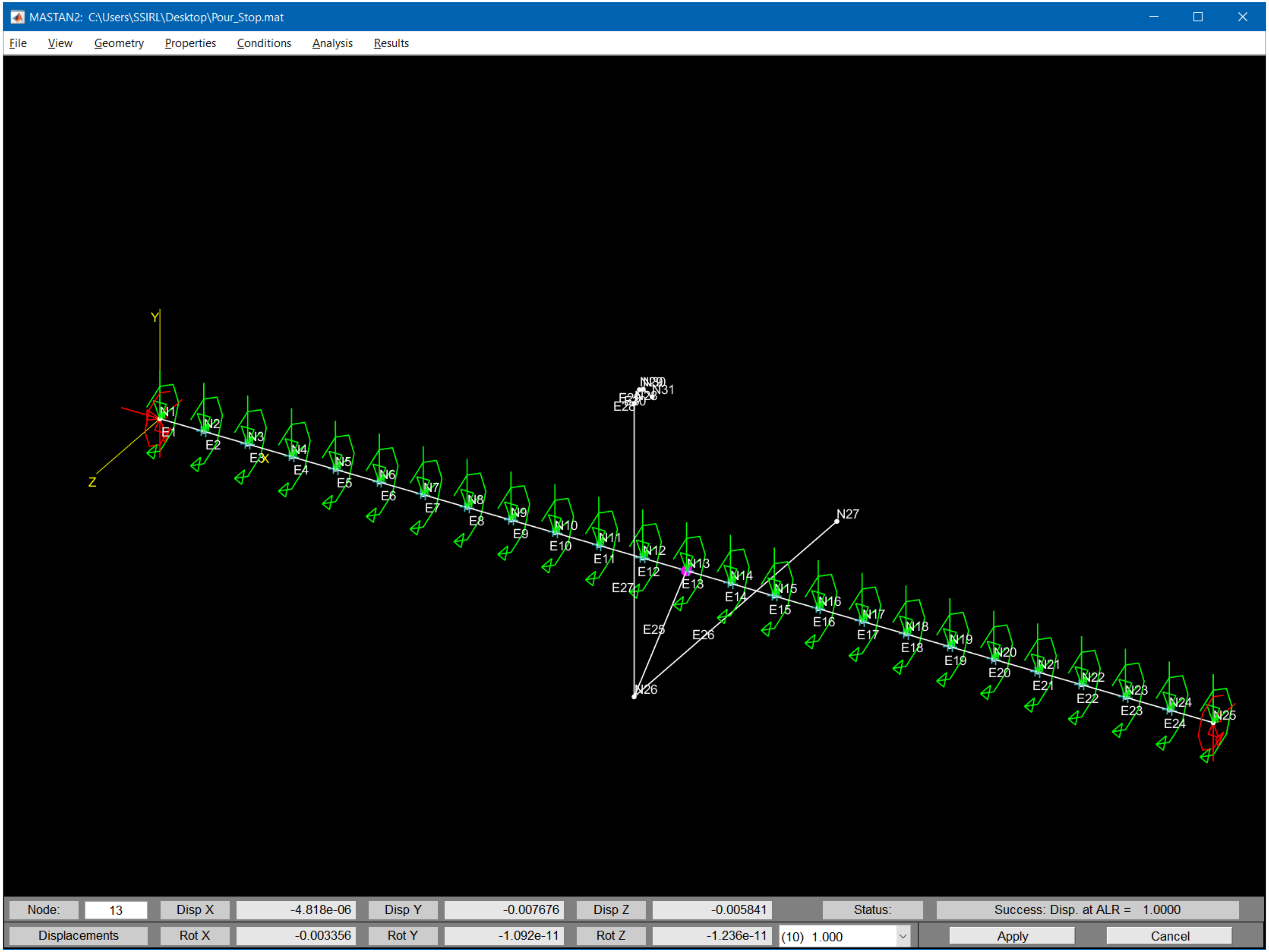
Results:

Disp X	Disp Y	Disp Z	Rot X	Rot Y	Rot Z
-4.818e-6	-0.007676	-0.005841	-3.356e-3	~0	~0

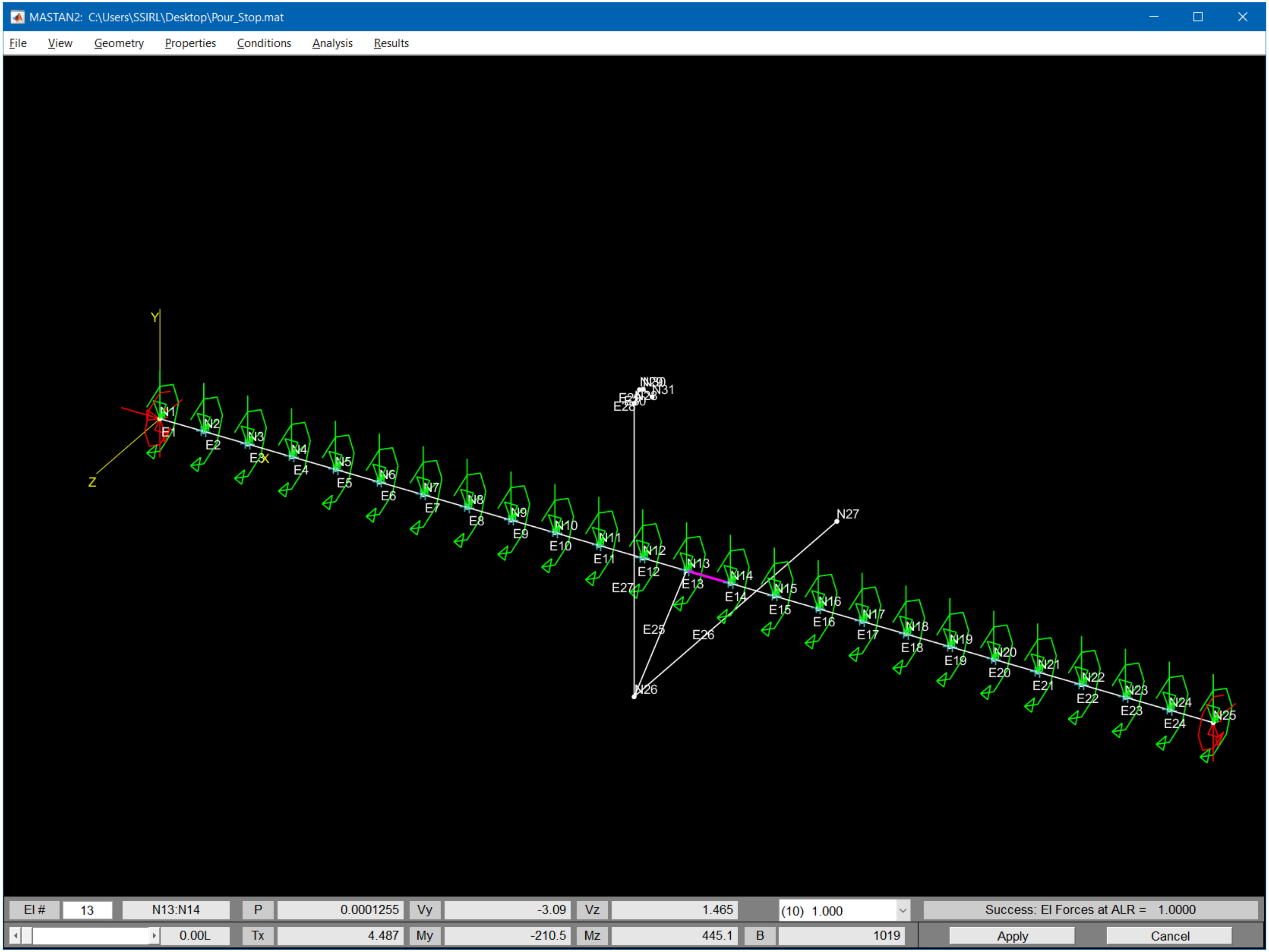
6) From the **Results** menu select **Element Forces**.

7) On the undeflected shape, click on the element of interest, element **13**, and the internal forces are provided in the bottom menu bar. These are the forces at the start of the member and the middle of the beam.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
X
N1
N2
N3
N4
N5
N6
N7
N8
N9
N10
N11
N12
N13
N14
N15
N16
N17
N18
N19
N20
N21
N22
N23
N24
N25
N26
N27
N31
E1
E2
E3
E4
E5
E6
E7
E8
E9
E10
E11
E12
E13
E14
E15
E16
E17
E18
E19
E20
E21
E22
E23
E24
E25
E26
E27
Node:
13
Disp X
-4.818e-06
Disp Y
-0.007676
Disp Z
-0.005841
Status:
Success: Disp. at ALR = 1.0000
Displacements
Rot X
-0.003356
Rot Y
-1.092e-11
Rot Z
-1.236e-11
(10) 1.000
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat
File View Geometry Properties Conditions Analysis Results
El # 13 N13:N14 P 0.0001255 Vy -3.09 Vz 1.465
(10) 1.000
Success: El Forces at ALR = 1.0000
0.00L Tx 4.487 My -210.5 Mz 445.1 B 1019
Apply
Cancel

3) Updating the Geometry -- Additional Sections

If other critical locations for the displacement of the cross section are needed, we can duplicate the existing cross section that was modeled.

1) From the **Geometry** menu select **Duplicate Element(s)**.

2) At the bottom menu bar, click in the edit box to the right of **Delta x =** and change **0** to **-2**. Ensure the check box next to **Include Attributes** is selected to include the property information is included.

3) Click the **Adv** button to open the pop-up menu. Ensure the check box next to **X-axis** is selected.

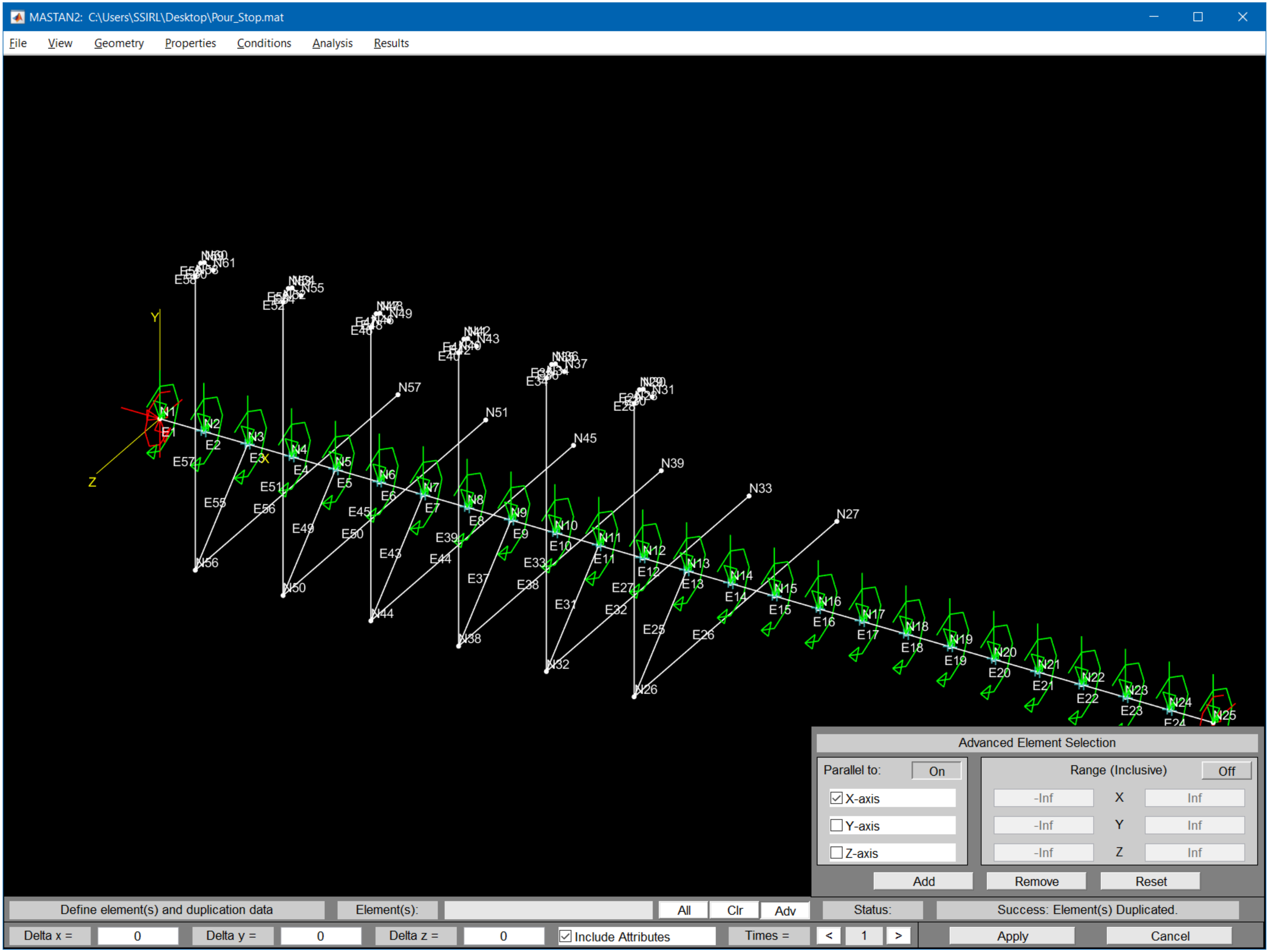
4) To select the illustrated cross section, click **All** next to the **Element(s):** window. Then click the **Remove** button.

5) Repeatedly click the **>** button to the right of **Times =** to increase **1** to **5**.

6) Click on the **Apply** button.

The analysis is not run again as the addition of these elements will not change the results. It would just provide more information on the displacement of the cross section.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
X
Advanced Element Selection
Parallel to:
On
X-axis
Y-axis
Z-axis
Range (Inclusive)
Off
-Inf
X
Inf
-Inf
Y
Inf
-Inf
Z
Inf
Add
Remove
Reset
Define element(s) and duplication data
Element(s):
All
Clr
Adv
Status:
Success: Element(s) Duplicated.
Delta x =
0
Delta y =
0
Delta z =
0
Include Attributes
Times =
<
1
>
Apply
Cancel

This completes the tutorial.