research report

Tutorials for MASTAN2 and Related Validation

RESEARCH REPORT RP21-03

MARCH 2021

Committee on Specifications
for the Design of Cold-Formed
Steel Structural Members

American Iron and Steel Institute

DISCLAIMER

The material contained herein has been developed by researchers based on their research findings. The material has also been reviewed by the American Iron and Steel Institute Committee on Specifications for the Design of Cold-Formed Steel Structural Members. The Committee acknowledges and is grateful for the contributions of such researchers.

The material herein is for general information only. The information in it should not be used without first securing competent advice with respect to its suitability for any given application. The publication of the information is not intended as a representation or warranty on the part of the American Iron and Steel Institute, or of any other person named herein, that the information is suitable for any general or particular use or of freedom from infringement of any patent or patents. Anyone making use of the information assumes all liability arising from such use.

PREFACE

This report creates the tutorials helping engineers to correctly model and analyze structural systems comprised of non-doubly symmetric sections using the latest version of MASTAN2. The report also validated a MASTAN2 model using non-doubly symmetric sections against results from a small test program. The following report and tutorials are provided:

AISI Small Project Summary – Tutorials for MASTAN2 and Related Validation

- Tutorial for MASTAN2 v5.1 - Introductory Frame
- Tutorial for MASTAN2 v5.1 - Pour Stop Beam
- Tutorial for MASTAN2 v5.1 – Steel Joist

The input files for tutorial examples can be downloaded from the link below:

1. Frame tutorial example:
   https://www.dropbox.com/s/u3gik4k8ww9bppi/Frame_End.mat?dl=0

2. Pour-stop example:
   https://www.dropbox.com/s/27qfsqsvqo3iliq/Pour_Stop.mat?dl=0

3. Joist tutorial example:
   https://www.dropbox.com/s/mzizxw5ki9byyff/Joist.mat?dl=0

Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

AISI Small Project Summary – Tutorials for MASTAN2 and Related Validation

Edward J. Sippel, Hannah B. Blum, Ronald D. Ziemian, Joe Pote, & Scott Morton

Sections

- Project Overview
- Tutorial Description
- Experimental Configuration
- Modeling with MASTAN2
- Results
- Summary

Project Overview

Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Project Objectives

- To create tutorials for engineers on how to correctly model and analyze structural systems comprised of non-doubly symmetric sections using the latest version of MASTAN2.

Result: Created 3 step-by-step tutorials with multiple screenshots.

- To validate a MASTAN2 model using non-doubly symmetric sections against results from a small test program.

Result: Assembled a small system with 2 open web steel joists. Subjected system to two loading sequences resulting in two sets of data to compare against MASTAN2 against. Results show good agreement.

Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Tutorial Description

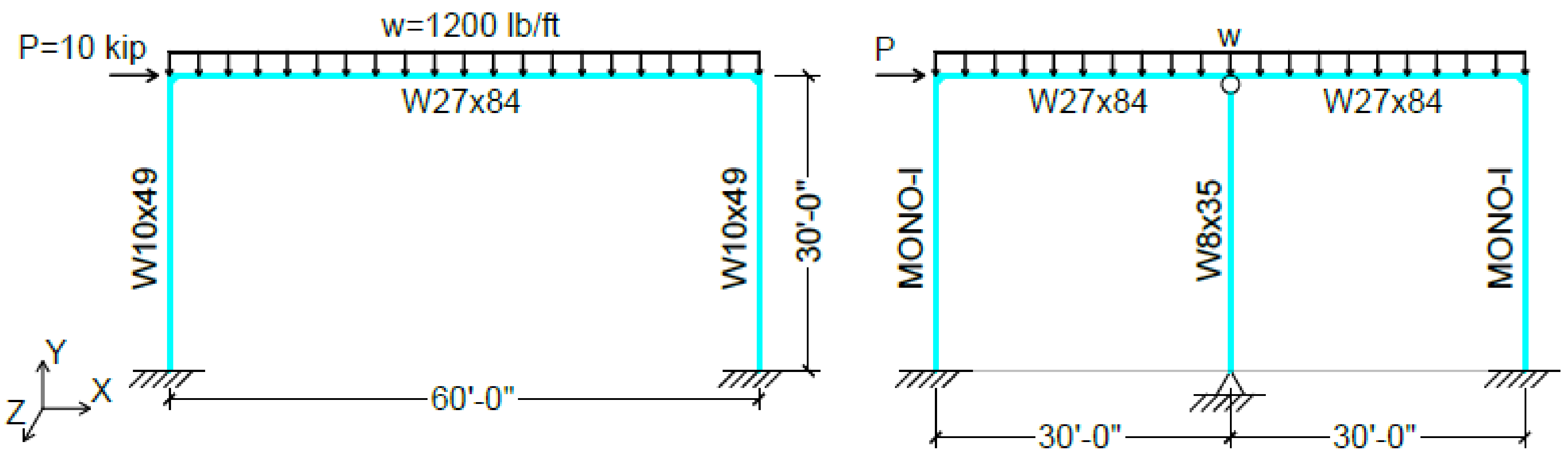
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

General Tutorial Structure

- Each tutorial starts with an introduction of MASTAN2 and the problem to be solved.
- The initial model is constructed, and an initial evaluation is completed.
- The base model is altered for a second scenario and analyzed again.
- Lastly, some additional piece of information related to the structural analysis is provided.
- No one tutorial covers all features in MASTAN2. As a group, these tutorials cover most of the new features offered.

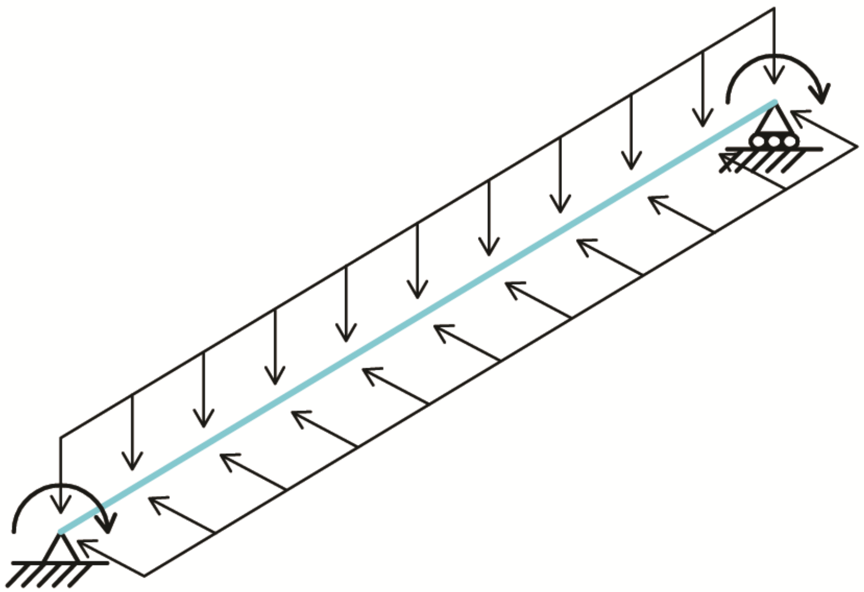
Tutorial 1 – Introductory Frame

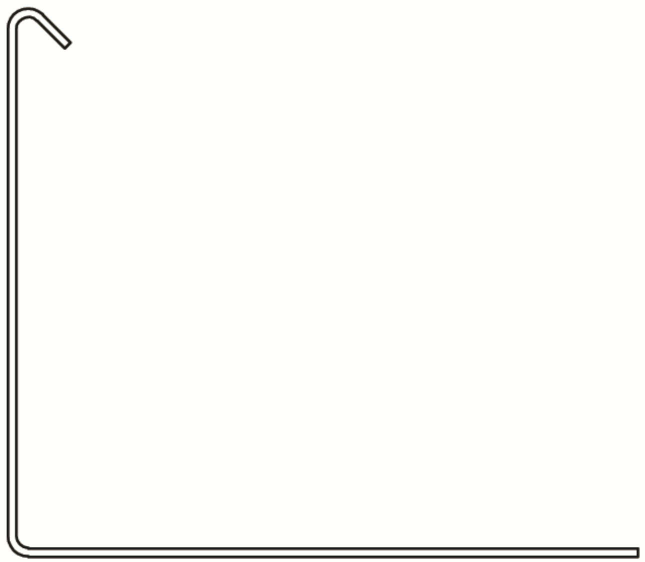
- A single bay, single story portal frame was analyzed in 2-D and then 3-D to show the impact of warping in the modeling process.
- An interior column was added, and non-doubly symmetric abilities were introduced.
- The impact of accounting for the non-doubly symmetric section properties in the overall stability of the frame was highlighted.



Tutorial 2 – Pour Stop Beam

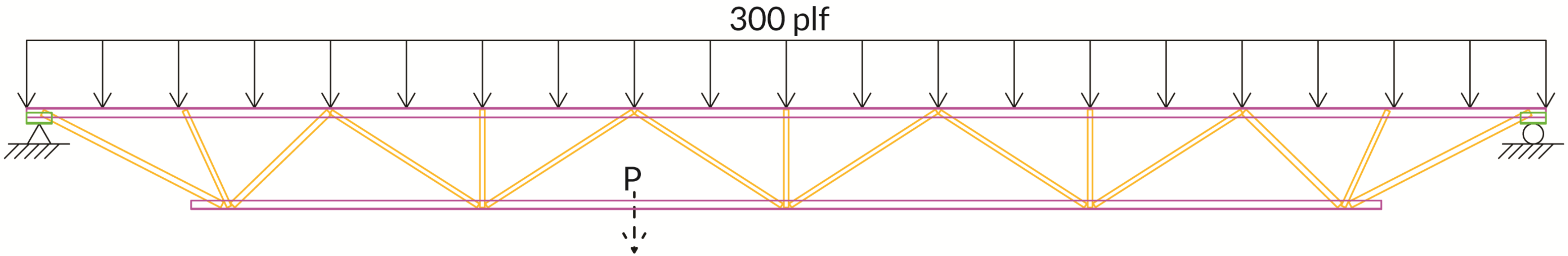
- A non-doubly symmetric beam was subjected to uniform distributed loading including both transverse shear and torsional moment.
- Details on calculating stresses from MASTAN2 results are provided.
- The abilities of multiple new features of MASTAN2 are presented





Tutorial 3 - Joist

- A joist model was subjected to uniform loading and then a hanging load.
- The model is built using the new import function to create the base geometry and then refined within MASTAN2.



- Note: It is similar to the experimental set-up.

Experimental Configuration

Department of Civil and
Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Experimental Set-up

- (2) 20' – 16K2A Joists with rolled angle top and bottom chords with rolled channel webs
- Joists braced with cross bracing at end of bottom chords and decking

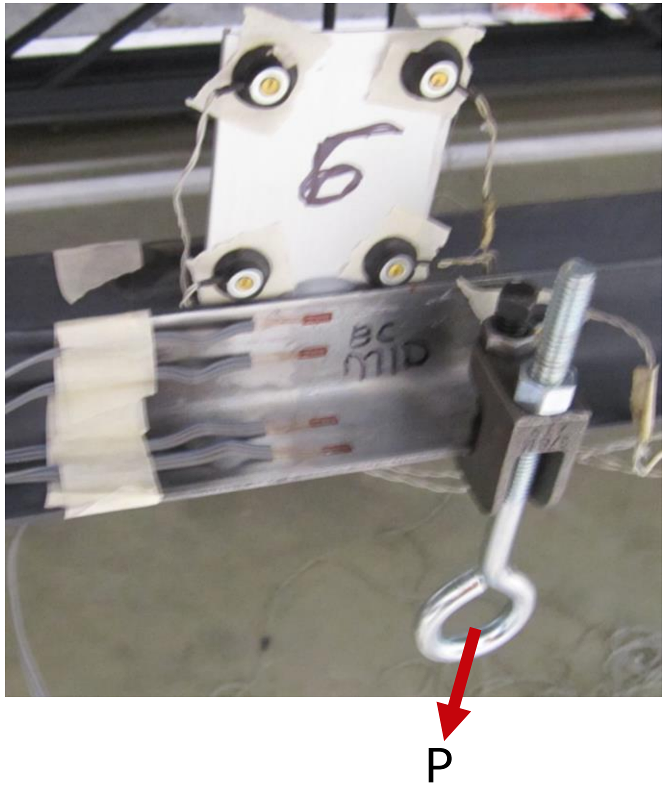
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Test Program

- Scenario 1:
  - Joist was subjected to a uniform distributed loaded applied via barrels with aggregate.
  - At ~0.5 design live load (152 plf) and ~1.0 design live load (304 plf), the bottom chord was subjected to 100 lbs of a hanging load, P

- Scenario 2:
  - No uniform load was applied to joist.
  - The bottom chord was subjected to 280 lbs of a hanging load, P

- In both cases the hanging load was applied near midspan away from the web intersections



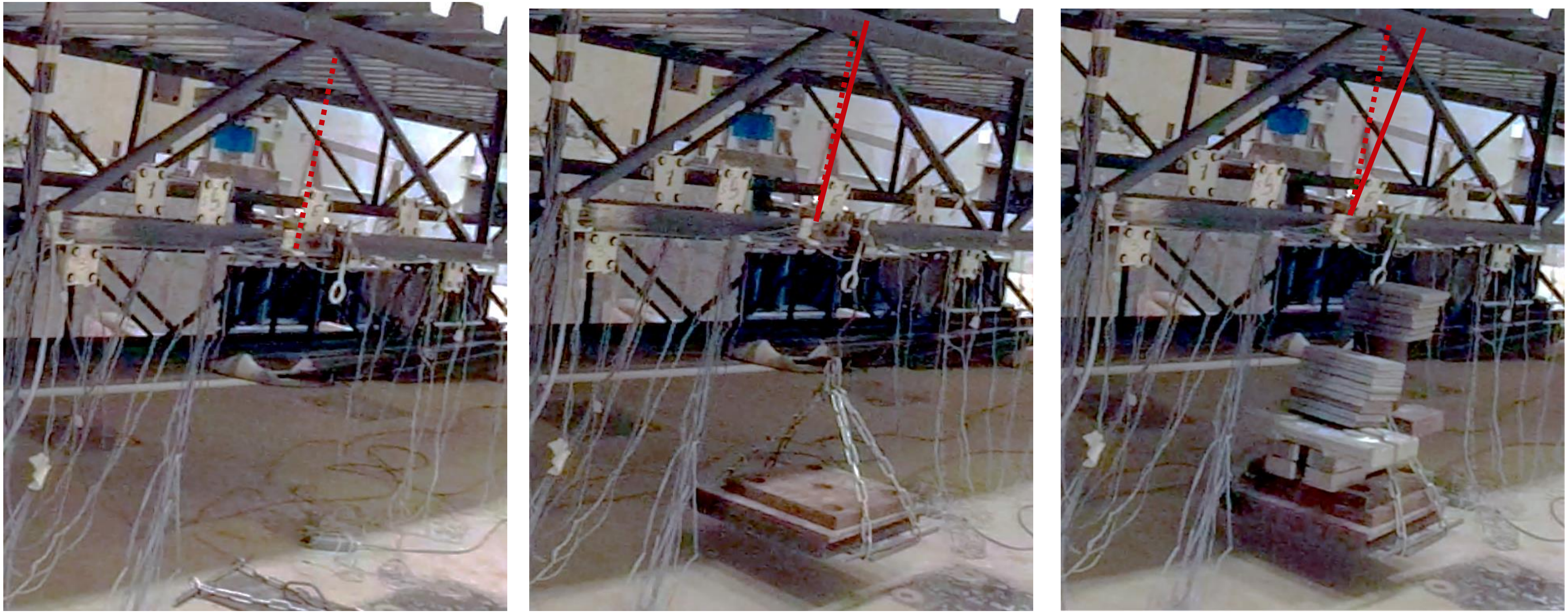
Testing Images – Scenario 1: Half Live Load

Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Testing Images – Scenario 1: Full Live Load

Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Testing Images – Scenario 2



Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Recorded Information

- Recorded displacements using Optotrak system for key translations and rotations.
- Recorded strain data at 5 locations. Each cross section included 4 linear strain gages and 1 shear strain gage to be able to approximate internal forces.

<u>Strain Measurement Locations</u>

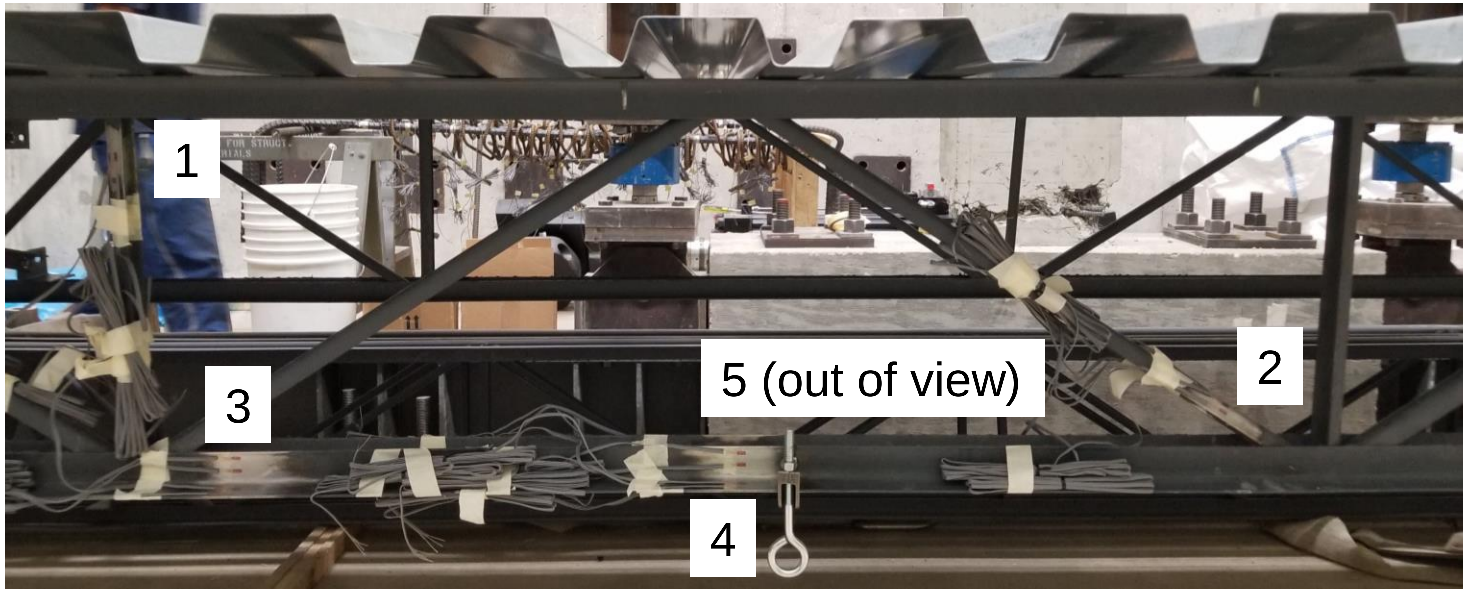
1 – Top of Vertical Web

2 – Bottom of Middle Web

3 – End of Loaded Chord

4 – Midspan of Loaded Chord

5 – Midspan on Other Chord



Modeling with MASTAN2

Department of Civil and
Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

MASTAN2 Model

- Joist members were modeled in an ideal location with a single node connection for the intersection of the webs. The ends of the webs were defined to be fixed for out-of-plane bending, released for in-plane bending (typical joist web pinned design condition), and fixed for warping at the end.

- All members were modeled with the nominal cross section properties in a principal orientation.

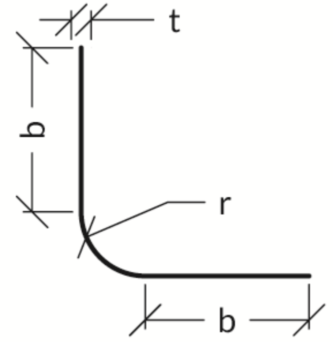
- Top chord was laterally supported at web intersections, ~24” o.c.

- Bottom chord was laterally supported 8” from end web intersection. Connection was inserted above the bottom chord to allow additional twist.

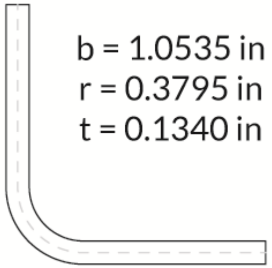
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Cross Section Information

Chord Centerline

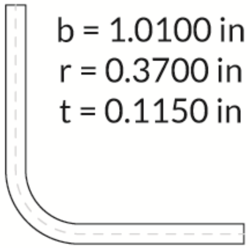


Top Chord

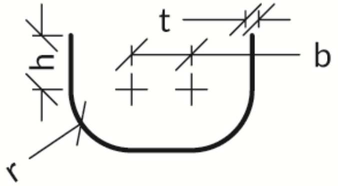


b = 1.0535 in
r = 0.3795 in
t = 0.1340 in

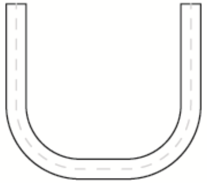
Bottom Chord



b = 1.0100 in
r = 0.3700 in
t = 0.1150 in

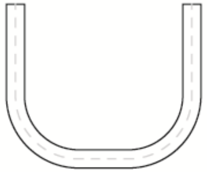


Web Centerline



Web 1

h = 0.5844 in
b = 0.2700 in
r = 0.3700 in
t = 0.1150 in

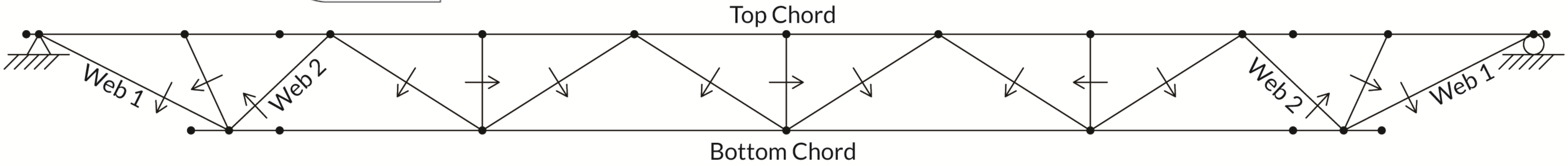


Web 2

h = 0.5258 in
b = 0.2900 in
r = 0.3650 in
t = 0.1050 in

Web 3

h = 0.3102 in
b = 0.3466 in
r = 0.3509 in
t = 0.0767 in



Arrows indicate the open side of the web channels. Web members not otherwise labeled are Web 3

Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

MASTAN2 Loading

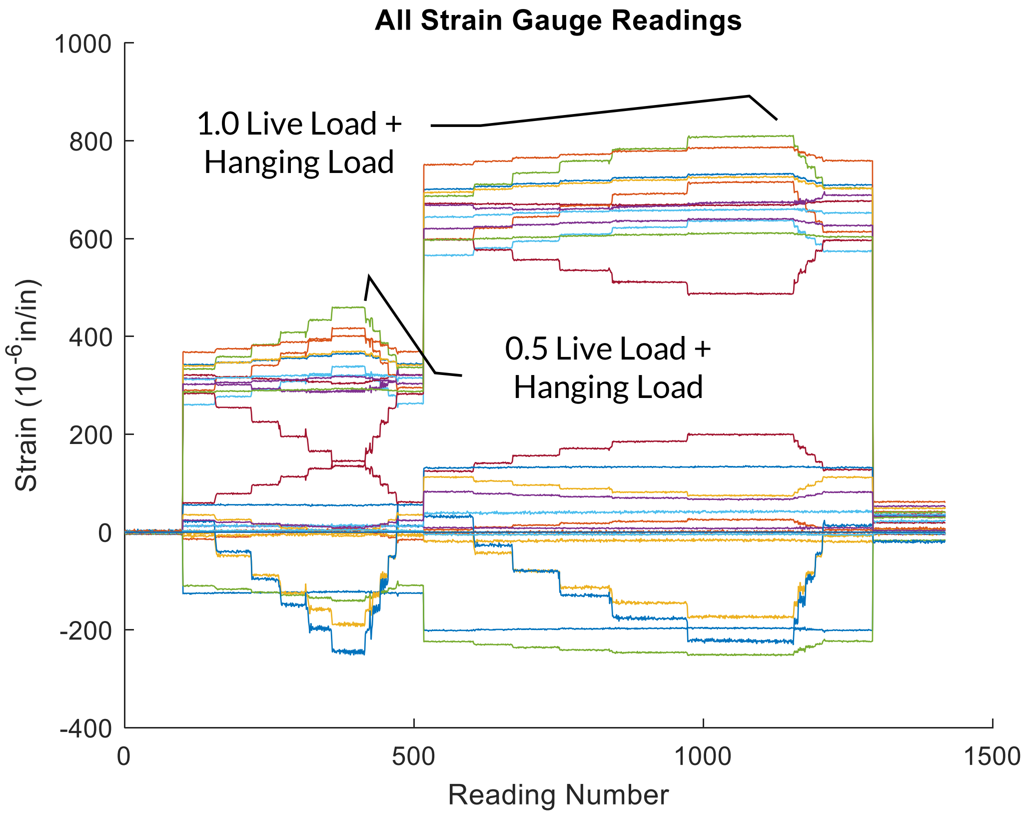
- The model includes uniform load applied to each top chord. The loaded load is applied in the global Y direction which causes it to be split into the member's principal orientation.
- The hanging load is applied via a vertical 2.5 inch 3/8" round steel rod modeled at the end of a horizontal 1.5 inch rigid link extending from the centroid of the bottom chord. The hanging load is applied at the end of the vertical rod to account for movement of the eye bolt.
- The results shown account for the uniform load being applied and then the hanging load being added to the model in a 2^{nd} order elastic analysis.

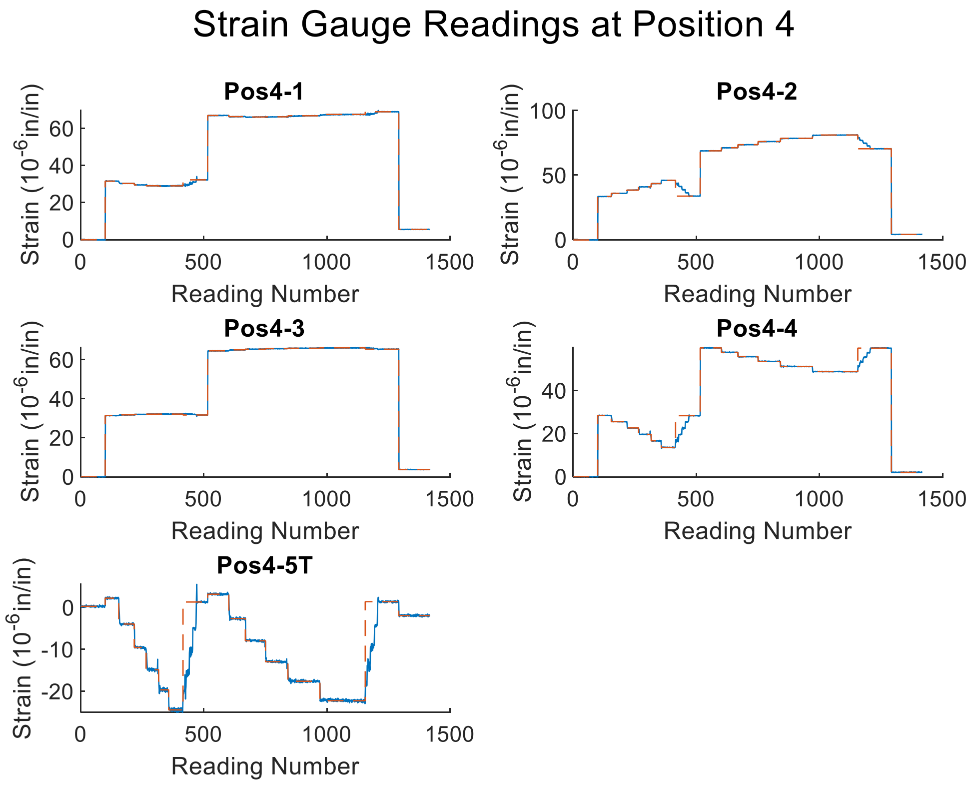
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Results

Example Recorded Information

- Strain gage information during Scenario 1





Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Calculations with Joist Experiment Results

- The displacement information was measured at a few critical locations for comparing translations.

- Most of the displacement information was used to calculate the twist of the bottom chord at the 10 sensor group locations.

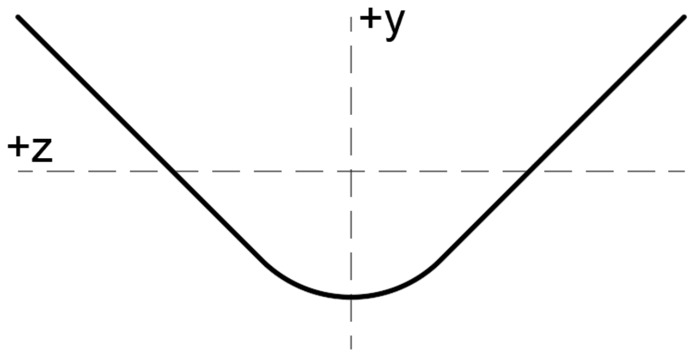
- At each sensor group, the Euler angle and axis was calculated using 3 of the 4 displacement measurements and the component of that angle about the original x-axis was determined. The final rotation presented was based on the average of the 4 combinations.

Calculations with Joist Experiment Results

- Using the strain gauge measurements, internal forces were calculated.

- Two scenarios were considered:
  1) Bimoment and warping were significant meaning all 4 linear strain measurements were needed to define the resulting internal forces.
  2) Warping was negligible and only 3 strain measurements were need to define the internal forces. The resulting forces were calculated excluding one gage and the 4 solutions were averaged.

- The internal forces considered are
  - Axial Force, P
  - Bending Moment about the y-axis, M_y
  - Bending Moment about the z-axis, M_z
  - Bimoment from warping, B



Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Stress Distributions

- Using the internal forces calculated from the experiment and pulled from MASTAN2, the centerline normal stress diagrams were calculated.

- Relationship between normal stress and the internal forces:
  - The same relationship was used for calculation on the previous page.

$$\sigma_x = \frac{P}{A} - \frac{M_z y}{I_{zz}} - \frac{M_y z}{I_{yy}} + \frac{B\omega_n}{C_\omega}$$

- A closely spaced path of nodes along the element centerline was defined. The above relationship was evaluated at each position.

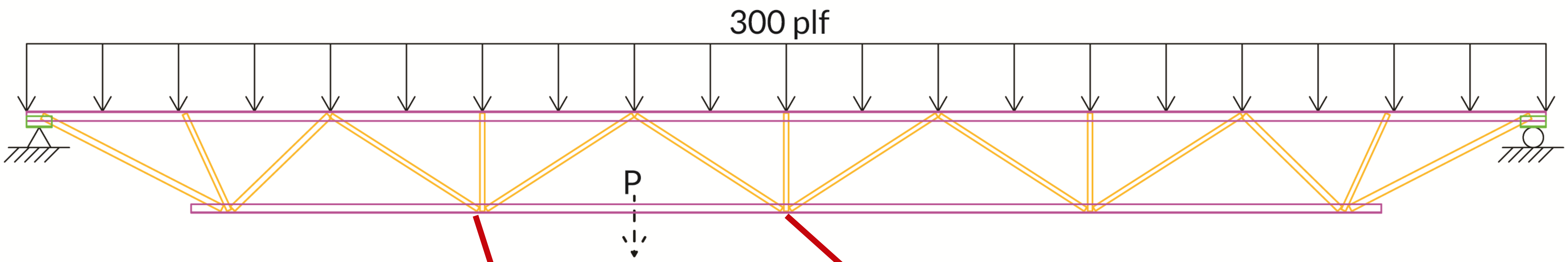
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

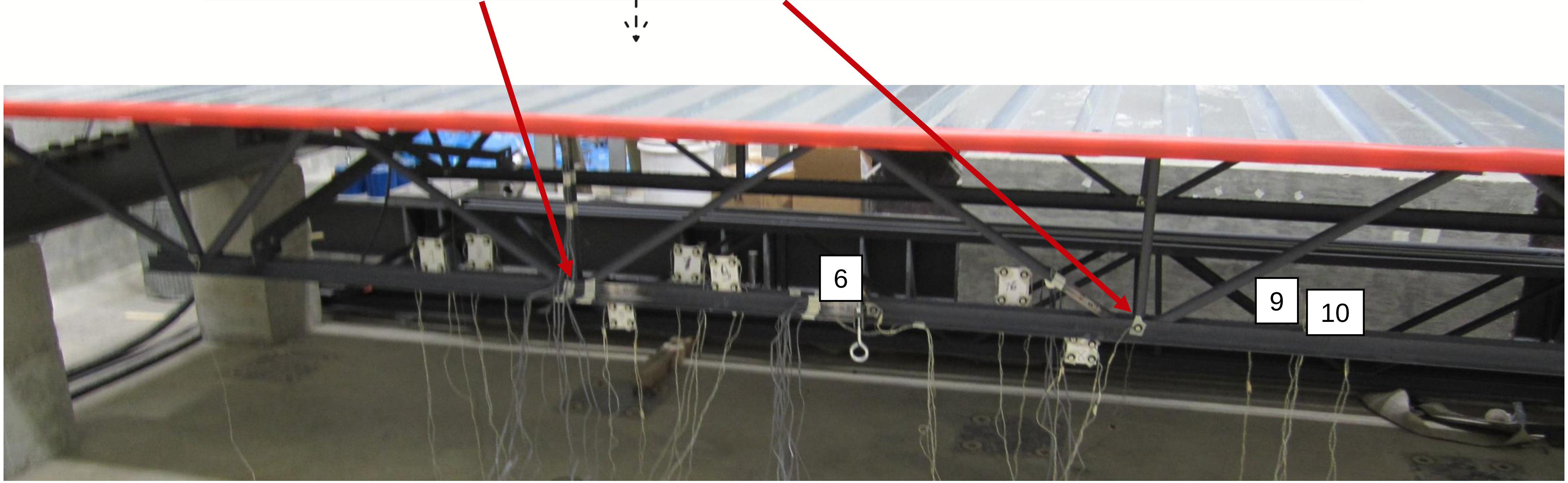
General Comment

- Results related to Position 1 and 2 are not shown. The measurements were taken 3 inches vertically from the extremes of the joist in an attempt to verify the end conditions of the web . This placed the measurements a little farther than the maximum dimension of the web channel away from the welded connection.

- While the individual strain gauge measurements displayed the anticipated stepwise changes in strain corresponding to the applied hanging load, the results indicated that there were likely stress concentration effects being captured from being too close to the welded connection. This effect cannot be captured in MASTAN2 and as such was not shown here.

Reference for Rotation Plots





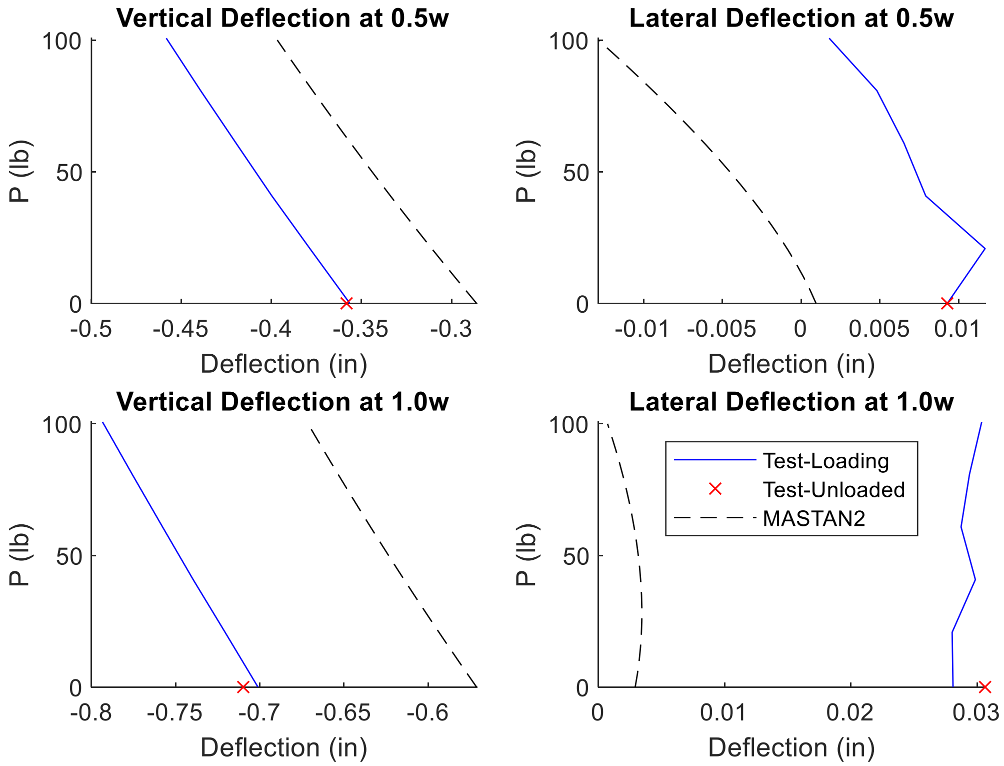
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Result Diagrams – Scenario 1

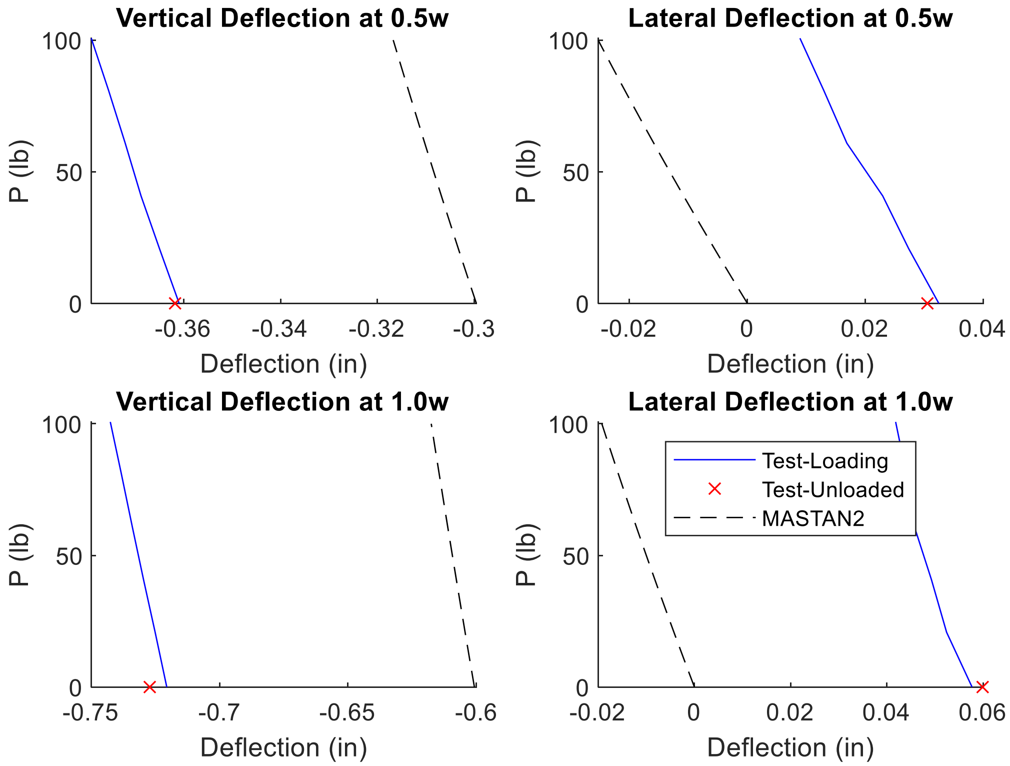
- This section is all diagrams from Scenario 1.
- Force plots are shown with both the test results considering bimoment from warping and from ignoring the effects of warping.
- The stress plots are shown only with the case considering bimoment since at the locations checked the values were small.
  - At the top of each plot, the range of stresses is provided.
  - Stress diagrams have MASTAN2 results on top and Test results on bottom for a given loading. A label is provided underneath the x-axis as a reminder.
  - Diagrams are scaled in each vertical pair, but not across remaining diagrams.
  - Red is tension (+) and blue is compression (-)

Measured Global Displacements

Deflection near Loading

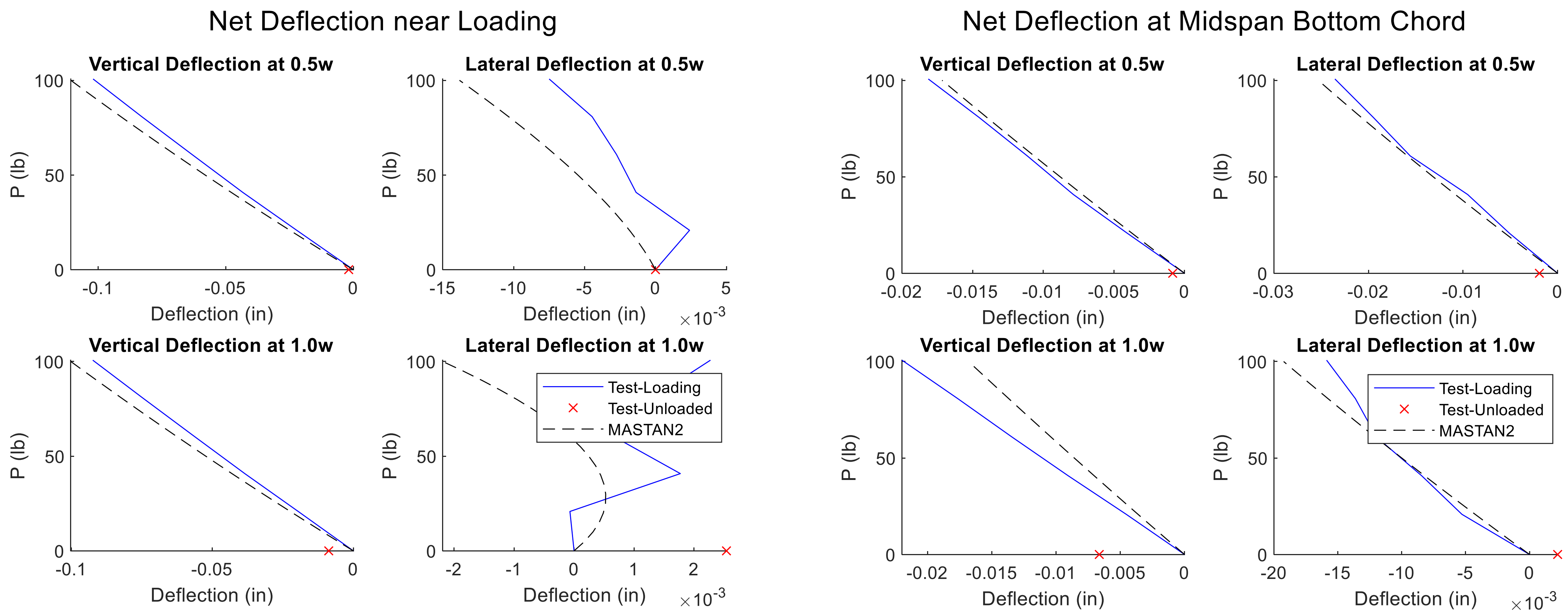


Deflection at Midspan Bottom Chord



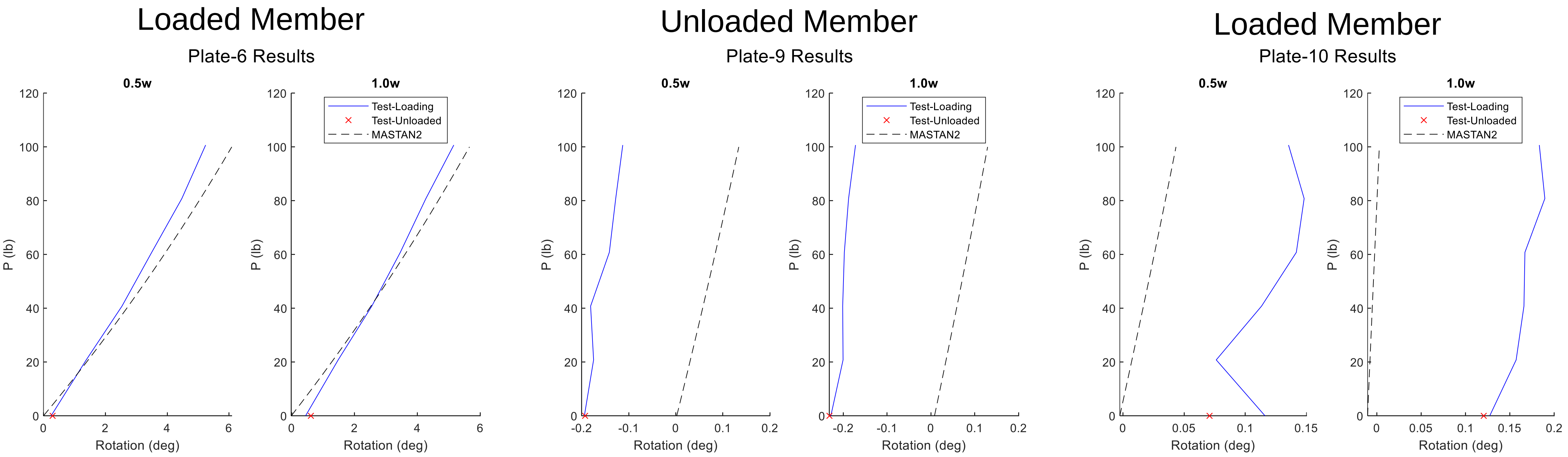
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Measured Net Displacements



Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Measured Rotations

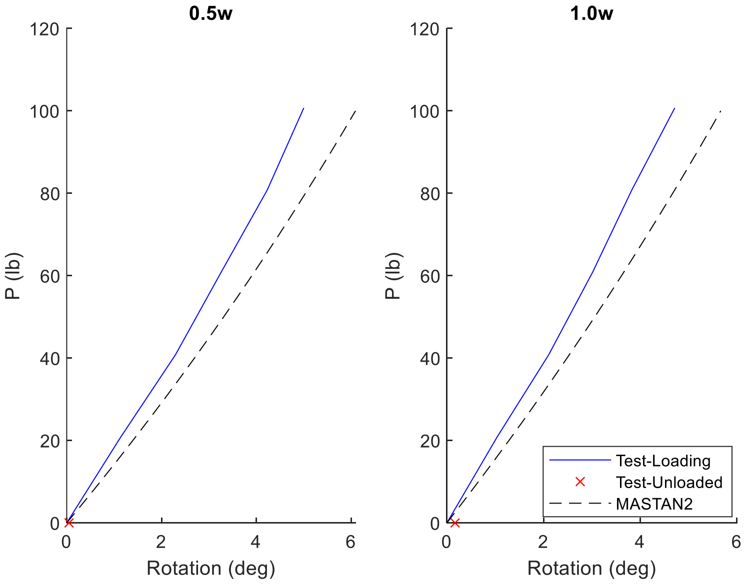


Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Measured Net Rotations

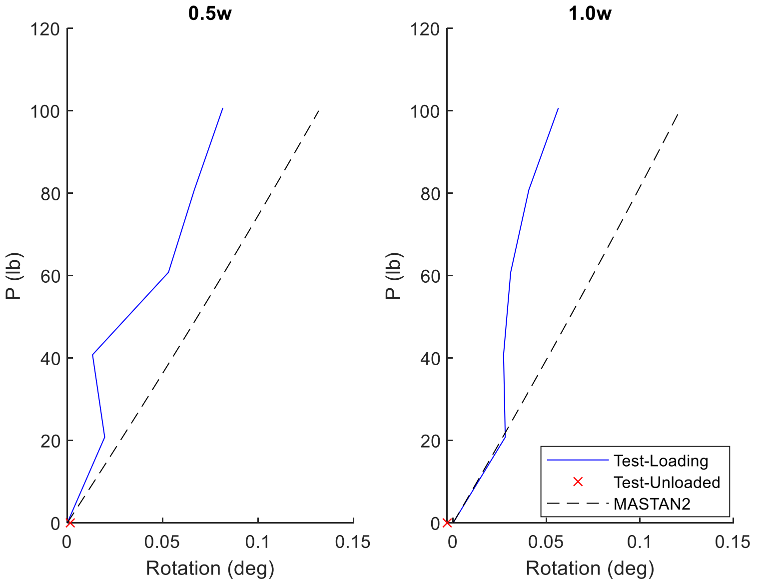
Loaded Member

Net Plate-6 Results



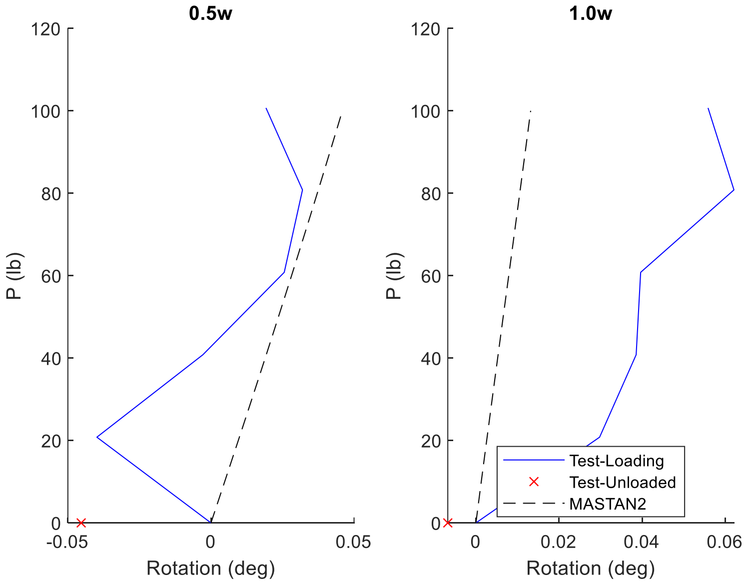
Unloaded Member

Net Plate-9 Results



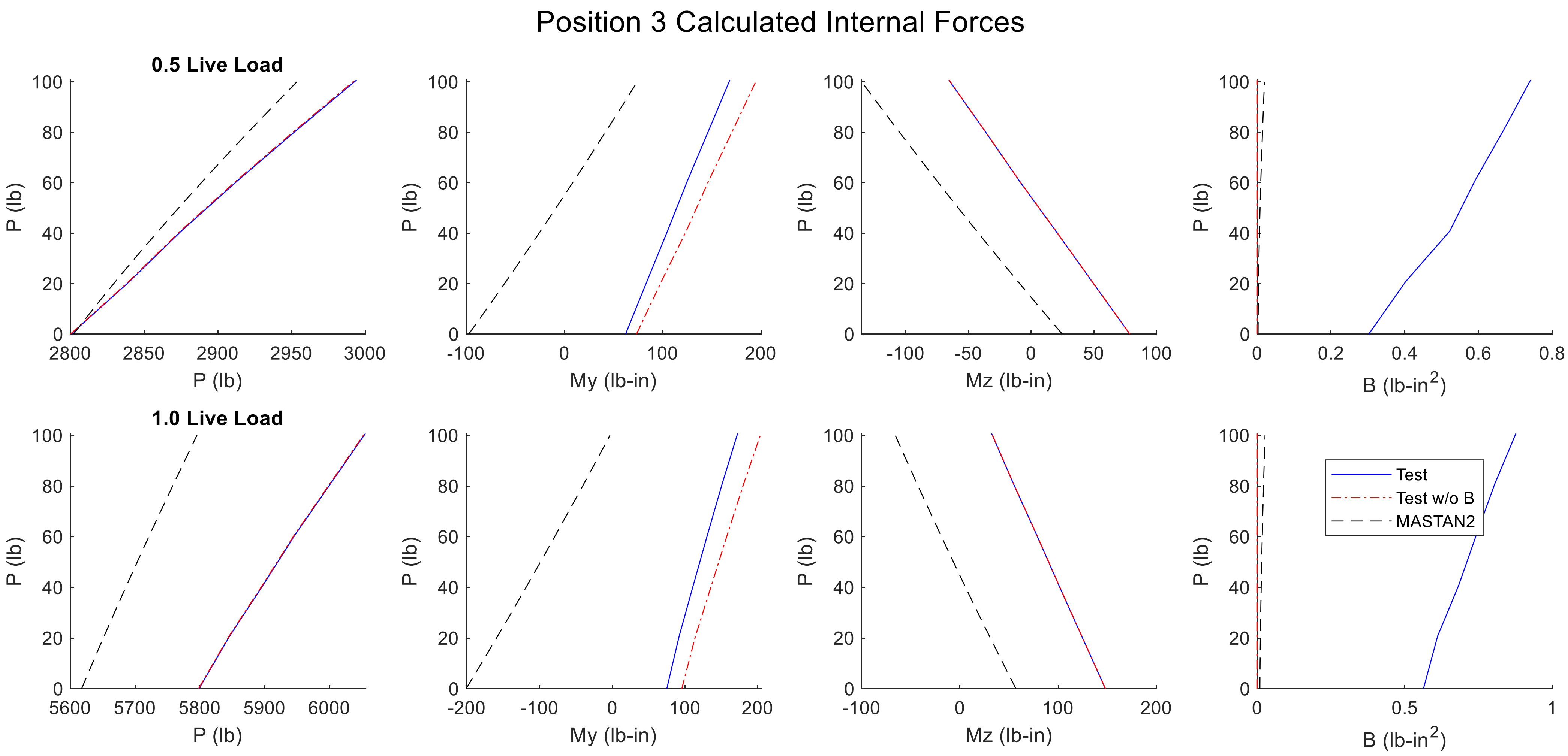
Loaded Member

Net Plate-10 Results



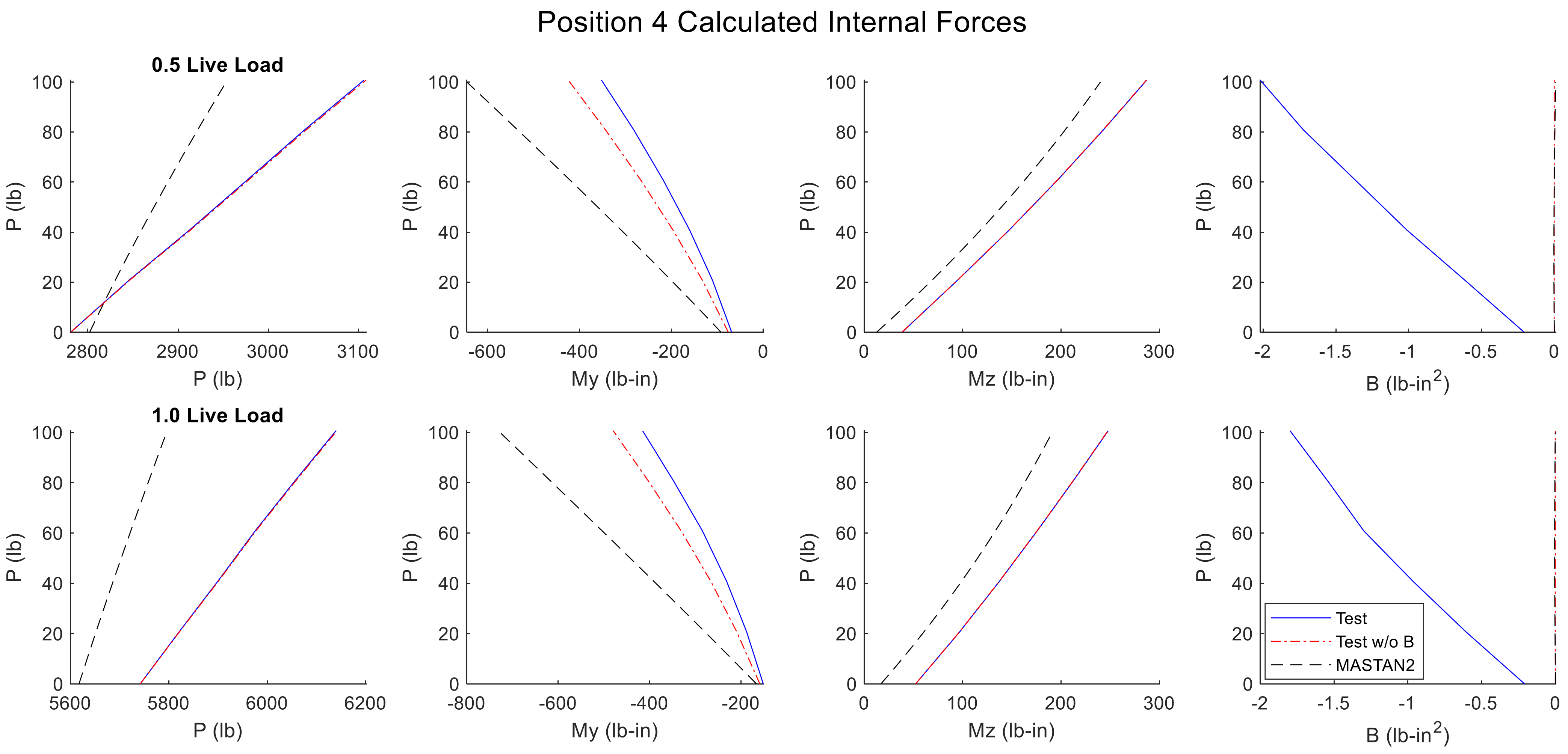
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Force Comparisons



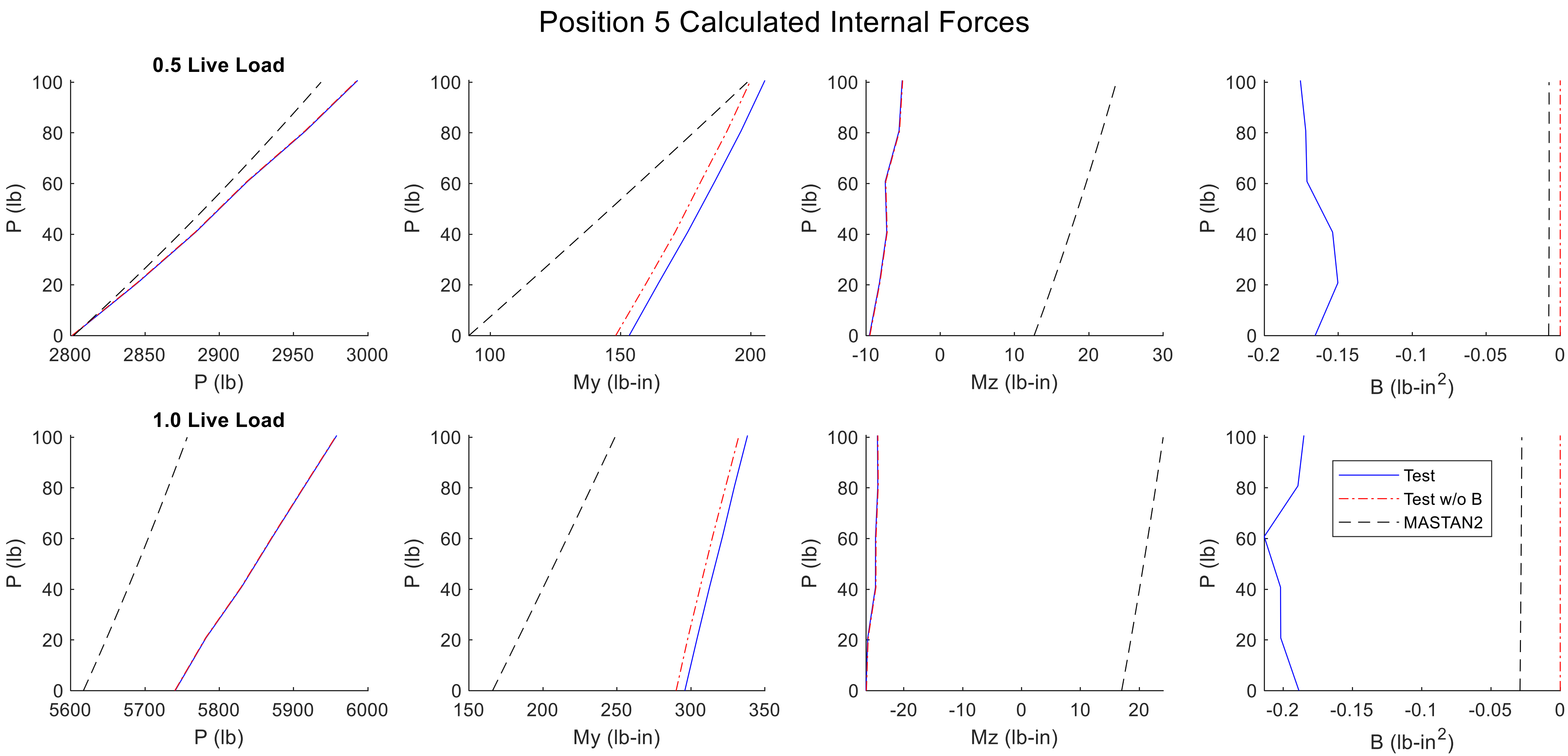
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Force Comparisons



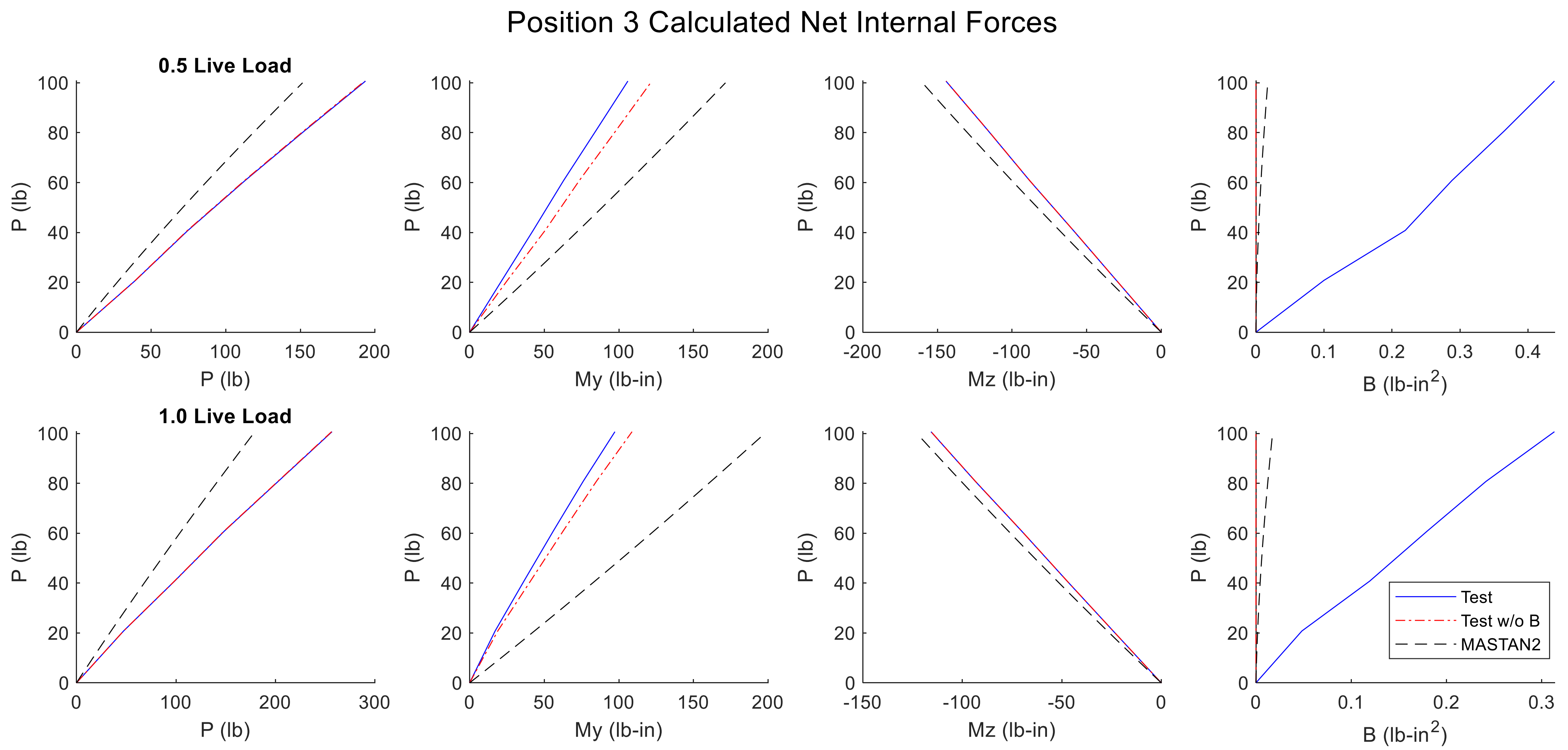
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Force Comparisons



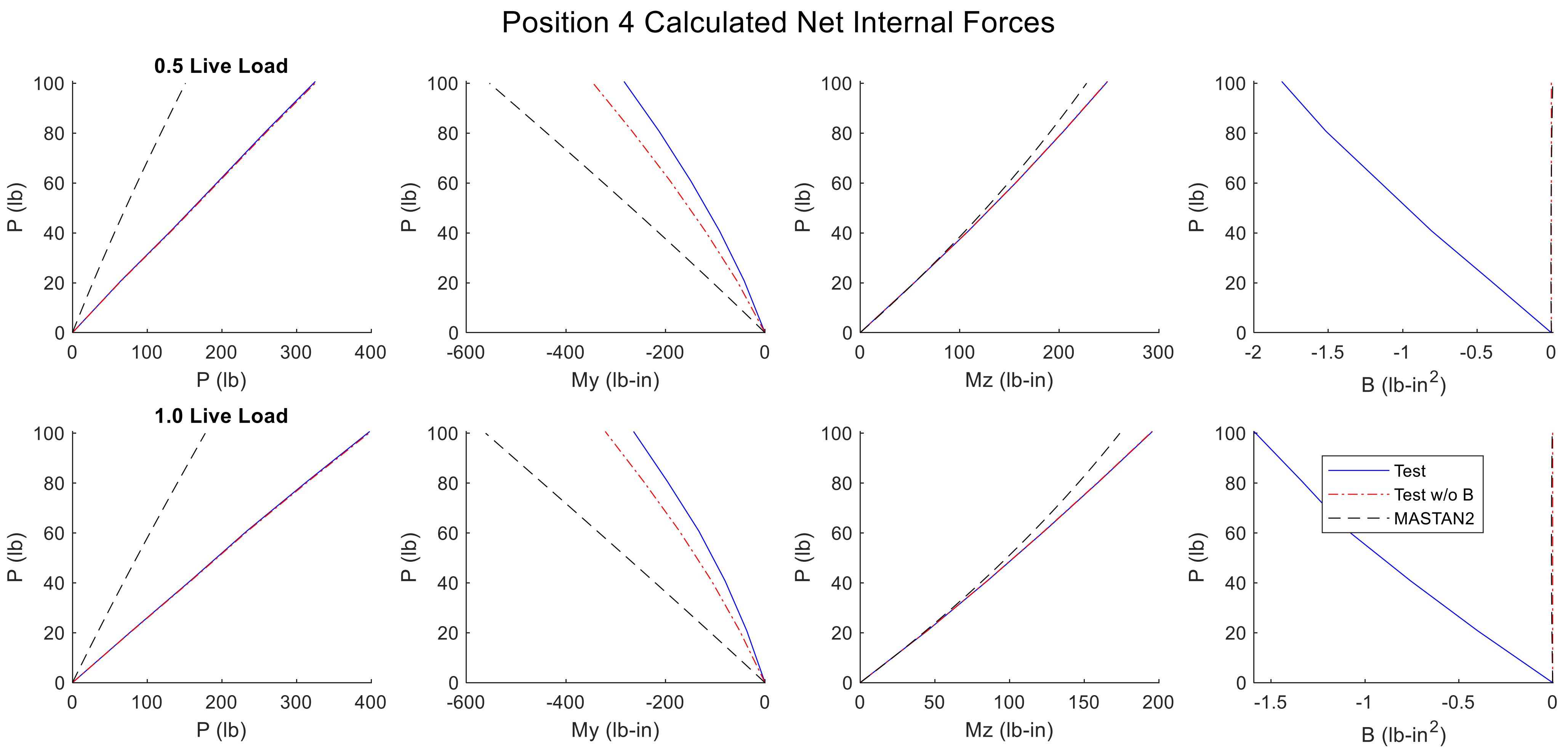
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Net Force Comparisons



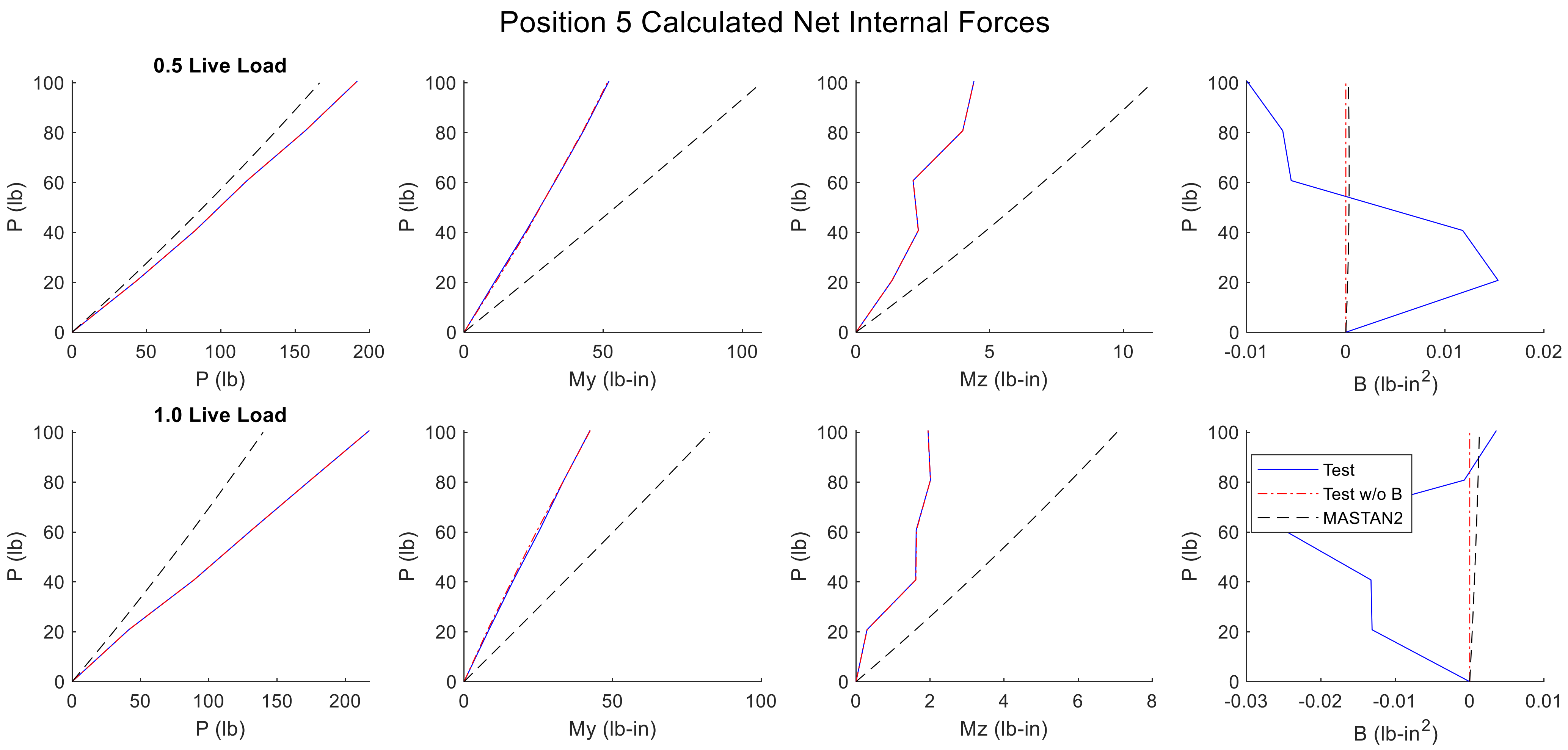
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Net Force Comparisons



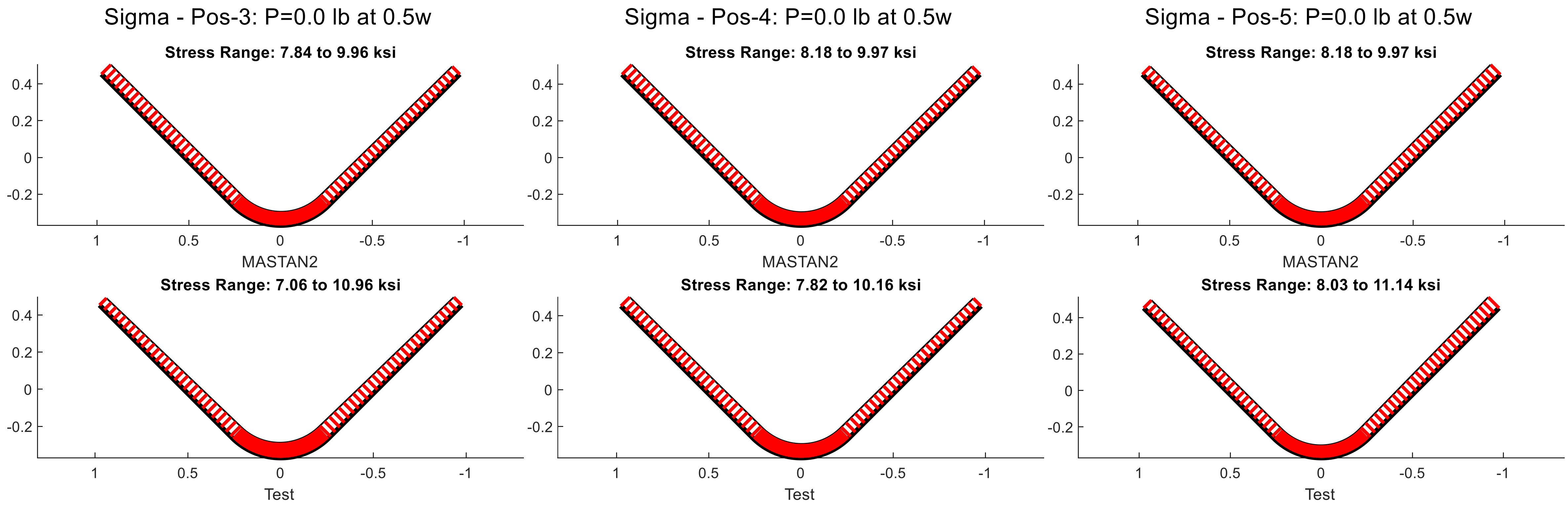
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Net Force Comparisons



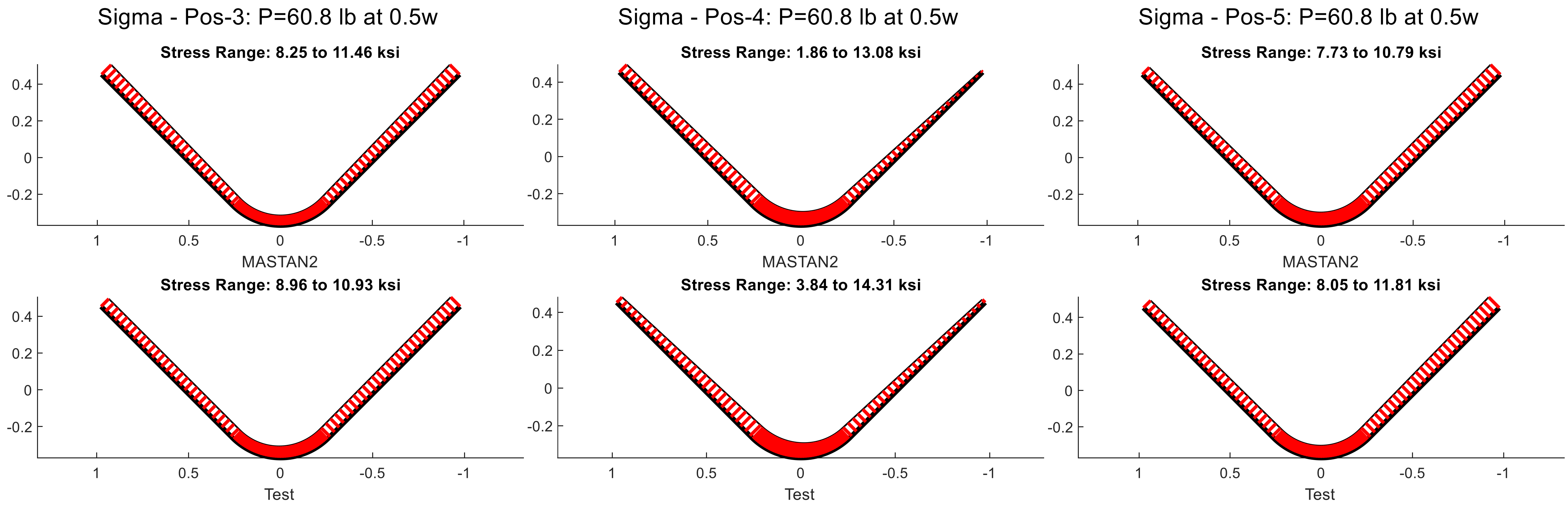
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Stress Comparisons



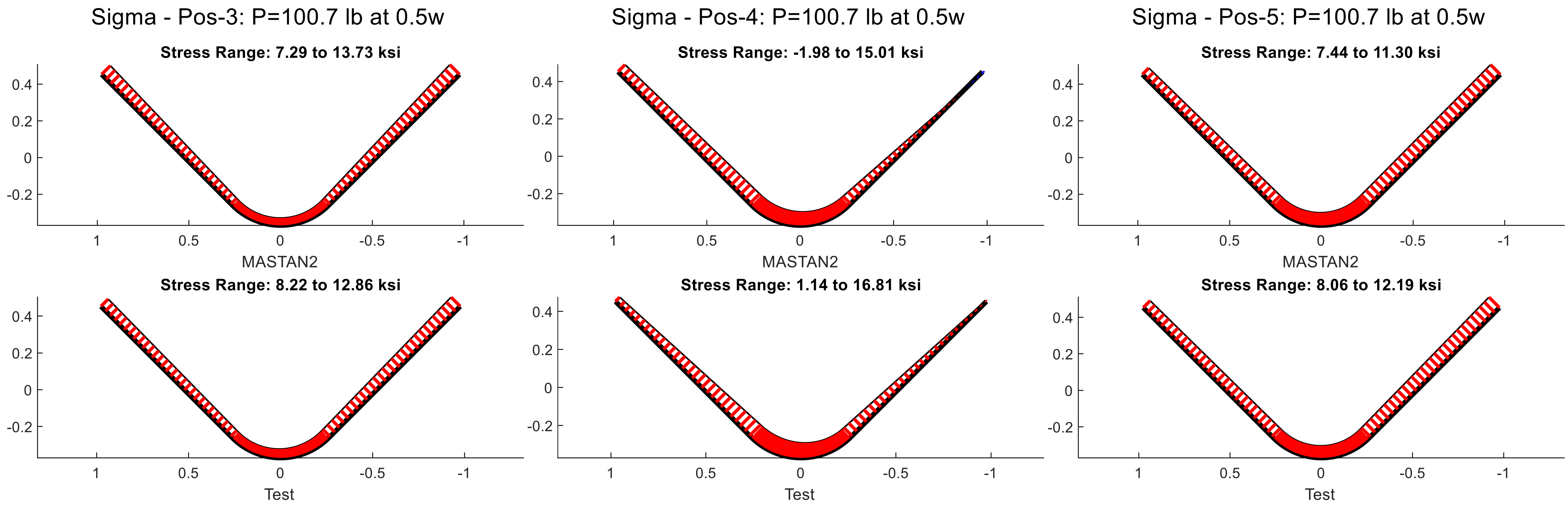
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Stress Comparisons



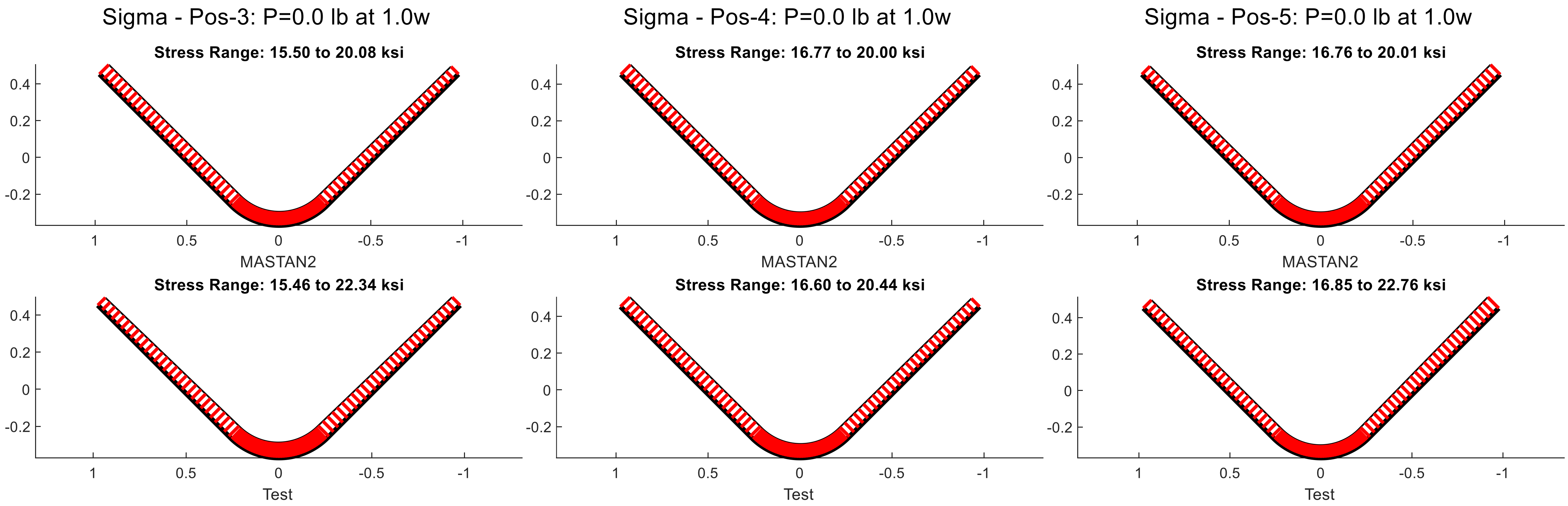
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Stress Comparisons

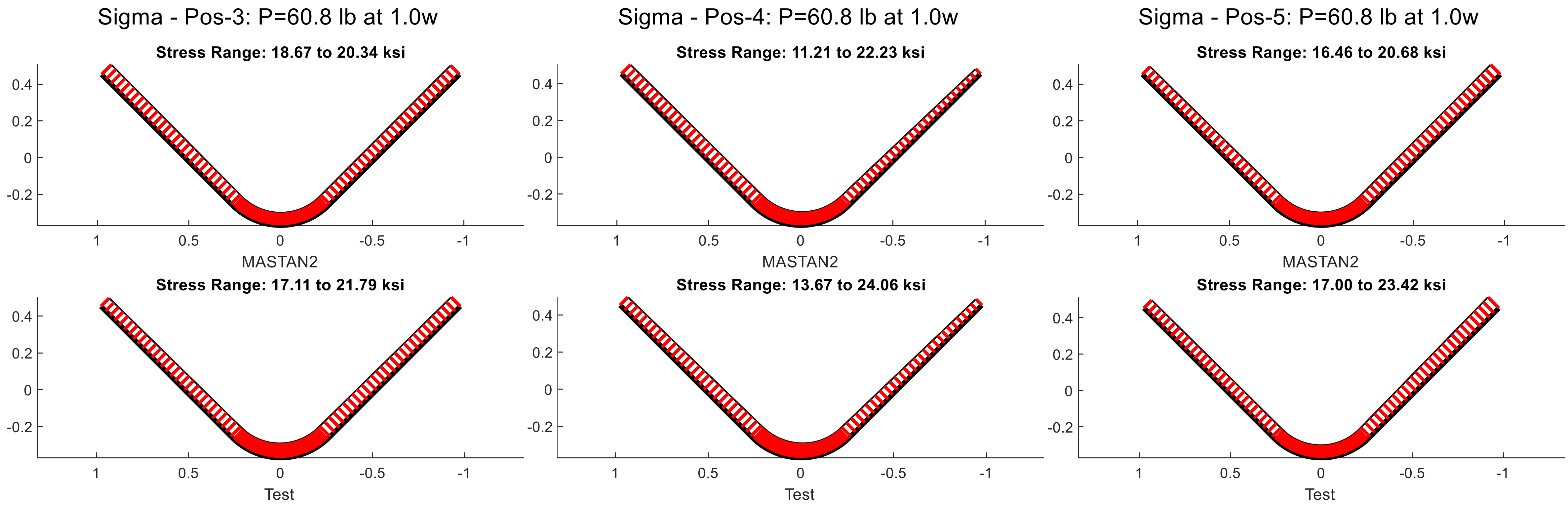


Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

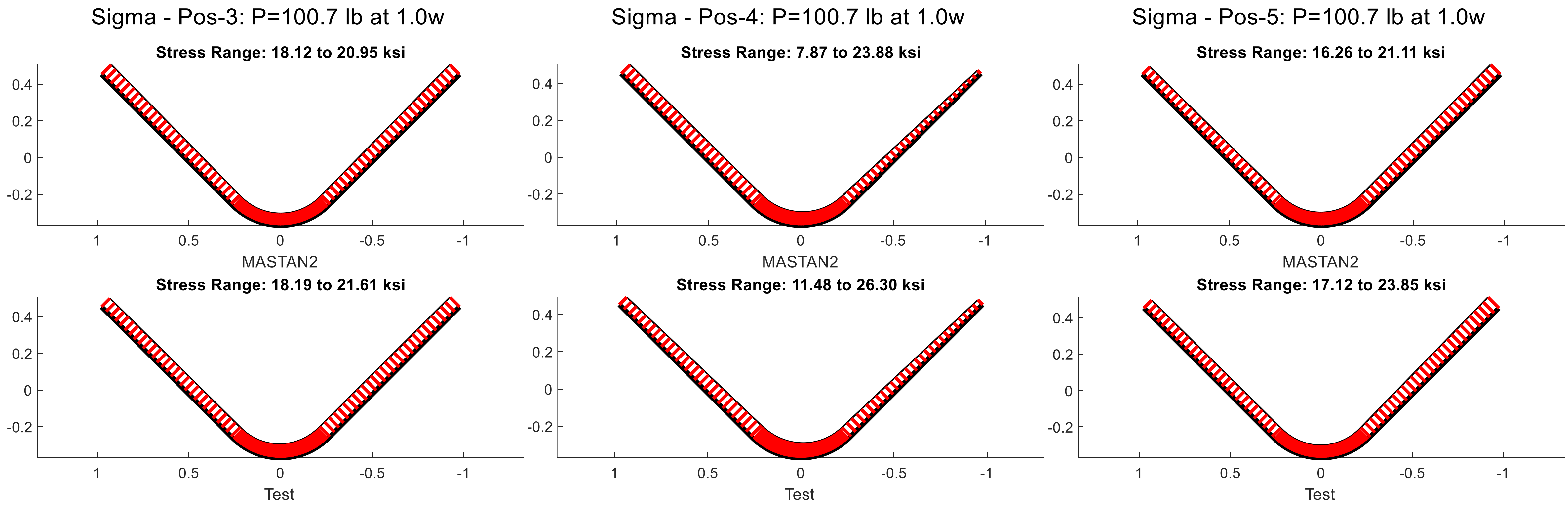
Stress Comparisons



Stress Comparisons

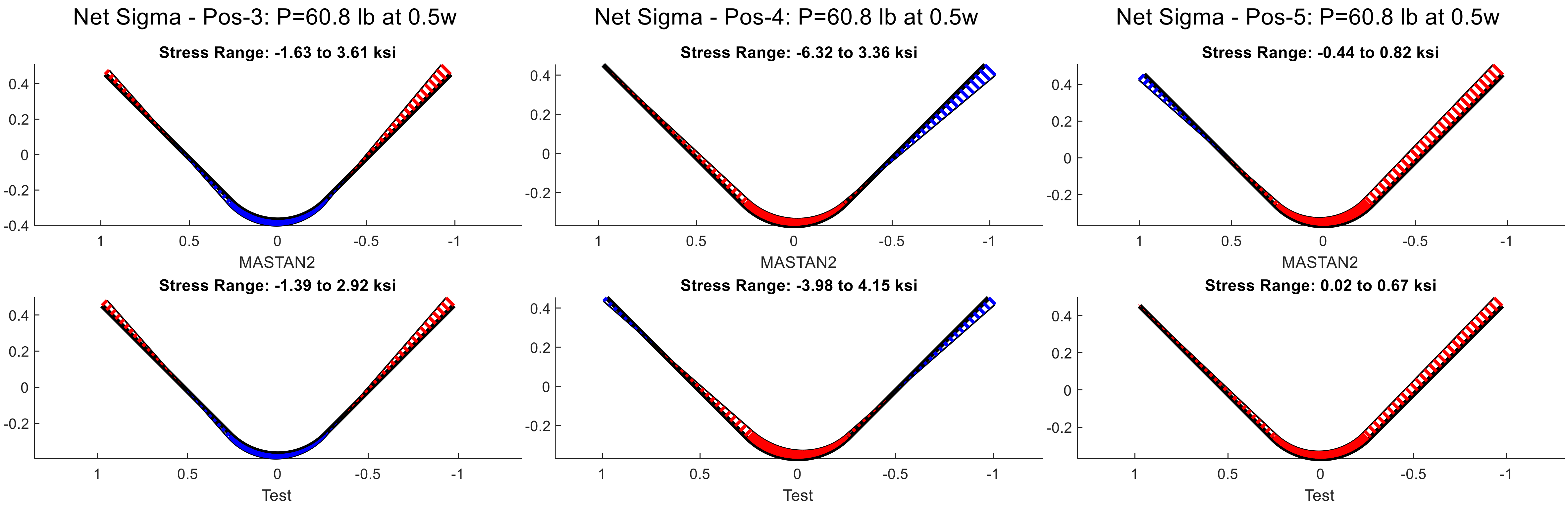


Stress Comparisons

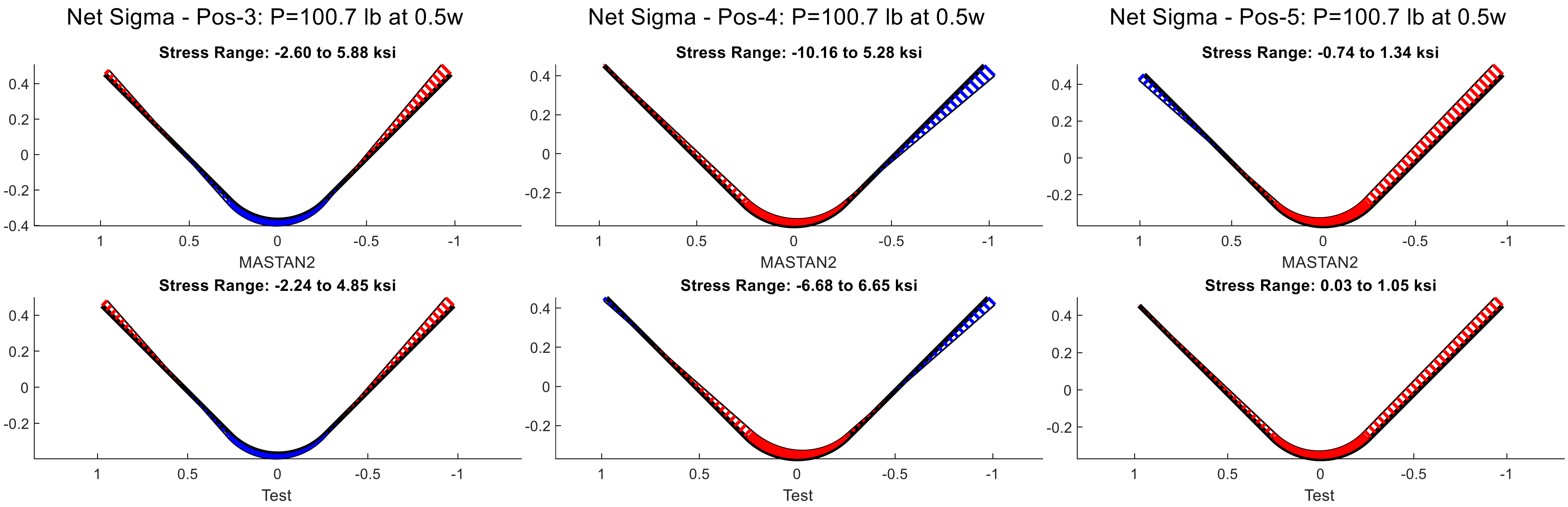


Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

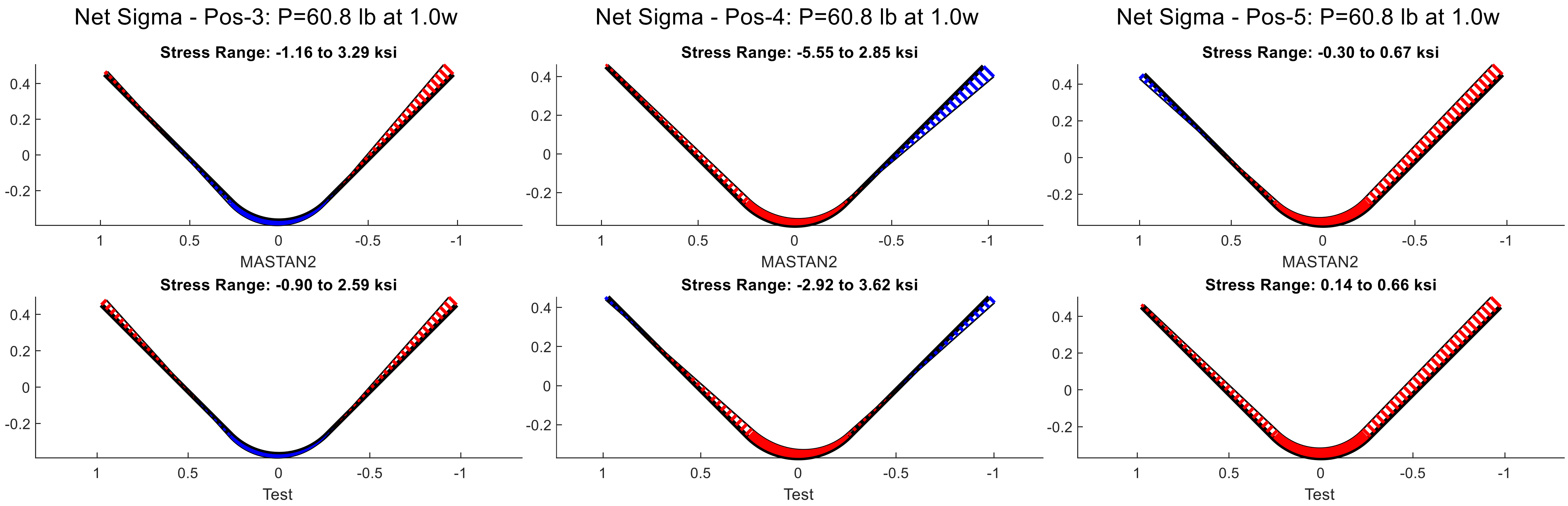
Stress Comparisons



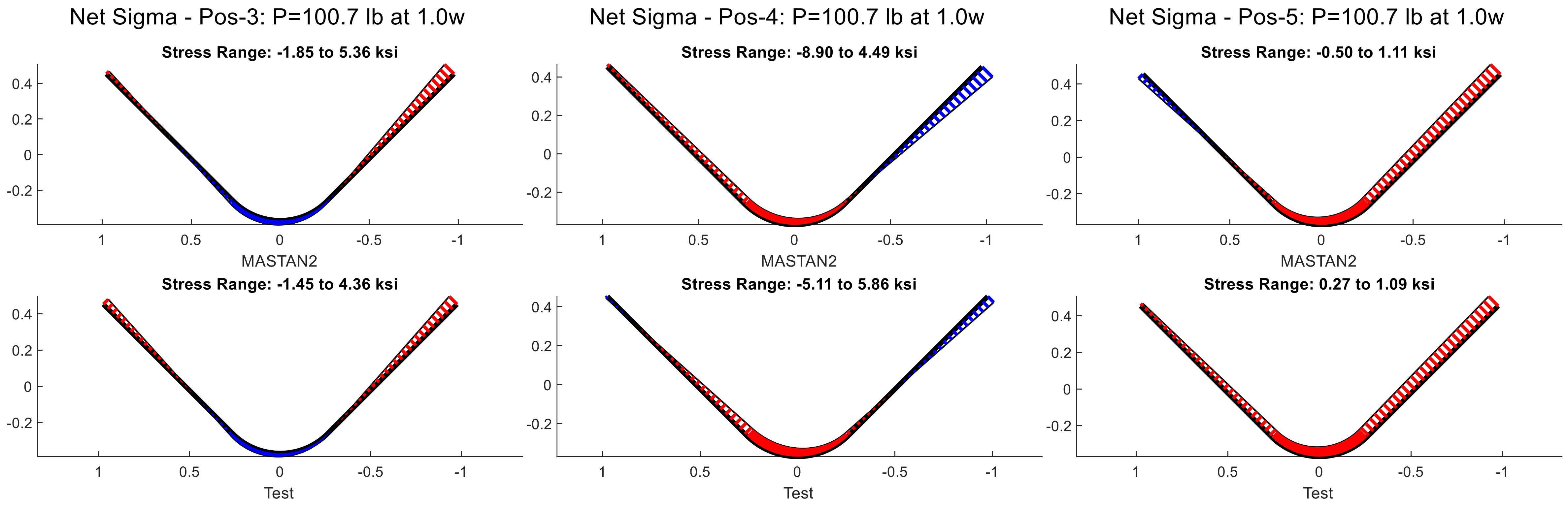
Stress Comparisons



Stress Comparisons



Stress Comparisons



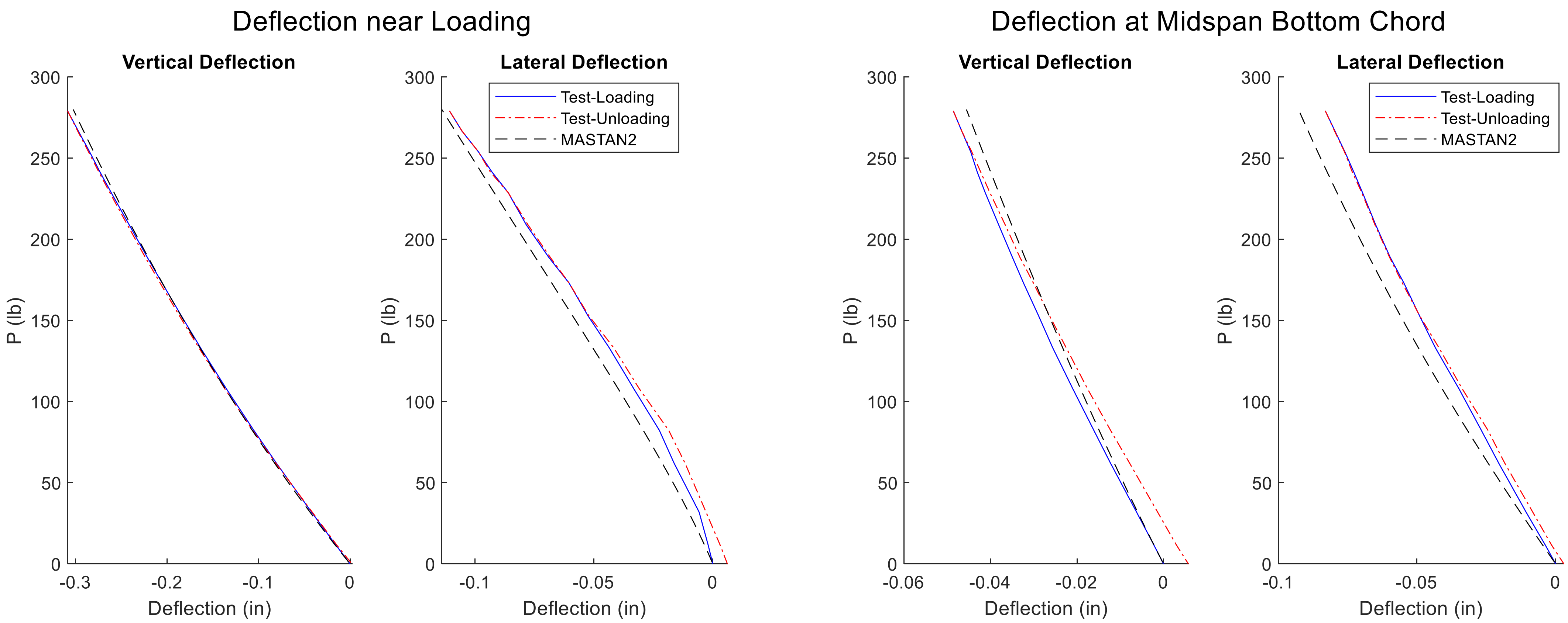
Result Diagrams – Scenario 2

- This section is all diagrams from Scenario 2.

- Force plots are shown with both the test results considering bimoment from warping and from ignoring the effects of warping.

- The stress plots are shown only with the case considering bimoment since at the locations checked the values were small.
  - Stress diagrams have MASTAN2 result on top and Test result on bottom for a given loading.
  - Diagrams are scaled in each vertical pair, but not across remaining calculations.
  - Red is tension (+) and blue is compression (-)

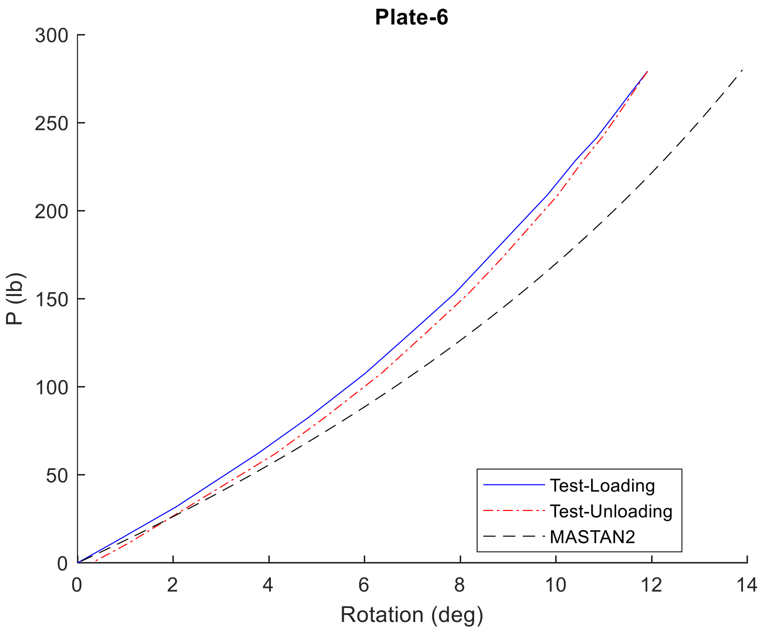
Measured Global Displacements



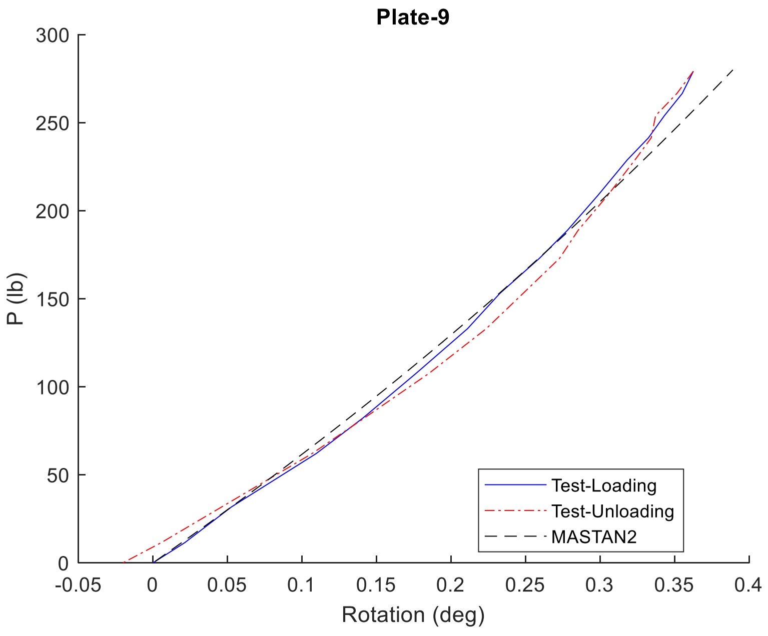
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Measured Bottom Chord Twist

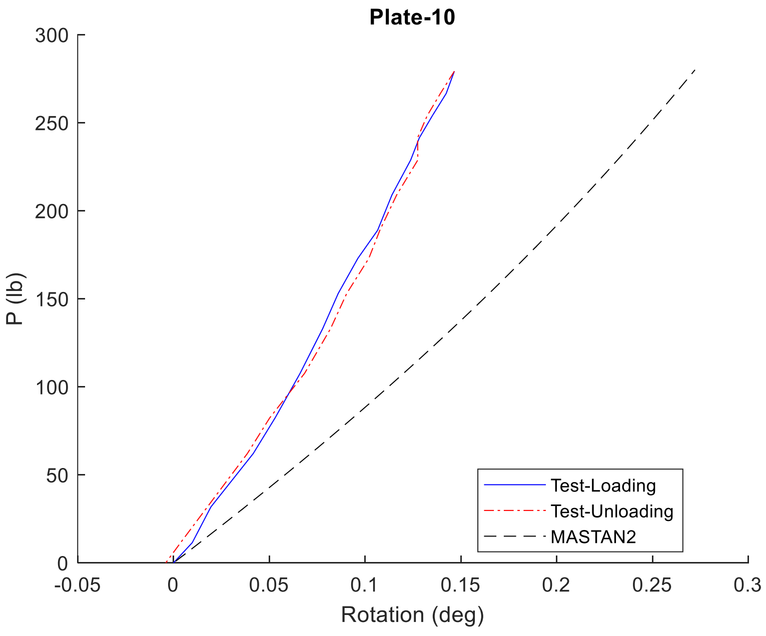
Loaded Member



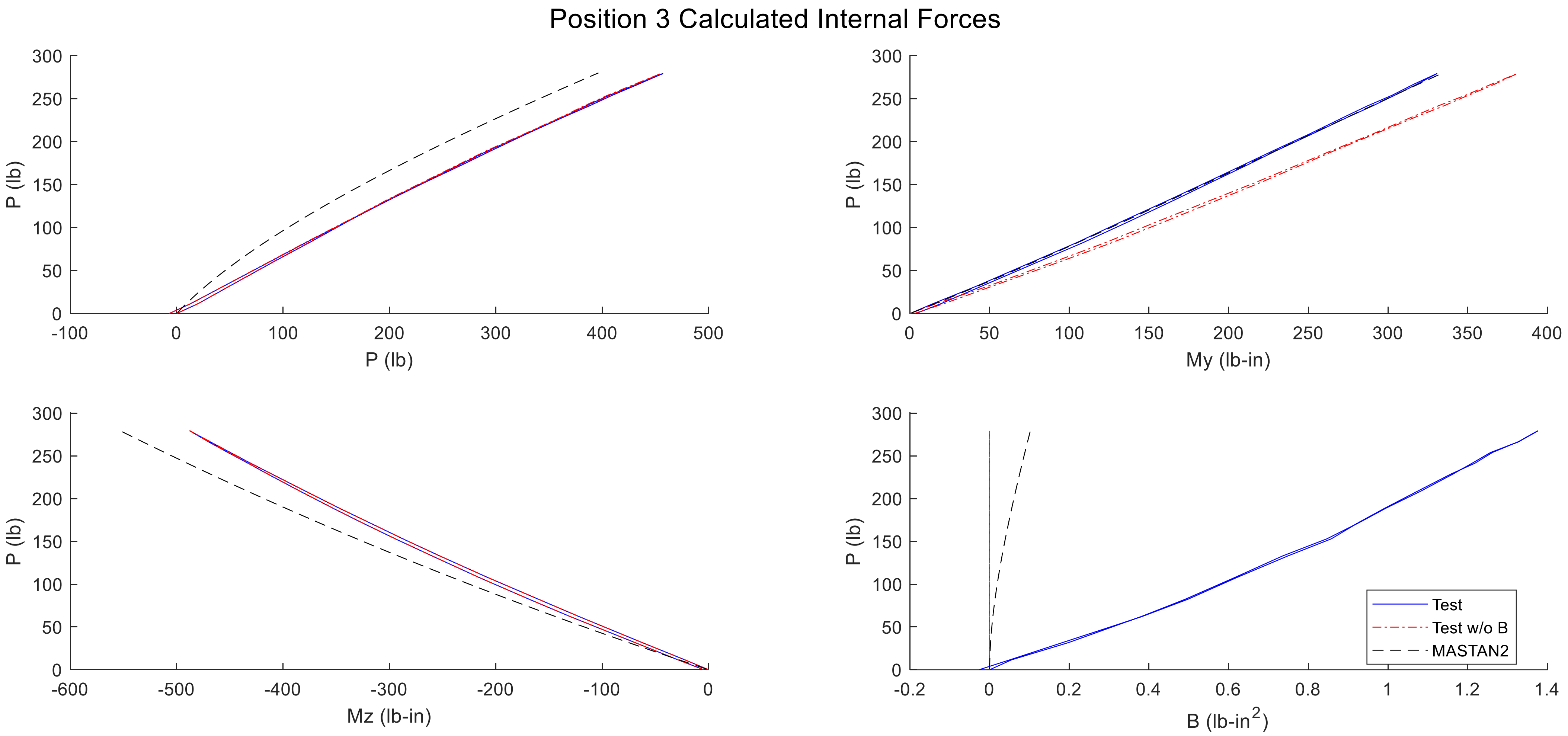
Unloaded Member



Loaded Member

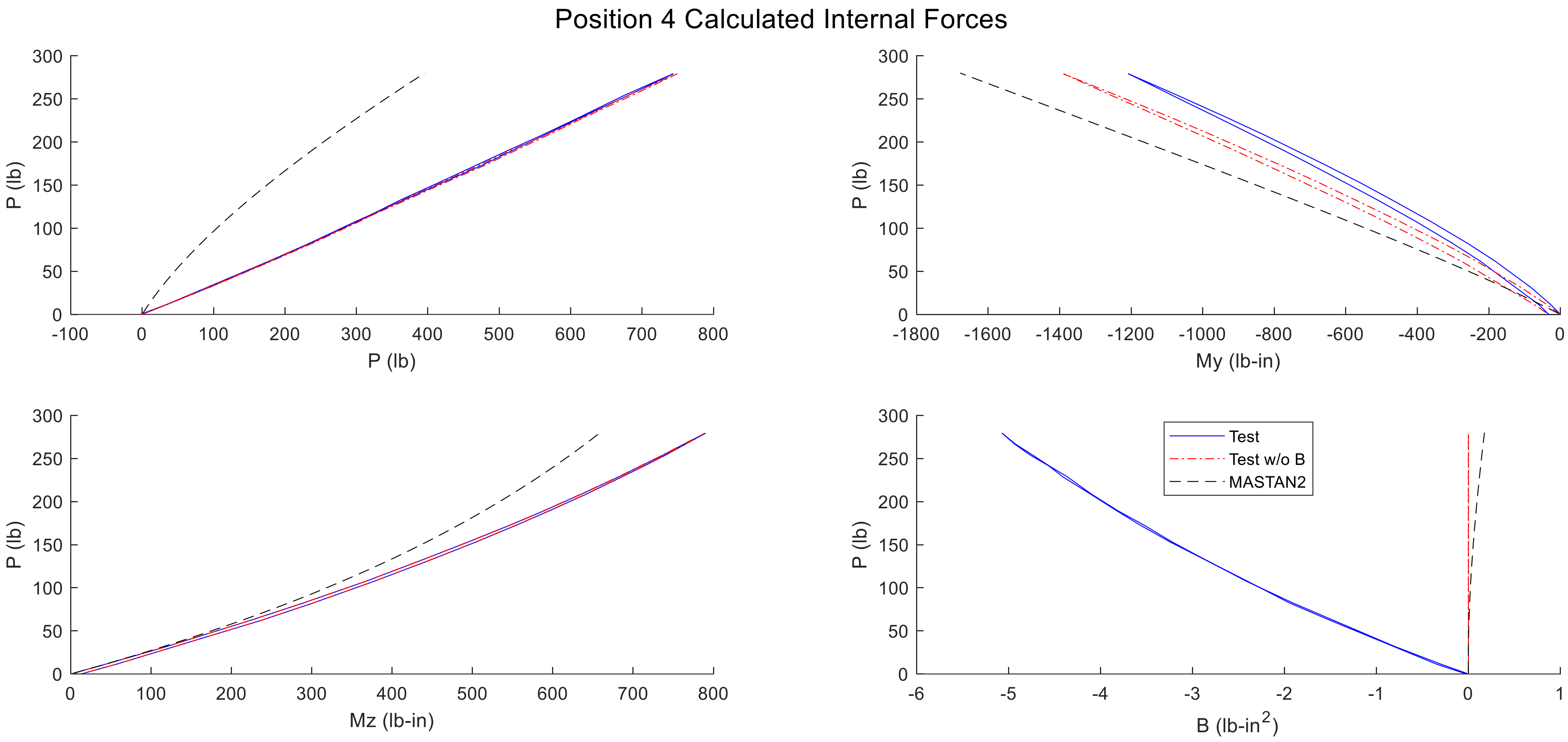


Force Comparisons

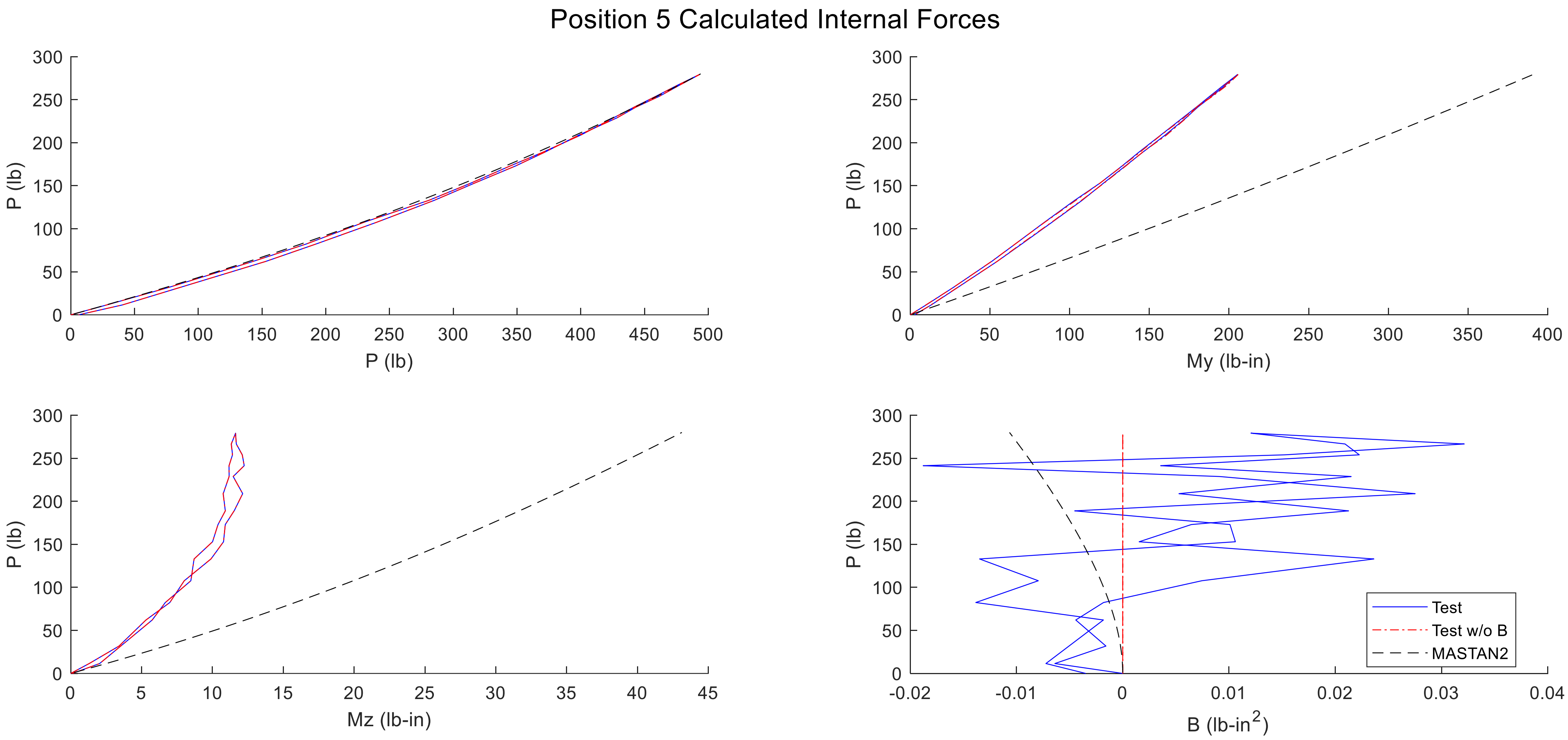


Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Force Comparisons

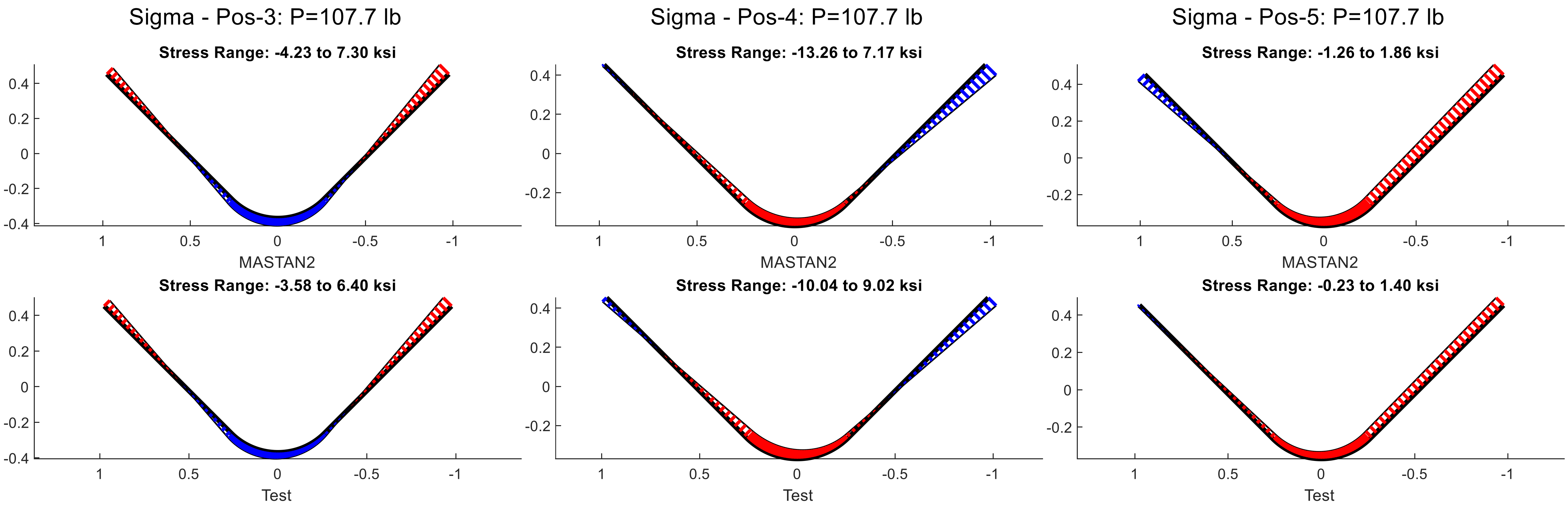


Force Comparisons



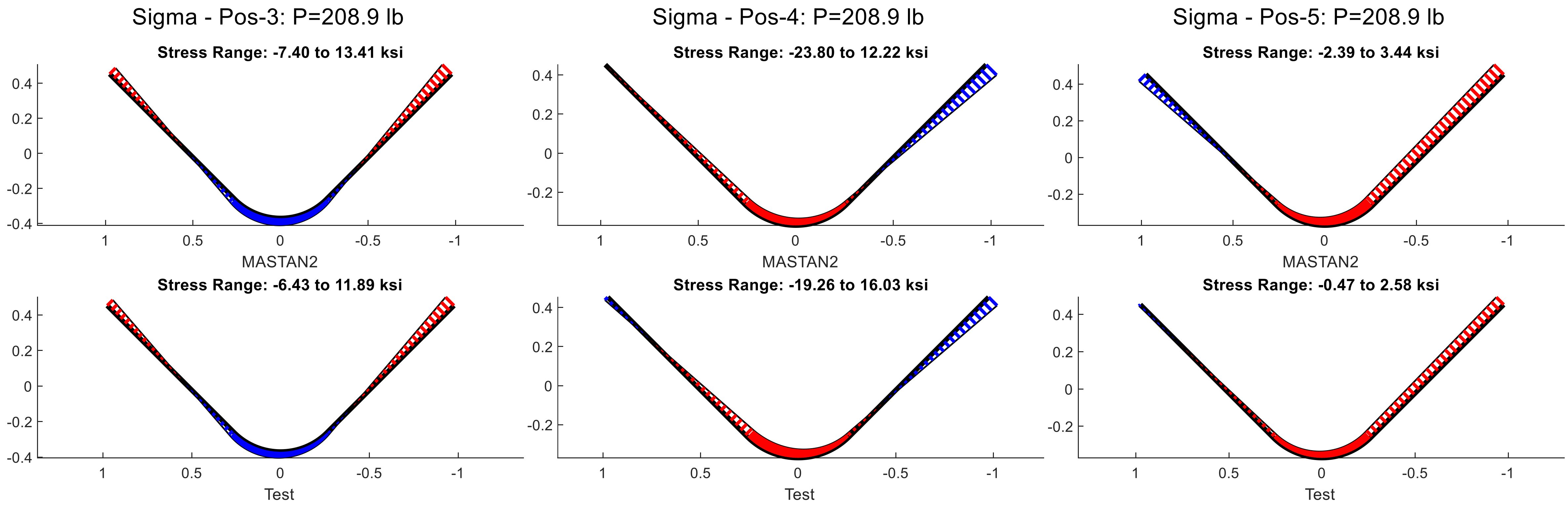
Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Stress Comparisons

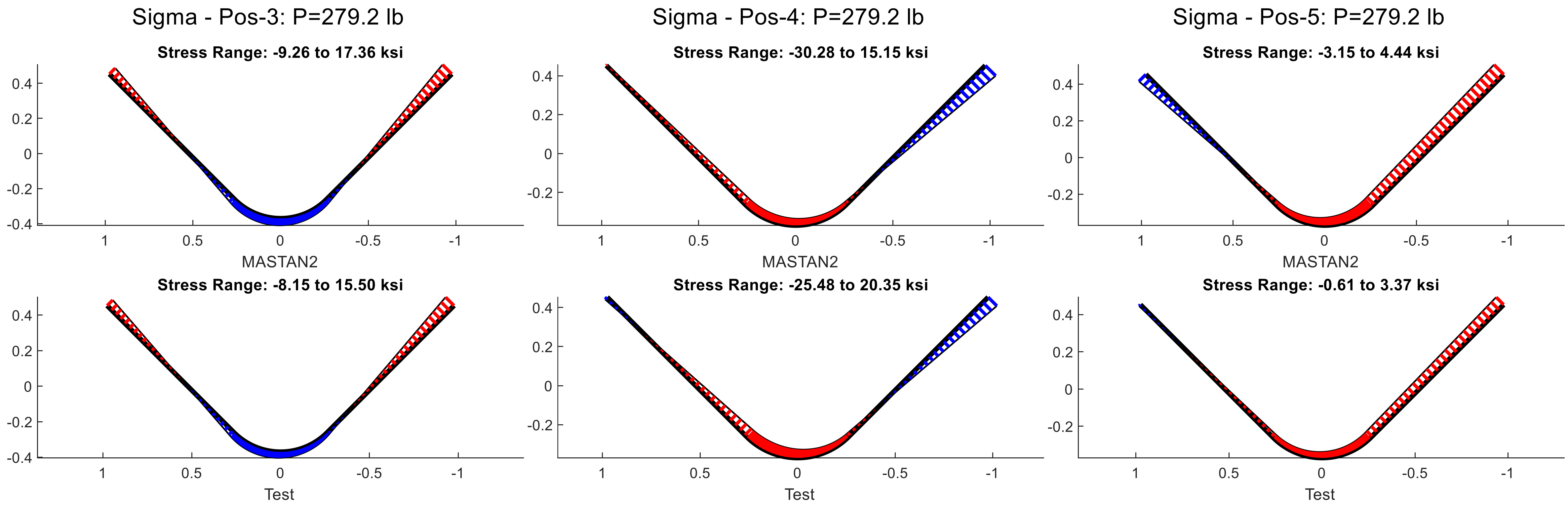


Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Stress Comparisons



Stress Comparisons



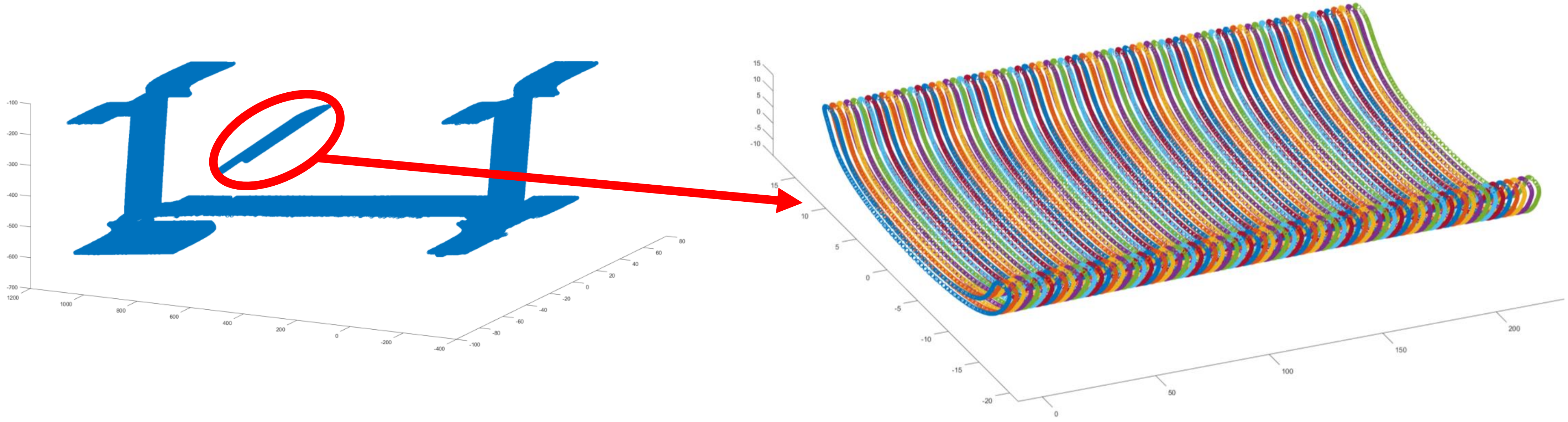
Extra Information

- While attempting to verify the results with the webs, it was noted that all the cross sections were not perfectly formed.
- One attempt to obtain better results required accounting for this altered geometry.
- The geometry was measured using the Artec Leo Scanner.
- A 3-D point cloud of the surface was created.

Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

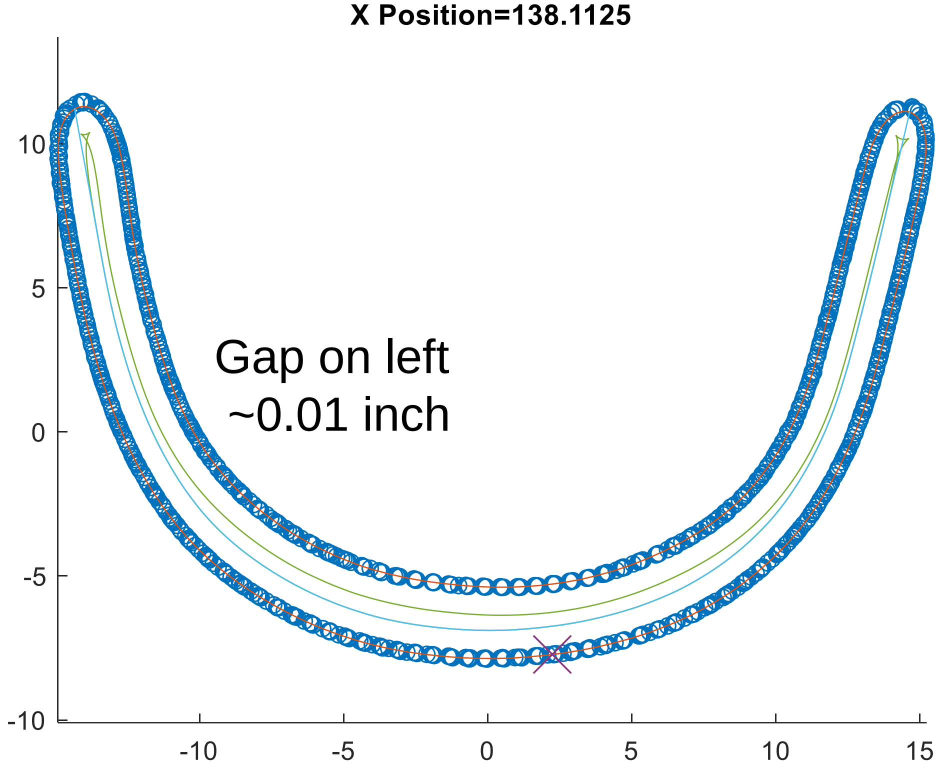
Extra Information

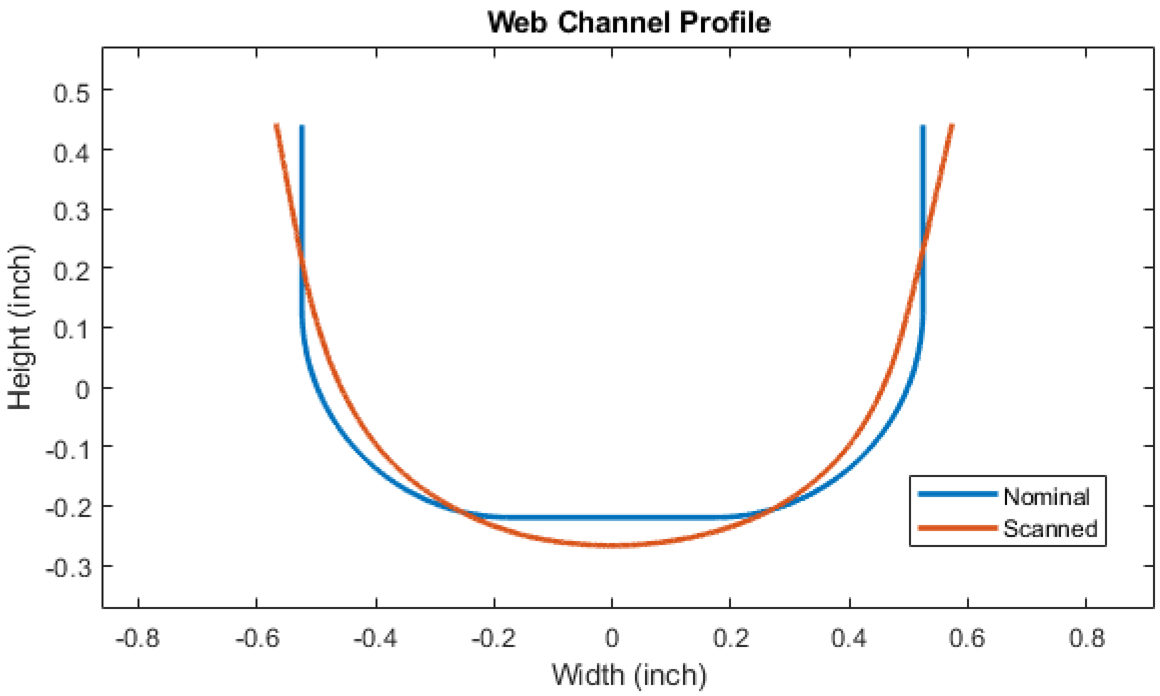
- The 3-D point cloud mesh could be segmented into separate chunks of information that could be used to create a profile.



Extra Information

- Each profile is of the exterior and a calculation of the interior centerline was created.

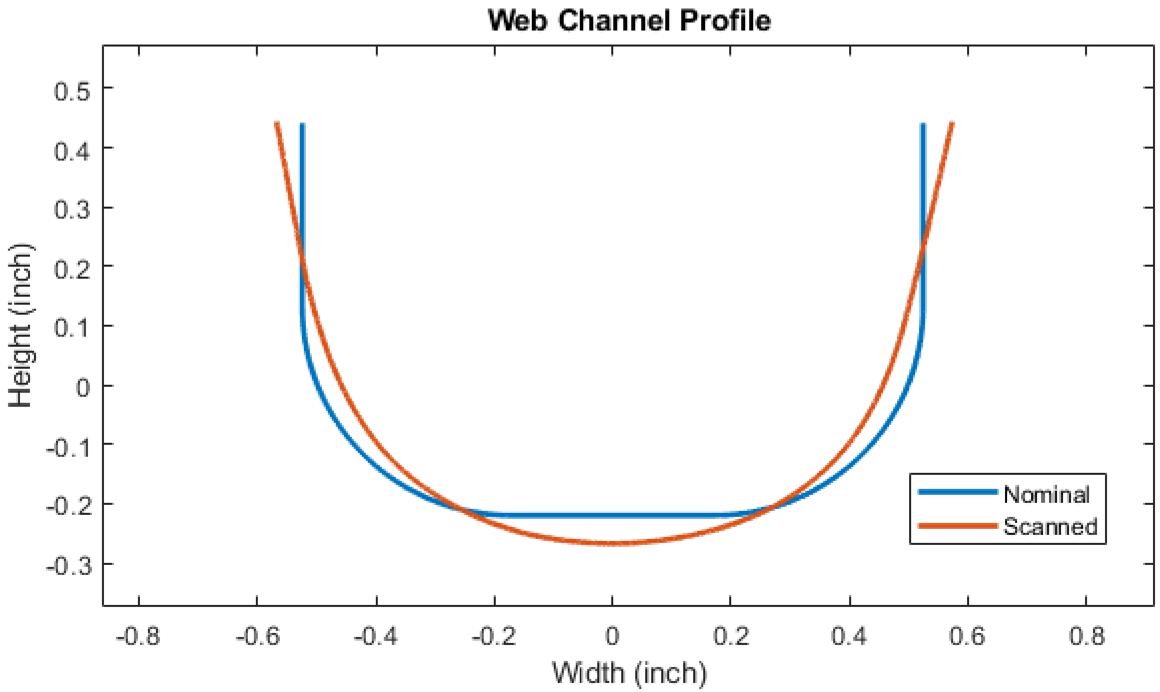




Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Extra Information

- The variation in the profile was found to have impact on the section properties; however, as mentioned earlier, it is not the controlling issue that was observed.



	Generic Section	Measured Section	Error Measured to Generic
A	0.1587	0.1567	-1.26%
J	3.11E-04	3.07E-04	-1.27%
I_{zz}	7.27E-03	7.90E-03	8.73%
I_{yy}	2.76E-02	2.65E-02	-4.19%
I_{yz}	0	0	-
y_s	-4.72E-01	-4.90E-01	3.82%
z_s	0	2.44E-04	-
C_w	7.62E-04	3.94E-04	-48.27%

Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Summary

Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

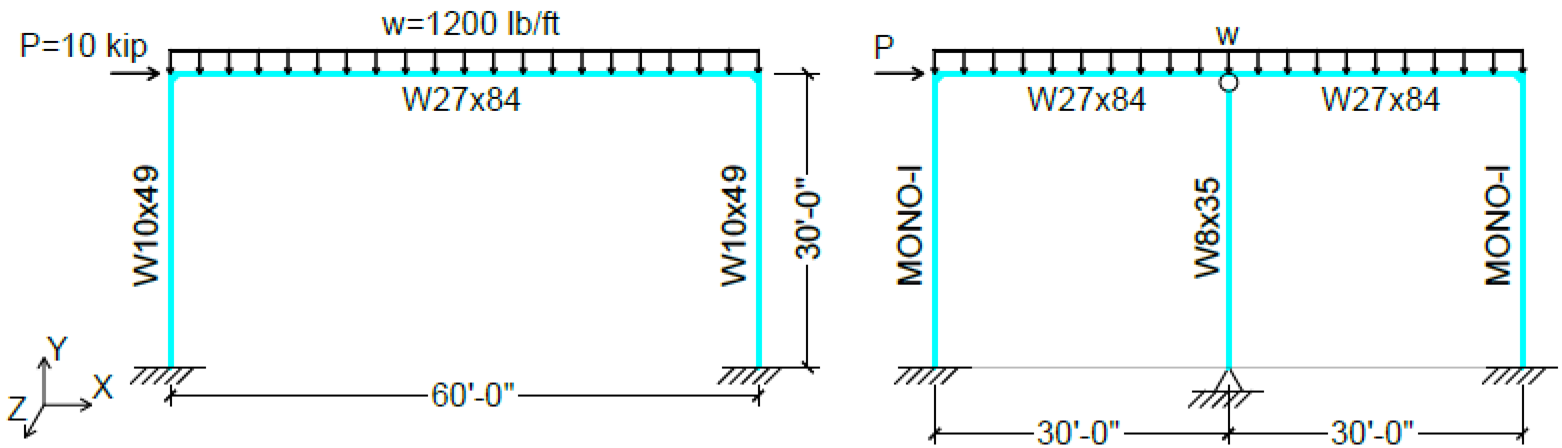
Summary

- MASTAN2 was able to determine reasonable stress distributions matching the experiment of a joist subjected to an eccentric hanging load.

- In the calculated results from Scenario 1, there was a consistent underprediction of stresses in the initial condition prior to the application of the hanging load. But looking at the net change in Scenario 1 or the total loading in Scenario 2, MASTAN2 was consistently finding conservative values for stresses while matching the desired distribution of stress for the effect of the torsion applied to the chord.

- The largest variation was observed on the leg of the angle where the hanging load was applied 2” away from the strain gauge measurements.

Tutorial for MASTAN2 v5.1 - Introductory Frame



Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Bucknell UNIVERSITY

NEW MILLENNIUM BUILDING SYSTEMS

American Iron and Steel Institute

STEEL JOIST INSTITUTE SJI

STEEL DECK INSTITUTE sdi ®

MASTAN2

Credits

Published 2021

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.

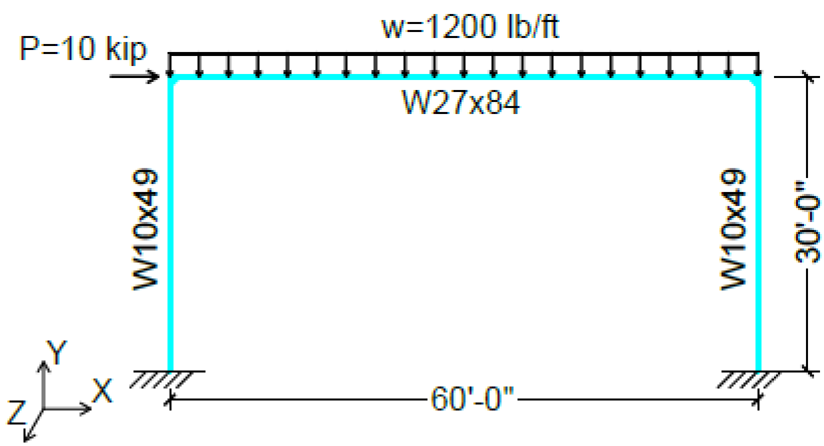
Section 1: Overview

Overview

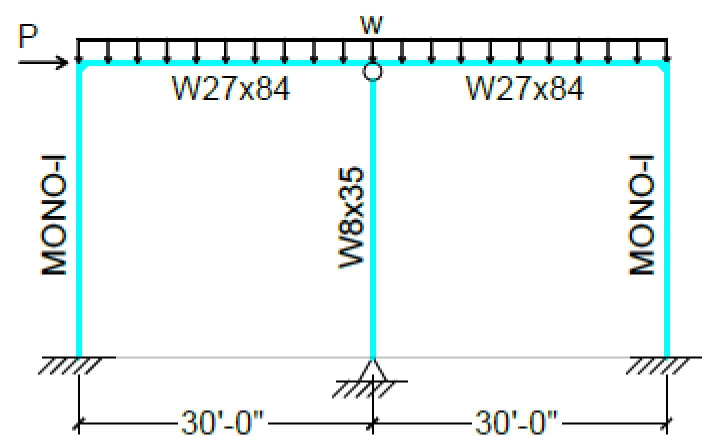
This tutorial provides step-by-step guidance for the sample frame structure. Enough details are provided that the example model with non-doubly symmetric sections can be completed following the instructions here. Not every feature available in MASTAN2 will be mentioned nor utilized in this tutorial.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

Problem Overview

This tutorial will start with the simple one-bay frame shown on the left. This model will then be altered to the two-bay frame shown on the right include non-doubly symmetric sections. Further details of each model will be provided in the corresponding section.



Starting Frame



Final Frame

Section 2: Getting Started

MASTAN2 General Information

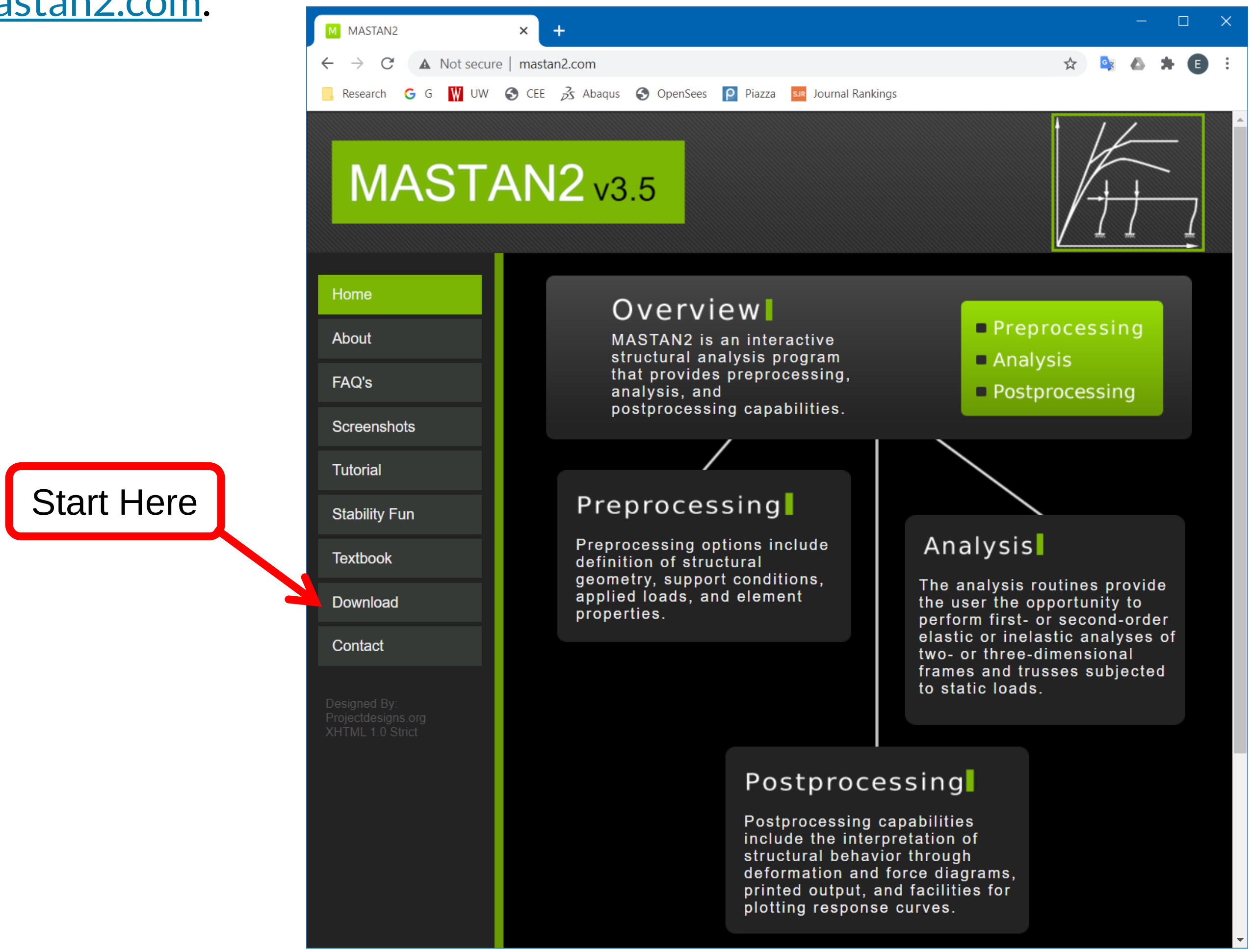
MASTAN2 is an interactive graphics program that provides preprocessing, analysis, and postprocessing capabilities. Preprocessing options include definition of structural geometry, support conditions, applied loads, and element properties. The analysis routines provide the user the opportunity to perform 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.

MATLAB is a registered trademark of The MathWorks, Inc., 3 Apple Hill Drive, Natick, MA 01760-2098.

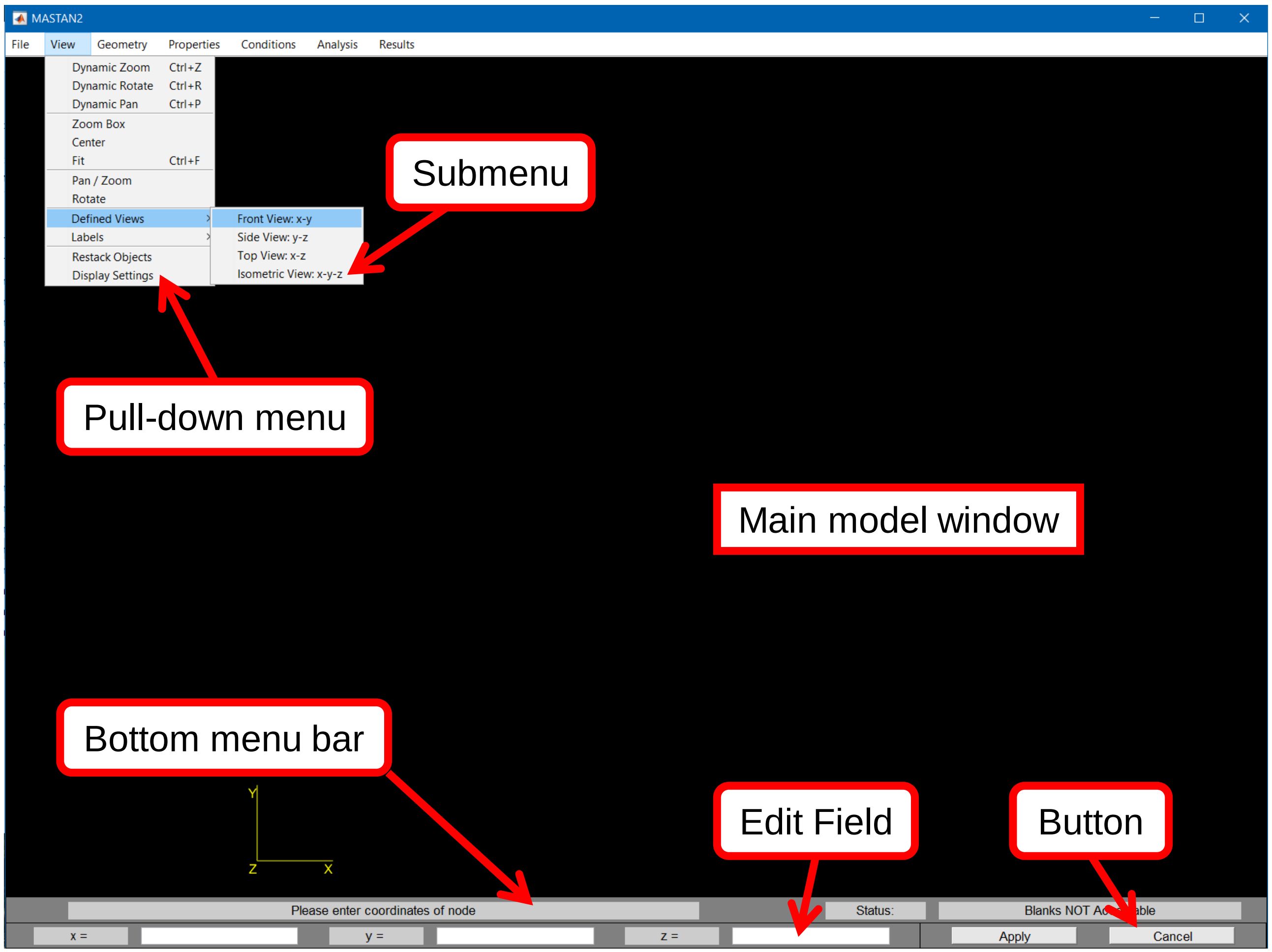
Launching MASTAN2

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

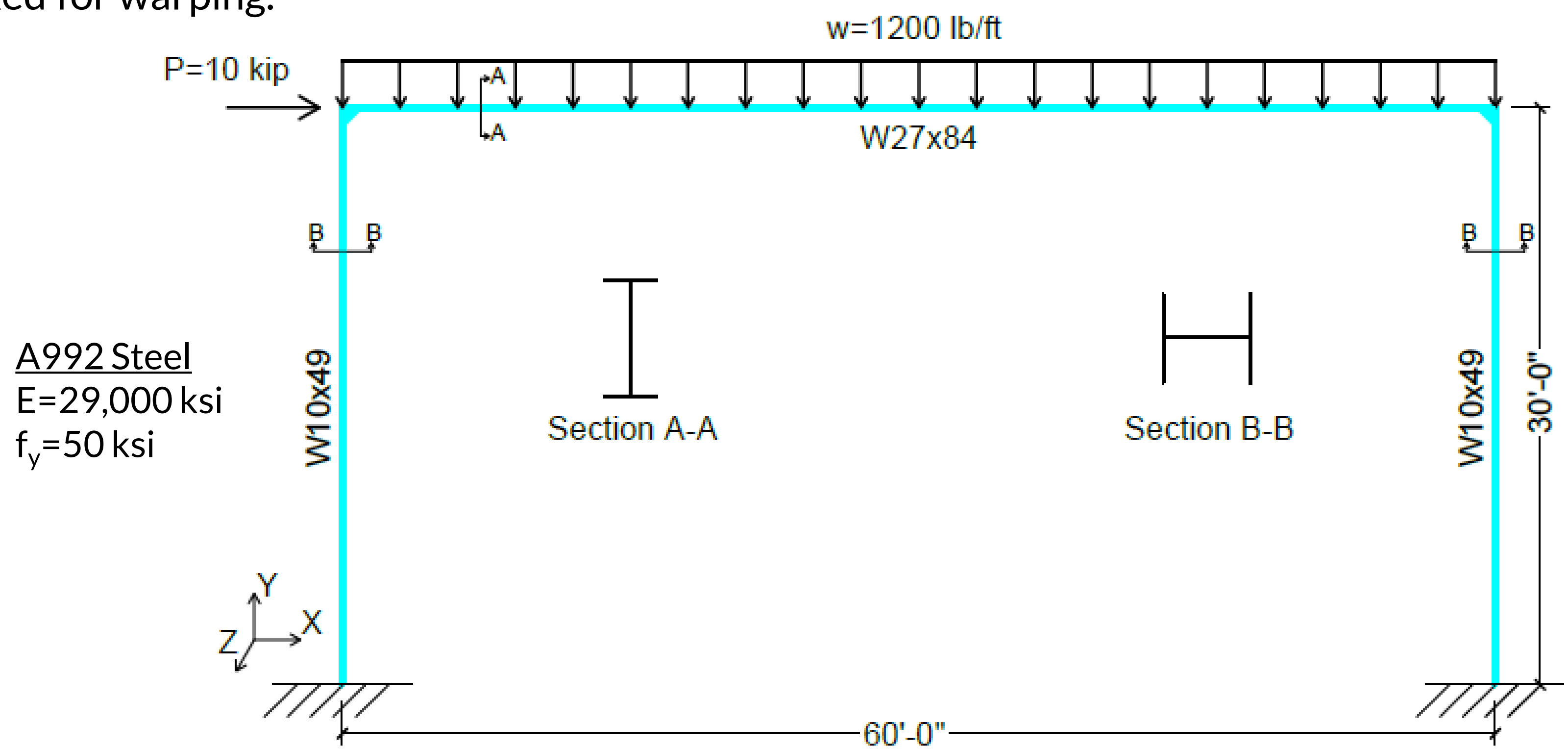
In order to minimize the learning time for MASTAN2, its graphical user interface (GUI) has been designed using a simple and consistent two menu approach. Using a pull-down menu at the top of the GUI, a command is selected. Parameters are then defined in the bottom menu bar and the command is executed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



Section 3: 2-D Frame Analysis

Problem Description - Figure

The frame is constructed of A992 steel with the properties indicated. The frame is also supported out of plane in the Z direction at the ends and middle of the beam. The connections are assumed to be fixed for warping.



MASTAN2 does not assume any unit system. Models in MASTAN2 require the use of any consistent set of units. This tutorial will use kip and inch.

A few steps completed as part of this segment of the tutorial are not specifically required for a 2-D analysis. Comments are provided to identify them.

Geometry Definition

1) Start with a new, empty model.

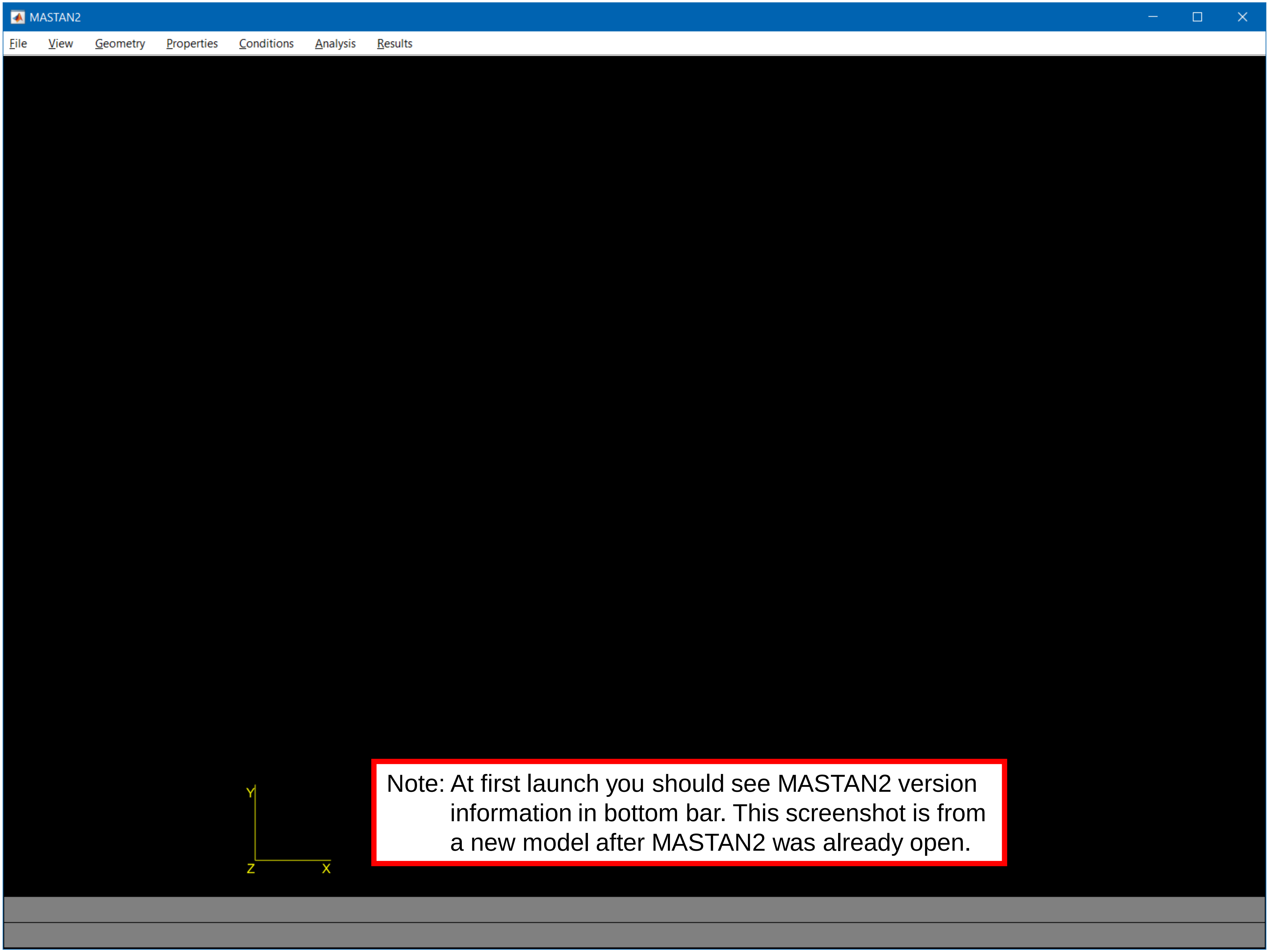
2) From the **Geometry** menu select **Define Frame**.

3) At the bottom menu bar, click the pop-up menu to the left of **bays @** and change **0** to **1**. Click in the edit box to the right of **bays @** and change **0** to **720**.

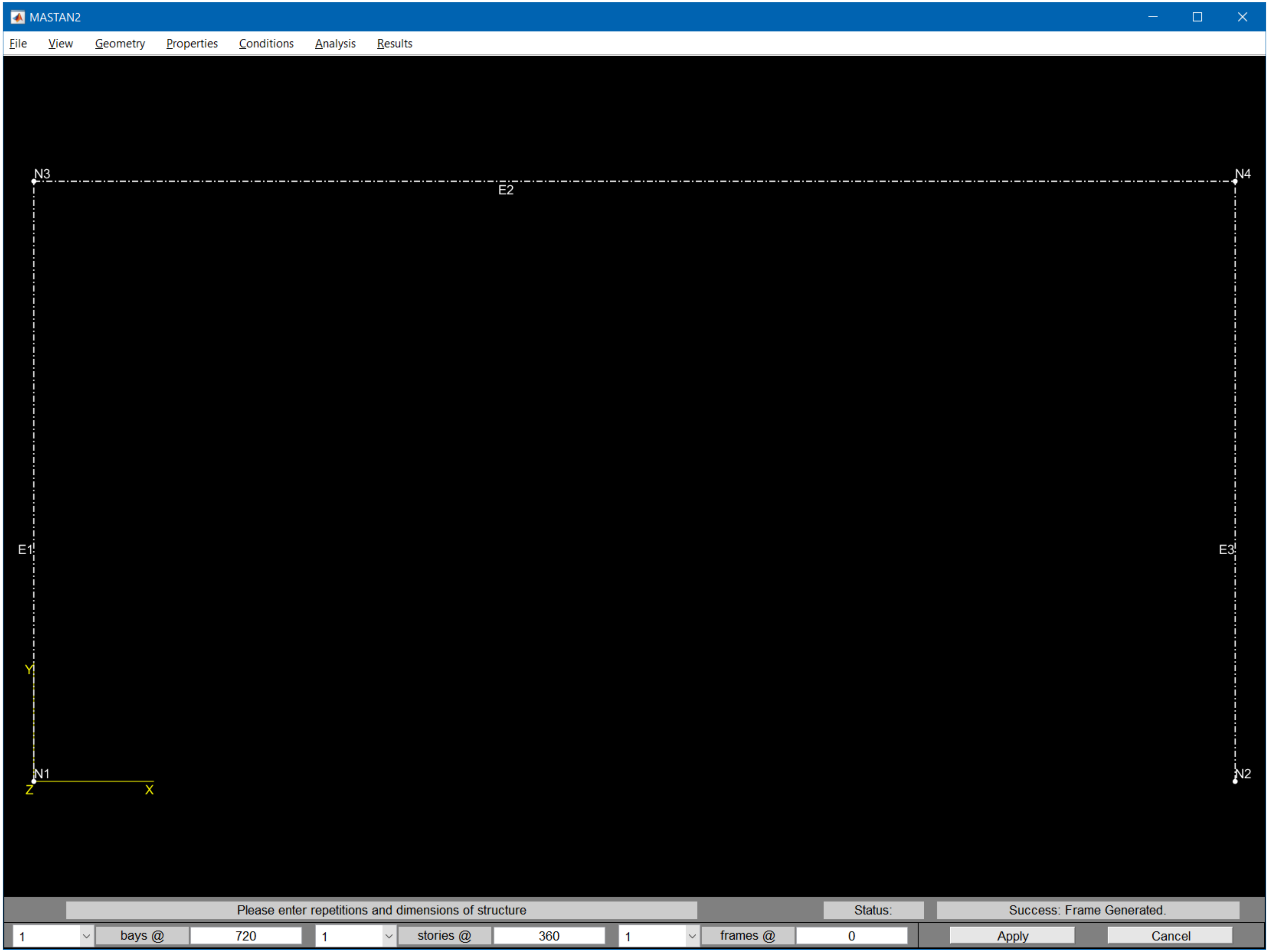
4) Click the pop-up menu to the left of **stories @** and change **0** to **1**. Click in the edit box to the right of **stories @** and change **0** to **360**.

5) Click on the **Apply** Button. A one-bay single story frame is now defined.

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.


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
X
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


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
N3
N4
E2
E1
E3
Y
N1
N2
Z
X
Please enter repetitions and dimensions of structure
Status:
Success: Frame Generated.
1
bays @
720
1
stories @
360
1
frames @
0
Apply
Cancel

Element Modification

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

2) Create the list of elements by clicking on the horizontal element.

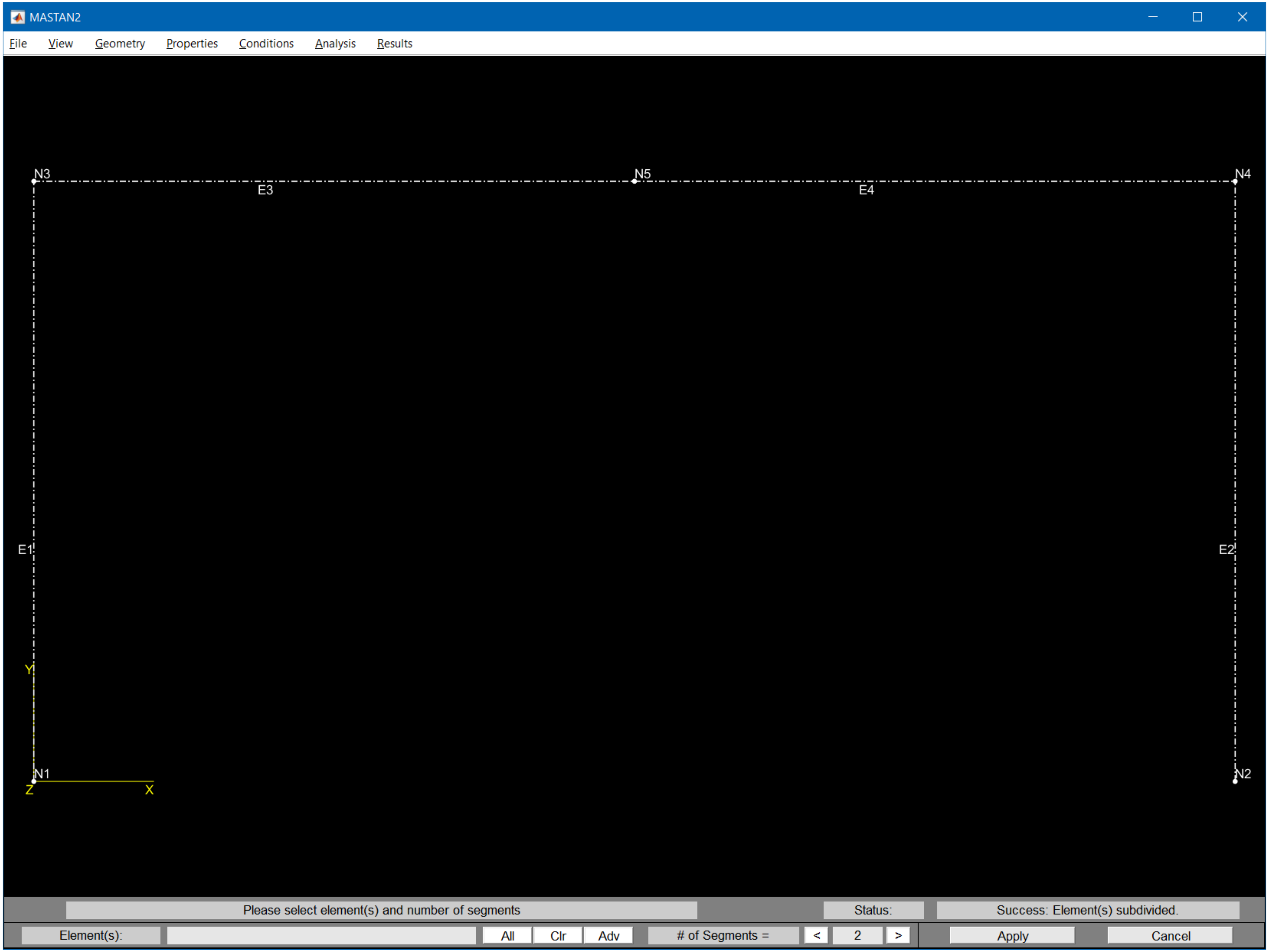
3) Since the number of segments is already set at **2**, click on the **Apply** button.

4) Create a new list of all elements by clicking the **All** button.

5) Click the **>** button to the right of **# of Segments =** to increase **2** to **8**.

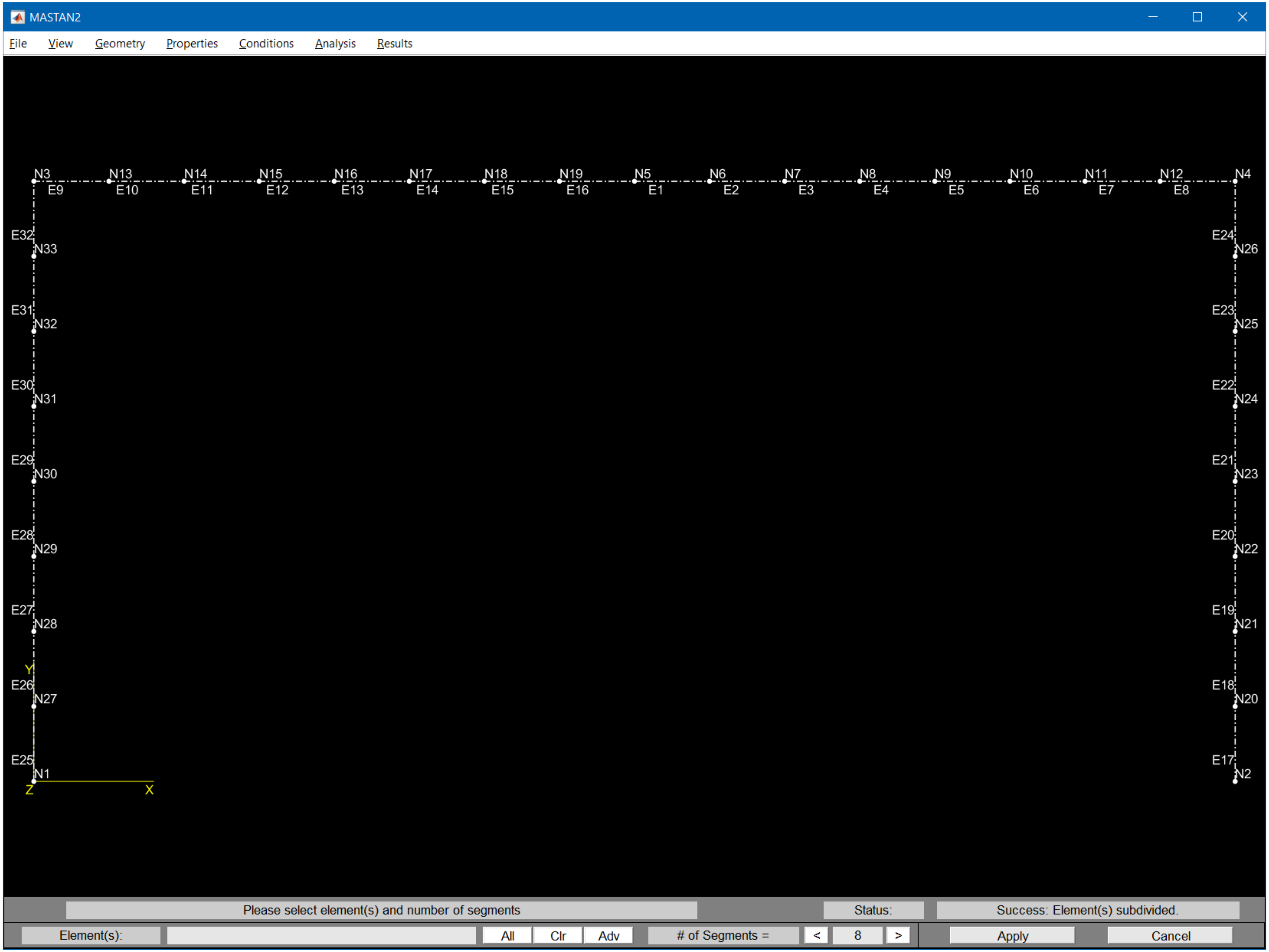
6) Click on the **Apply** button.

MASTAN2



MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
N3
N5
N4
E3
E4
E1
E2
Y
N1
Z
X
N2
Please select element(s) and number of segments
Status:
Success: Element(s) subdivided.
Element(s):
All
Clr
Adv
# of Segments =
<
2
>
Apply
Cancel

MASTAN2


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
N3 N13 N14 N15 N16 N17 N18 N19 N5 N6 N7 N8 N9 N10 N11 N12 N4
E9 E10 E11 E12 E13 E14 E15 E16 E1 E2 E3 E4 E5 E6 E7 E8
E32 N33
E31 N32
E30 N31
E29 N30
E28 N29
E27 N28
E26 N27
E25 N1
E24 N26
E23 N25
E22 N24
E21 N23
E20 N22
E19 N21
E18 N20
E17 N2
Y
Z
X
Please select element(s) and number of segments
Status:
Success: Element(s) subdivided.
Element(s):
All
Clr
Adv
# of Segments =
<
8
>
Apply
Cancel

Model Cleanup

These steps are not technically required; however, it will help makes it easier to find results in the model. Additionally, any reference to node or element number will be using this updated reference.

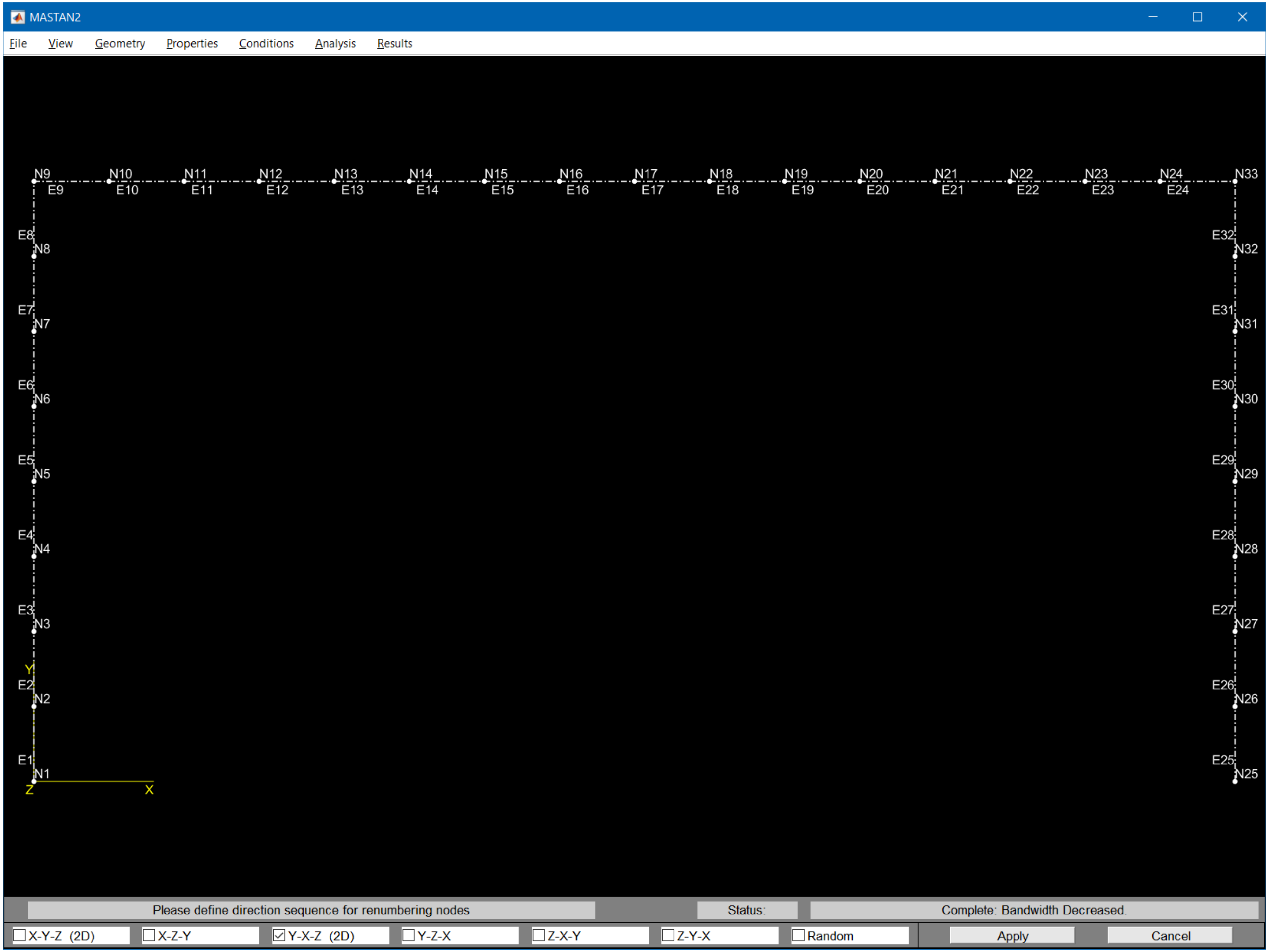
1) From the **Geometry** menu select **Renumber Elements**.

2) Click the checkbox to the left of **Y-X-Z (2D)**. Click on the **Apply** button.

3) From the **Geometry** menu select **Renumber Nodes**.

4) Click the checkbox to the left of **Y-X-Z (2D)**. Click on the **Apply** button.

MASTAN2


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N33
E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2 E1 N1
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26 E25 N25
Y
Z
X
Please define direction sequence for renumbering nodes
Status:
Complete: Bandwidth Decreased.
X-Y-Z (2D)
X-Z-Y
Y-X-Z (2D)
Y-Z-X
Z-X-Y
Z-Y-X
Random
Apply
Cancel

Section Properties - Creating

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

2) At the bottom menu bar, click on the **Database** button.

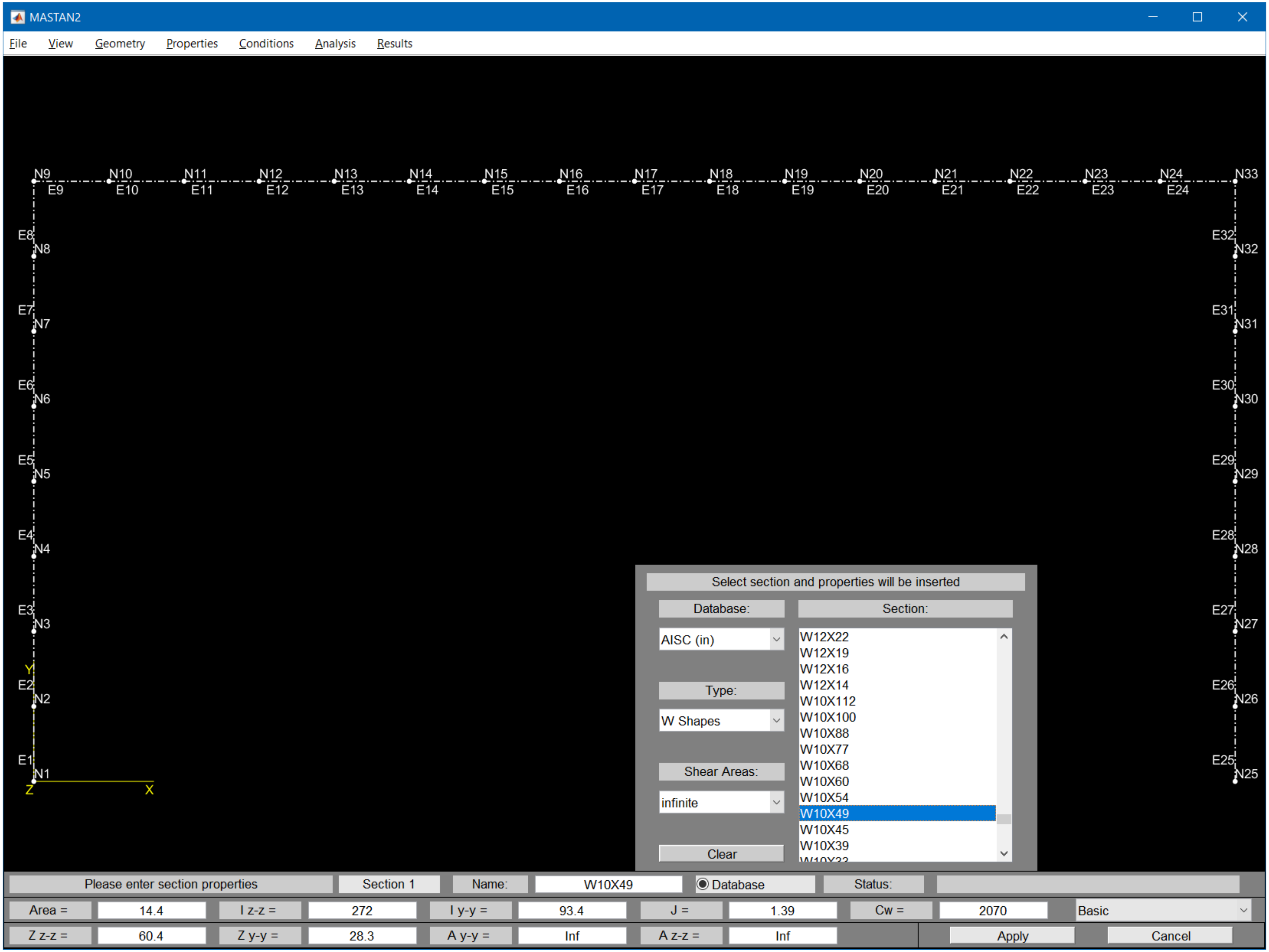
3) In the pop-up menu, scroll to find Section: **W10x49** and click on it.

4) Then click on the **Apply** button. Section 1 is now defined with the properties of W10x49.

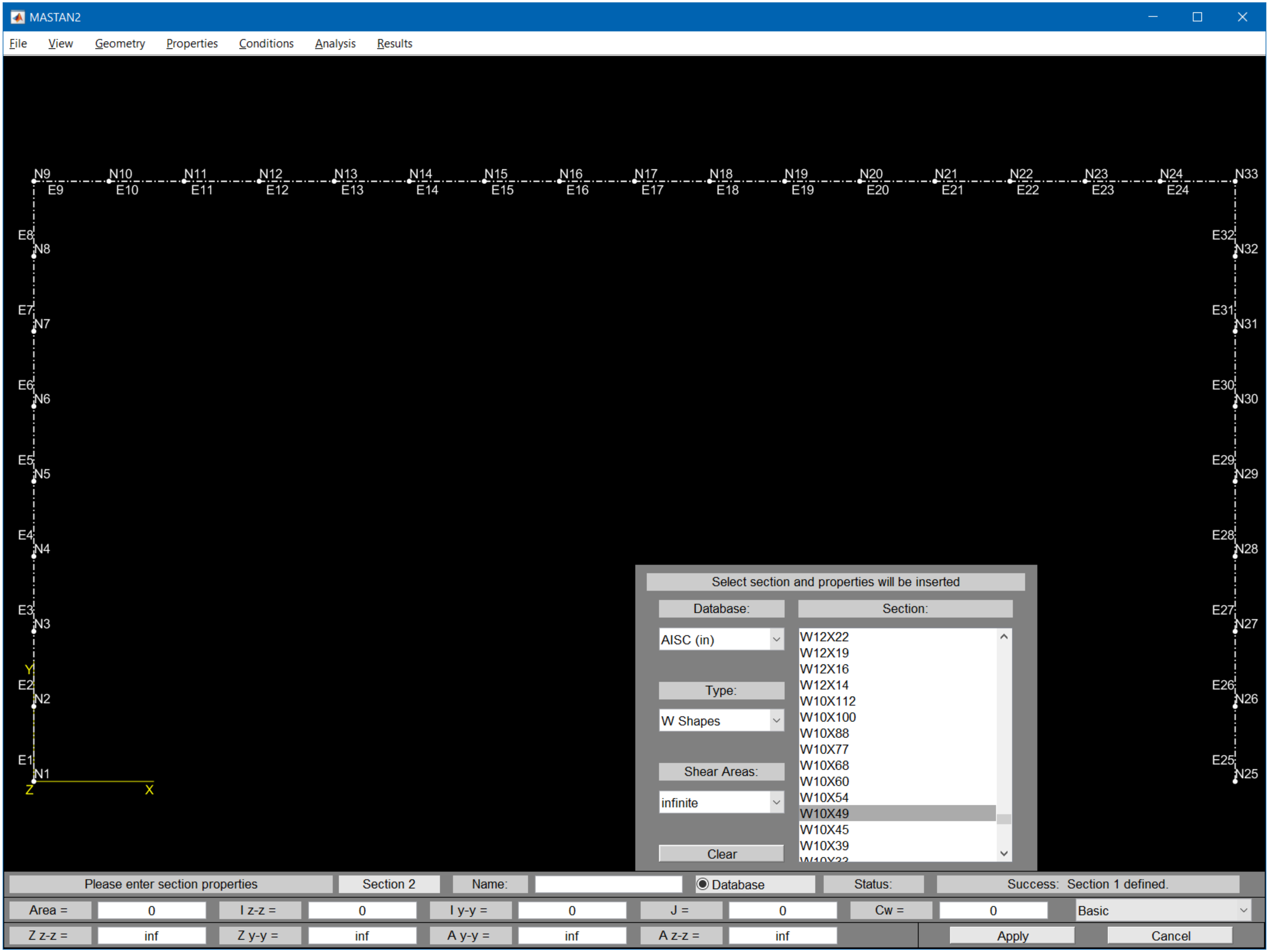
5) Repeat step 3 with Section: **W27x84**. After clicking the **Apply** button, Section 2 will be defined.

For the initial 2-D analysis, only **Area**, **I z-z**, and **Z z-z** would be required. The other section properties are only needed when moving to 3-D analysis.

MASTAN2



MASTAN2
File View Geometry Properties Conditions Analysis Results
Select section and properties will be inserted
Database:
AISC (in)
Type:
W Shapes
Shear Areas:
infinite
Clear
Section:
W12X22
W12X19
W12X16
W12X14
W10X112
W10X100
W10X88
W10X77
W10X68
W10X60
W10X54
W10X49
W10X45
W10X39
Please enter section properties
Section 1
Name:
W10X49
Database
Status:
Area = 14.4
I z-z = 272
I y-y = 93.4
J = 1.39
Cw = 2070
Basic
Z z-z = 60.4
Z y-y = 28.3
A y-y = Inf
A z-z = Inf
Apply
Cancel


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N33
E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2 E1 N1
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26 E25 N25
Y
Z
X
Select section and properties will be inserted
Database:
Section:
AISC (in)
W12X22
W12X19
W12X16
W12X14
W10X112
W10X100
W10X88
W10X77
W10X68
W10X60
W10X54
W10X49
W10X45
W10X39
Type:
W Shapes
Shear Areas:
infinite
Clear
Please enter section properties
Section 2
Name:
Database
Status:
Success: Section 1 defined.
Area =
0
I z-z =
0
I y-y =
0
J =
0
Cw =
0
Basic
Z z-z =
inf
Z y-y =
inf
A y-y =
inf
A z-z =
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) Click the **Adv** button to open pop-up menu. To select all the vertical elements, click the check box next to the **Y-axis** option. Click **Add** to add all vertical elements to the element list.

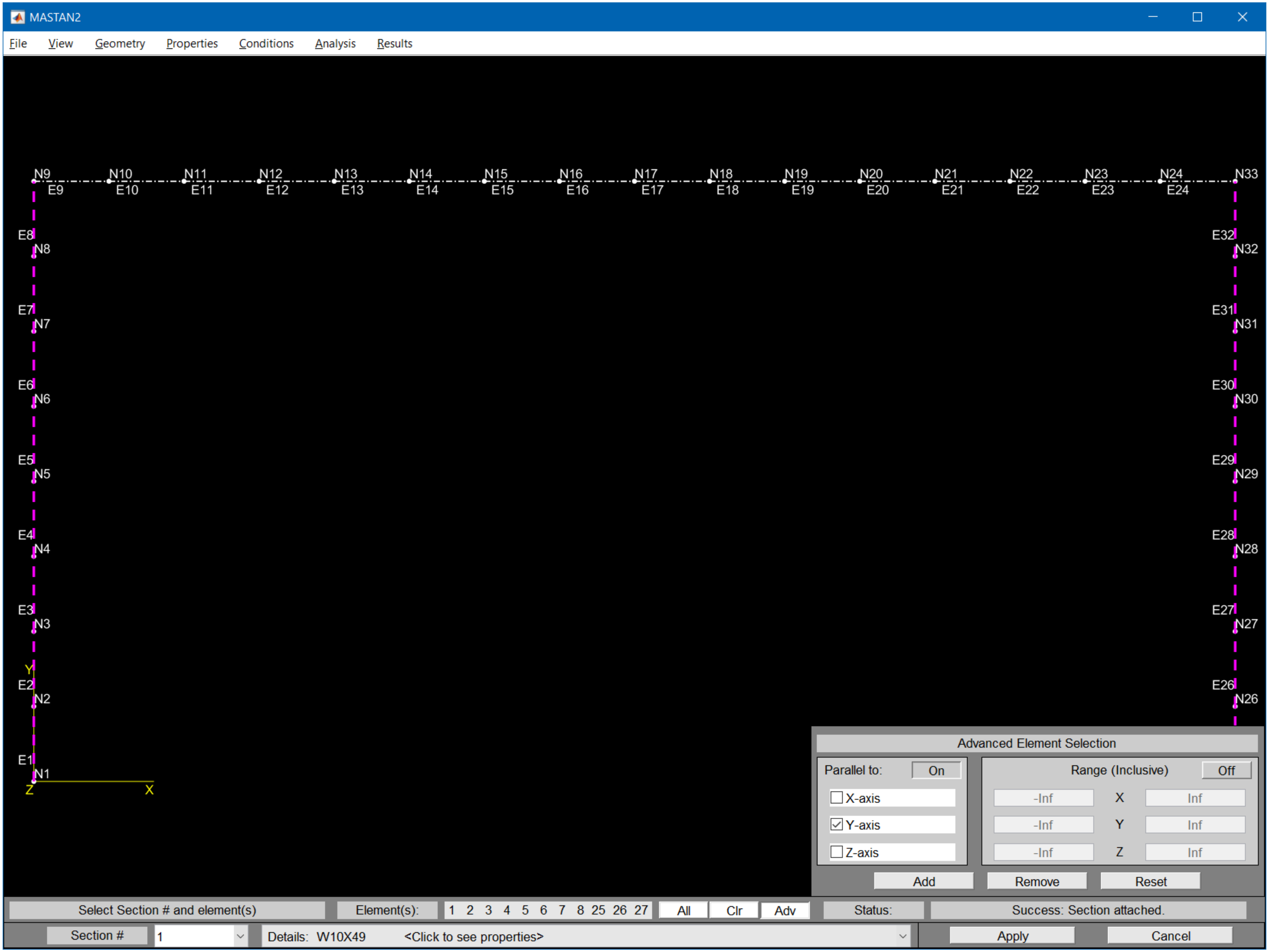
4) Click on the **Apply** button to assign Section 1. (Note that the element line style has changed from dash-dot to dashed.)

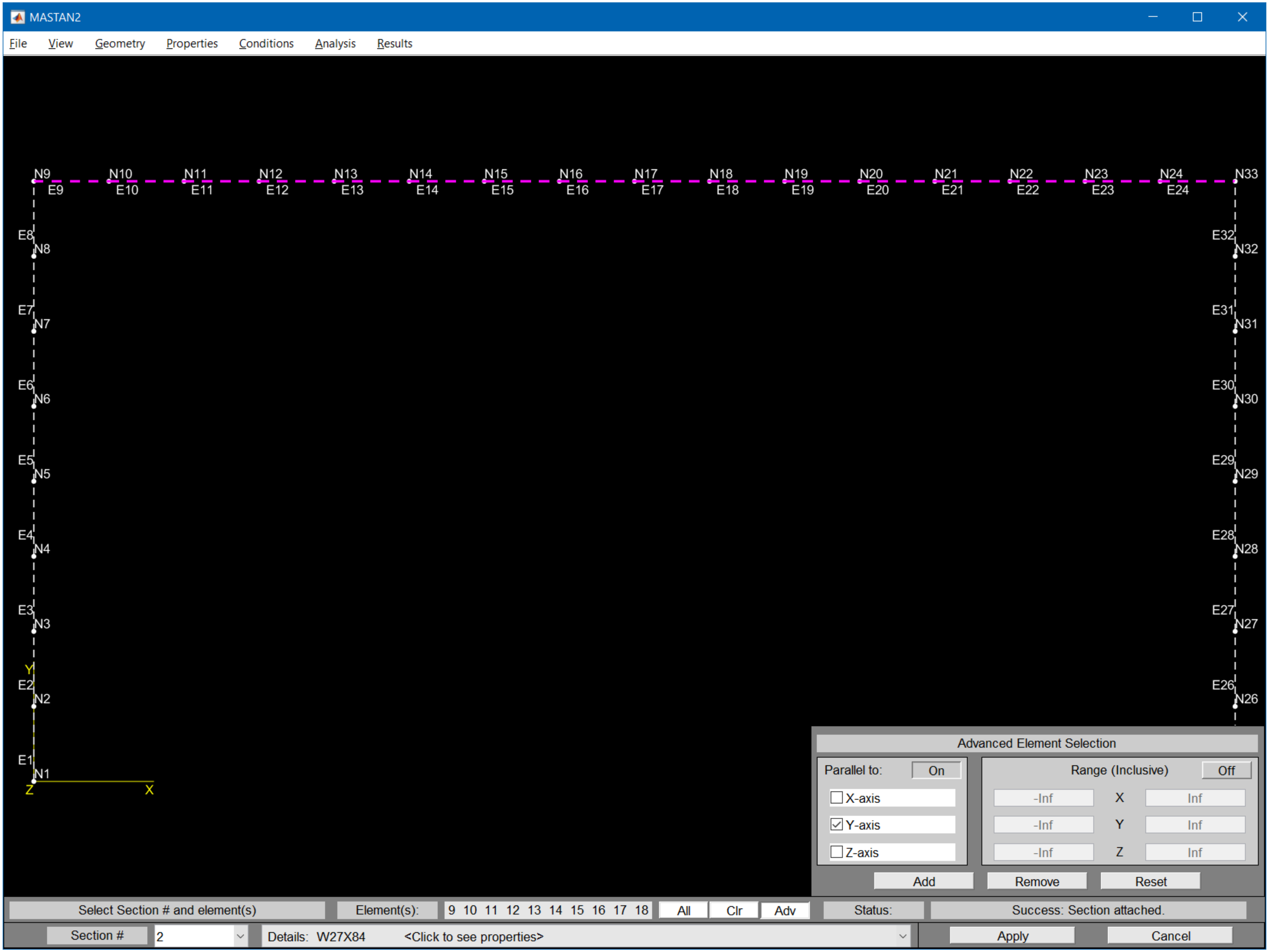
5) Select the **Clr** button located to the right of **Elements:** to clear the list of elements.

6) Create a list of the remaining elements by clicking the **All** button and then the **Remove** button in the pop-up menu. This should leave only the horizontal members selected.

7) Change the **Section #** by clicking on the current section number, **1**, just to the right to open a pop-up menu with all section numbers. Click on **2** to select the W27x84 section.

8) Assign Section #2 properties by clicking the **Apply** button.


MASTAN2
File View Geometry Properties Conditions Analysis Results
N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N33
E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2 E1 N1
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26
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
Select Section # and element(s)
Element(s): 1 2 3 4 5 6 7 8 25 26 27
All Clr Adv
Status: Success: Section attached.
Section # 1
Details: W10X49 <Click to see properties>
Apply Cancel


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N33
E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2 E1 N1
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26
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
Select Section # and element(s)
Element(s):
9 10 11 12 13 14 15 16 17 18
All
Clr
Adv
Status:
Success: Section attached.
Section #
2
Details: W27X84 <Click to see properties>
Apply
Cancel

Material Properties

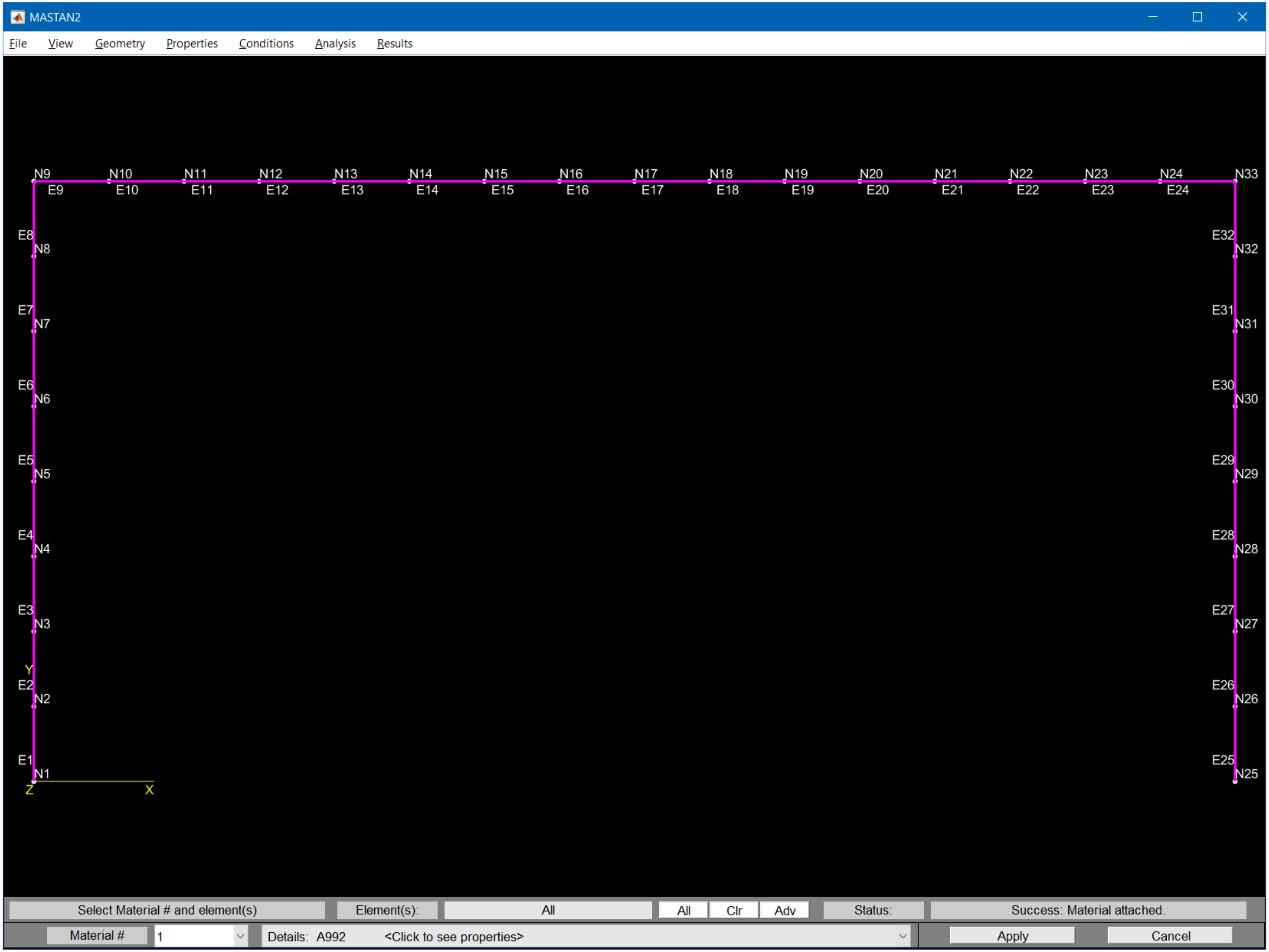
1) From the **Properties** menu select **Define Material**.

2) At the bottom menu bar, click in the edit box just to the right of **E=** and change the **0** to **29000** (not 29,000). Similarly, click in the edit box just to the right of **Fy=** and change the **inf** to **50**. Next, click in the edit box to the right of **Name:** and type **A992**. Click on the **Apply** button. (Material #1 is now defined with the properties of A992 steel.)

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

4) 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. (Note that elements with assigned section and material properties turn solid.)

MASTAN2


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Select Material # and element(s)
Element(s):
All
All
Clr
Adv
Status:
Success: Material attached.
Material #
1
Details: A992 <Click to see properties>
Apply
Cancel

Support Conditions

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

2) At the bottom menu bar, define a fixed support by clicking in the **check boxes** just to the left of all six degrees of freedom: **X-disp**, **Y-disp**, **Z-disp**, **X-rot**, **Y-rot**, and **Z-rot**.

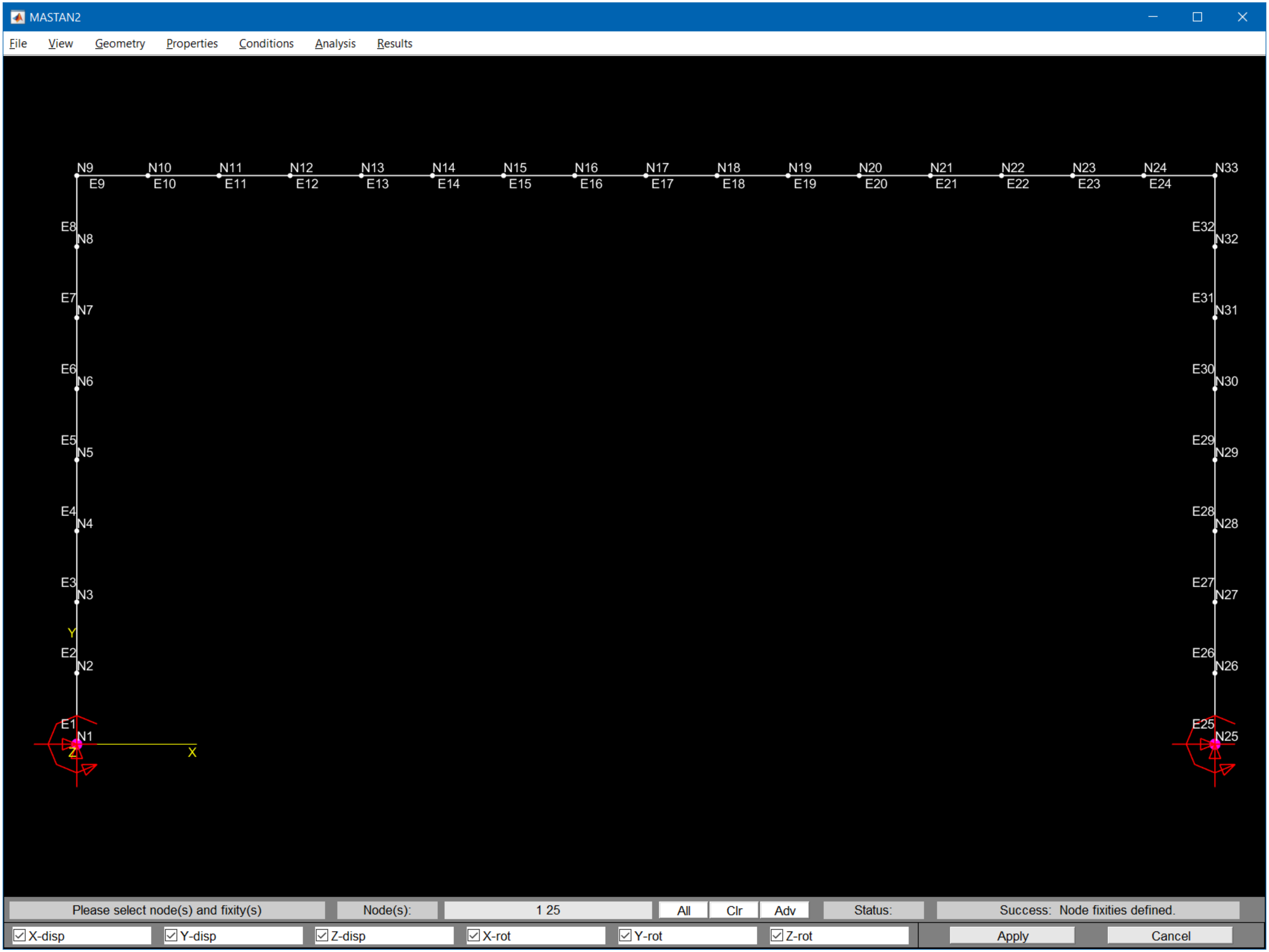
3) Create the list of nodes to be assigned these fixities by clicking on the bottom two nodes of the model, **1** and **25**.

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

5) From the **View** menu select **Fit**.

For the initial 2-D analysis, only **X-disp**, **Y-disp**, and **Z-rot** would need to be constrained for full fixity. The other fixities are only needed when moving to 3-D analysis.

MASTAN2


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Please select node(s) and fixity(s)
Node(s):
1 25
All
Clr
Adv
Status:
Success: Node fixities defined.
X-disp
Y-disp
Z-disp
X-rot
Y-rot
Z-rot
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 **PX =** and change the **0** to **10**.

3) Create the list of nodes to be assigned these forces by clicking on the upper left-hand node, **9**.

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

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

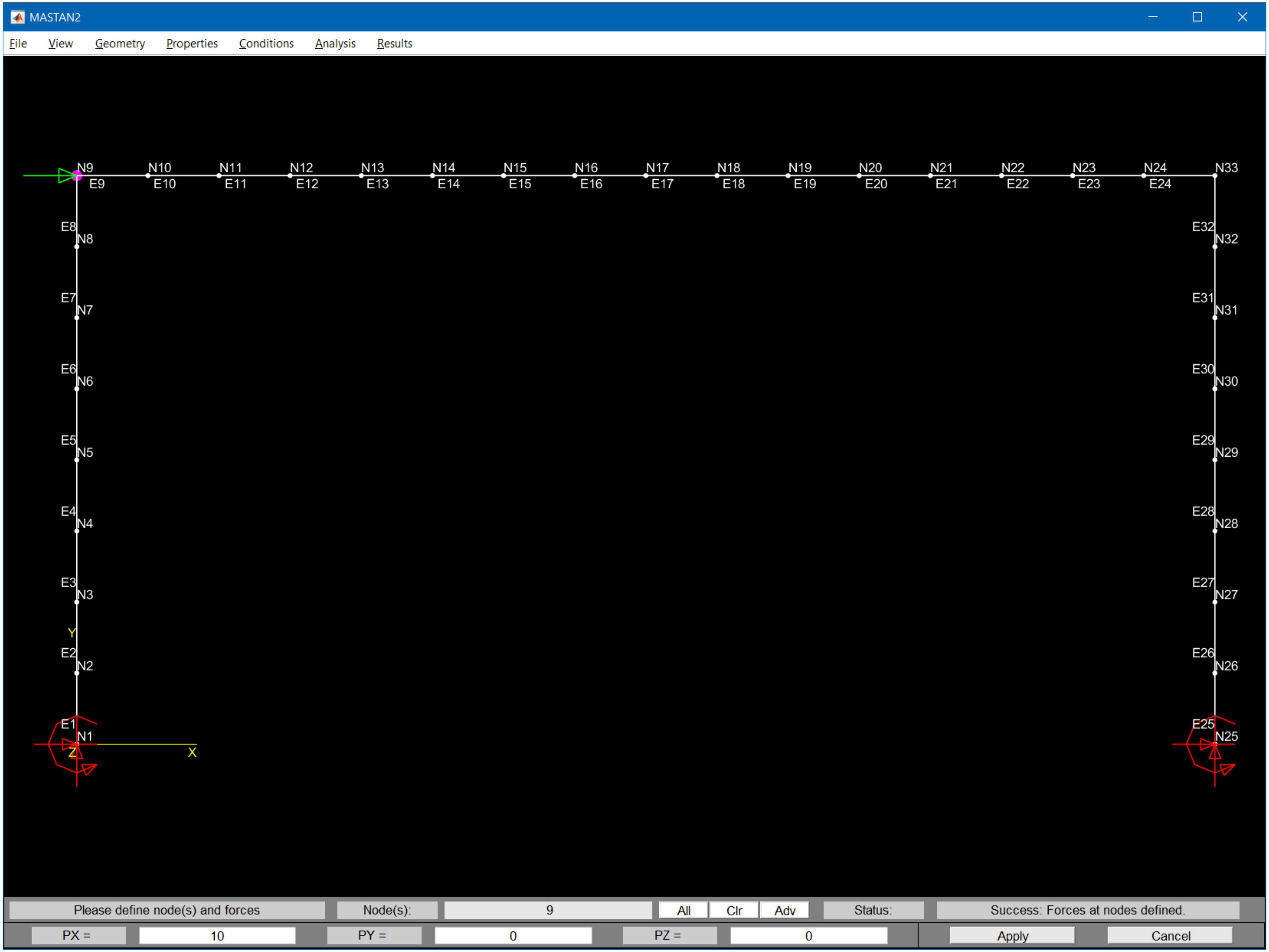
6) Since the loading input is already **Element(s) local x'-y'-z'**, click in the edit box just to the right of **wy' =** and change **0** to **-0.1**.

7) Click the **Adv** button to open pop-up menu. Create a list of the horizontal elements by clicking the **All** button and then the **Remove** button in the pop-up menu.

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

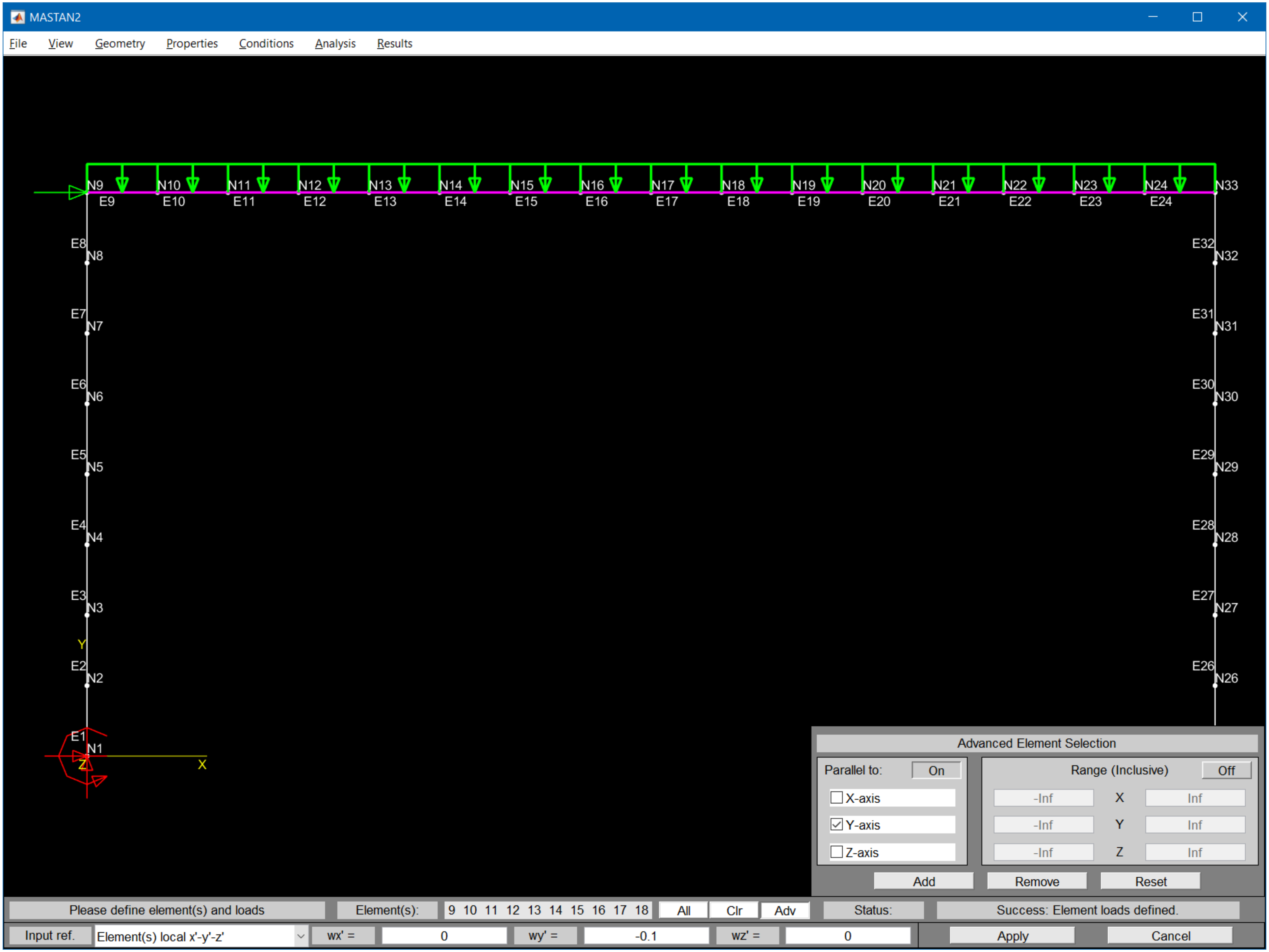
9) From the **View** menu select **Fit**.

MASTAN2



MASTAN2
File View Geometry Properties Conditions Analysis Results
N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N33
E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2 E1 N1
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26 E25 N25
Y
X
Z
Please define node(s) and forces
Node(s):
9
All
Clr
Adv
Status:
Success: Forces at nodes defined.
PX =
10
PY =
0
PZ =
0
Apply
Cancel

MASTAN2



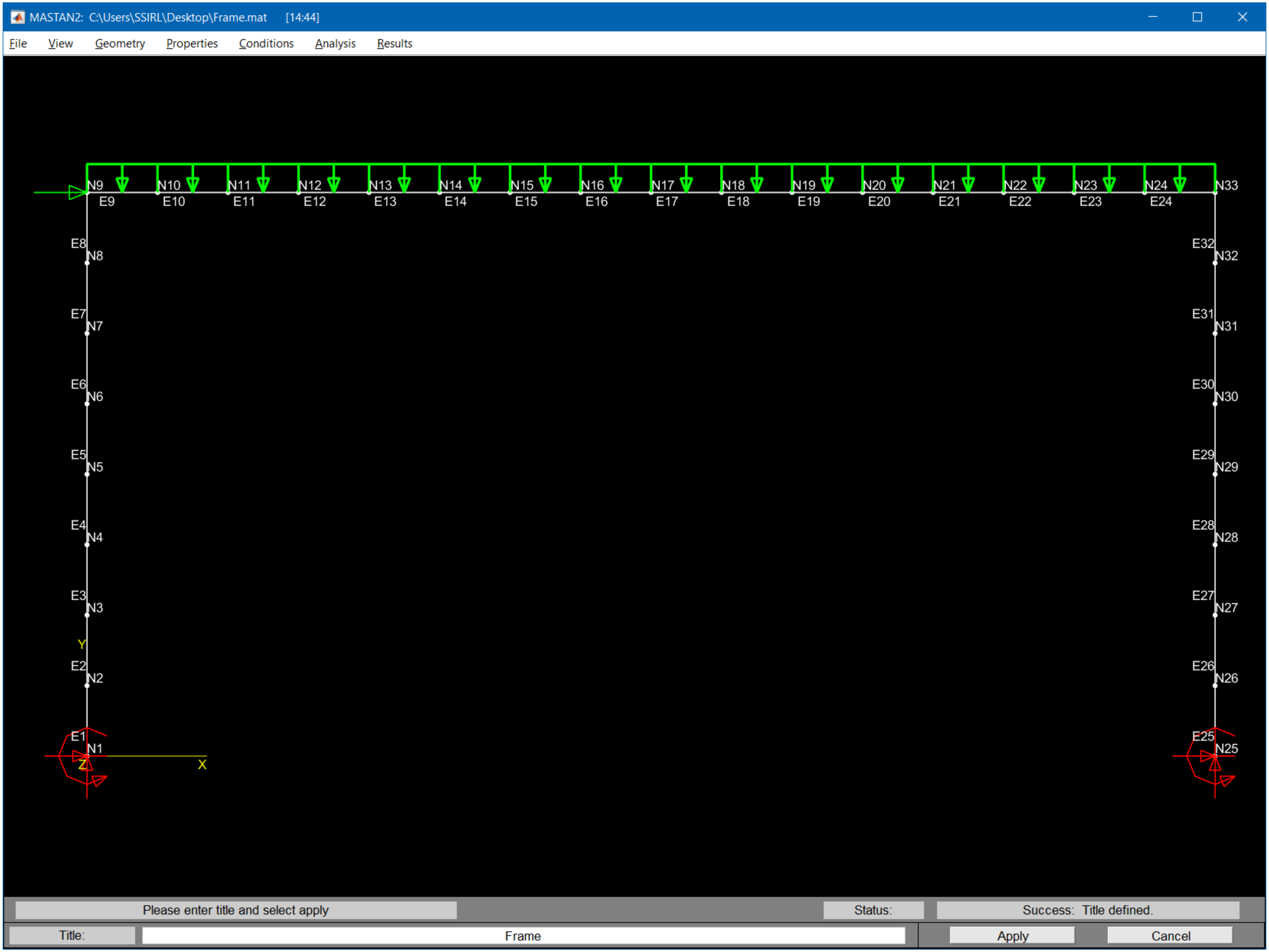
MASTAN2
File View Geometry Properties Conditions Analysis Results
N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N33
E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2 E1 N1
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26
Y
X
Z
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
Please define element(s) and loads
Element(s): 9 10 11 12 13 14 15 16 17 18
All Clr Adv
Status: Success: Element loads defined.
Input ref. Element(s) local x'-y'-z'
wx' = 0
wy' = -0.1
wz' = 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 **Frame** 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\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N33
E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
E8 N8
E7 N7
E6 N6
E5 N5
E4 N4
E3 N3
Y
E2 N2
E1 N1
Z
X
E32 N32
E31 N31
E30 N30
E29 N29
E28 N28
E27 N27
E26 N26
E25 N25
Please enter title and select apply
Status:
Success: Title defined.
Title:
Frame
Apply
Cancel

2-D First-Order Elastic Analysis

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

2) At the bottom menu bar, click on the pop-up menu just to the right of **Analysis Type:** and Select **Planar Frame (x-y)**.

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**.

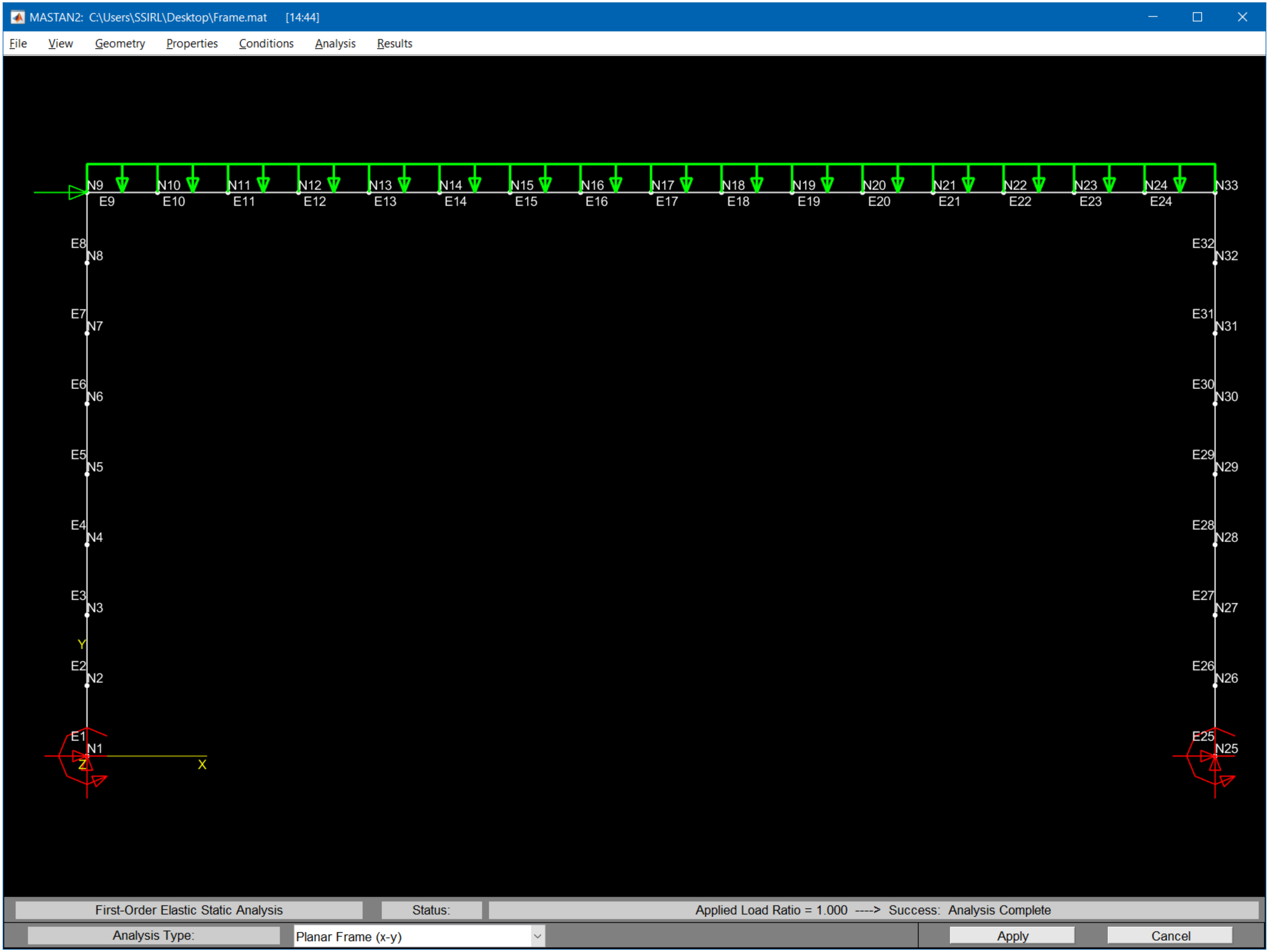
7) On the undeflected shape, click on the node of interest in the upper right corner, **33**, and its components are provided in the bottom menu bar.

Results:

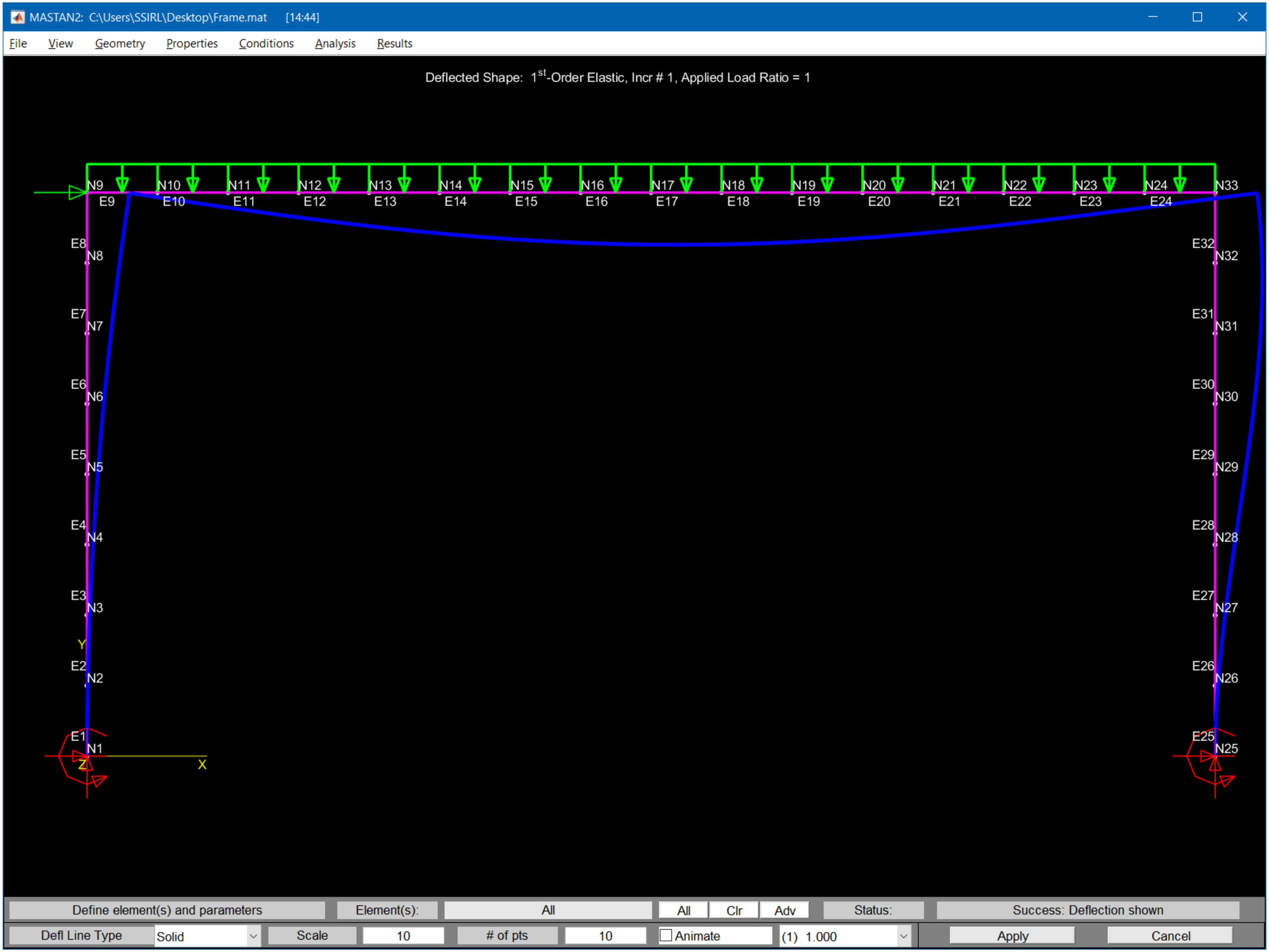
Disp X	Disp Y	Disp Z	Rot X	Rot Y	Rot Z
2.688	-0.03312	N/A	N/A	N/A	0.01235

This can be repeated for other nodes by clicking on them or click in the edit box to the right of **Node:**, enter the value, and click **Apply**.

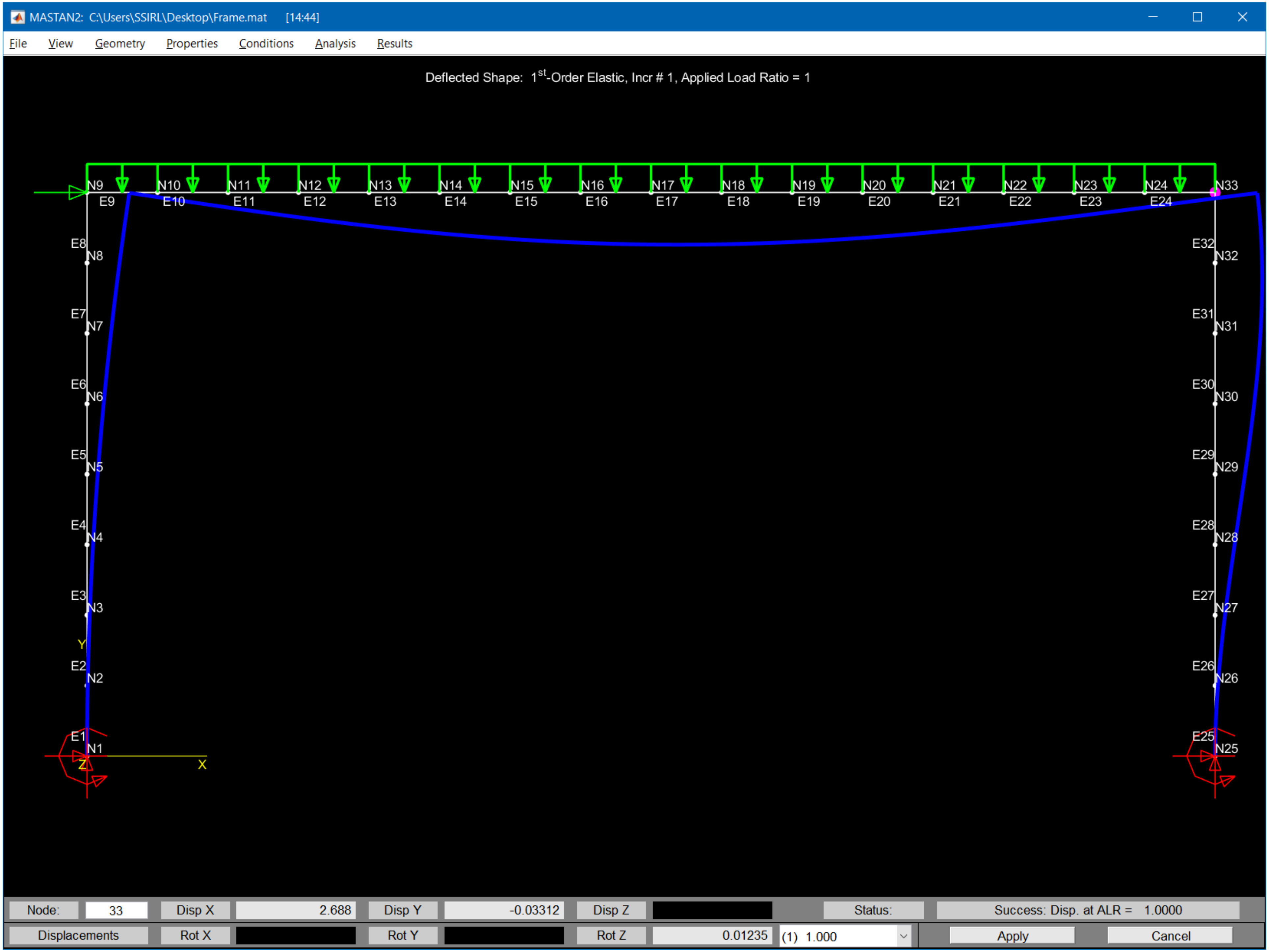
MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N33
E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2 E1 N1
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26 E25 N25
Y
X
Z
First-Order Elastic Static Analysis
Status:
Applied Load Ratio = 1.000 ----> Success: Analysis Complete
Analysis Type:
Planar Frame (x-y)
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
Deflected Shape: 1st-Order Elastic, Incr # 1, Applied Load Ratio = 1
N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N33
E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 Y E2 N2 E1 N1 Z X
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26 E25 N25
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\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
Deflected Shape: 1st-Order Elastic, Incr # 1, Applied Load Ratio = 1
N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N33
E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2 E1 N1
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26 E25 N25
Y
X
Z
Node: 33
Disp X 2.688
Disp Y -0.03312
Disp Z
Status: Success: Disp. at ALR = 1.0000
Displacements
Rot X
Rot Y
Rot Z 0.01235
(1) 1.000
Apply
Cancel

2-D Second-Order Elastic Analysis

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

2) At the bottom menu bar, click on the pop-up menu just to the right of **Analysis Type:** and Select **Planar Frame (x-y)**.

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

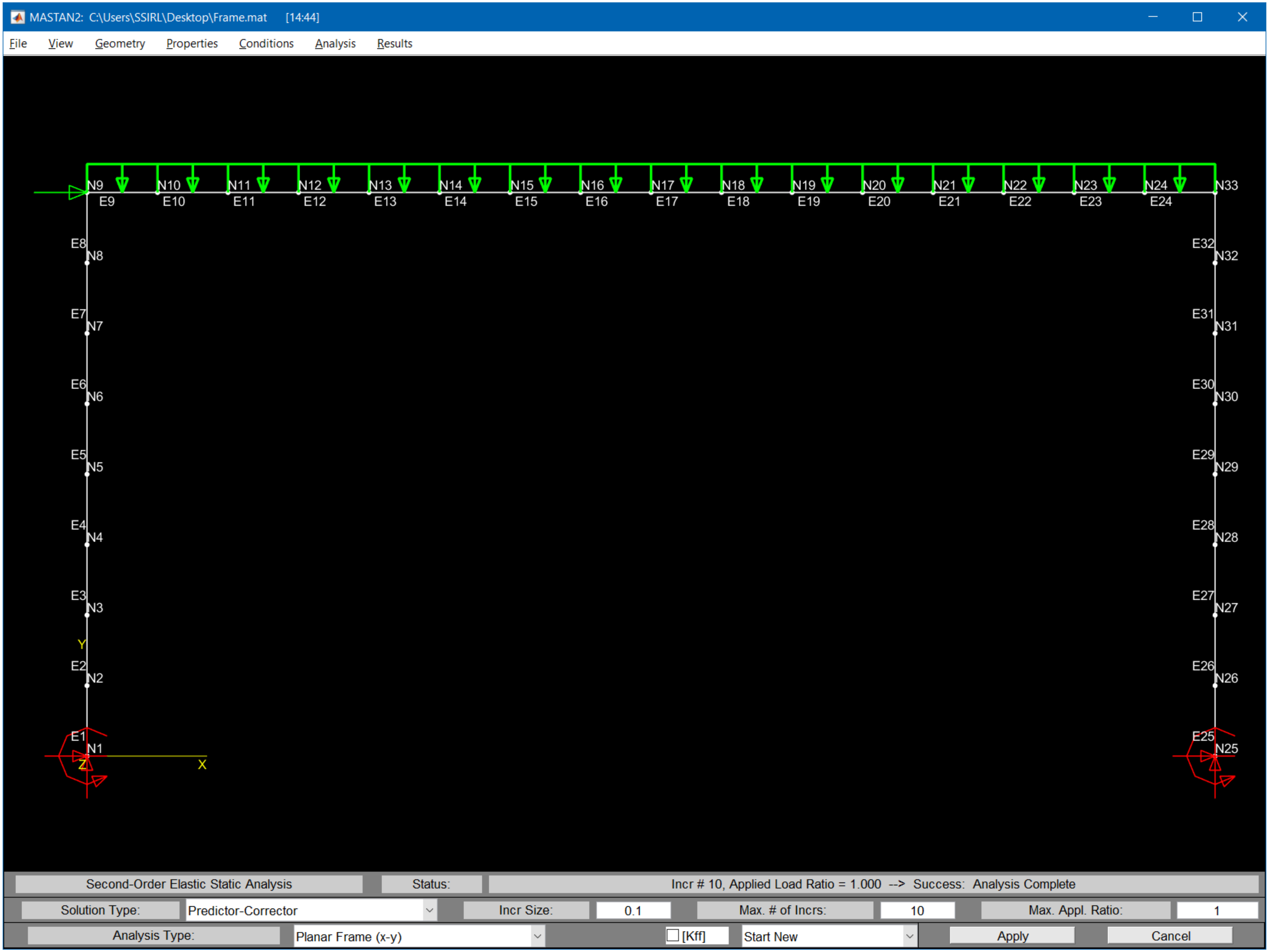
4) From the **Results** menu select **Node Displacements**.

5) On the undeflected shape, click on the node of interest in the upper right corner, **33**, and its components are provided in the bottom menu bar.

Results:

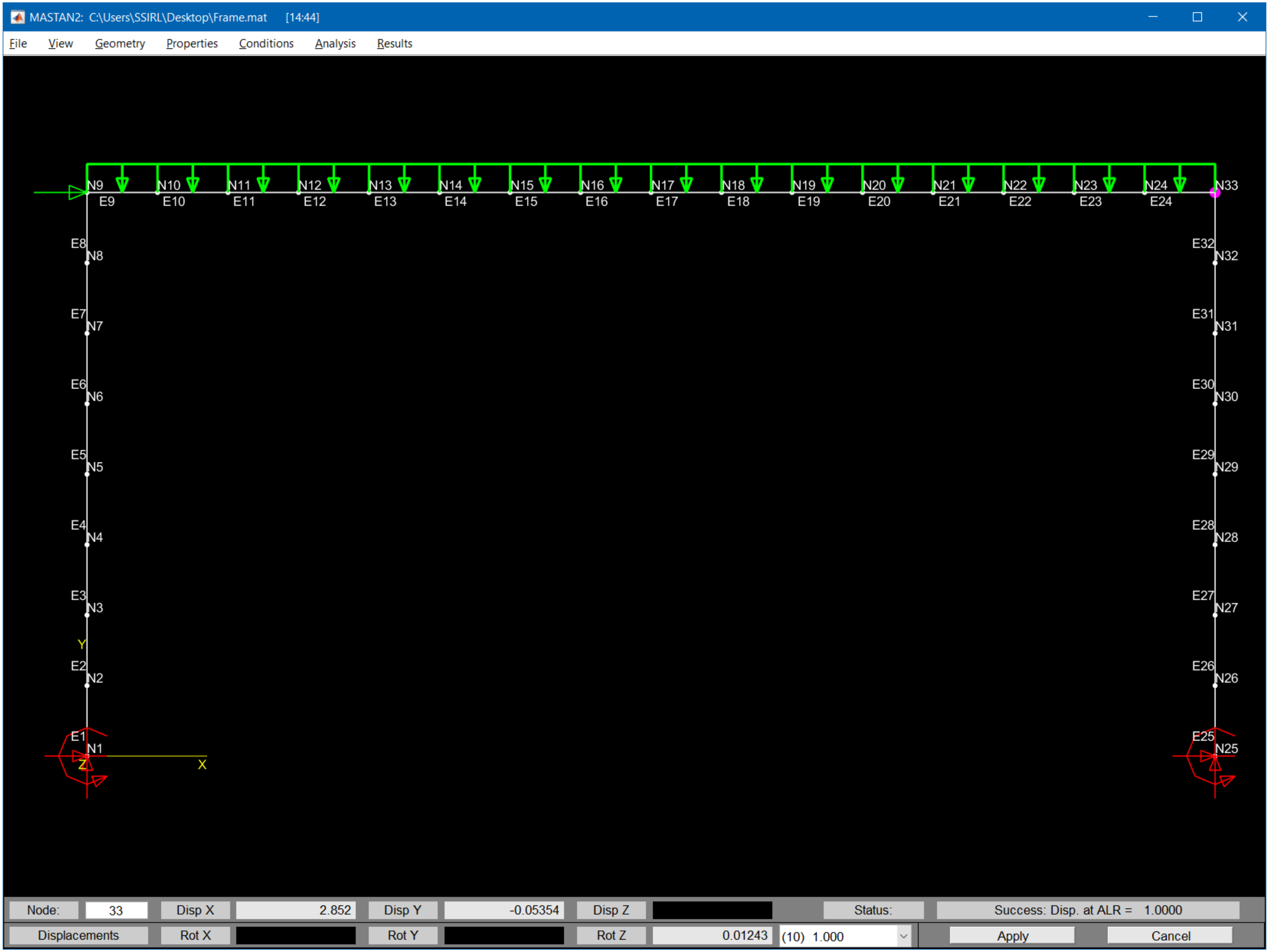
Disp X	Disp Y	Disp Z	Rot X	Rot Y	Rot Z
2.852	-0.05354	N/A	N/A	N/A	0.01243

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
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:
Planar Frame (x-y)
[Kff]
Start New
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N33
E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2 E1 N1
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26 E25 N25
Y
X
Z
Node: 33
Disp X 2.852
Disp Y -0.05354
Disp Z
Status:
Success: Disp. at ALR = 1.0000
Displacements
Rot X
Rot Y
Rot Z 0.01243
(10) 1.000
Apply
Cancel

Section 4: 3-D Frame Analysis

Updating for 3-D Analysis

As is, the model could be run in 3-D. Previously having entered the complete section properties and applying full fixity to the base support nodes would be satisfactory to meet the requirements to run a 3-D analysis. However, this model would be missing the lateral support of the beam previously mentioned in the problem statement. Before proceeding, we will add that support to the frame through additional boundary conditions.

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

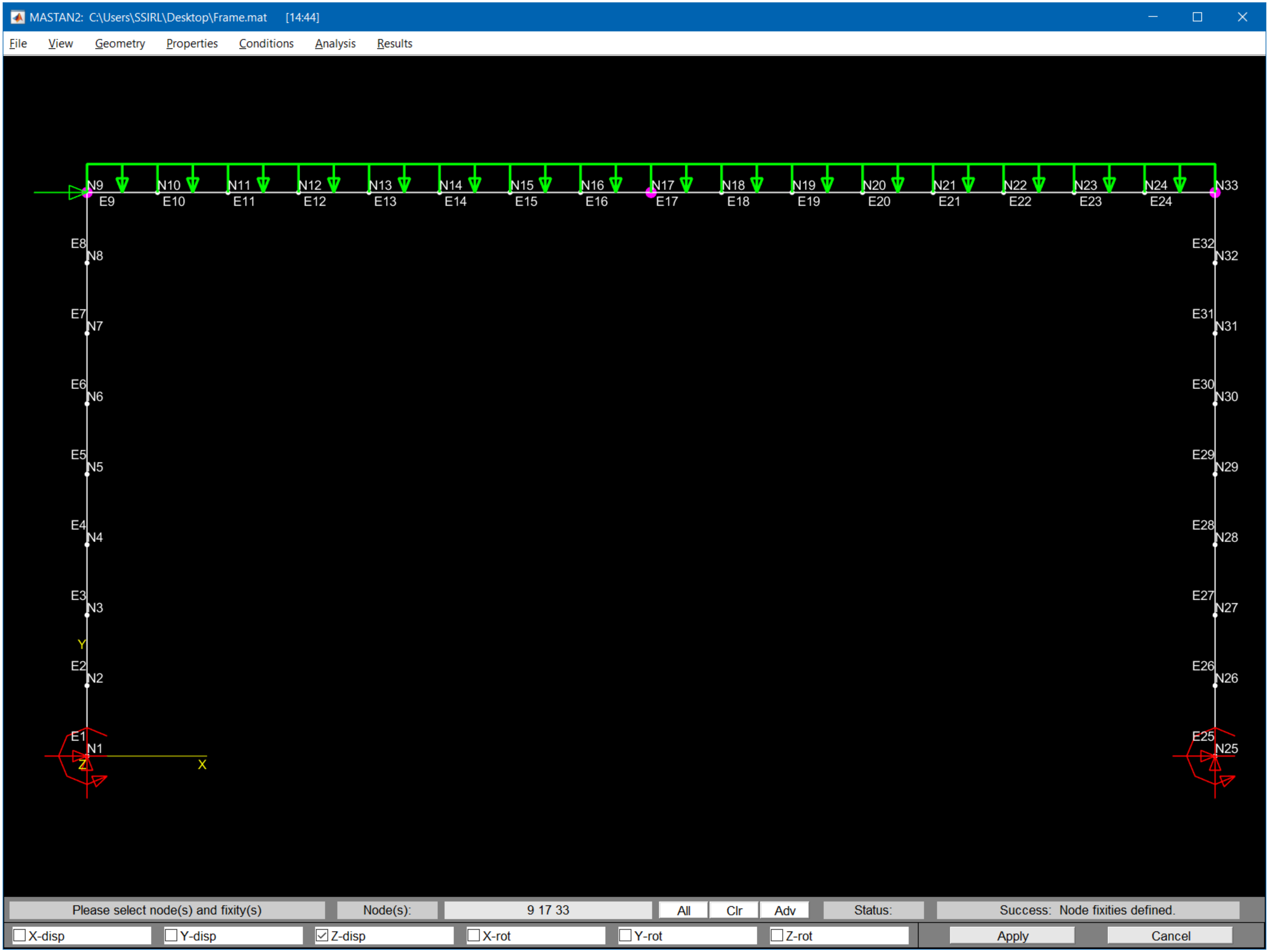
2) At the bottom menu bar, define the lateral support by clicking in the **check box** to the left of **Z-disp**.

3) Create the list of nodes to be assigned these fixities by clicking on the top corner and middle nodes of the model: **9**, **17**, and **33**.

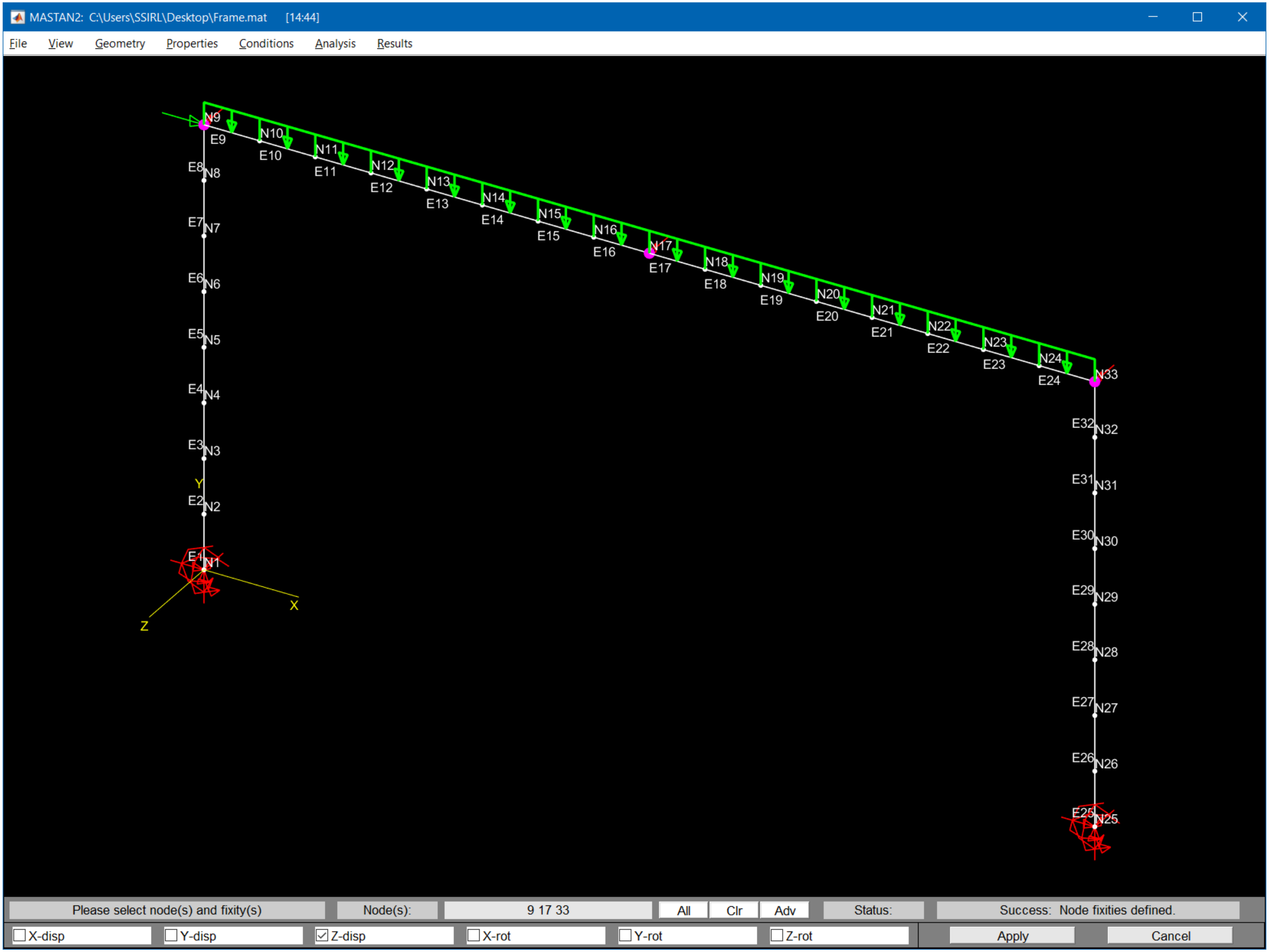
4) Click on the **Apply** button.

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

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
Please select node(s) and fixity(s)
Node(s):
9 17 33
All
Clr
Adv
Status:
Success: Node fixities defined.
X-disp
Y-disp
Z-disp
X-rot
Y-rot
Z-rot
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
Please select node(s) and fixity(s)
Node(s):
9 17 33
All
Clr
Adv
Status:
Success: Node fixities defined.
X-disp
Y-disp
Z-disp
X-rot
Y-rot
Z-rot
Apply
Cancel

3-D Second-Order Elastic Analysis

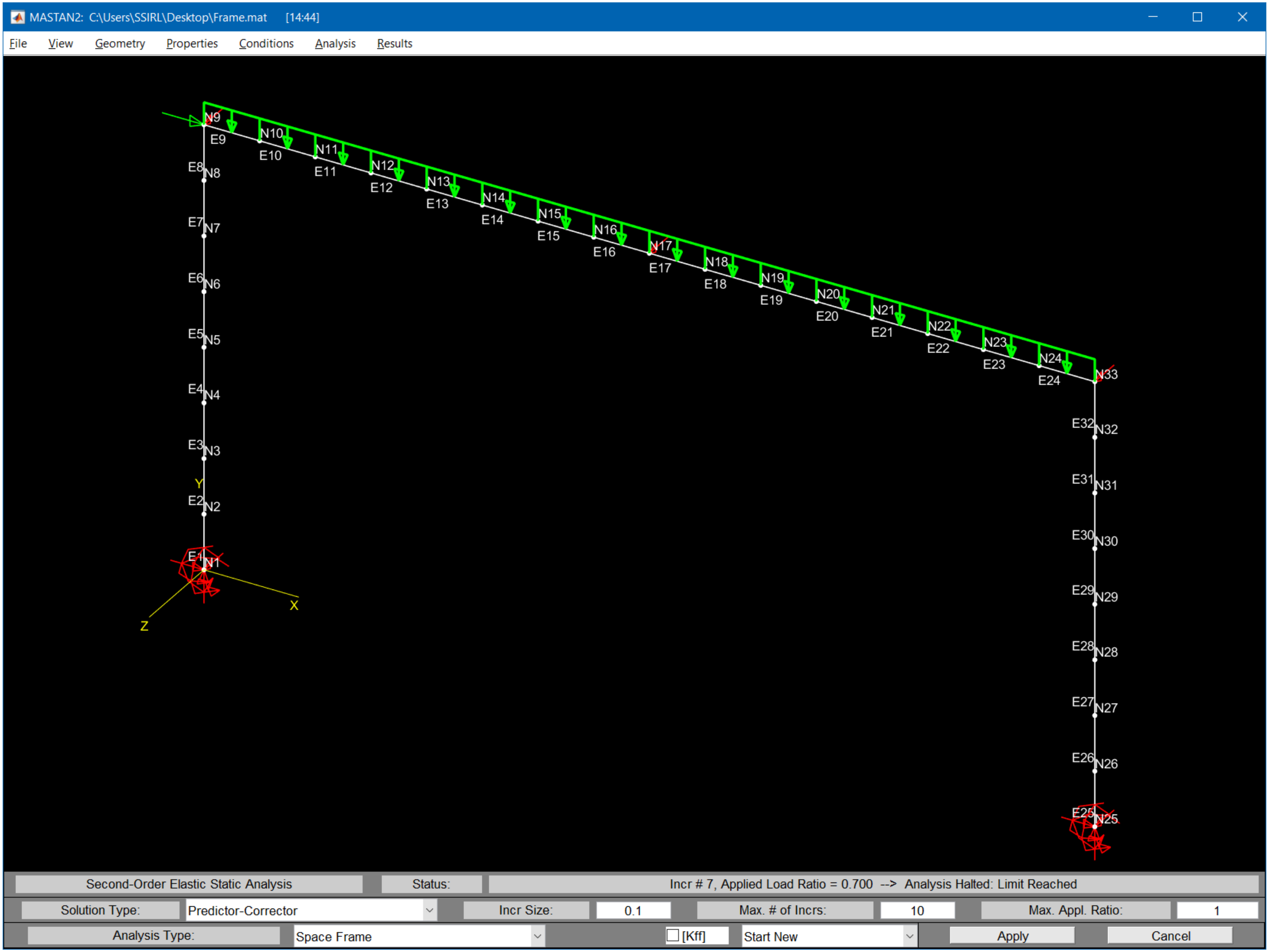
1) From the **Analysis** menu select **Static** and submenu option **2nd-Order Elastic**.

2) At the bottom menu bar, click on the pop-up menu just to the right of **Analysis Type:** and Select **Space Frame**.

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

The analysis should stop with the message **Analysis Halted: Limit Reached**. Often this message is related to the analysis encountering a stability limit. The use of the eigen-buckling tool may help identify the problem.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
N9 E9 N10 E10 N11 E11 N12 E12 N13 E13 N14 E14 N15 E15 N16 E16 N17 E17 N18 E18 N19 E19 N20 E20 N21 E21 N22 E22 N23 E23 N24 E24 N33
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2 E1 N1
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26 E25 N25
Y X Z
Second-Order Elastic Static Analysis
Status:
Incr # 7, Applied Load Ratio = 0.700 --> Analysis Halted: Limit Reached
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

3-D Elastic Critical Load

1) From the **Analysis** menu select **Eigen-Buckling** and submenu option **Elastic Critical Load**.

2) At the bottom menu bar, the **Analysis Type:** should already be set to **Space Frame** with the **Max. # of Modes:** set to **1** as desired.

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

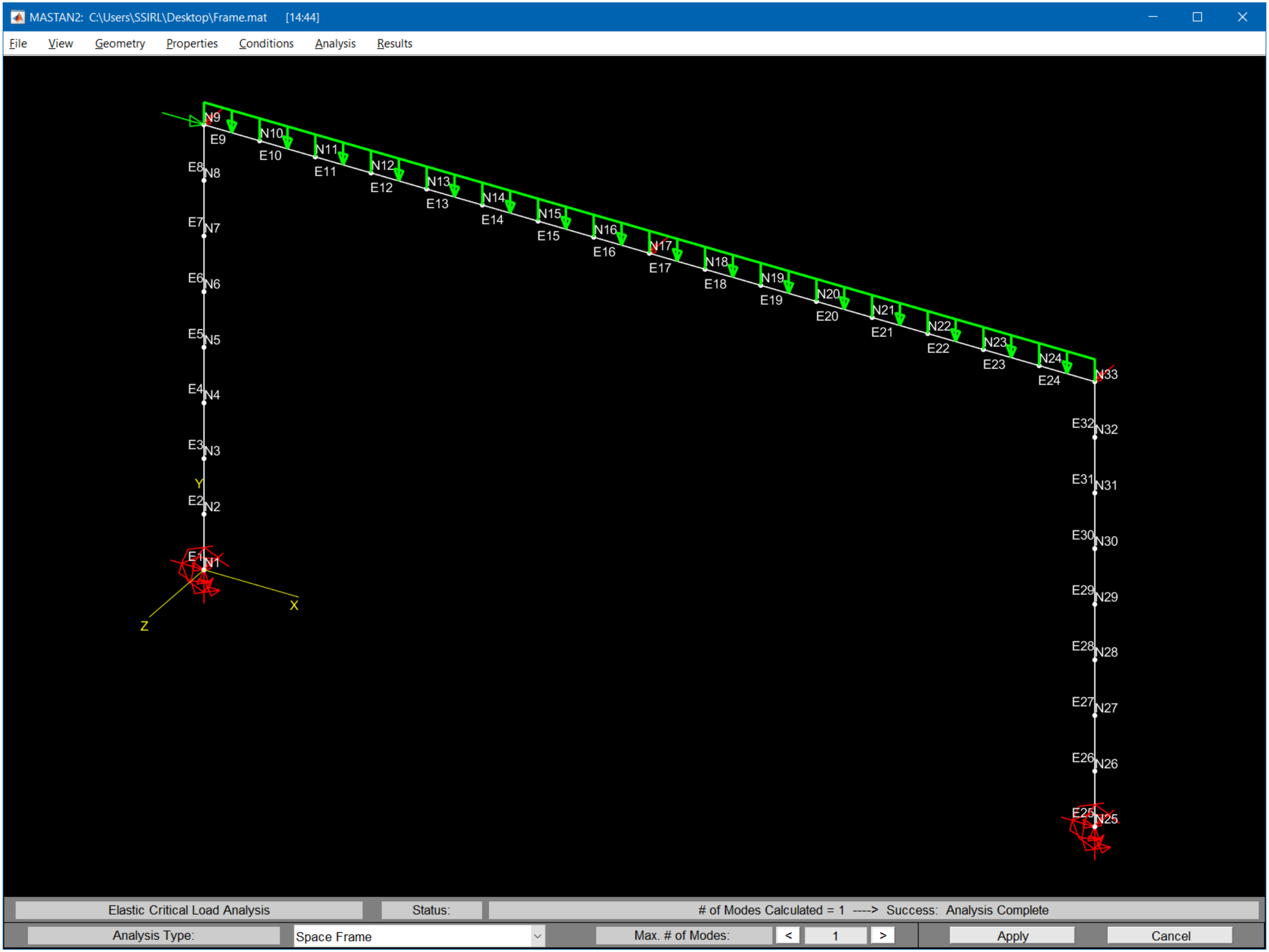
4) From the **Results** menu select **Diagrams** and submenu option **Deflected Shape**.

5) At the bottom menu bar, click the edit box to the right of **Scale**. Change **10** to **30** to amplify the deformed geometry in the visualization.

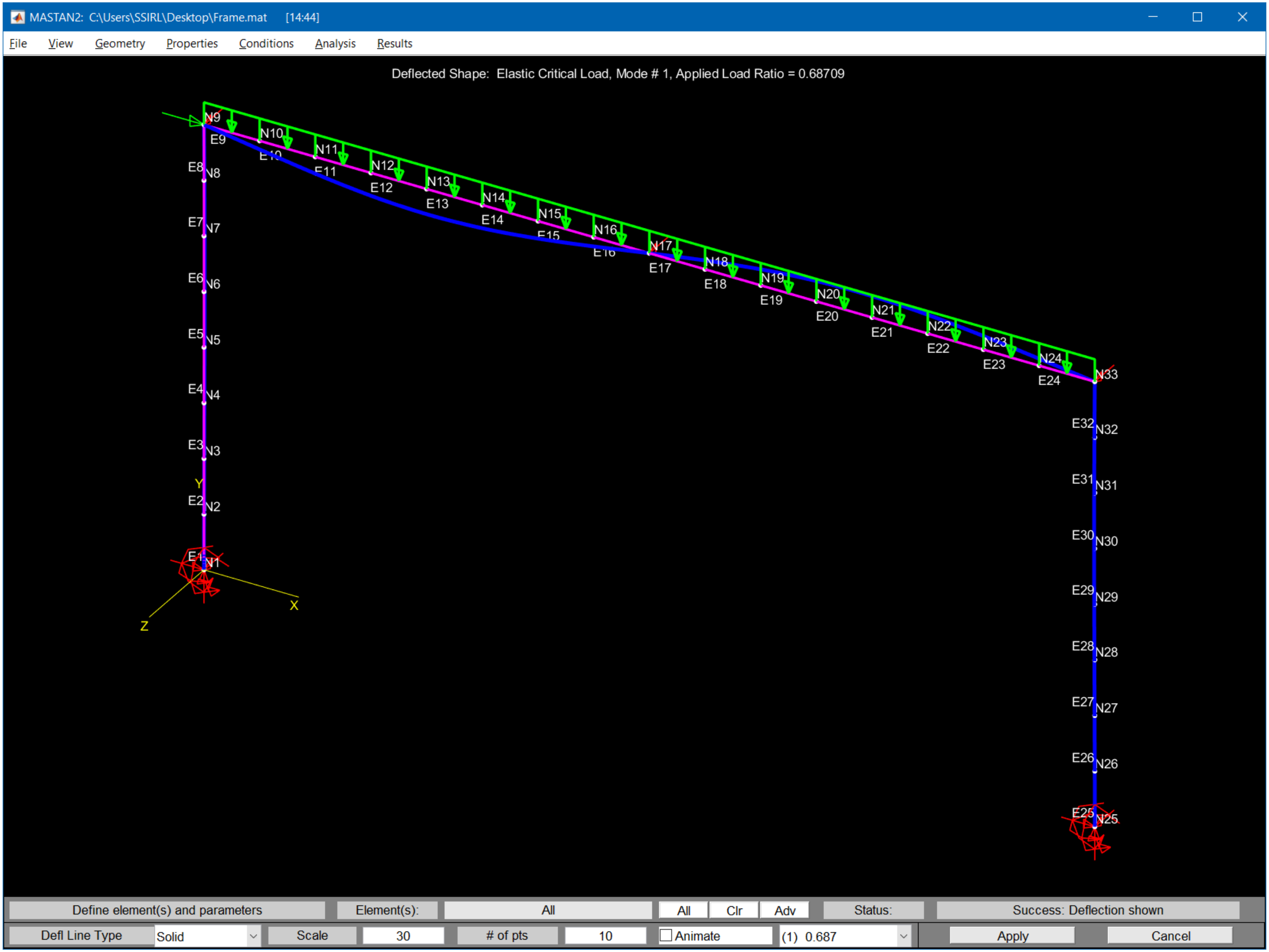
6) Click on the **Apply** button and the first mode is shown with the Applied Load Ratio identified at the top of the screen.

The result indicates that the beam is failing in lateral torsional buckling at only 0.687 times the applied load. Currently, the analysis does not include the warping stiffness which increases the buckling capacity of the beam. MASTAN2 can account for warping effects if the element's warping end conditions are changed.

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
Elastic Critical Load Analysis
Status:
# of Modes Calculated = 1 ----> Success: Analysis Complete
Analysis Type:
Space Frame
Max. # of Modes:
<
1
>
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
Deflected Shape: Elastic Critical Load, Mode # 1, Applied Load Ratio = 0.68709
Define element(s) and parameters
Element(s):
All
All Clr Adv
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
30
# of pts
10
Animate
(1) 0.687
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) Create the list of elements to be assigned continuous warping by clicking on the **All** button to the right of **Elements:**. Click on the **Apply** button. Note: no symbol indicates the end is free to warp, a blue + indicates continuous warping, and a blue ✳ indicates fixed warping.

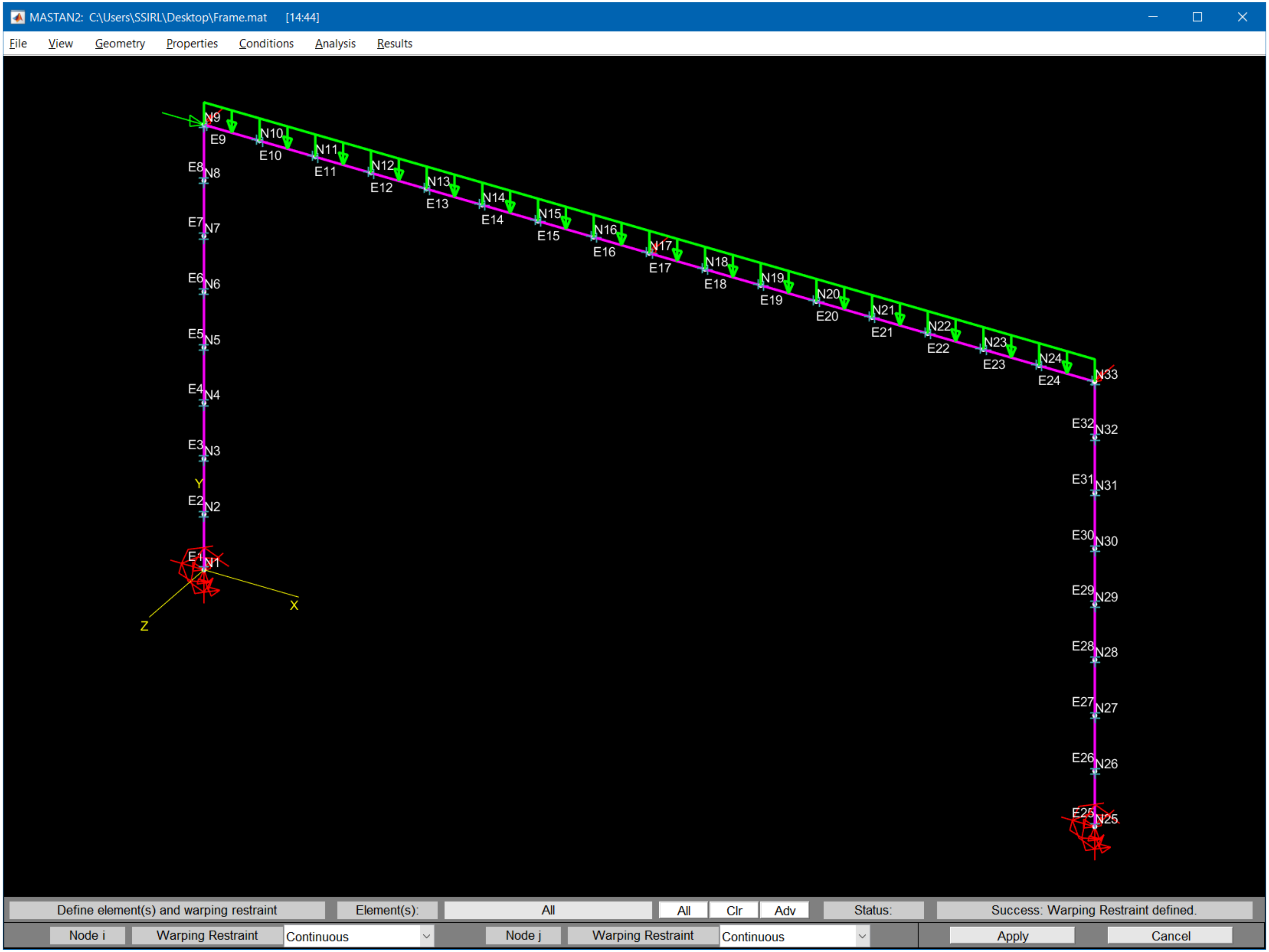
4) Click **Clr** to empty the list of elements. Click on the bottom element of each column and left end element of the beam to define the members that start with warping fixed and are continuous.

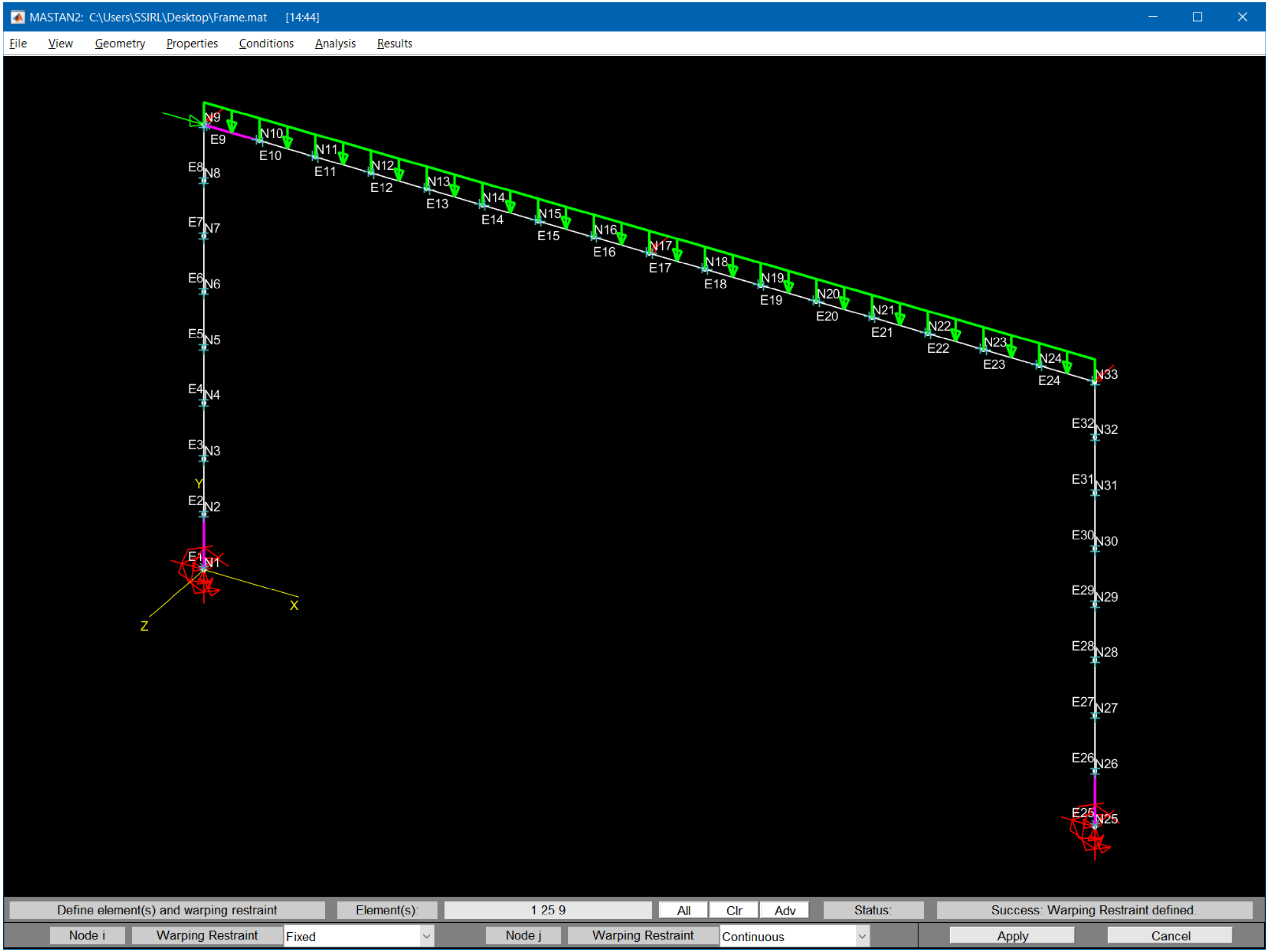
5) Click on the menu to the right of **Warping Restraint for Node i** and set the value to **Fixed**. Node j is set from the previous step. Click on the **Apply** button.

6) Click **Clr** to empty the list of elements. Click on the top element of each column and right end element of the beam.

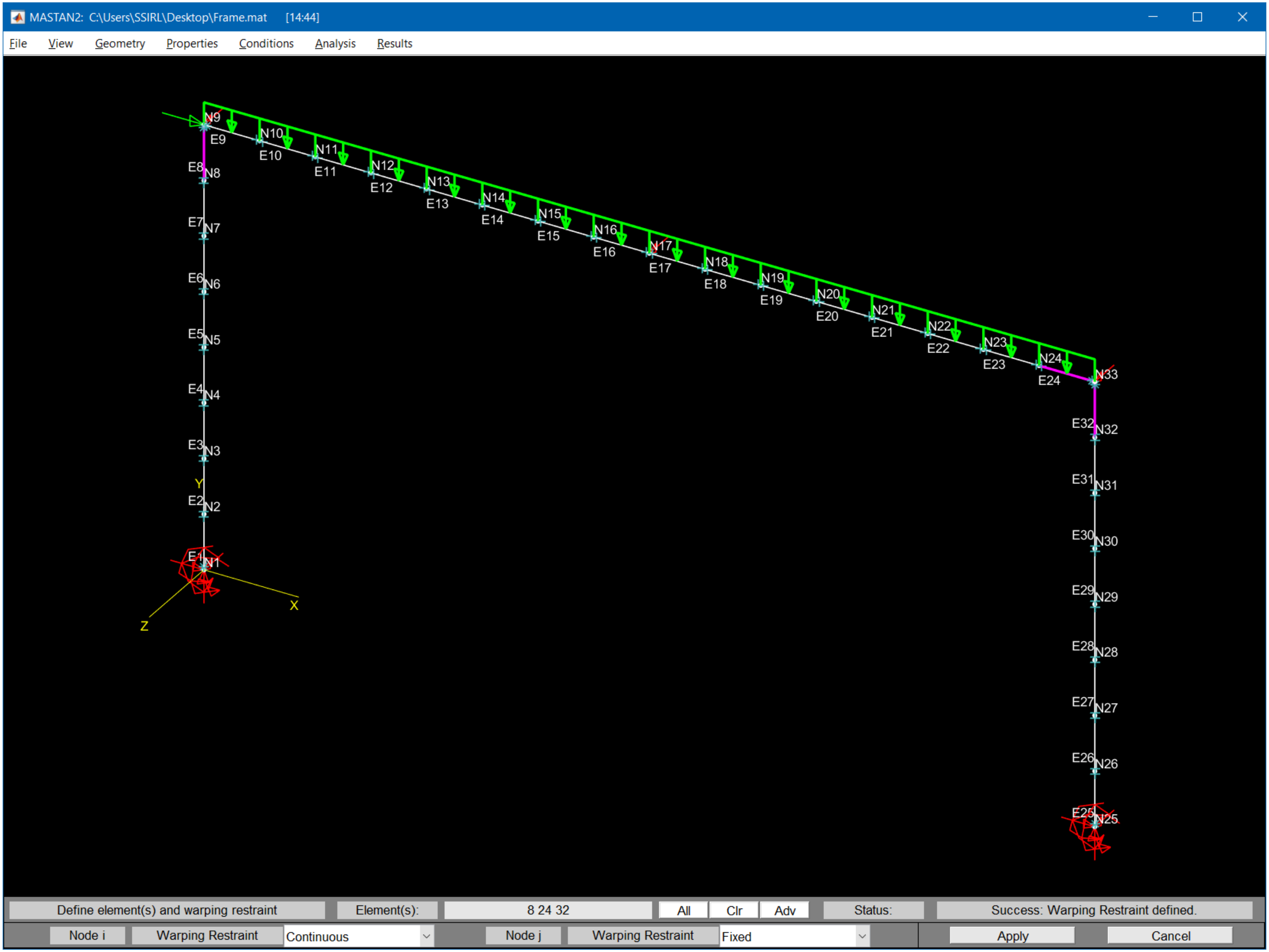
7) Click on the menu to the right of **Warping Restraint for Node i** and set the value to **Continuous**. Click on the menu to the right of **Warping Restraint for Node j** and set the value to **Fixed**.

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


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
Define element(s) and warping restraint
Element(s):
All
All
Clr
Adv
Status:
Success: Warping Restraint defined.
Node i
Warping Restraint
Continuous
Node j
Warping Restraint
Continuous
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
Define element(s) and warping restraint
Element(s):
1 25 9
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\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
Define element(s) and warping restraint
Element(s):
8 24 32
All Clr Adv
Status:
Success: Warping Restraint defined.
Node i
Warping Restraint
Continuous
Node j
Warping Restraint
Fixed
Apply
Cancel

3-D Elastic Critical Load

1) From the **Analysis** menu select **Eigen-Buckling** and submenu option **Elastic Critical Load**.

2) At the bottom menu bar, the **Analysis Type:** should already be set to **Space Frame** with the **Max. # of Modes:** set to **1** as desired.

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

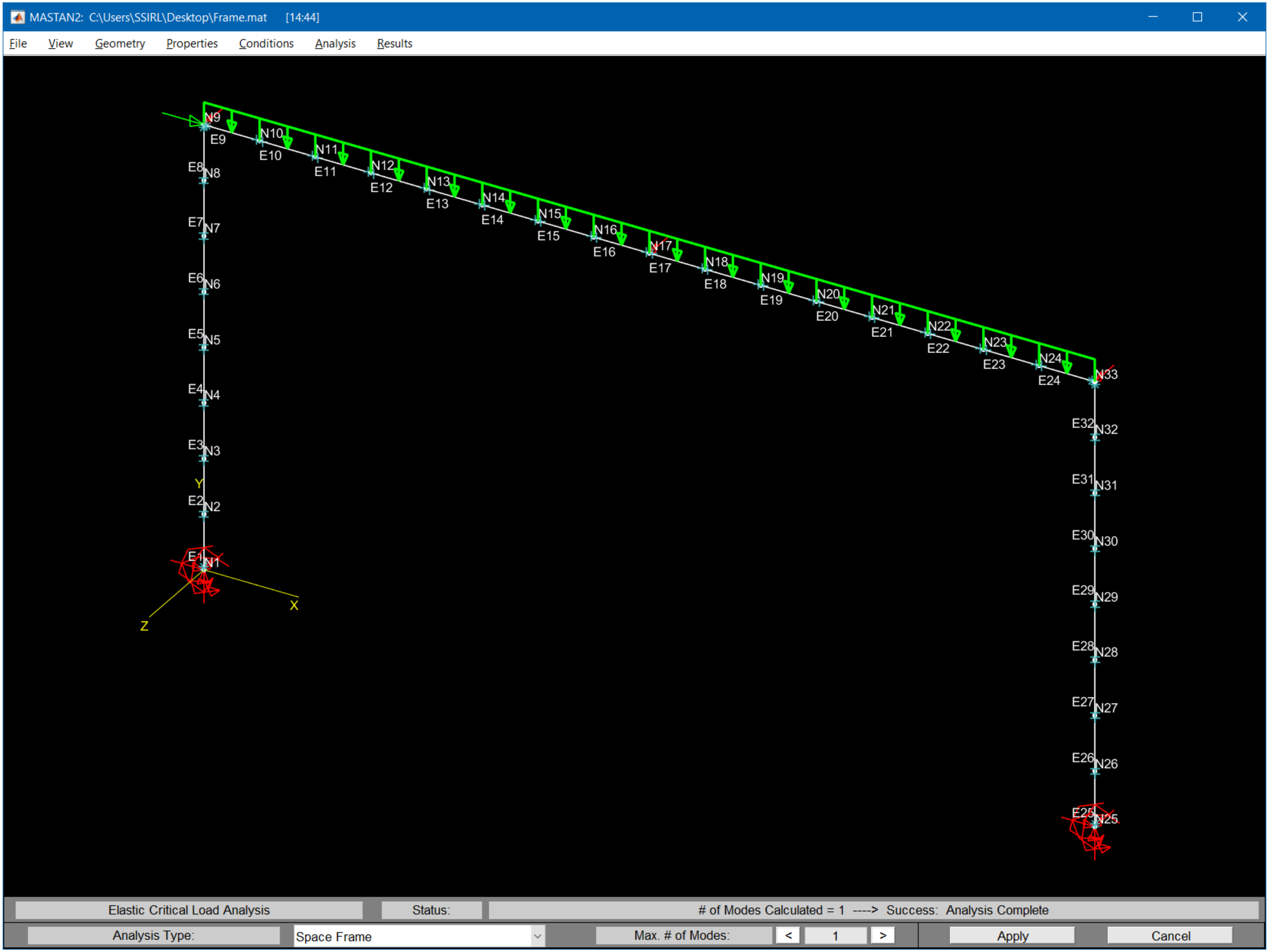
4) From the **Results** menu select **Diagrams** and submenu option **Deflected Shape**.

5) At the bottom menu bar, the **Scale** should still be set to **30** from previous analysis.

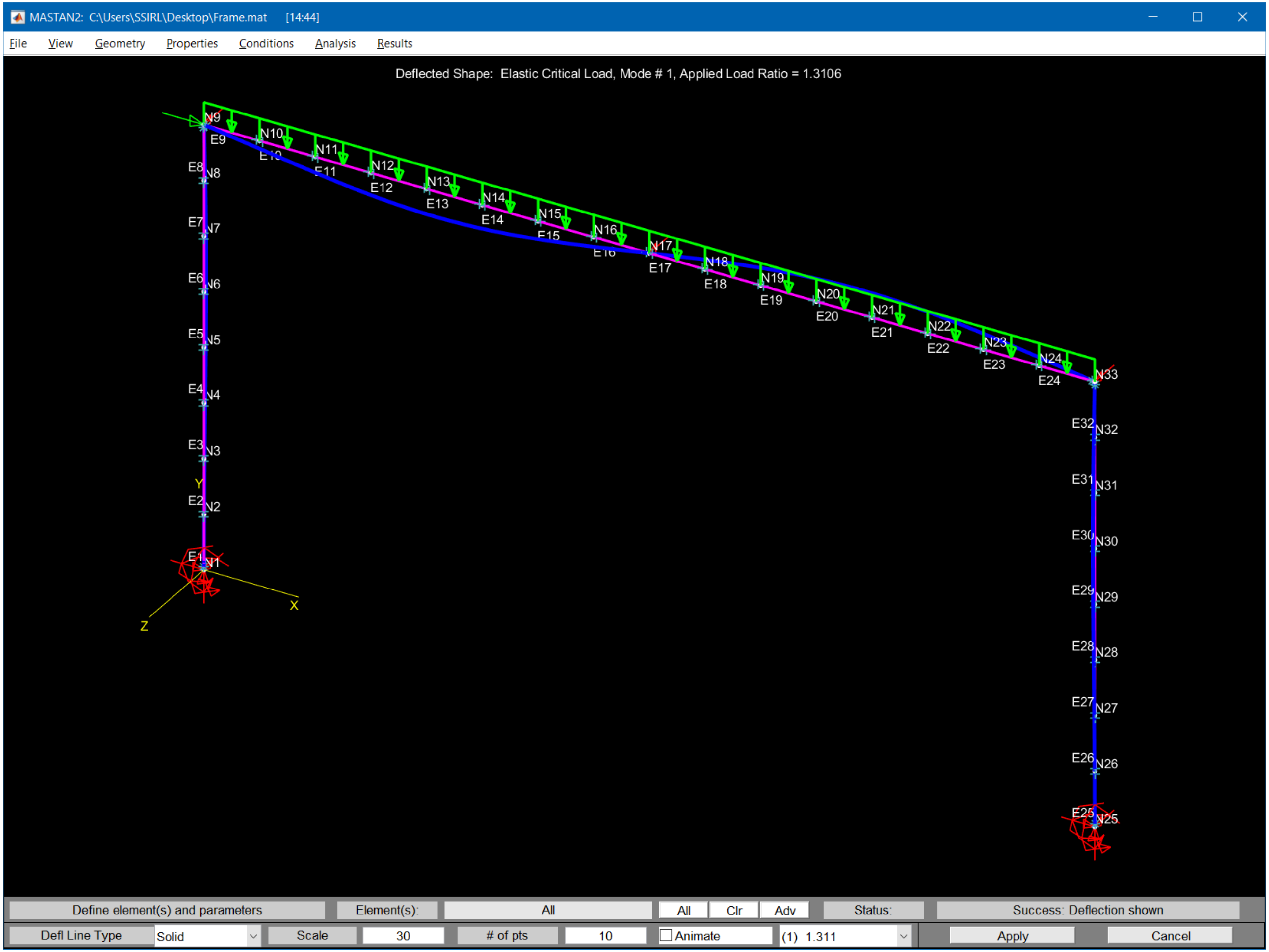
6) Click on the **Apply** button and the first mode is shown with the Applied Load Ratio identified at the top of the screen.

The result indicates that the beam is failing in lateral torsional buckling at 1.31 times the applied load. This value is 1.9 times the result when ignoring the effects of warping stiffness. The fact that the Applied Load Ratio is greater than 1 means it should now be possible to complete the desired 3-D 2nd order analysis.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N33
E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2 E1 N1
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26 E25 N25
Y
X
Z
Elastic Critical Load Analysis
Status:
# of Modes Calculated = 1 ----> Success: Analysis Complete
Analysis Type:
Space Frame
Max. # of Modes:
<
1
>
Apply
Cancel

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
Deflected Shape: Elastic Critical Load, Mode # 1, Applied Load Ratio = 1.3106
Define element(s) and parameters
Element(s):
All
All
Clr
Adv
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
30
# of pts
10
Animate
(1) 1.311
Apply
Cancel

3-D 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**.

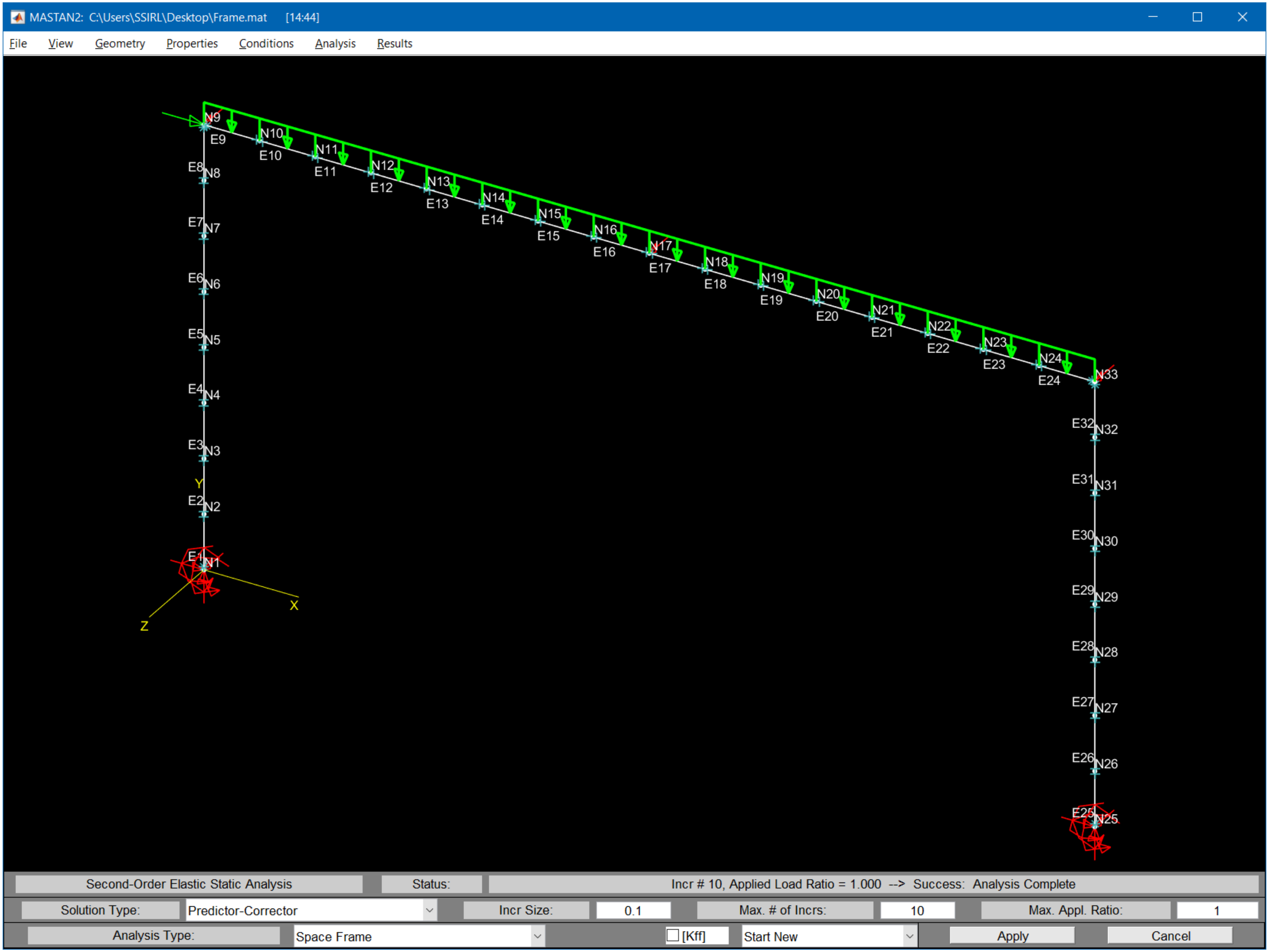
5) On the undeflected shape, click on the node of interest in the upper right corner, **33**, and its components are provided in the bottom menu bar.

Results:

Disp X	Disp Y	Disp Z	Rot X	Rot Y	Rot Z
2.852	-0.05354	0	0	0	0.01243

The deflection response is the same as 2-D as no out-of-plane loading or displacements were added. The same axial and flexural deformations are being modeled. The introduction of the 3-D analysis highlighted the existing out-of-plane instability and the analysis could not proceed past the bifurcation load in the perfect model.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N33
E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2 E1 N1
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26 E25 N25
Y
X
Z
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\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
N9 E9 N10 E10 N11 E11 N12 E12 N13 E13 N14 E14 N15 E15 N16 E16 N17 E17 N18 E18 N19 E19 N20 E20 N21 E21 N22 E22 N23 E23 N24 E24 N33
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2 E1 N1
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26 E25 N25
Y
X
Z
Node: 33
Disp X 2.852
Disp Y -0.05354
Disp Z 0
Status: Success: Disp. at ALR = 1.0000
Displacements
Rot X 0
Rot Y 0
Rot Z 0.01243
(10) 1.000
Apply
Cancel

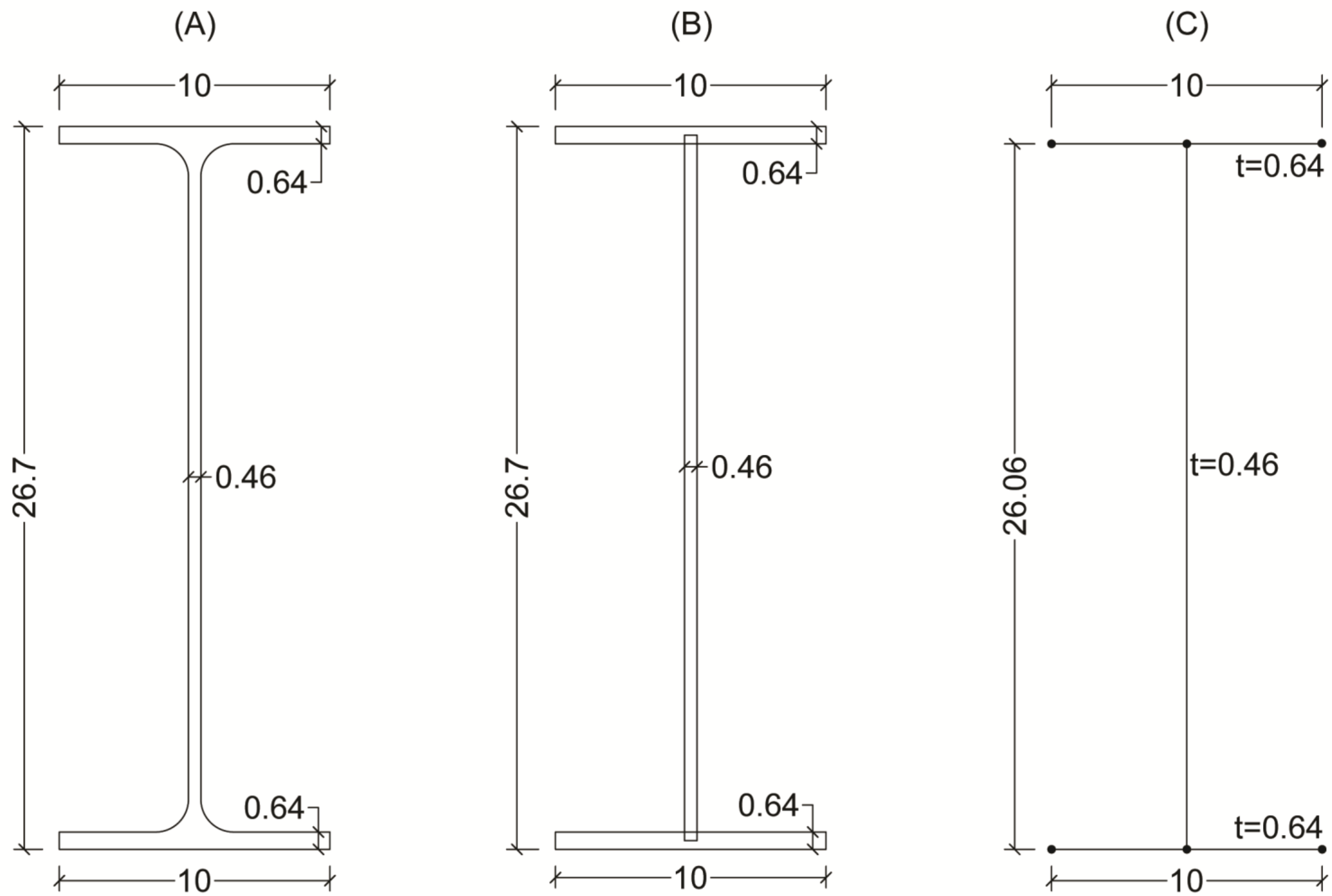
Section 5: Using MSASect

MSASect

The section properties used so far have been for doubly symmetric cross sections where we would have looked up the values or calculated them outside the program for ourselves. The updated version of MASTAN2 includes a new tool MSASect that can calculate section properties for thin wall cross sections. MSASect can be used with open and closed cross sections whether symmetric or not. In addition to the section properties used thus far, MSASect will calculate the necessary non-doubly symmetric section properties. The tool is found within the **Define Section** and **Modify Section** menu. As a demonstration, the section properties of a W27x84 cross section will be found.

Cross Section Geometry

The W27x84 cross section is shown below. Figure A illustrates the full cross section with fillets that is associated with the AISC table values. Figure B illustrates the simplified section with overlap and no fillets that represents the cross section to be calculated by MSASect. These are the dimensions to be entered when working with the template. Figure C illustrates the resulting node to node model created when using the template that will be used for calculations in MSASect.



Using MSA Sect

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** and new edit boxes and buttons should appear.

3) Click on **MSASect**.

4) As the I-beam cross-section is selected by default, click the edit box to the right of **B1=** and enter **10**. Repeat to define **B2=10**, **D=26.7**, **t1=0.64**, **t2=0.64**, and **t3=0.46**.

   Note: The dimensions to enter in the template correspond to Figure B on the previous page. While the section property calculations need to be completed using the dimensions shown in Figure C, this information is automatically generated based on the assumption that the numbers provided followed Figure B.

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

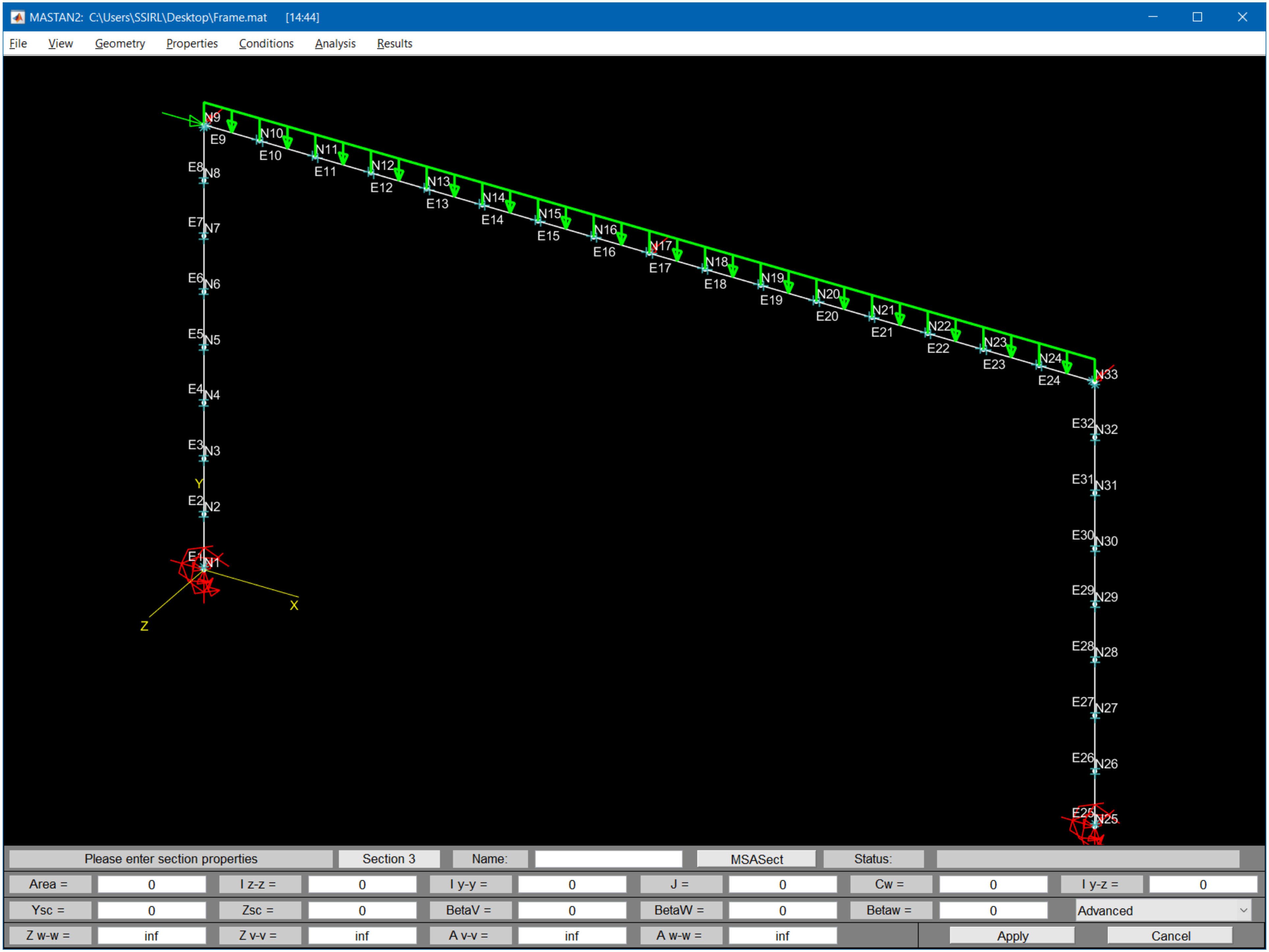
6) Click edit box to right of **Name:** and enter **W27x84Hand**.

7) Click **Export to MASTAN2** to copy values to main program.

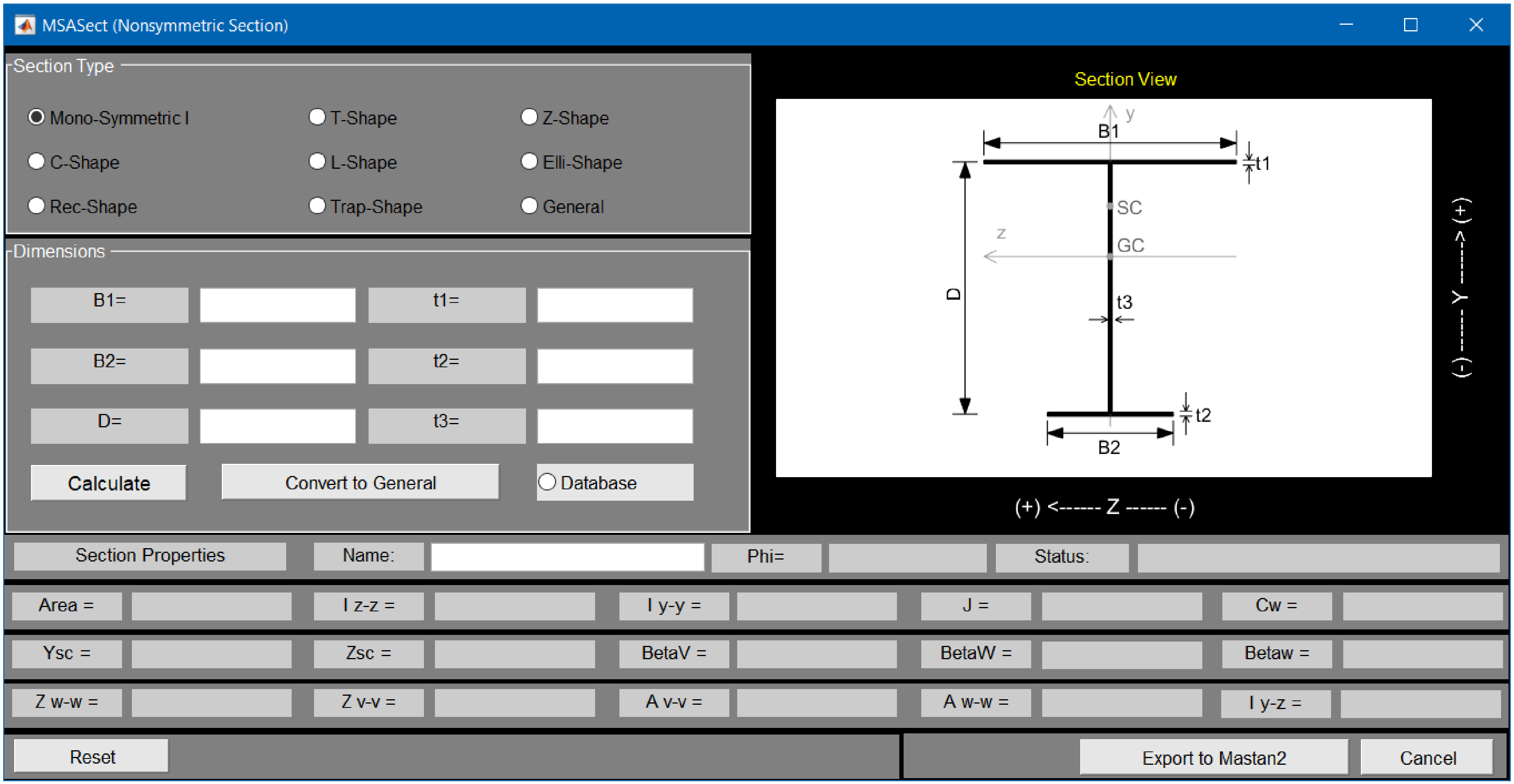
8) Click **Close** to return to the main window. There will often be a confirmation when closing it.

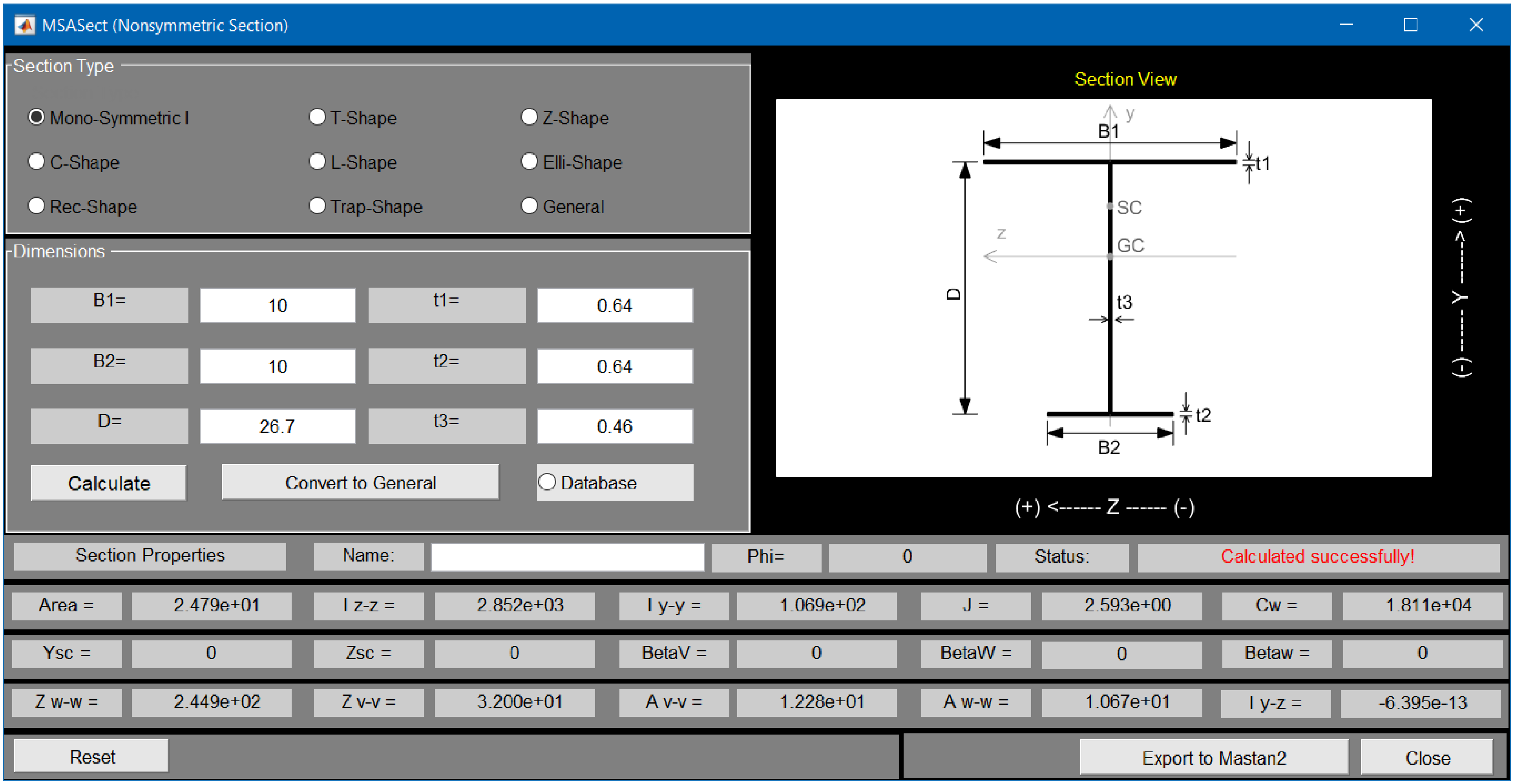
9) Click **Apply** to define Section 3.

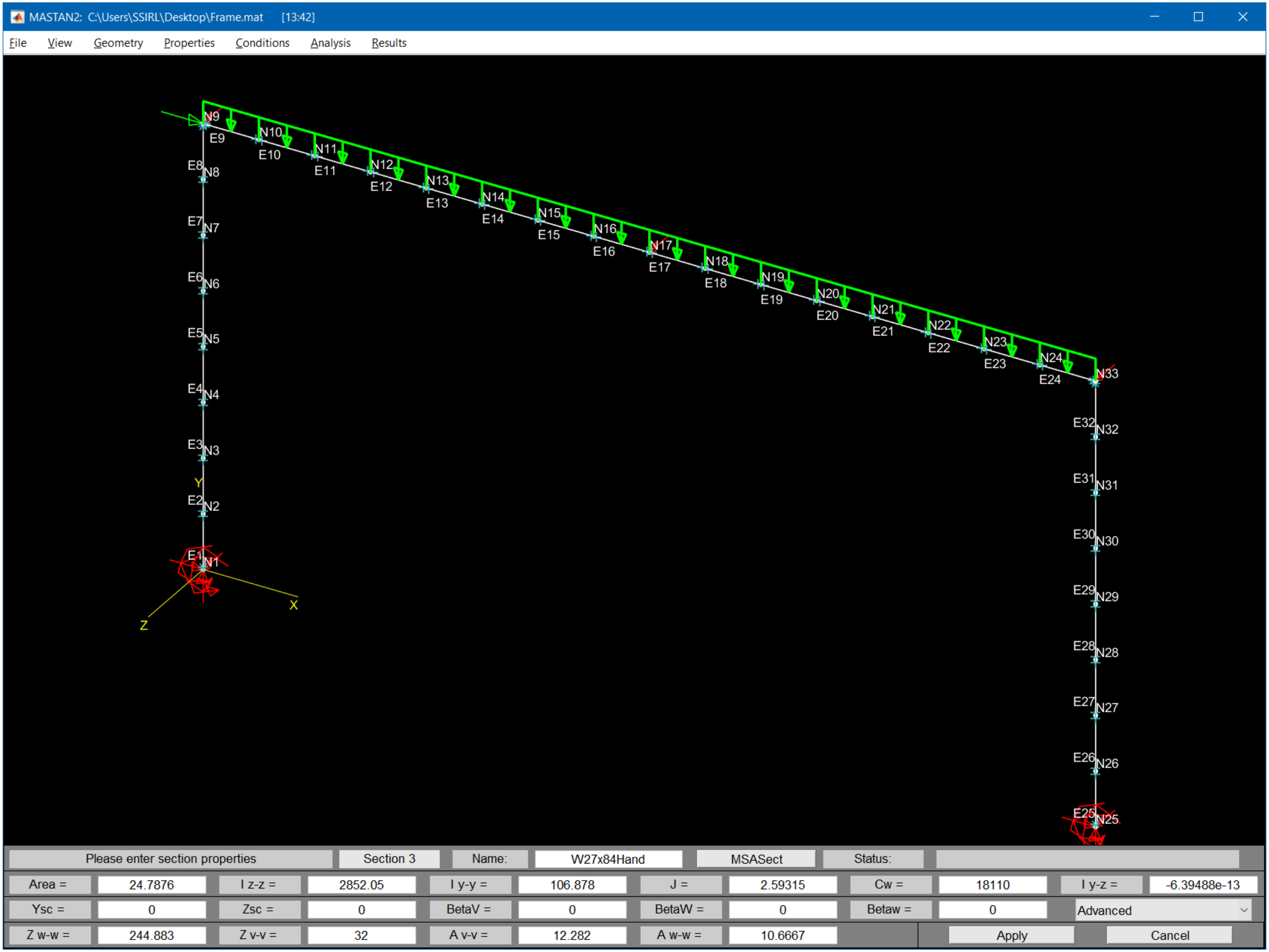
MASTAN2



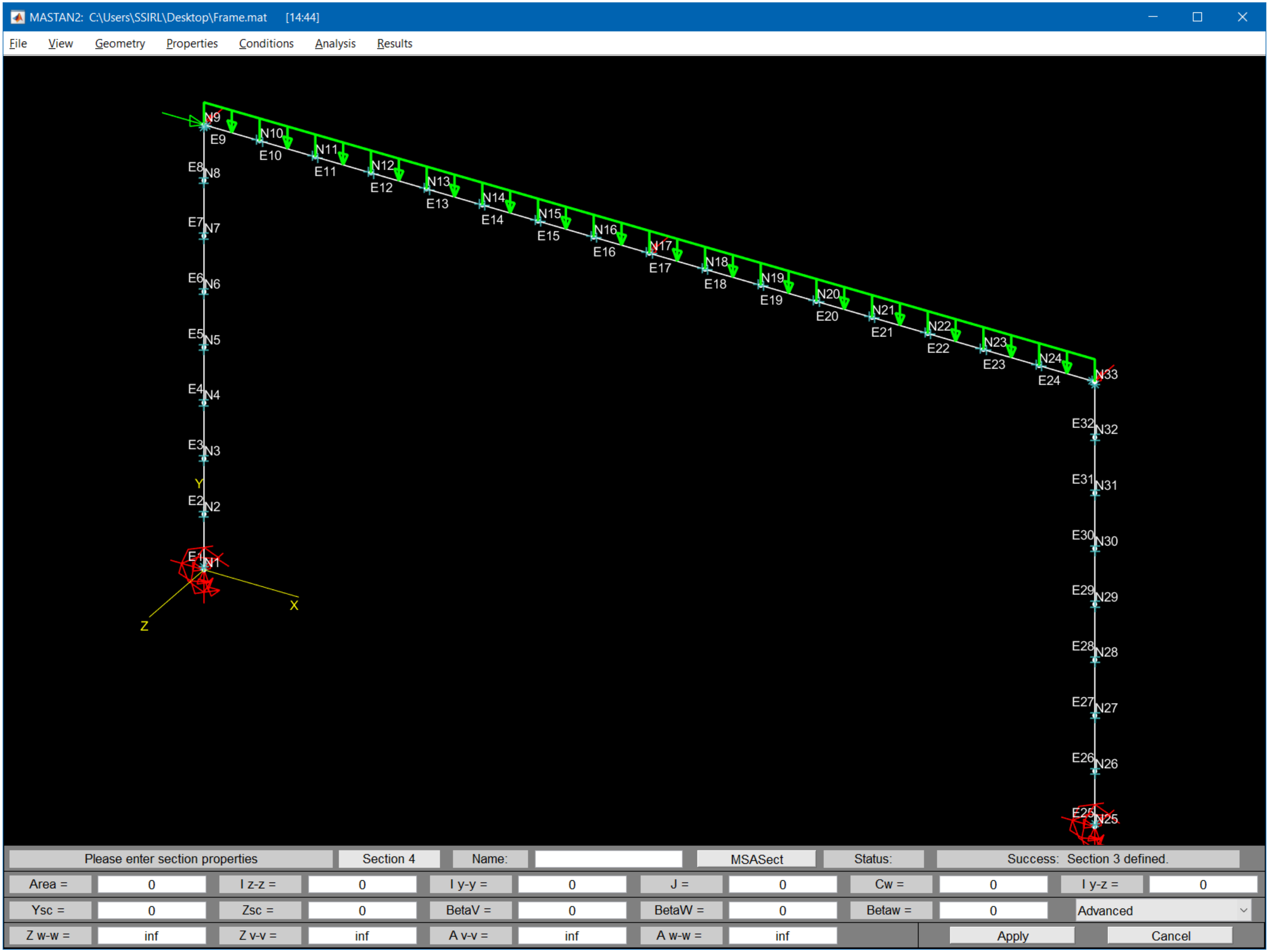
MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
N9
E9
N10
E10
N11
E11
N12
E12
N13
E13
N14
E14
N15
E15
N16
E16
N17
E17
N18
E18
N19
E19
N20
E20
N21
E21
N22
E22
N23
E23
N24
E24
N33
E8
N8
E7
N7
E6
N6
E5
N5
E4
N4
E3
N3
Y
E2
N2
E1
N1
X
Z
E32
N32
E31
N31
E30
N30
E29
N29
E28
N28
E27
N27
E26
N26
E25
N25
Please enter section properties
Section 3
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
B1=
t1=
B2=
t2=
D=
t3=
Calculate
Convert to General
Database
Section View
y
B1
t1
SC
z
GC
D
t3
t2
B2
(+) <------ 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
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
B1=
10
t1=
0.64
B2=
10
t2=
0.64
D=
26.7
t3=
0.46
Calculate
Convert to General
Database
Section View
y
B1
t1
SC
z
GC
D
t3
t2
B2
(+) <------ Y ------ (-)
(+) <------ Z ------ (-)
Section Properties
Name:
Phi=
0
Status:
Calculated successfully!
Area =
2.479e+01
I z-z =
2.852e+03
I y-y =
1.069e+02
J =
2.593e+00
Cw =
1.811e+04
Ysc =
0
Zsc =
0
BetaV =
0
BetaW =
0
Betaw =
0
Z w-w =
2.449e+02
Z v-v =
3.200e+01
A v-v =
1.228e+01
A w-w =
1.067e+01
I y-z =
-6.395e-13
Reset
Export to Mastan2
Close


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [13:42]
File
View
Geometry
Properties
Conditions
Analysis
Results
Please enter section properties
Section 3
Name:
W27x84Hand
MSASect
Status:
Area =
24.7876
I z-z =
2852.05
I y-y =
106.878
J =
2.59315
Cw =
18110
I y-z =
-6.39488e-13
Ysc =
0
Zsc =
0
BetaV =
0
BetaW =
0
Betaw =
0
Advanced
Z w-w =
244.883
Z v-v =
32
A v-v =
12.282
A w-w =
10.6667
Apply
Cancel

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N33
E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2 E1 N1
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26 E25 N25
Y
X
Z
Please enter section properties
Section 4
Name:
MSASect
Status:
Success: Section 3 defined.
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

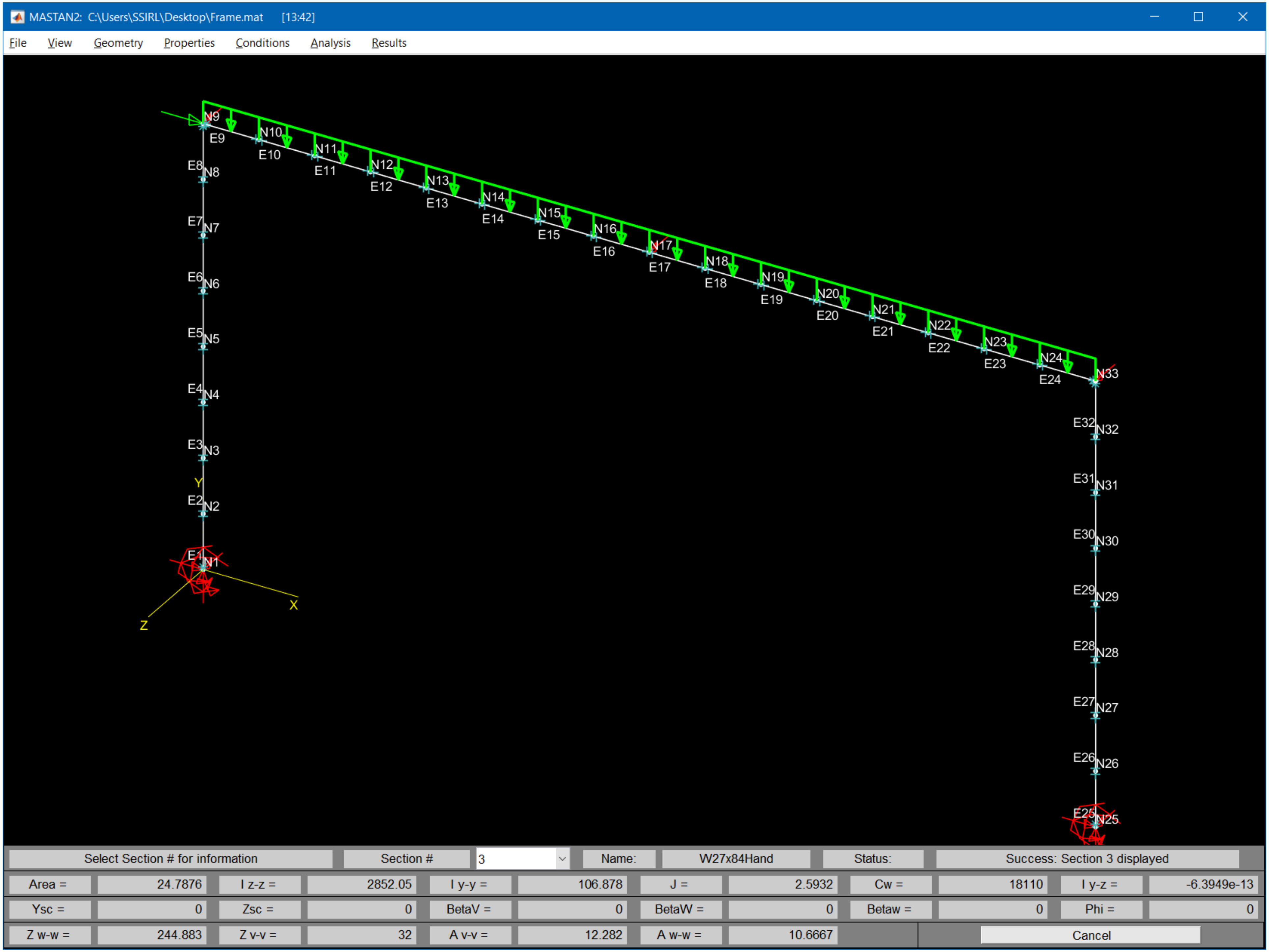
MSASect Results

1) From the **Properties** menu select **Information** and submenu option **Section**.

2) Change the **Section #** by clicking on the current section number just to the right to open a pop-up menu with all section numbers. Click on **2** to view the Section Properties based on the AISC database. Repeat with clicking on **3** to see the MSASect calculated values.

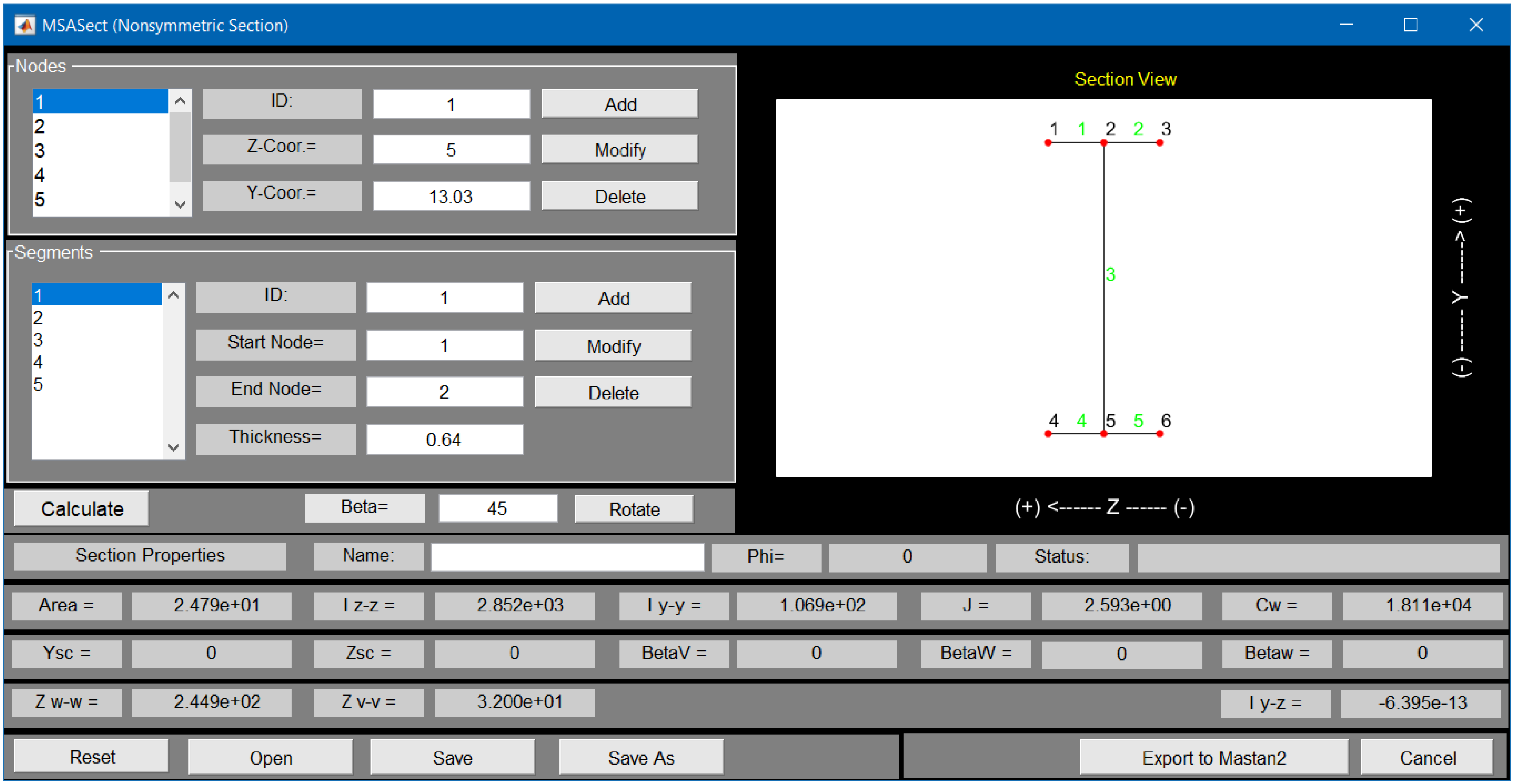
Property	Units	AISC	MSASect	Difference
A	in^2	24.7	24.79	0.4 %
Izz	in^4	2850	2852	0.1 %
Iyy	in^4	106	106.9	0.8 %
J	in^4	2.81	2.59	-7.7 %
Cw	in^6	18000	18110	0.6 %
Zzz	in^3	244	244.9	0.4 %
Zyy	in^3	33.2	32	-3.6 %

From the comparison of section properties from AISC and the values calculated by MSASect, most of the calculated properties match well. Take note that some of the template shapes calculate standard shear area values. To match the previous analysis, the **A v-v** and **A w-w** would need to be set to **inf**.


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [13:42]
File View Geometry Properties Conditions Analysis Results
Select Section # for information
Section # 3
Name: W27x84Hand
Status: Success: Section 3 displayed
Area = 24.7876
I z-z = 2852.05
I y-y = 106.878
J = 2.5932
Cw = 18110
I y-z = -6.3949e-13
Ysc = 0
Zsc = 0
BetaV = 0
BetaW = 0
Betaw = 0
Phi = 0
Z w-w = 244.883
Z v-v = 32
A v-v = 12.282
A w-w = 10.6667
Cancel

Using MSA Sect

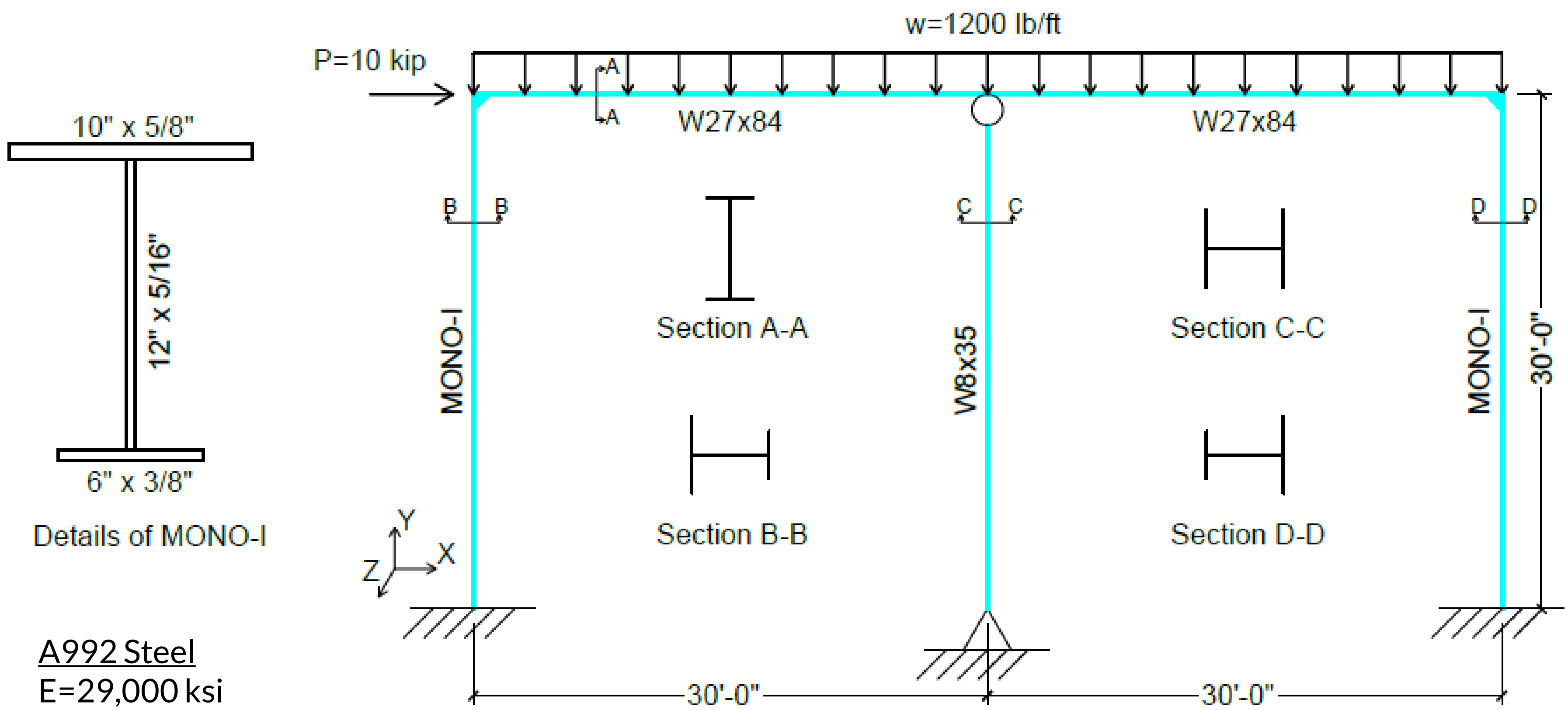
If one of the default cross sections does not cover your situation the General option allows for the input of nodes and line segments by the user. Clicking the radio button next to **General** and then the **Next** button will open an interface that allows for the input of nodes and line segments directly. If you want to verify the final node coordinates used or tweak a default geometry, click **Convert to General** to gain access to the list of nodes and line segments automatically created in the MSASect interface. The following is an example of what the W24x87 would look like. Note that the coordinates correspond with Figure C shown previously.


MSASect (Nonsymmetric Section)
Nodes
1
2
3
4
5
ID:
1
Add
Z-Coor.=
5
Modify
Y-Coor.=
13.03
Delete
Segments
1
2
3
4
5
ID:
1
Add
Start Node=
1
Modify
End Node=
2
Delete
Thickness=
0.64
Calculate
Beta=
45
Rotate
Section View
1 1 2 2 3
3
4 4 5 5 6
(+) <------ Y ------ (-)
(+) <------ Z ------ (-)
Section Properties
Name:
Phi=
0
Status:
Area =
2.479e+01
I z-z =
2.852e+03
I y-y =
1.069e+02
J =
2.593e+00
Cw =
1.811e+04
Ysc =
0
Zsc =
0
BetaV =
0
BetaW =
0
Betaw =
0
Z w-w =
2.449e+02
Z v-v =
3.200e+01
I y-z =
-6.395e-13
Reset
Open
Save
Save As
Export to Mastan2
Cancel

Section 6: Frame Analysis with Non-Doubly Symmetric Sections

Problem Description - Figure

The frame is constructed of A992 steel with the properties indicated. The frame is also supported out of plane in the Z direction on the beam at the column locations. The outer columns and top beam are assumed to be fixed for warping at the end. The beam is also continuous for warping over the middle column. The middle column is assumed to be free to warp at each end.



A992 Steel
E=29,000 ksi
f_y=50 ksi

Adding Interior Column

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

2) At the bottom menu bar, click in the edit box to the right of **x =** and enter **360**. 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**.

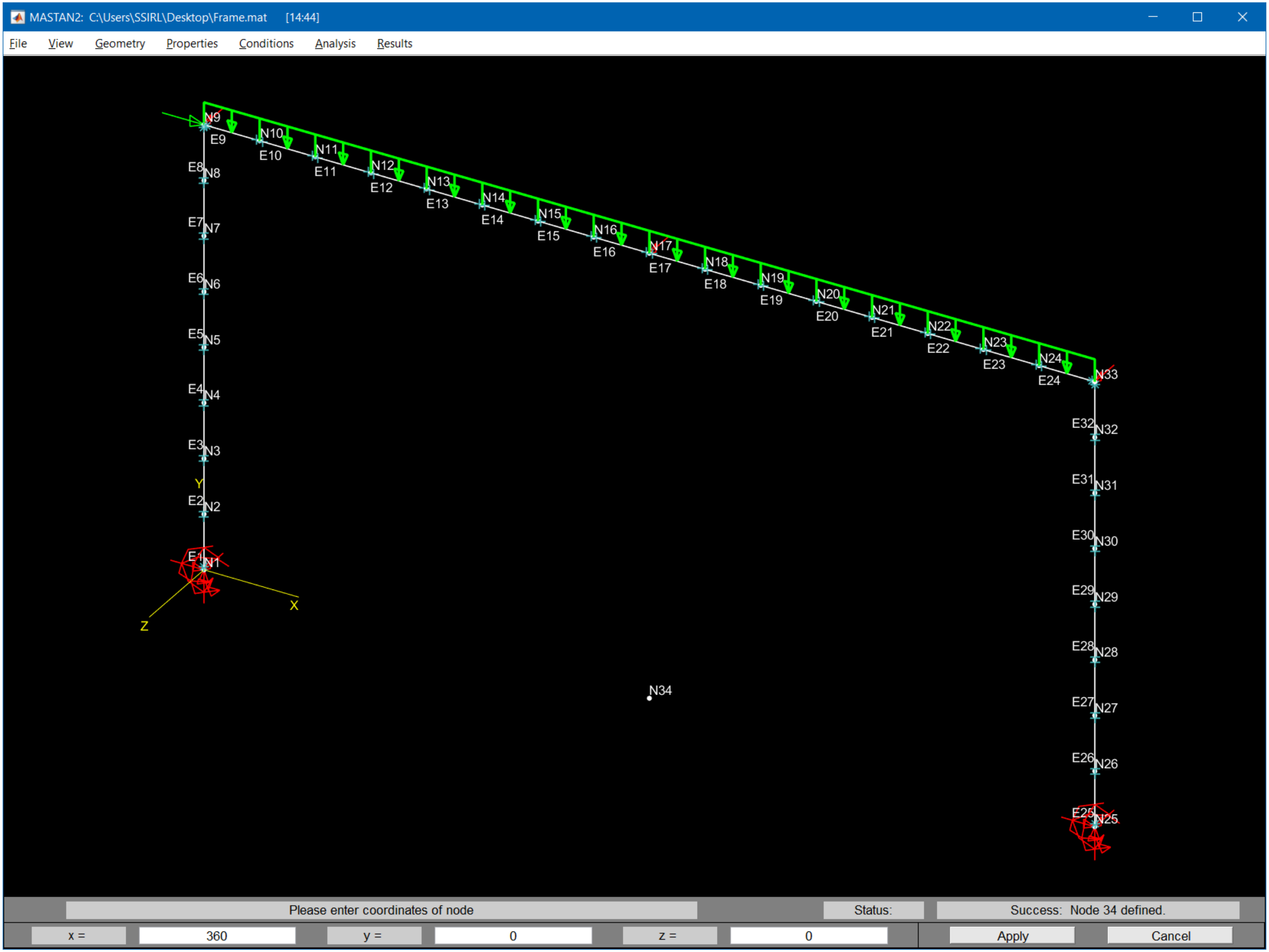
3) Click on the **Apply** Button.

4) From the **Geometry** menu select **Define Element**.

5) On the model, click the newly created node to define Node i. Then click the middle node of the top beam to define Node j. These nodes should be **34** and **17**, respectively.

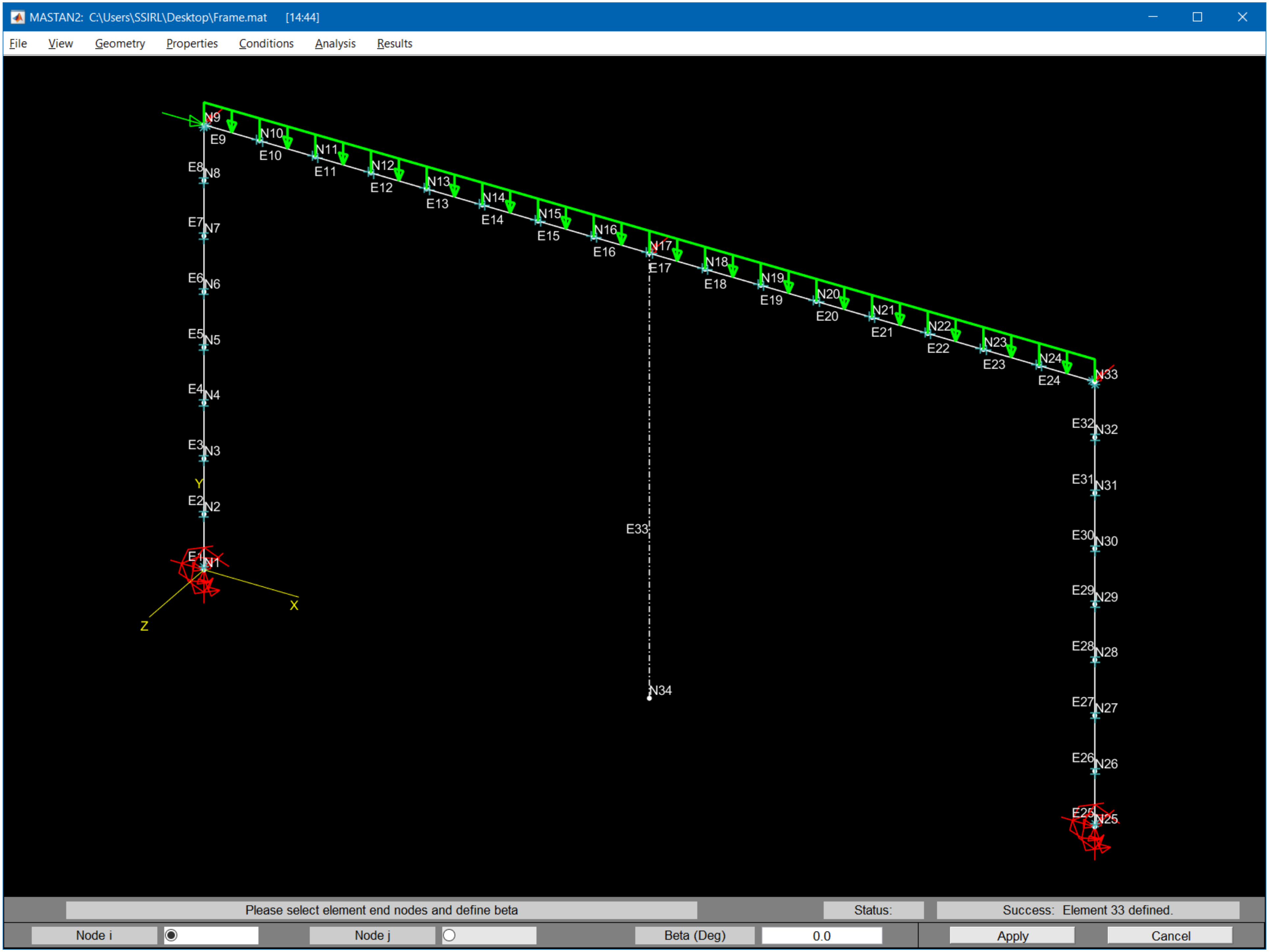
6) Click on the **Apply** Button.

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
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
N28
N29
N30
N31
N32
N33
N34
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
E28
E29
E30
E31
E32
X
Y
Z
Please enter coordinates of node
Status:
Success: Node 34 defined.
x =
360
y =
0
z =
0
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
N9
E9
N10
E10
N11
E11
N12
E12
N13
E13
N14
E14
N15
E15
N16
E16
N17
E17
N18
E18
N19
E19
N20
E20
N21
E21
N22
E22
N23
E23
N24
E24
N33
E8 N8
E7 N7
E6 N6
E5 N5
E4 N4
E3 N3
Y
E2 N2
E1 N1
X
Z
E33
N34
E32 N32
E31 N31
E30 N30
E29 N29
E28 N28
E27 N27
E26 N26
E25 N25
Please select element end nodes and define beta
Status:
Success: Element 33 defined.
Node i
Node j
Beta (Deg)
0.0
Apply
Cancel

Applying Material and Section Properties

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

2) Create the list of elements to be assigned the properties of Material 1 by clicking on the new column. Click on the **Apply** button. (Note that elements with assigned just material properties turn dotted.)

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

4) At the bottom menu bar, click on the **Database** button.

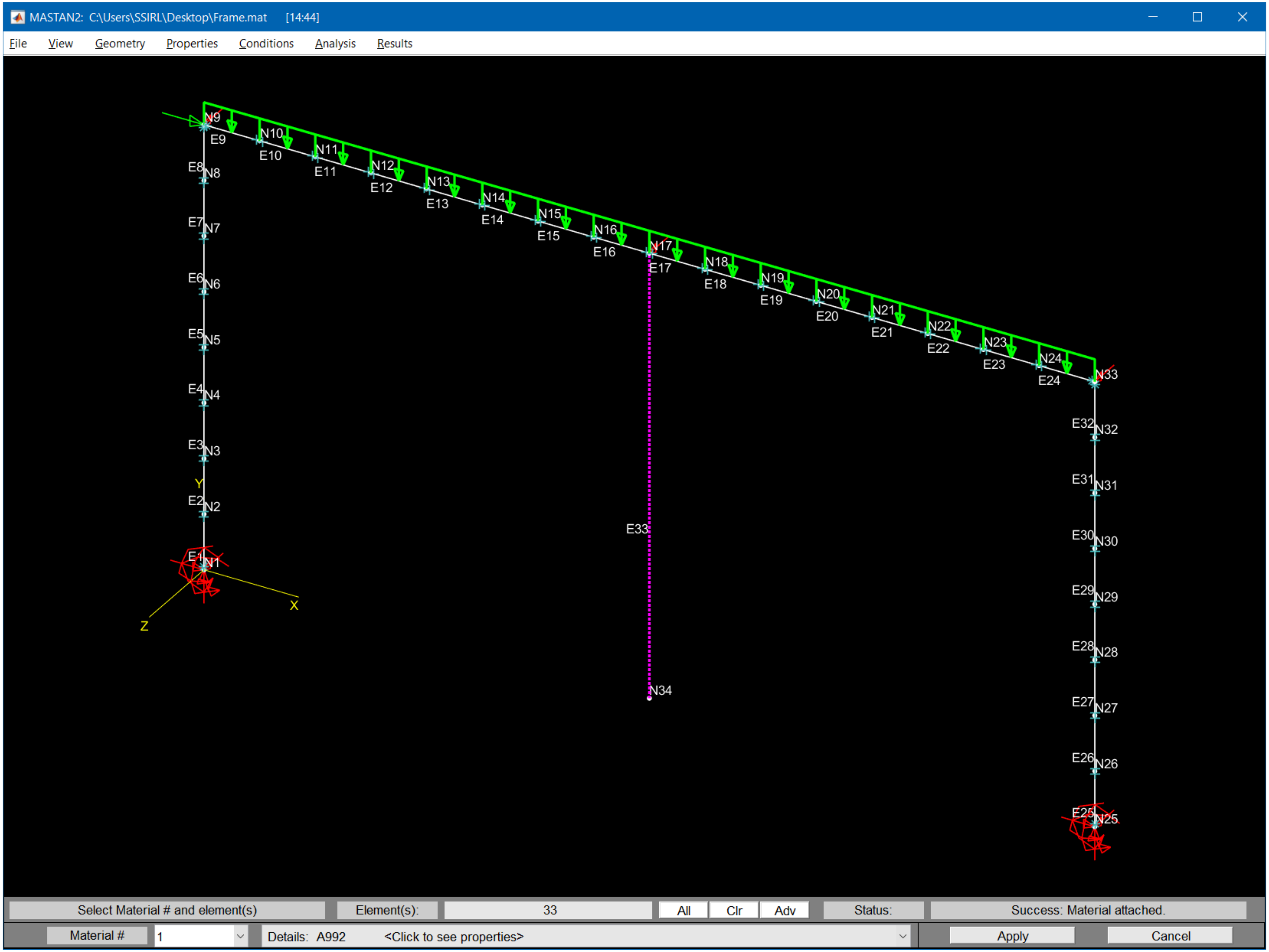
5) In the pop-up menu, scroll to find section **W8x35** and click on it. Then click on the **Apply** button. (Section 4 is now defined with the properties of W8x35.)

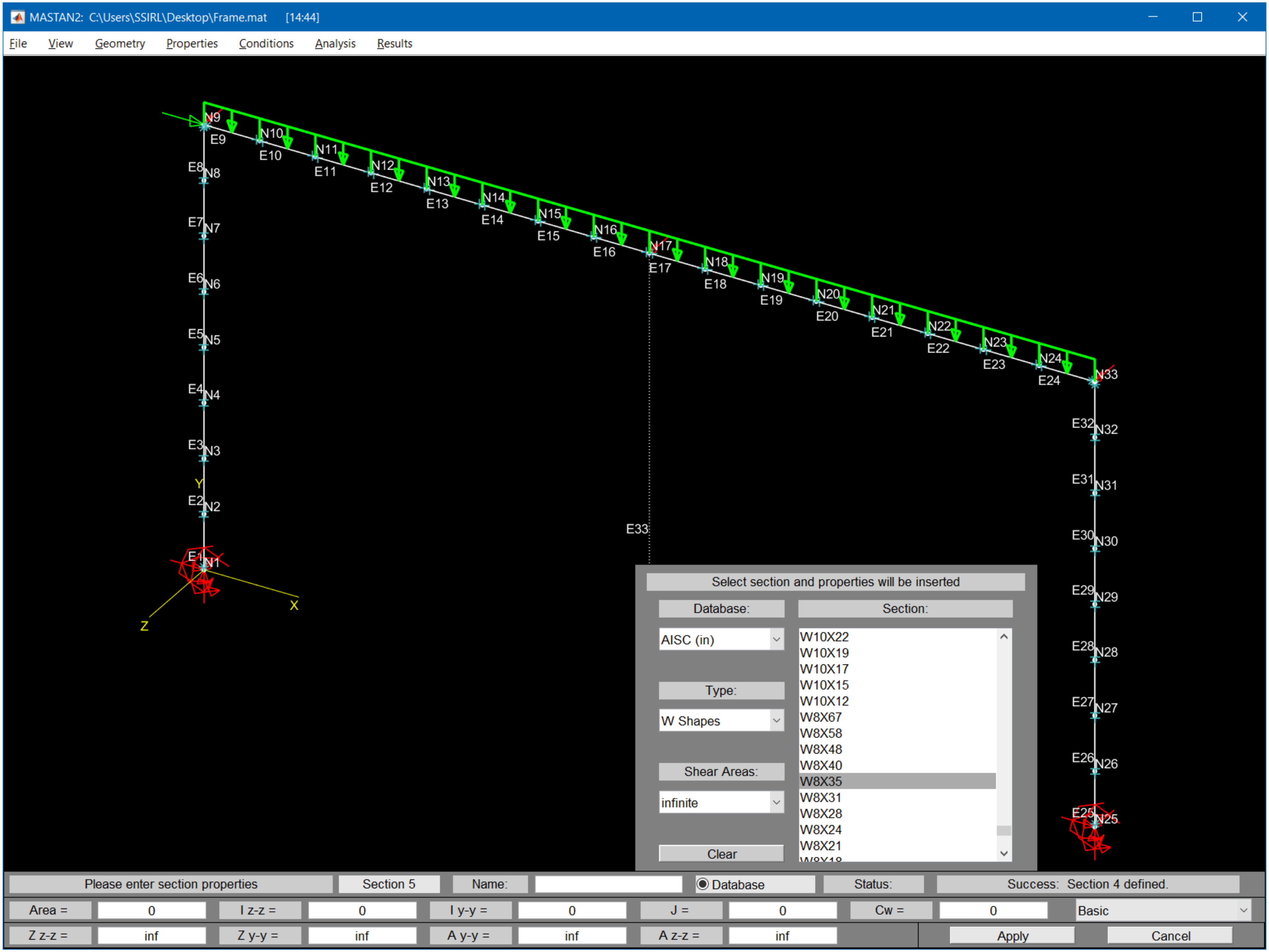
6) From the **Properties** menu select **Attach Section**.

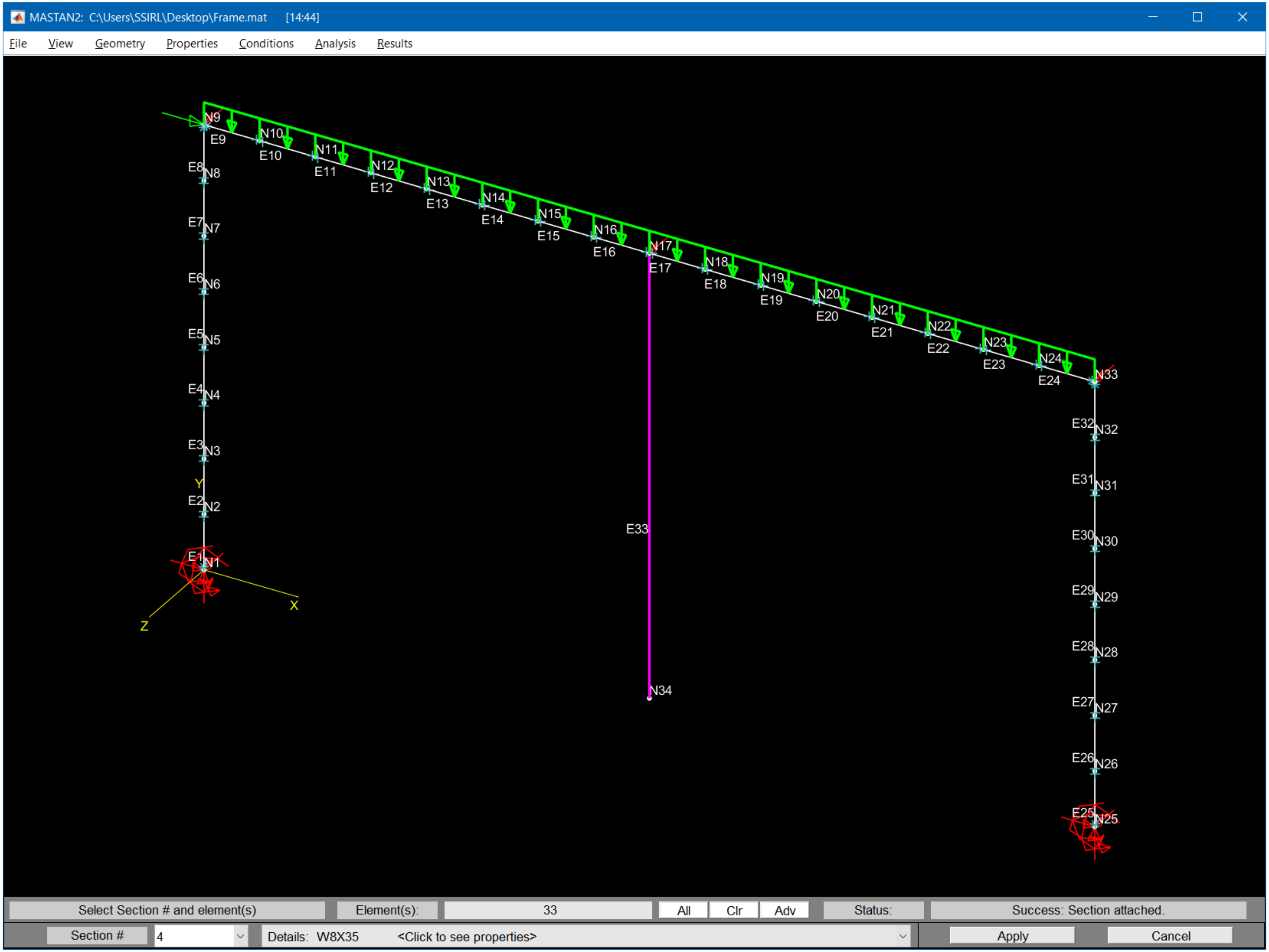
7) Create the list of elements to be assigned the properties of Section 4 by clicking on the new column, element **33**.

8) Change the **Section #** by clicking on the current section number just to the right to open a pop-up menu with all section numbers. Click on **4** to select Section #4, W8x35.

9) Assign Section 4 properties by clicking the **Apply** button.


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N33
E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24
E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2 E1 N1
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26 E25 N25
E33
N34
Y
X
Z
Select Material # and element(s)
Element(s):
33
All
Clr
Adv
Status:
Success: Material attached.
Material #
1
Details: A992 <Click to see properties>
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
Select section and properties will be inserted
Database:
AISC (in)
Type:
W Shapes
Shear Areas:
infinite
Clear
Section:
W10X22
W10X19
W10X17
W10X15
W10X12
W8X67
W8X58
W8X48
W8X40
W8X35
W8X31
W8X28
W8X24
W8X21
Please enter section properties
Section 5
Name:
Database
Status:
Success: Section 4 defined.
Area = 0
I z-z = 0
I y-y = 0
J = 0
Cw = 0
Basic
Z z-z = inf
Z y-y = inf
A y-y = inf
A z-z = inf
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
N9
N10
N11
N12
N13
N14
N15
N16
N17
N18
N19
N20
N21
N22
N23
N24
N33
E9
E10
E11
E12
E13
E14
E15
E16
E17
E18
E19
E20
E21
E22
E23
E24
E8 N8
E7 N7
E6 N6
E5 N5
E4 N4
E3 N3
E2 N2
E1 N1
Y
X
Z
E33
N34
E32 N32
E31 N31
E30 N30
E29 N29
E28 N28
E27 N27
E26 N26
E25 N25
Select Section # and element(s)
Element(s):
33
All
Clr
Adv
Status:
Success: Section attached.
Section #
4
Details: W8X35 <Click to see properties>
Apply
Cancel

Element Modification

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

2) Create the list of elements by clicking on the new column.

3) Click the **>** box to the right of **# of Segments =** to increase **2** to **8**.

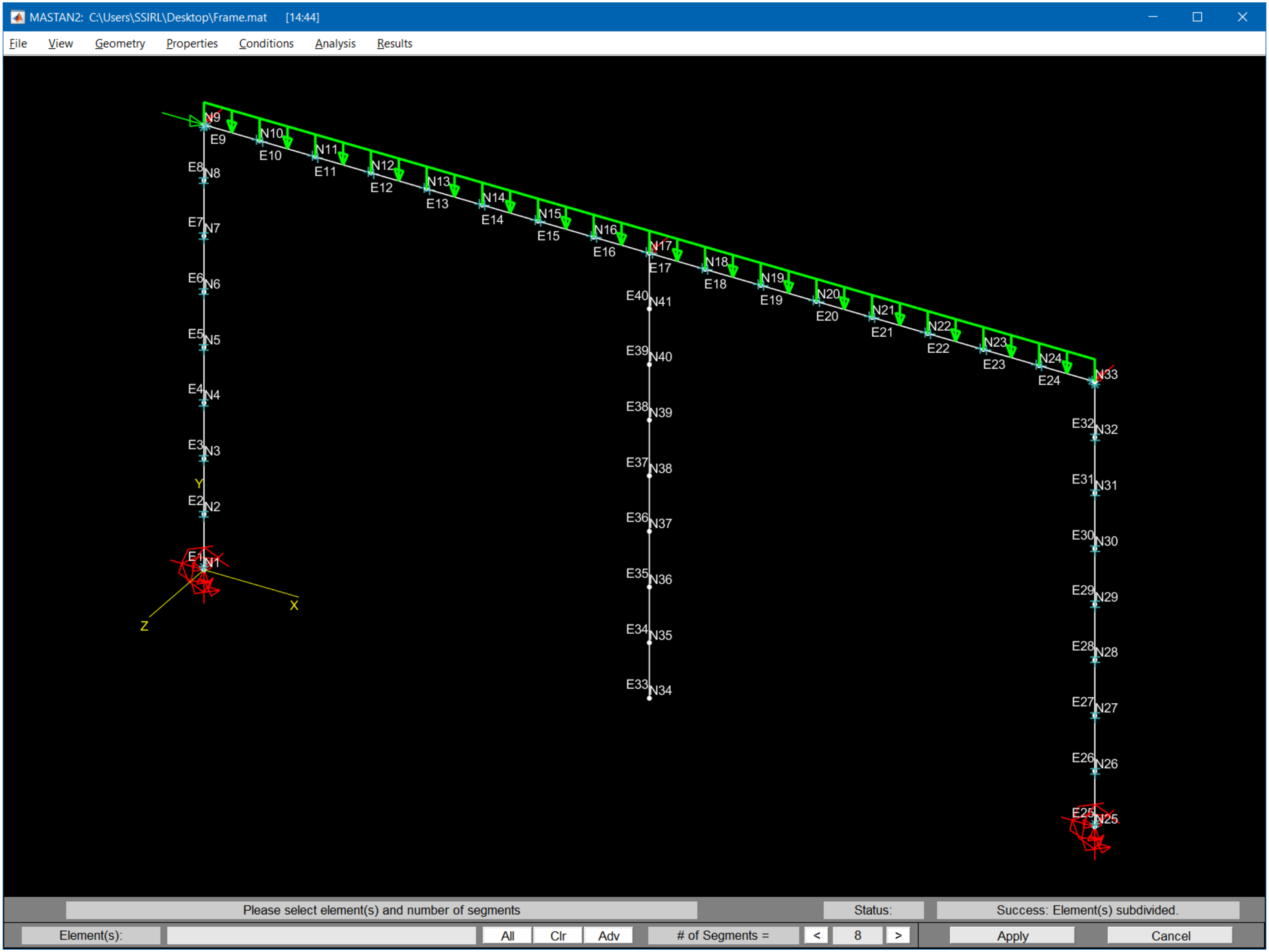
4) Click on the **Apply** button. (Note that same the section and material property information is given to all new elements.)

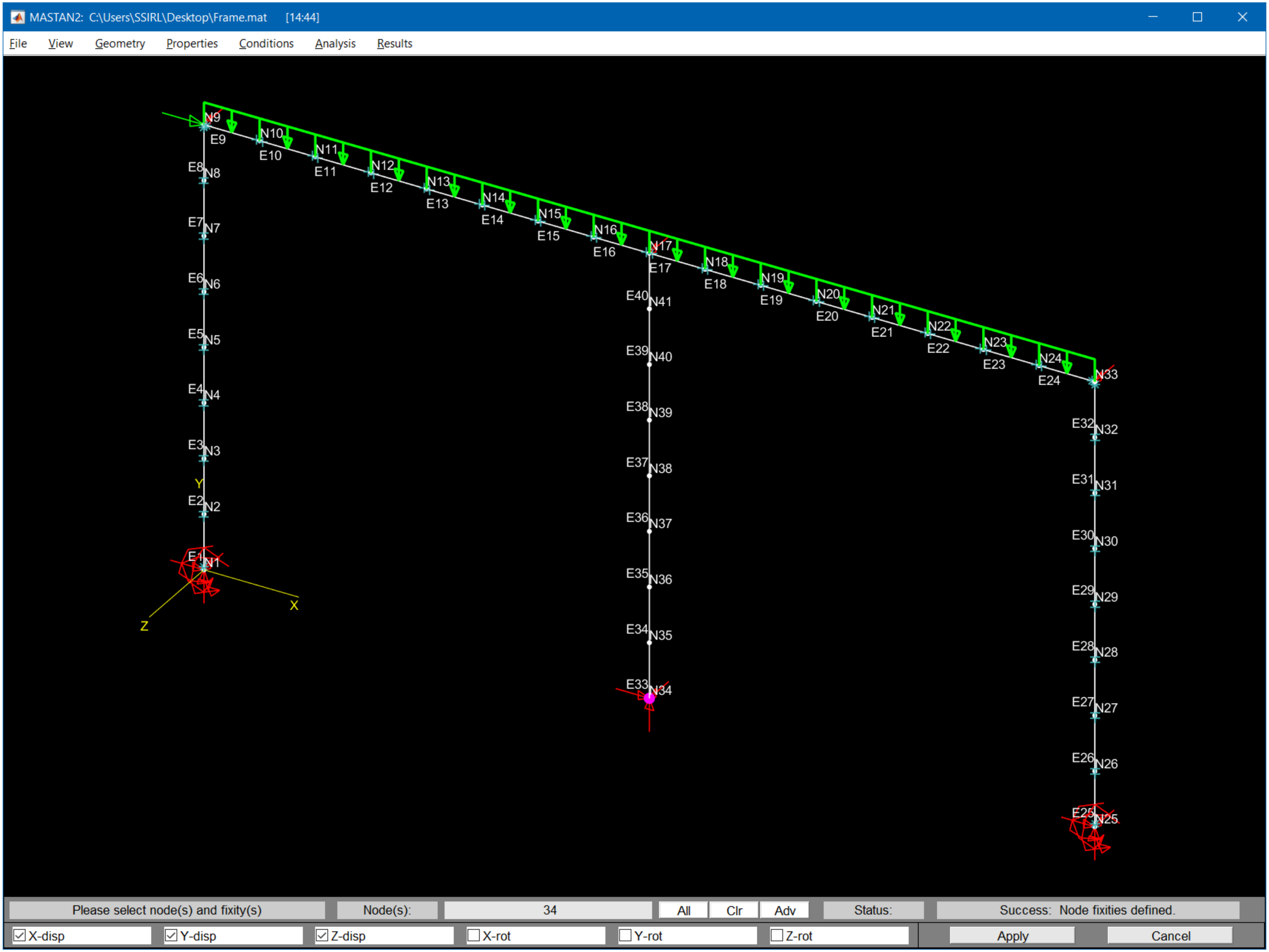
5) From the **Conditions** menu select Define **Fixities**.

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

7) Create the list of nodes to be assigned this fixity by clicking on the middle bottom node, **34**.

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


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
Please select element(s) and number of segments
Status:
Success: Element(s) subdivided.
Element(s):
All
Clr
Adv
# of Segments =
<
8
>
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
Please select node(s) and fixity(s)
Node(s):
34
All
Clr
Adv
Status:
Success: Node fixities defined.
X-disp
Y-disp
Z-disp
X-rot
Y-rot
Z-rot
Apply
Cancel

Warping Continuity

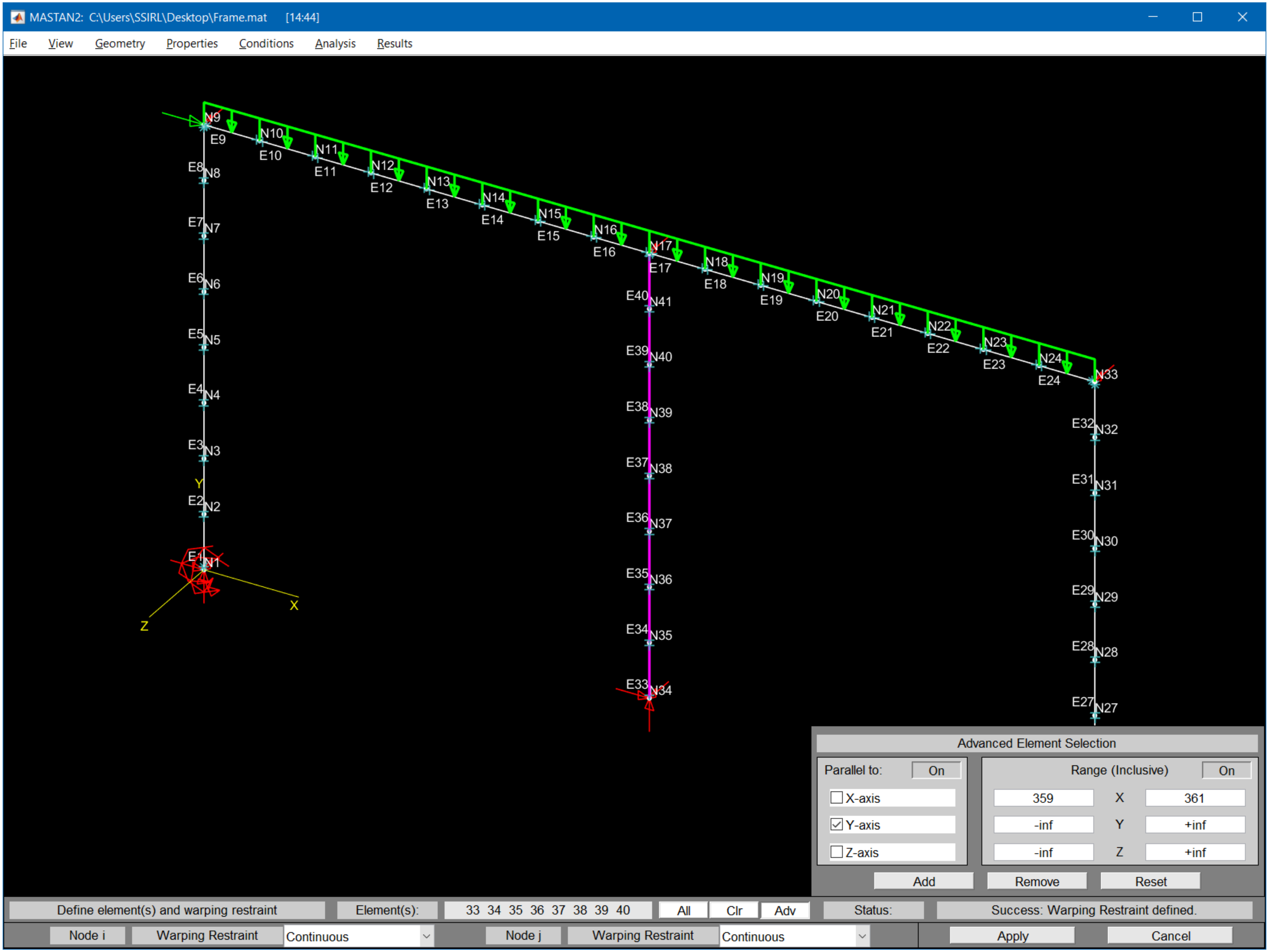
1) From the **Geometry** menu select **Define Connections** and submenu option **Torsion**.

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) Use the buttons to the right of **Element(s)**: to make the list of elements. Click the **Adv** button to open the pop-up menu. To select all the middle column elements, click the **Off** button to the right of **Range (Inclusive)** to turn this tool **On**. Click the edit box to the left of **X** and change **–Inf** to **359**. Click the edit box to the right of **X** and change **Inf** to **361**.

4) Click **Add** to add all these elements to the element list. Click on the **Apply** button to assign continuous warping.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
Advanced Element Selection
Parallel to: On
X-axis
Y-axis
Z-axis
Range (Inclusive) On
359 X 361
-inf Y +inf
-inf Z +inf
Add Remove Reset
Define element(s) and warping restraint
Element(s):
33 34 35 36 37 38 39 40
All Clr Adv
Status:
Success: Warping Restraint defined.
Node i Warping Restraint Continuous
Node j Warping Restraint Continuous
Apply Cancel

Warping Boundary Conditions

1) Click **Adv** to close the pop-up menu.

2) Click **Clr** to empty the list of elements. Click on the bottom element of the middle column to define the member that start with warping free and is continuous.

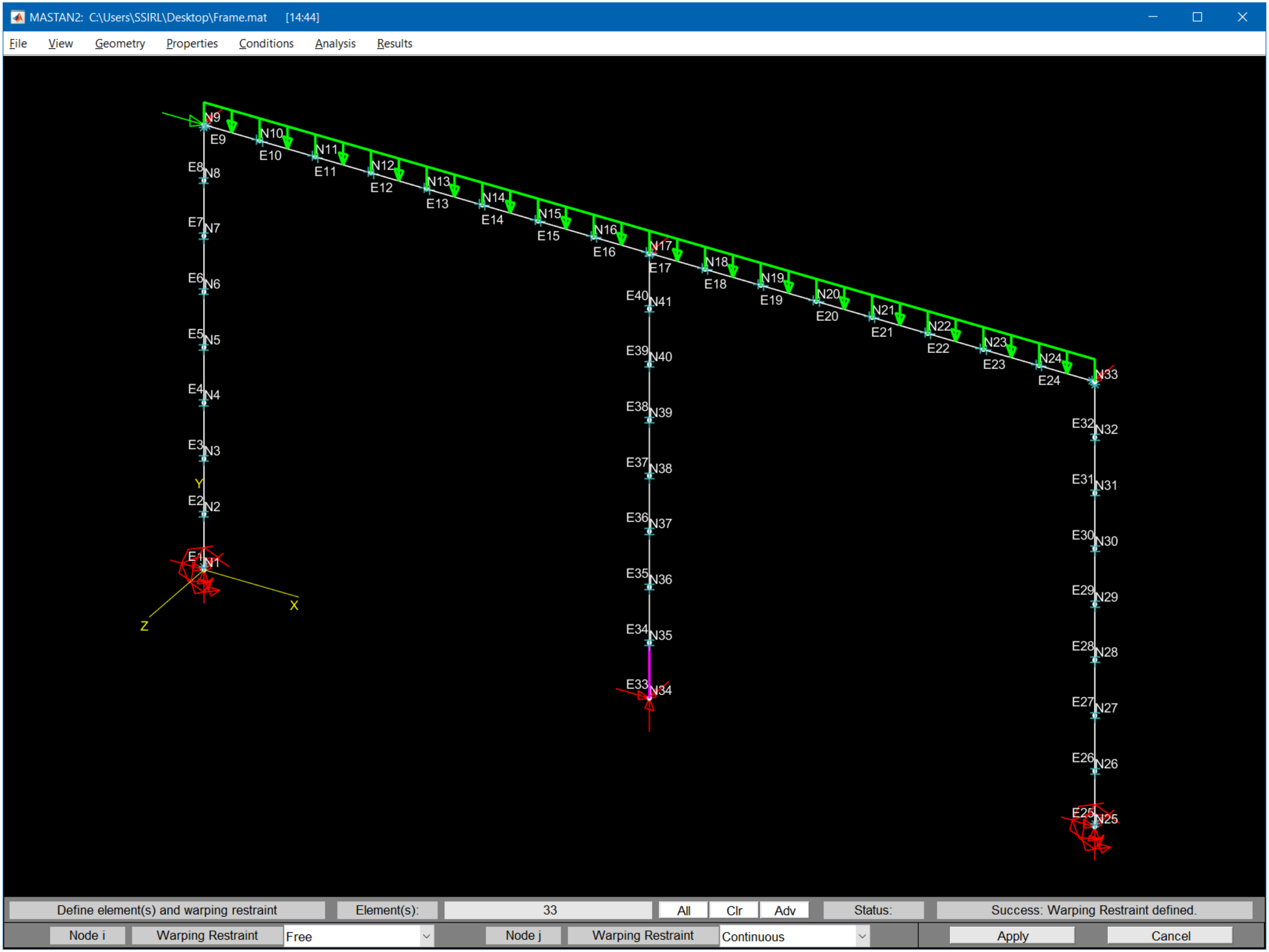
3) Click on the menu to the right of **Warping Restraint for Node i** and set the value to **Free**. Node j is set from the previous step.

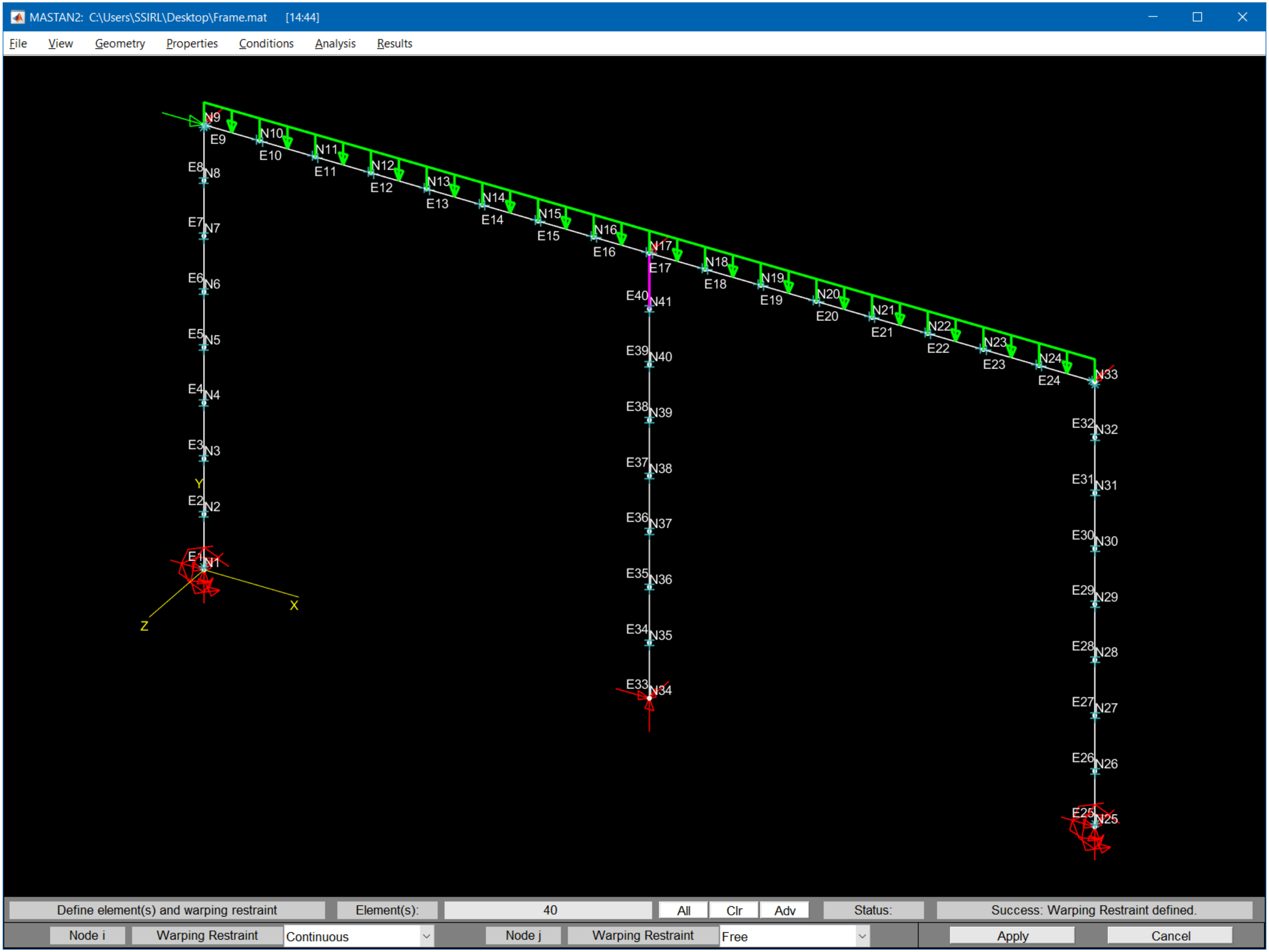
4) Click on the **Apply** button.

5) Click **Clr** to empty the list of elements. Click on the top element of the middle column.

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 **Free**.

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


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
Define element(s) and warping restraint
Element(s):
33
All
Clr
Adv
Status:
Success: Warping Restraint defined.
Node i
Warping Restraint
Free
Node j
Warping Restraint
Continuous
Apply
Cancel



MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
Define element(s) and warping restraint
Element(s):
40
All
Clr
Adv
Status:
Success: Warping Restraint defined.
Node i
Warping Restraint
Continuous
Node j
Warping Restraint
Free
Apply
Cancel

End Moment Release

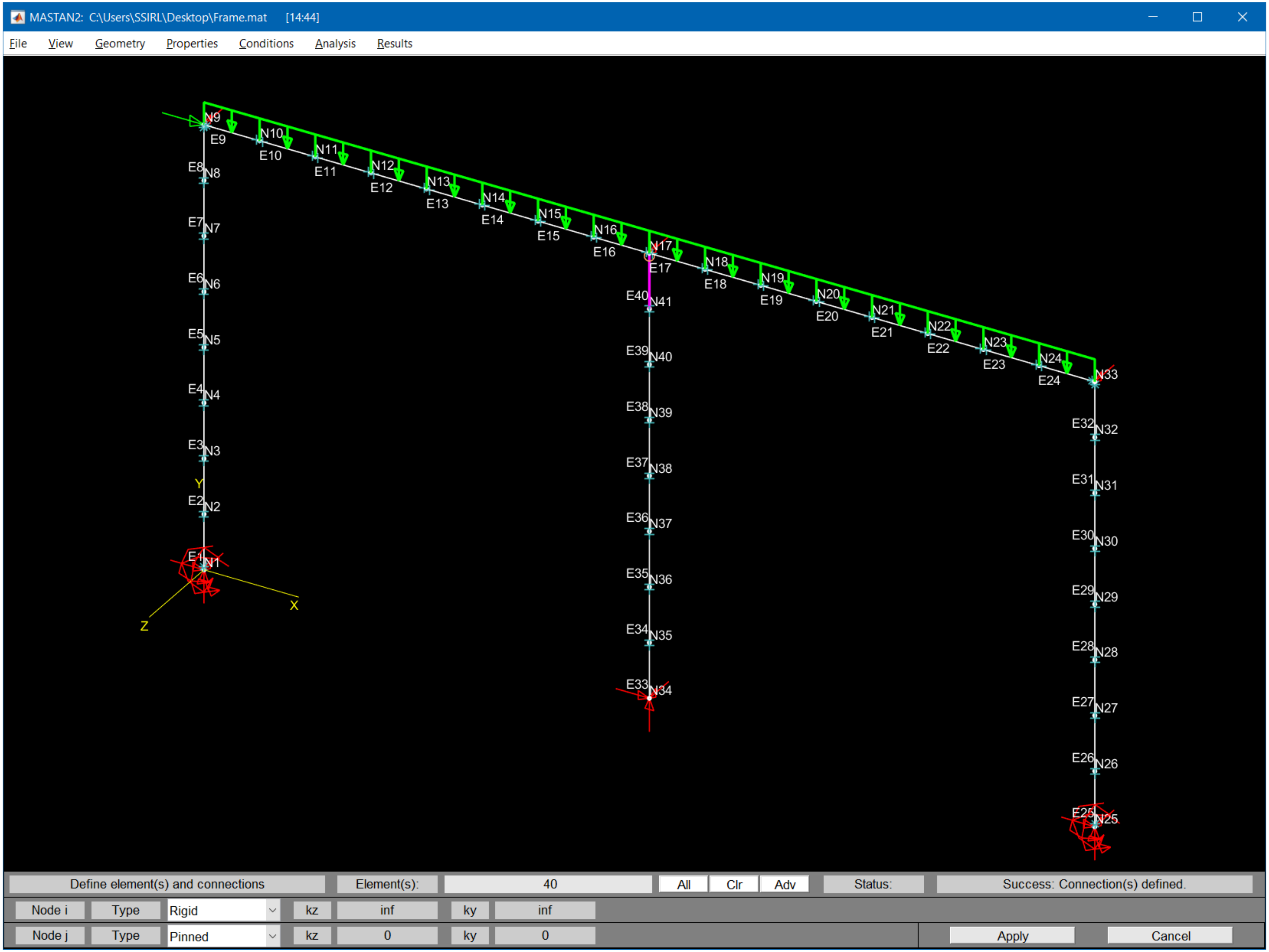
1) From the **Geometry** menu select **Define Connections** and submenu option **Flexure**.

2) At the bottom menu bar, click on the menu to the right of **Type** for **Node j** and set the value to **Pinned**.

3) Create the list of elements by clicking on the top element of the middle column.

4) Click on the **Apply** button to apply the pin connection. Note the orange circle is displayed to signify the end that has the Mx and My moment released. Torsion cannot be released.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
Define element(s) and connections
Element(s):
40
All
Clr
Adv
Status:
Success: Connection(s) defined.
Node i
Type
Rigid
kz
inf
ky
inf
Node j
Type
Pinned
kz
0
ky
0
Apply
Cancel

Modifying Section Properties

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

2) At the bottom menu bar, Section #**1** should be selected already. Click on the pop-up menu on the far right that current displays **Basic**. Click on **Advanced**.

3) Click on **MSASect**.

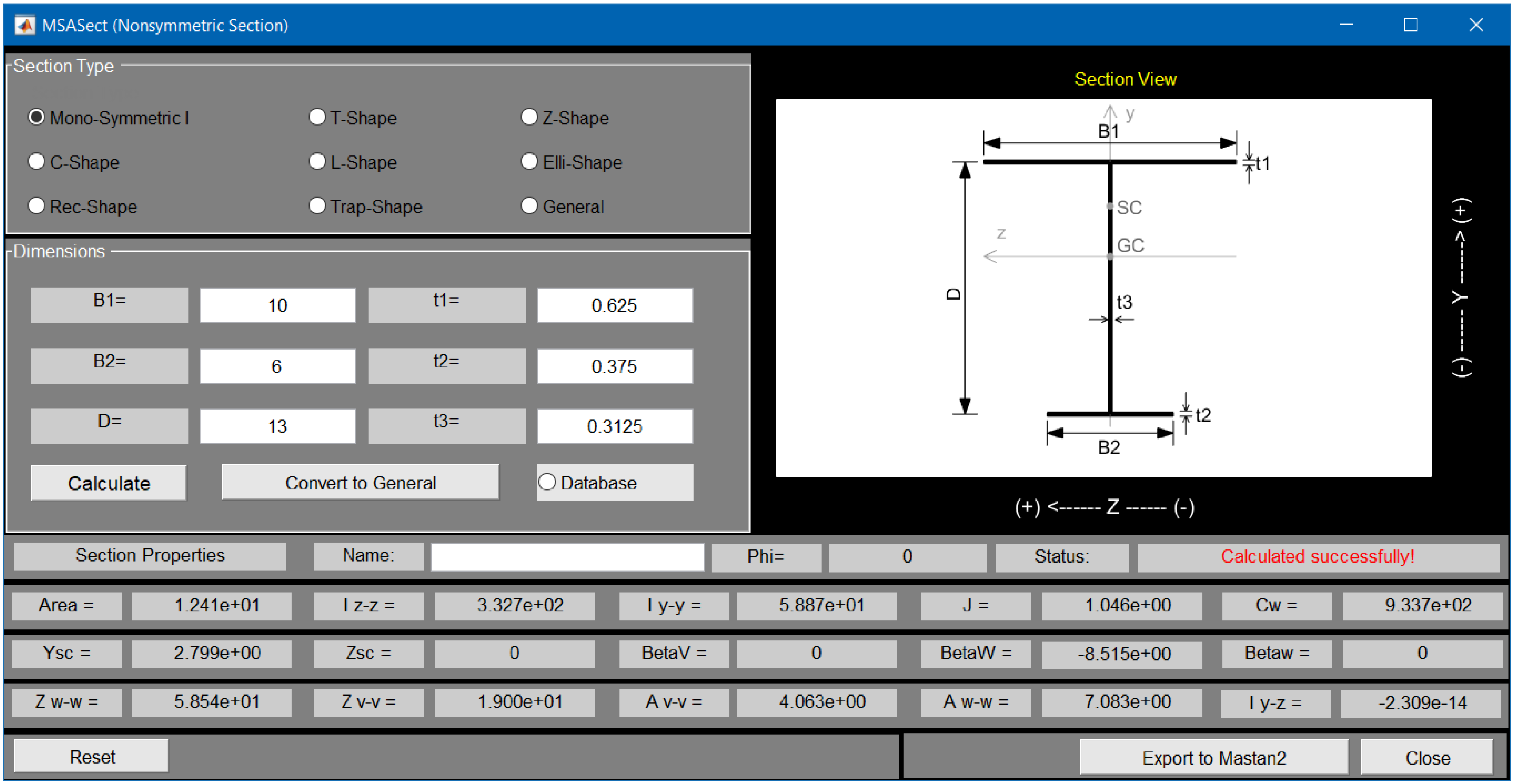
4) As the I-beam cross-section is selected by default, click the edit box to the right of **B1=** and enter **10**. Repeat to define **B2=6**, **D=13**, **t1=0.625**, **t2=0.375**, and **t3=0.3125**.

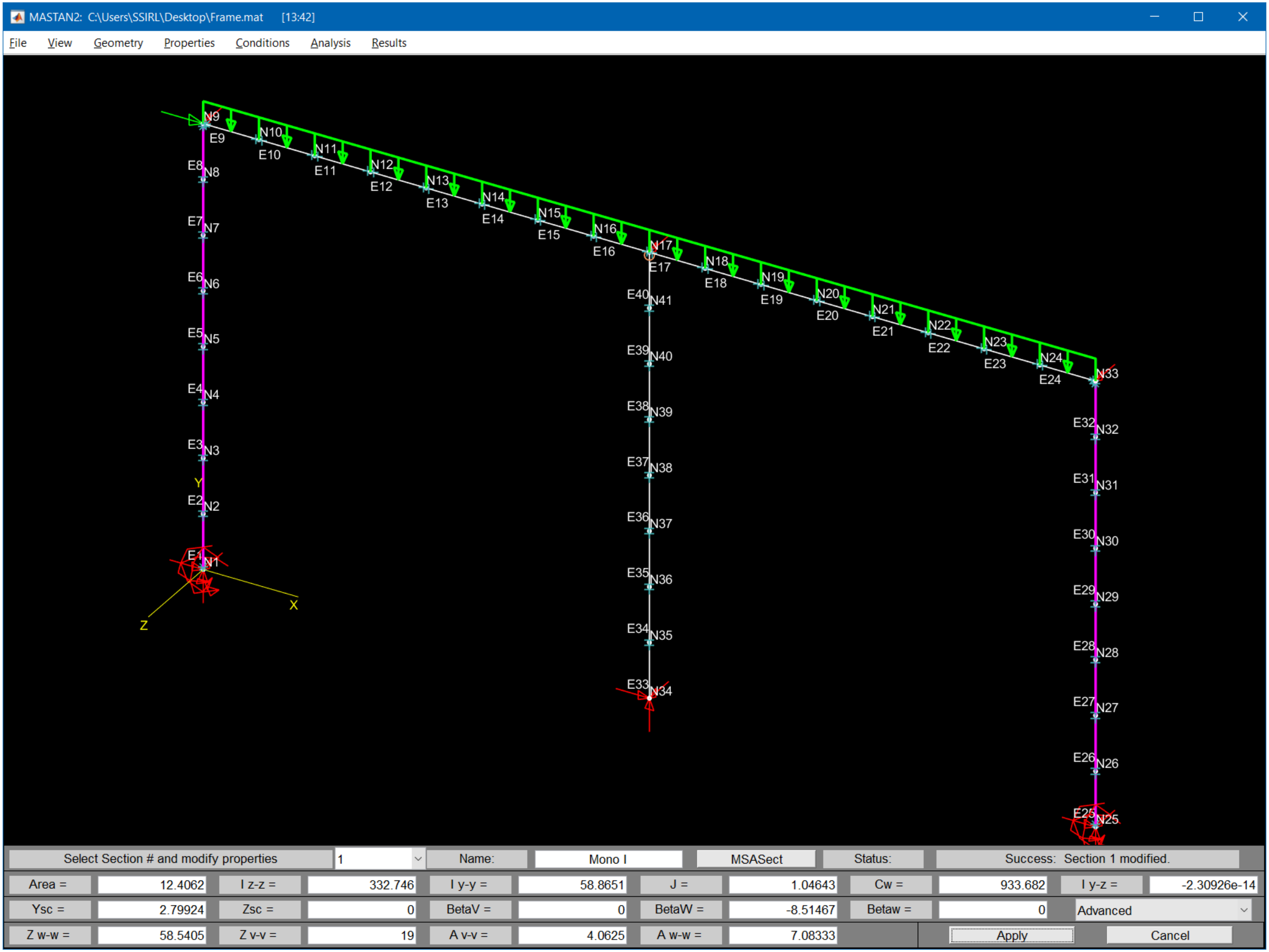
5) Click **Calculate** to determine the properties.

6) Click edit box to right of **Name:** and enter **Mono I**

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

8) Click **Apply** to modify Section 1.


MSASect (Nonsymmetric Section)
Section Type
Mono-Symmetric I
T-Shape
Z-Shape
C-Shape
L-Shape
Elli-Shape
Rec-Shape
Trap-Shape
General
Dimensions
B1= 10
t1= 0.625
B2= 6
t2= 0.375
D= 13
t3= 0.3125
Calculate
Convert to General
Database
Section View
y
B1
t1
SC
z
GC
D
t3
t2
B2
(+) <------ Y ------ (-)
(+) <------ Z ------ (-)
Section Properties
Name:
Phi= 0
Status: Calculated successfully!
Area = 1.241e+01
I z-z = 3.327e+02
I y-y = 5.887e+01
J = 1.046e+00
Cw = 9.337e+02
Ysc = 2.799e+00
Zsc = 0
BetaV = 0
BetaW = -8.515e+00
Betaw = 0
Z w-w = 5.854e+01
Z v-v = 1.900e+01
A v-v = 4.063e+00
A w-w = 7.083e+00
I y-z = -2.309e-14
Reset
Export to Mastan2
Close


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [13:42]
File View Geometry Properties Conditions Analysis Results
Select Section # and modify properties
1
Name:
Mono I
MSASect
Status:
Success: Section 1 modified.
Area = 12.4062
I z-z = 332.746
I y-y = 58.8651
J = 1.04643
Cw = 933.682
I y-z = -2.30926e-14
Ysc = 2.79924
Zsc = 0
BetaV = 0
BetaW = -8.51467
Betaw = 0
Advanced
Z w-w = 58.5405
Z v-v = 19
A v-v = 4.0625
A w-w = 7.08333
Apply
Cancel

Column Orientation

1) Since the section was modified, the exterior columns are already assigned the appropriate section. The orientation just needs to be verified.

2) From the **Geometry** menu select **Re-orient Element(s)**.

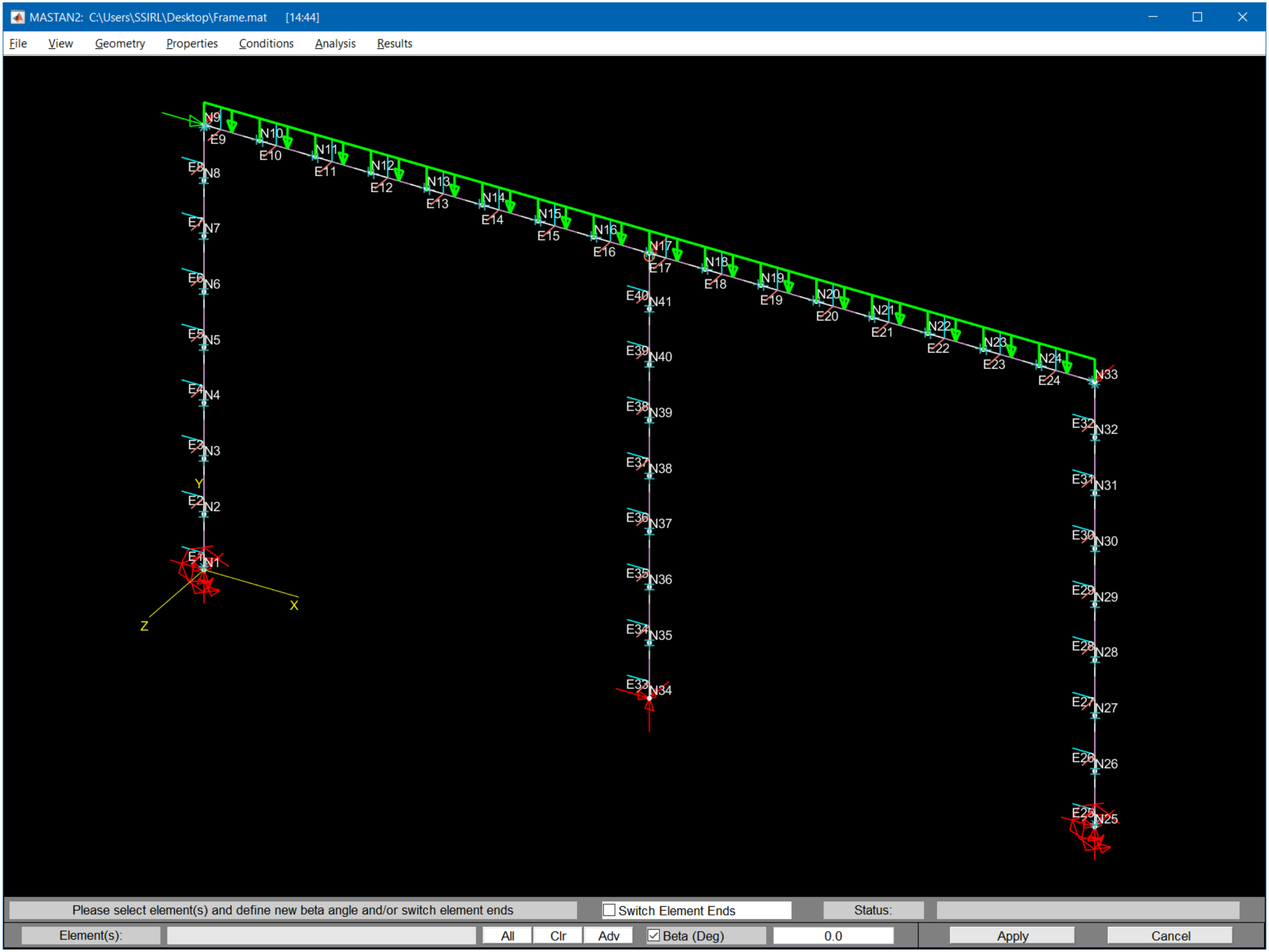
3) From the **View** menu select **Labels** and submenu option **Element local x'-y'-z' axes**. Each axis is shown with a different color line drawn in the positive direction. The x axis is purple, the y axis is blue, and the z axis is red.

4) At the bottom menu bar, click in the edit box to the right of **Beta (Deg)** and change **0.0** to **180**.

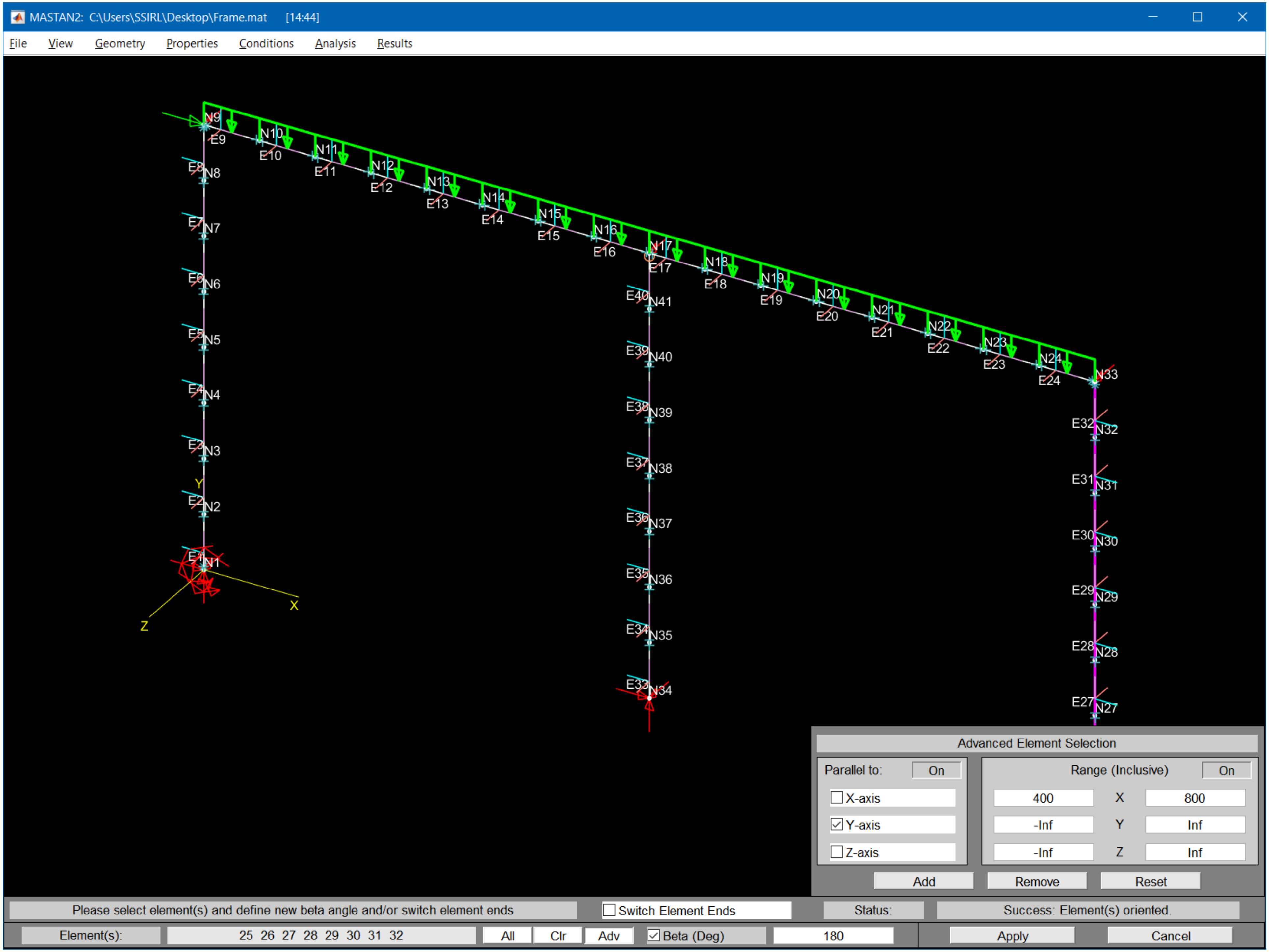
5) Use the buttons to the right of **Element(s):** to make the list of elements. Click the **Adv** button to open the pop-up menu. To select all the right column elements, click the edit box to the left of **X** and change **359** to **400**. Click the edit box to the right of **X** and change **361** to **800**.

6) Click **Add** to add all these elements to the element list. Click on the **Apply** button to re-orient the elements.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
Please select element(s) and define new beta angle and/or switch element ends
Switch Element Ends
Status:
Element(s):
All
Clr
Adv
Beta (Deg)
0.0
Apply
Cancel

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
Advanced Element Selection
Parallel to: On
X-axis
Y-axis
Z-axis
Range (Inclusive) On
400 X 800
-Inf Y Inf
-Inf Z Inf
Add Remove Reset
Please select element(s) and define new beta angle and/or switch element ends
Switch Element Ends
Status:
Success: Element(s) oriented.
Element(s): 25 26 27 28 29 30 31 32
All Clr Adv
Beta (Deg) 180
Apply Cancel

3-D 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.

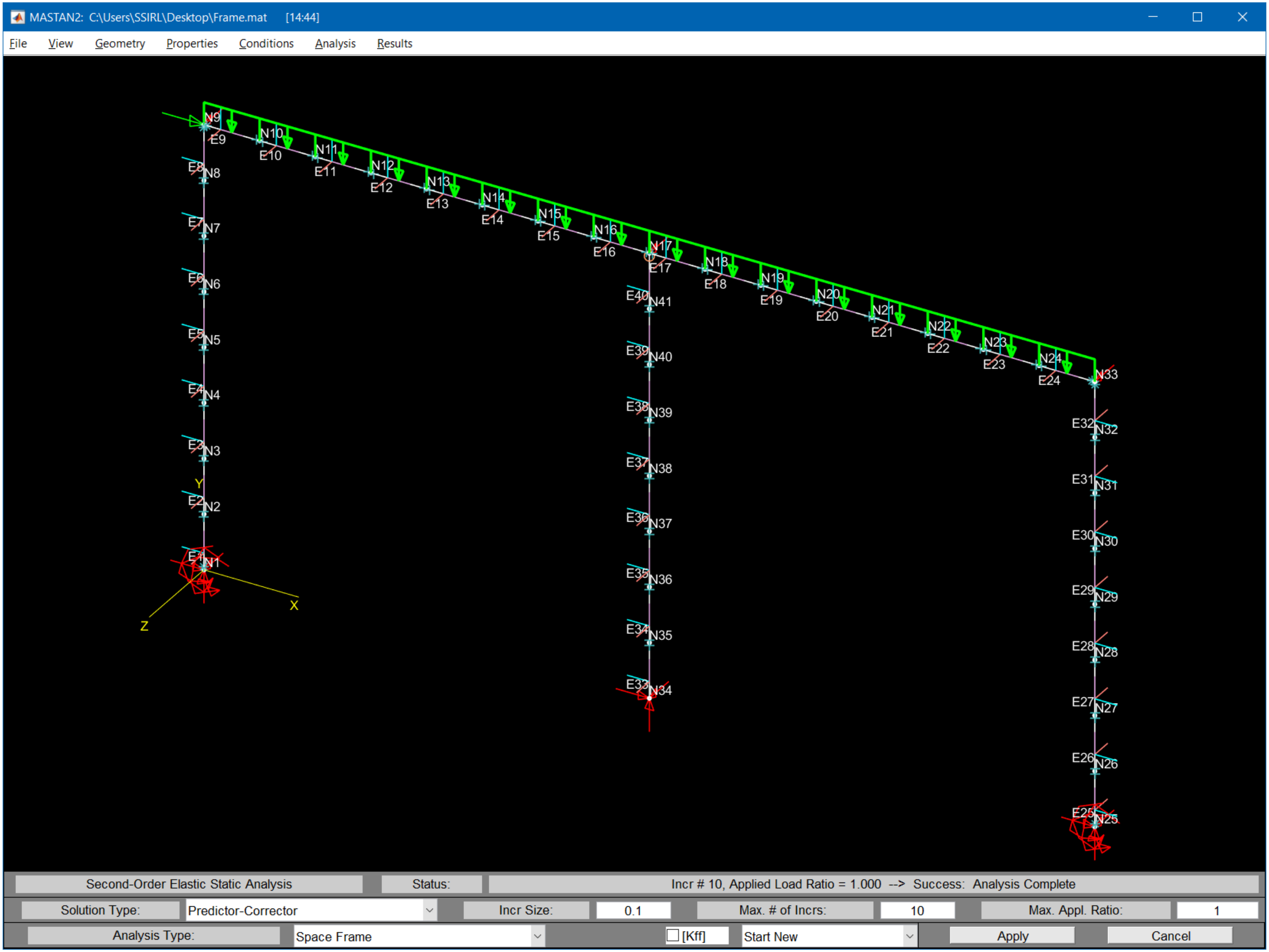
4) From the **Results** menu select **Node Displacements**.

5) On the undeflected shape, click on the node of interest in the upper right corner, **33**, and its components are provided in the bottom menu bar.

Results:

Disp X	Disp Y	Disp Z	Rot X	Rot Y	Rot Z
2.395	-0.02608	0	~0	~0	-1.304e-4

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
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\Frame.mat [13:42]
File View Geometry Properties Conditions Analysis Results
Node: 33
Disp X 2.395
Disp Y -0.02608
Disp Z 0
Status: Success: Disp. at ALR = 1.0000
Displacements
Rot X 5.776e-18
Rot Y -6.983e-21
Rot Z -0.0001304
(10) 1.000
Apply
Cancel

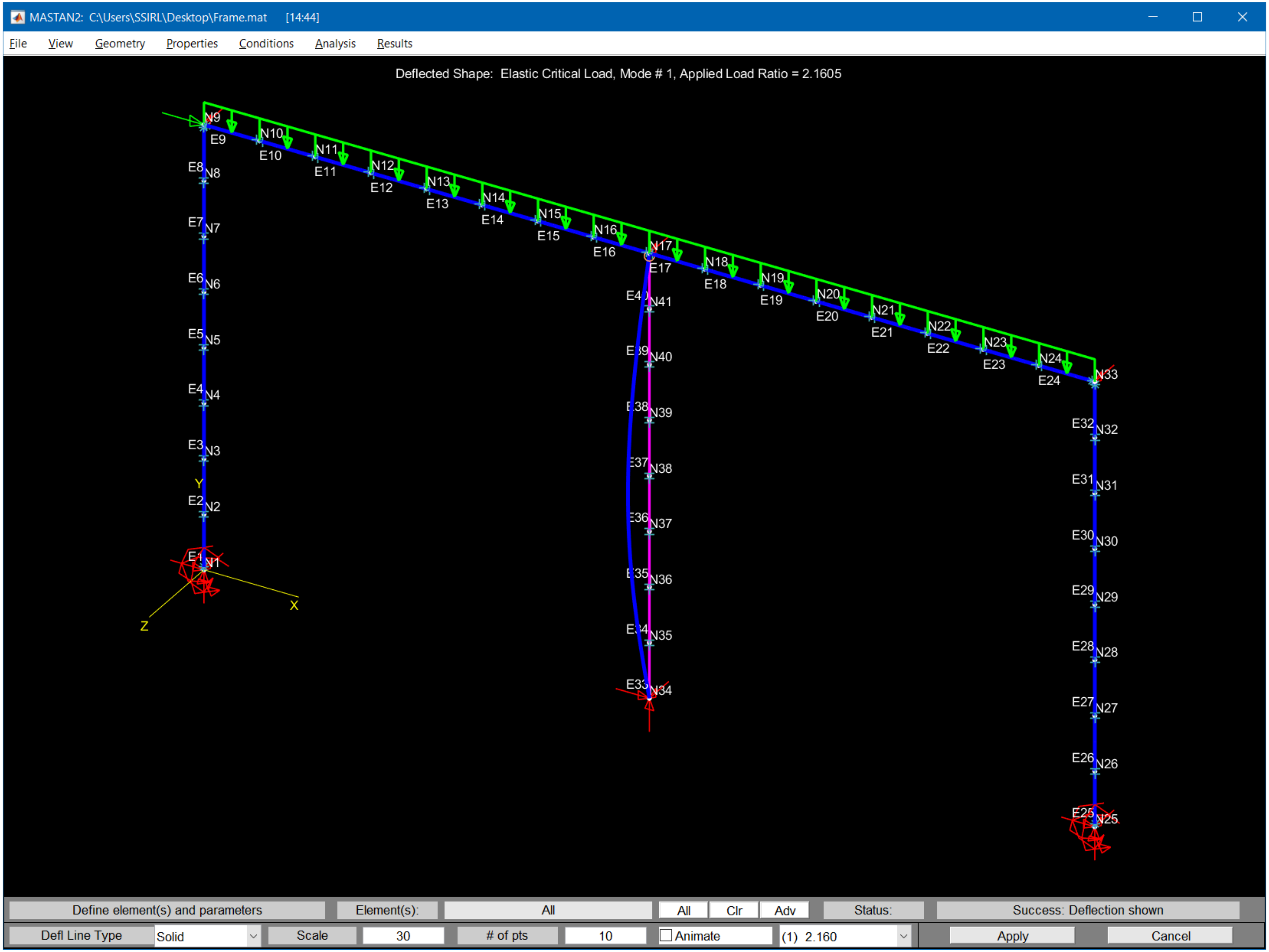
Additional Analysis

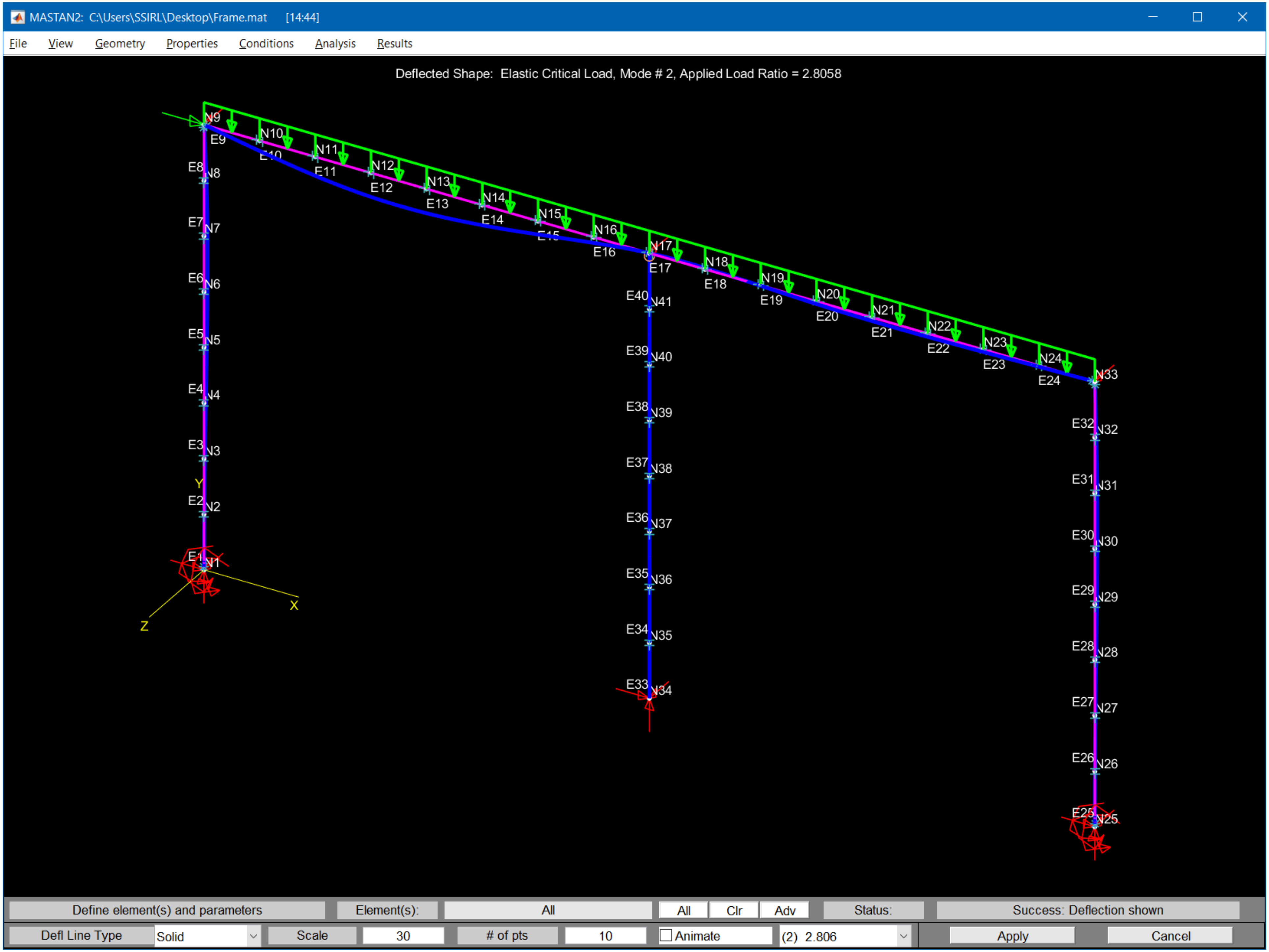
This final frame could also have been modeled with MASTAN2 using only the symmetric section properties. Since the frame was loaded only in plane and the non-doubly symmetric effects were not activated, the user would find that it is possible to recreate the frame without the use of advanced section properties and only input the basic section properties and calculate similar displacements.

	Disp X	Disp Y	Rot Z
Basic	2.395	-0.02608	-1.304e-4
Advanced	2.395	-0.02608	-1.304e-4

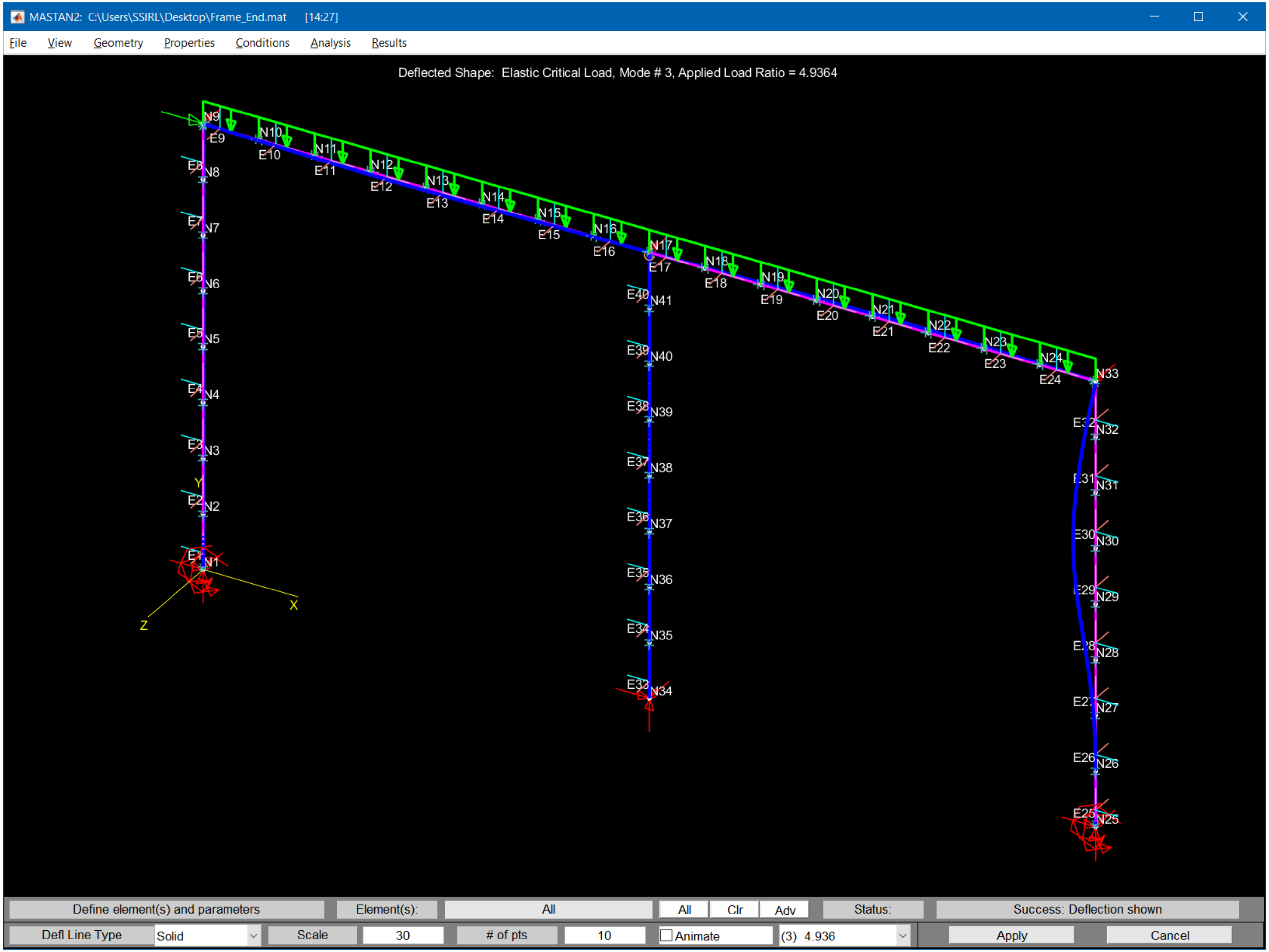
However, the evaluation of the critical buckling loads of the structure does capture the non-doubly symmetric effects. Different behavior could be observed if the user were to compare such an analysis on the frame with basic and advanced section properties. The first mode and second mode are very similar as the buckling behavior is controlled by the doubly symmetric elements. The third mode displays distinctly different behavior as the column is weaker considering singly symmetric behavior.

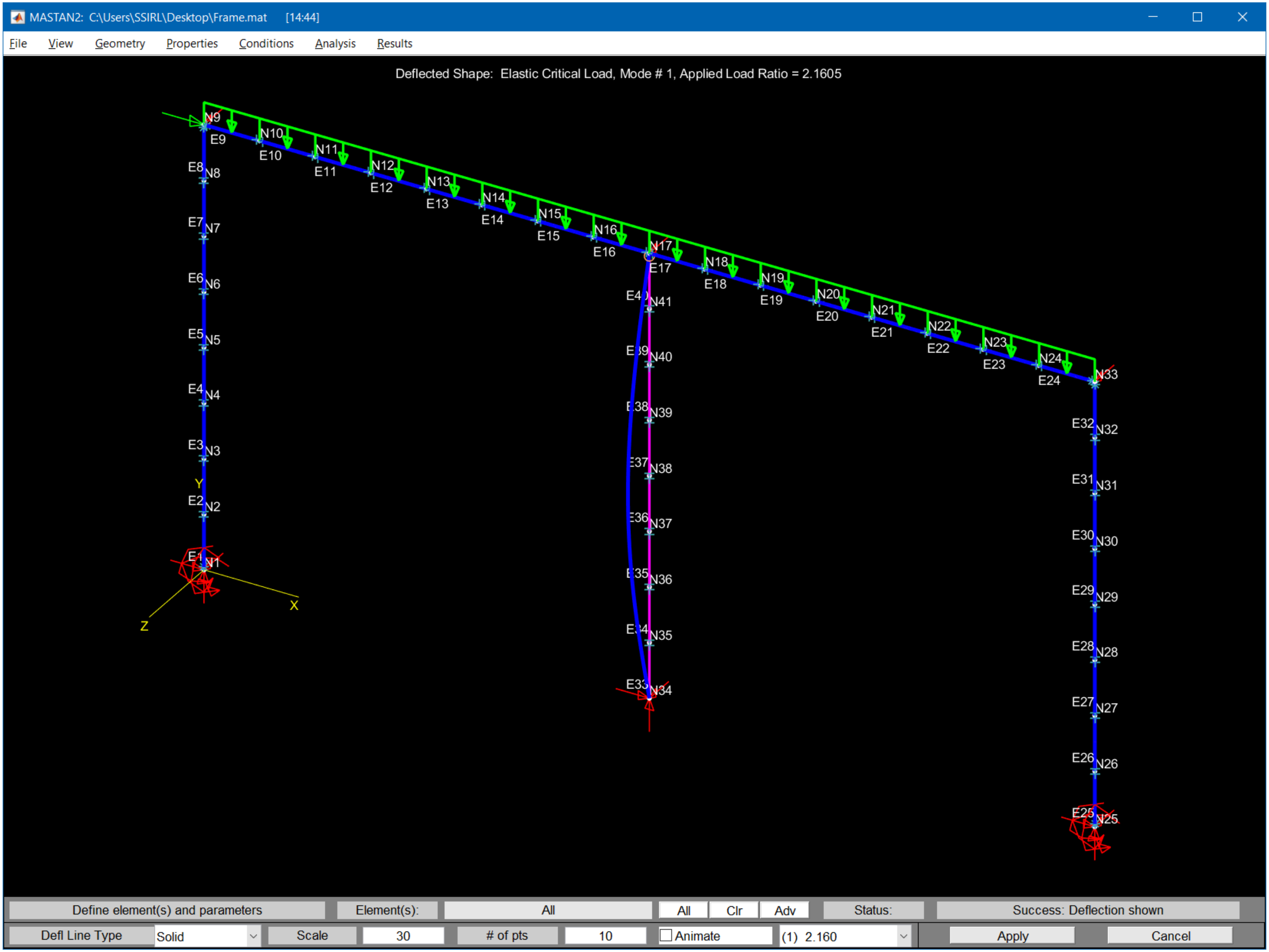
	Mode #1	Mode #2	Mode #3
Basic	2.160	2.806	4.936
Advanced	2.160	2.805	4.040

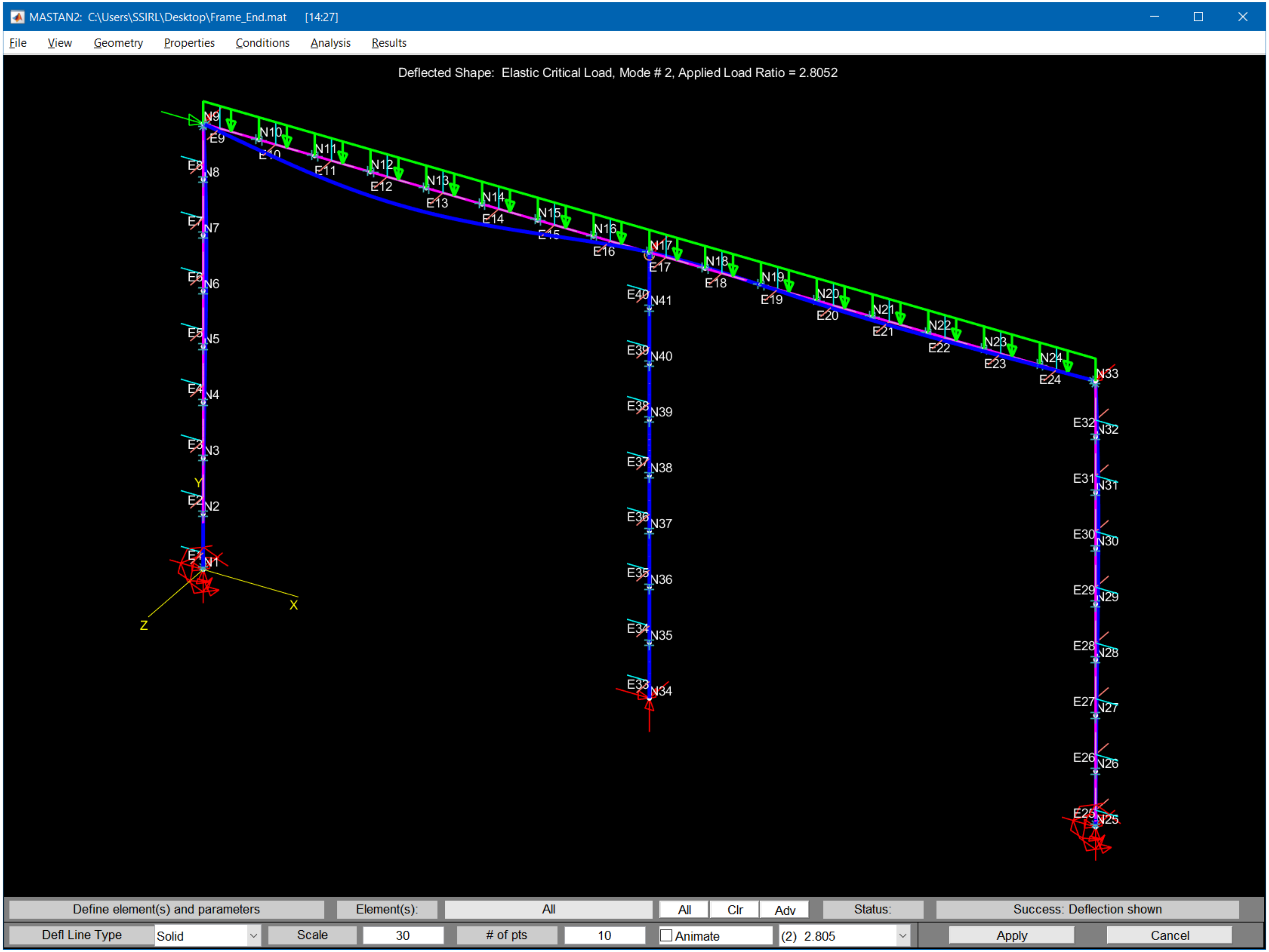

MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
Deflected Shape: Elastic Critical Load, Mode # 1, Applied Load Ratio = 2.1605
Define element(s) and parameters
Element(s):
All
All Clr Adv
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
30
# of pts
10
Animate
(1) 2.160
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
Deflected Shape: Elastic Critical Load, Mode # 2, Applied Load Ratio = 2.8058
Define element(s) and parameters
Element(s):
All
All
Clr
Adv
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
30
# of pts
10
Animate
(2) 2.806
Apply
Cancel

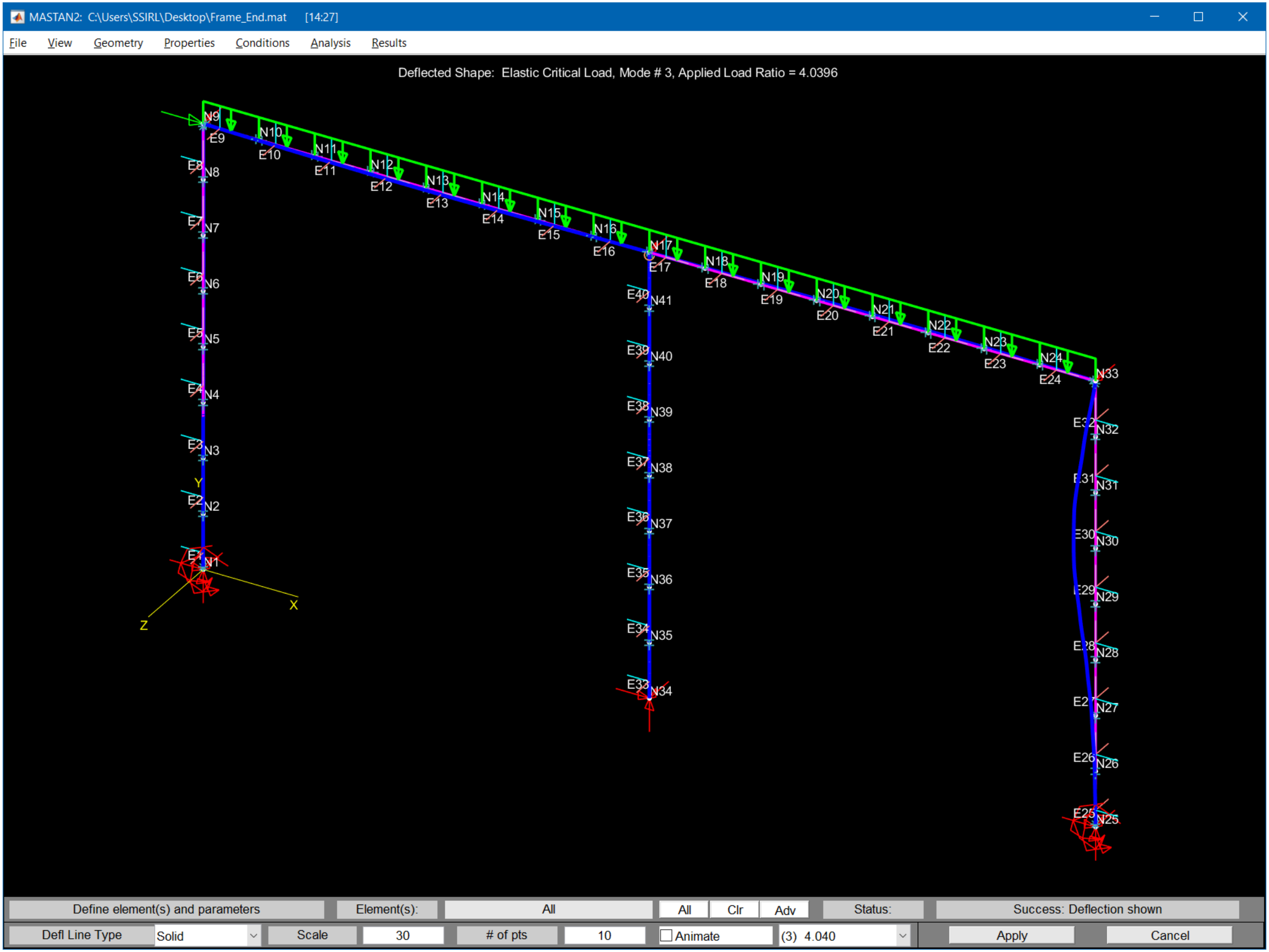
MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Frame_End.mat [14:27]
File View Geometry Properties Conditions Analysis Results
Deflected Shape: Elastic Critical Load, Mode # 3, Applied Load Ratio = 4.9364
Y
X
Z
Define element(s) and parameters
Element(s):
All
All
Clr
Adv
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
30
# of pts
10
Animate
(3) 4.936
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
Deflected Shape: Elastic Critical Load, Mode # 1, Applied Load Ratio = 2.1605
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 N28 N29 N30 N31 N32 N33 N34 N35 N36 N37 N38 N39 N40 N41
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 E28 E29 E30 E31 E32 E33
X
Y
Z
Define element(s) and parameters
Element(s):
All
All
Clr
Adv
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
30
# of pts
10
Animate
(1) 2.160
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Frame_End.mat [14:27]
File View Geometry Properties Conditions Analysis Results
Deflected Shape: Elastic Critical Load, Mode # 2, Applied Load Ratio = 2.8052
Define element(s) and parameters
Element(s):
All
All
Clr
Adv
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
30
# of pts
10
Animate
(2) 2.805
Apply
Cancel

MASTAN2

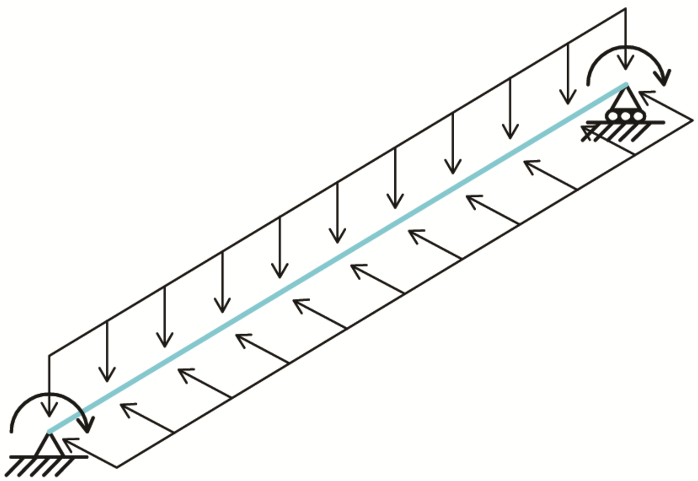


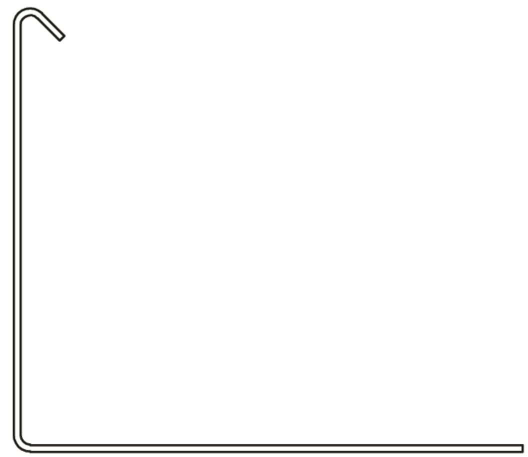
MASTAN2: C:\Users\SSIRL\Desktop\Frame_End.mat [14:27]
File View Geometry Properties Conditions Analysis Results
Deflected Shape: Elastic Critical Load, Mode # 3, Applied Load Ratio = 4.0396
Define element(s) and parameters
Element(s):
All
All
Clr
Adv
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
30
# of pts
10
Animate
(3) 4.040
Apply
Cancel

This completes the tutorial.

MASTAN2

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 INSTITUTE
SJI

STEEL DECK INSTITUTE
sdi ®

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.

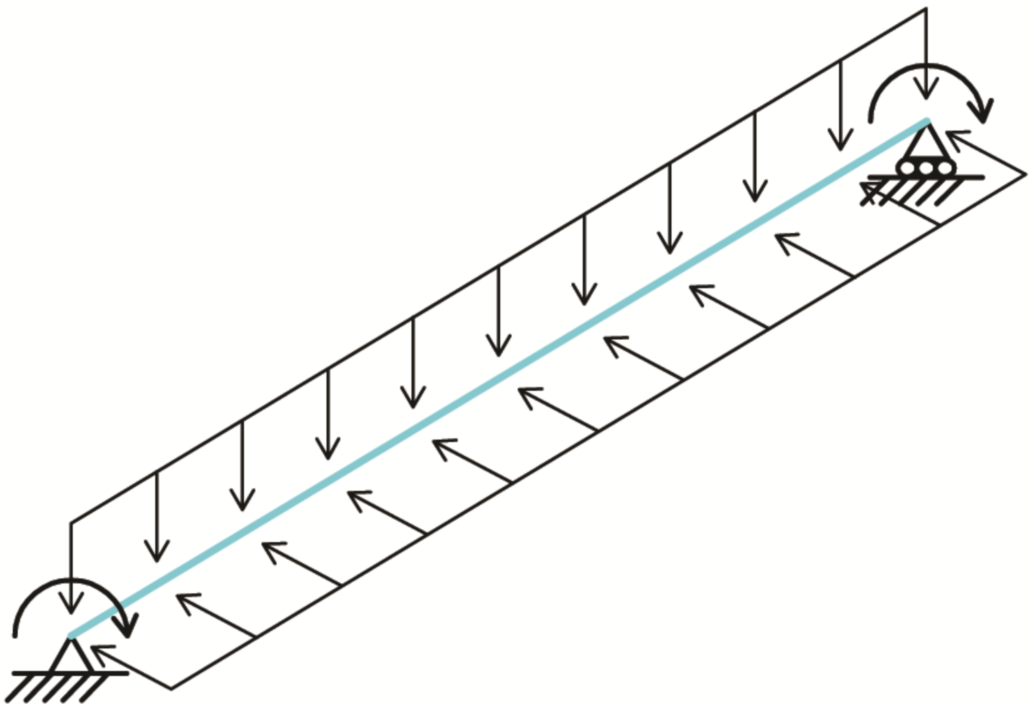
Section 1: Overview

Overview

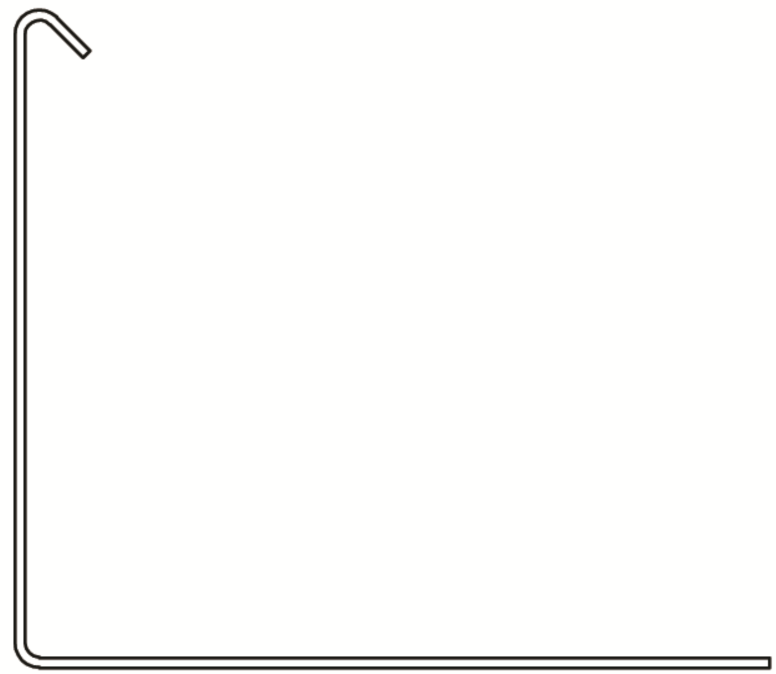
This tutorial provides step-by-step guidance for the sample pour stop beam evaluation. Enough details are provided that the example model with non-doubly symmetric sections can be completed following the instructions here. Not every feature available in MASTAN2 will be mentioned nor utilized in this 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

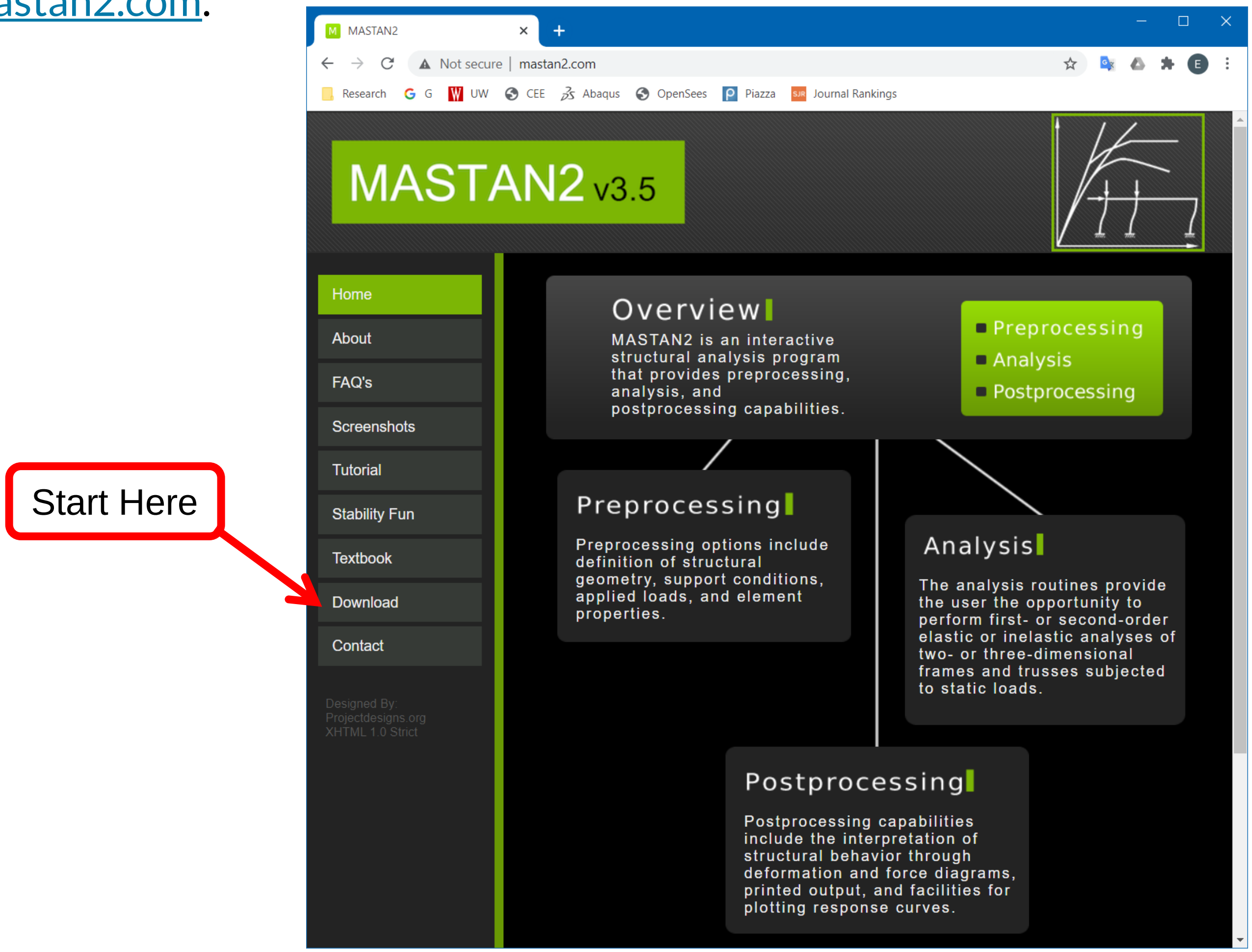
MASTAN2 General Information

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.

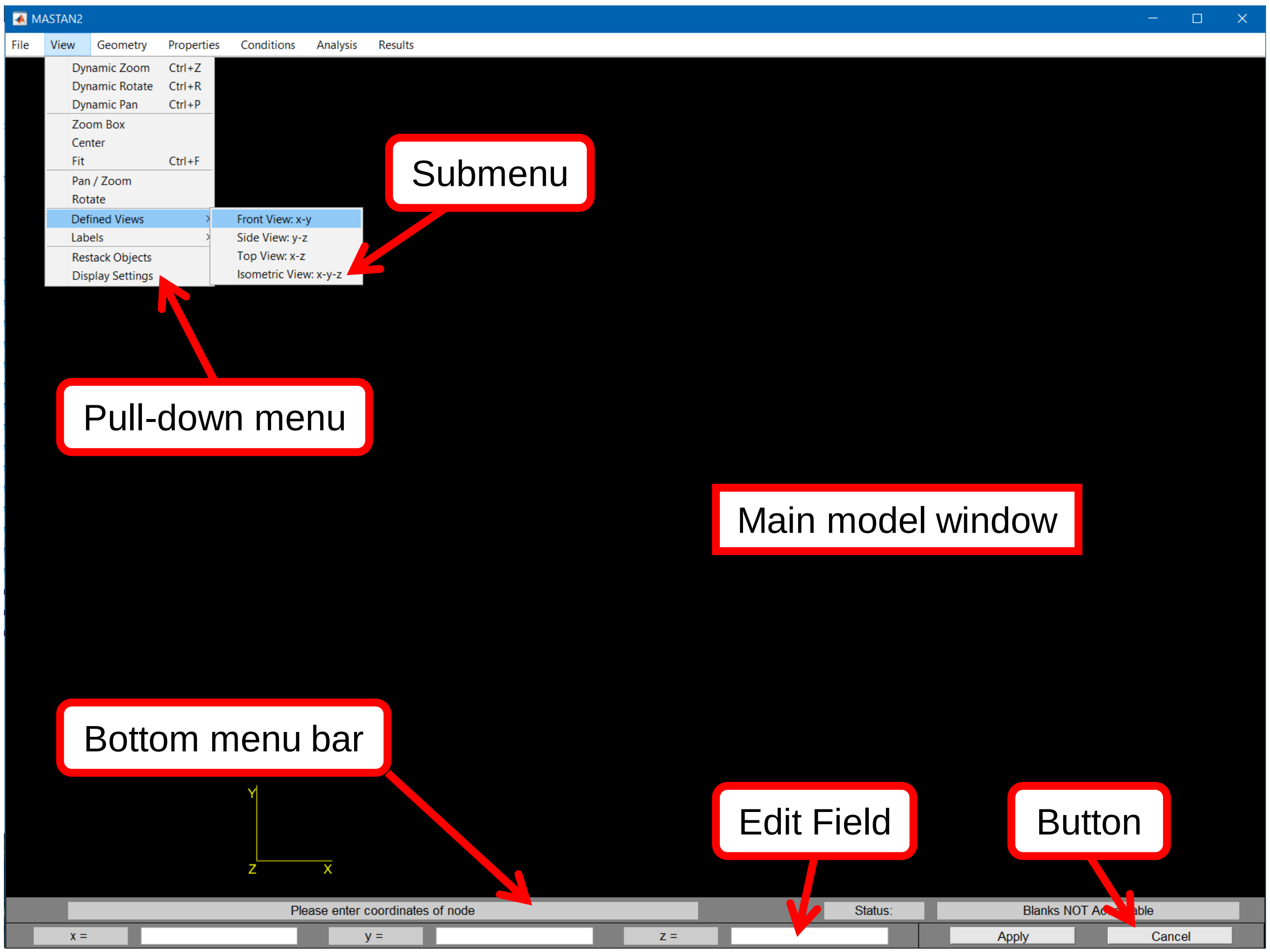
Launching MASTAN2

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

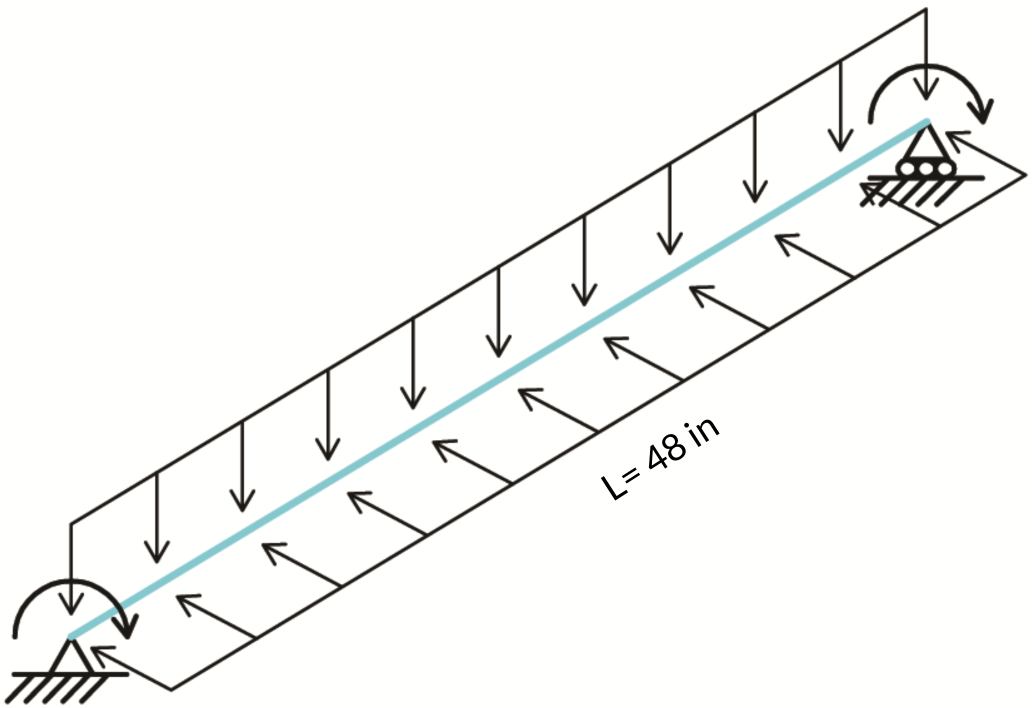
In order to minimize the learning time for MASTAN2, its graphical user interface (GUI) has been designed using a simple and consistent two menu approach. Using a pull-down menu at the top of the GUI, a command is selected. Parameters are then defined in the bottom menu bar and the command is executed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

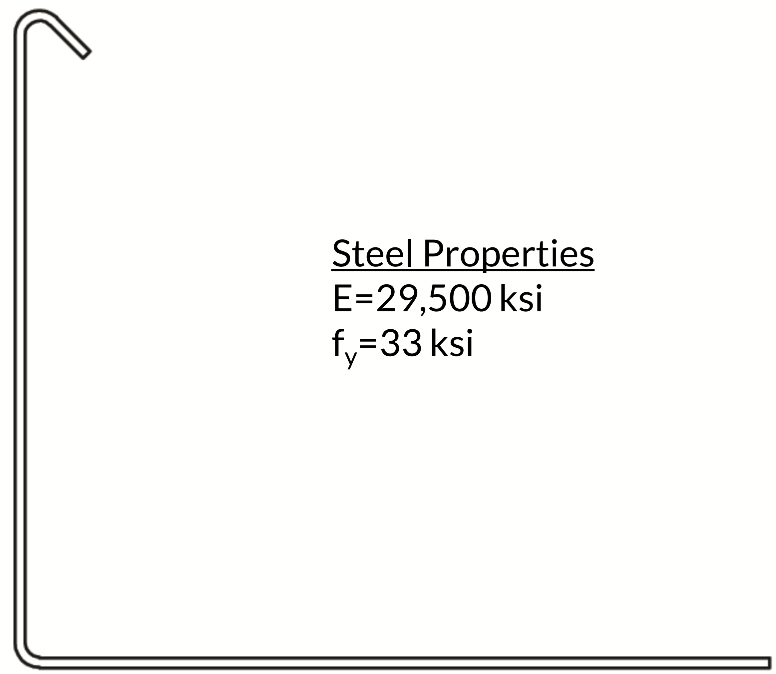


Section 3: Beam Modeling

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.

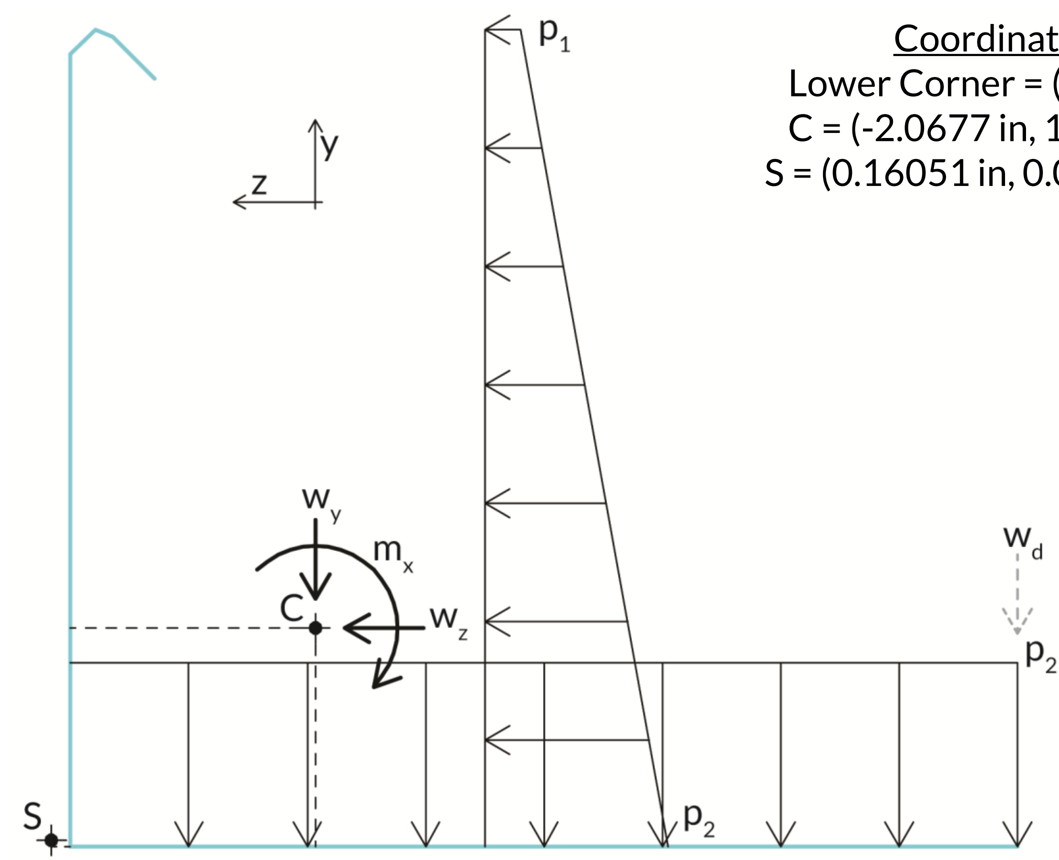




MASTAN2 does not assume any unit system. Models in MASTAN2 require the use of a consistent set of units. This tutorial will use pound and inch for the model. The later section of the tutorial that determines the internal stresses does include a unit conversion to show stresses in ksi.

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



<u>Coordinates</u>

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

<u>Load Details</u>

See SDI Tech No. 3 for additional information

p_1 = 20 psf
p_2 = 103.3 psf

w_d = 0 plf ↓

<u>Resulting Loads</u>

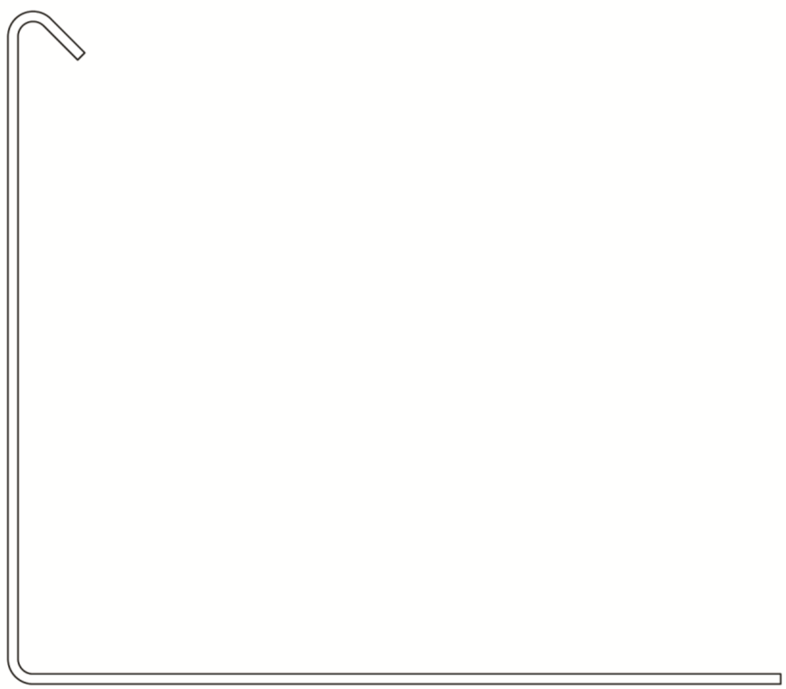
w_y = 74.0 plf ↓
= 6.17 pli ↓

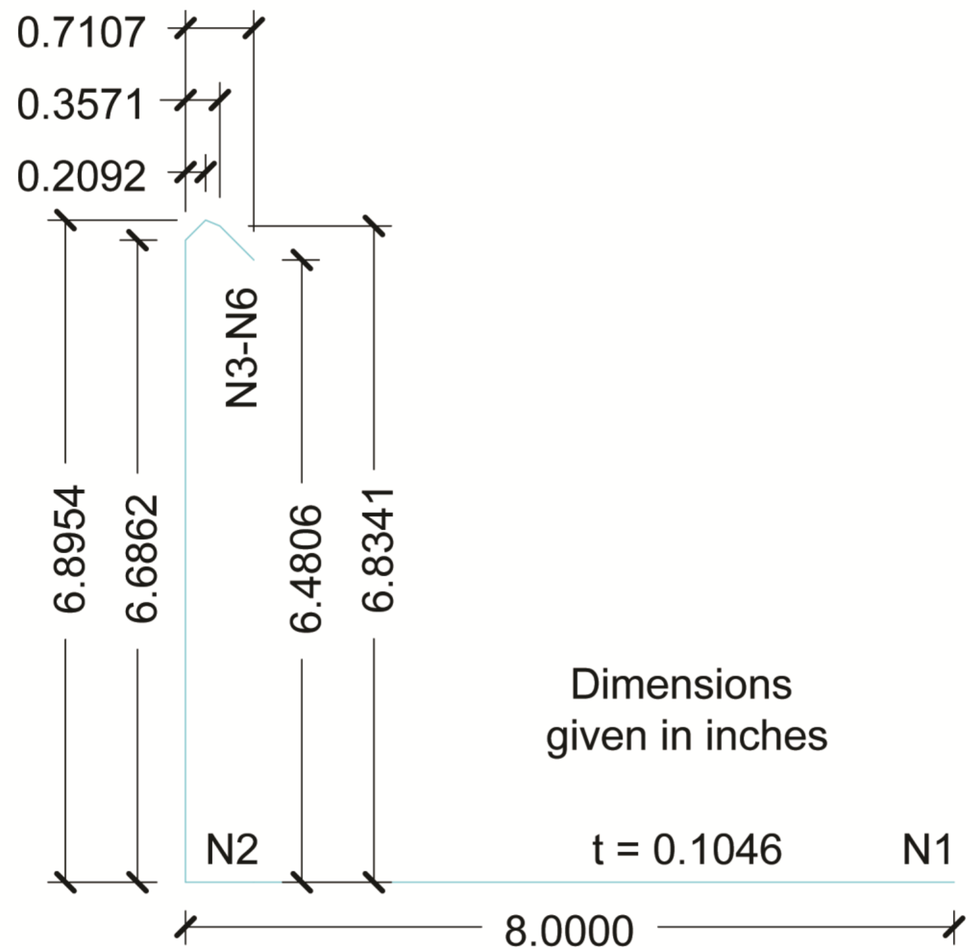
w_z = 35.4 plf ←
= 2.95 pli ←

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.

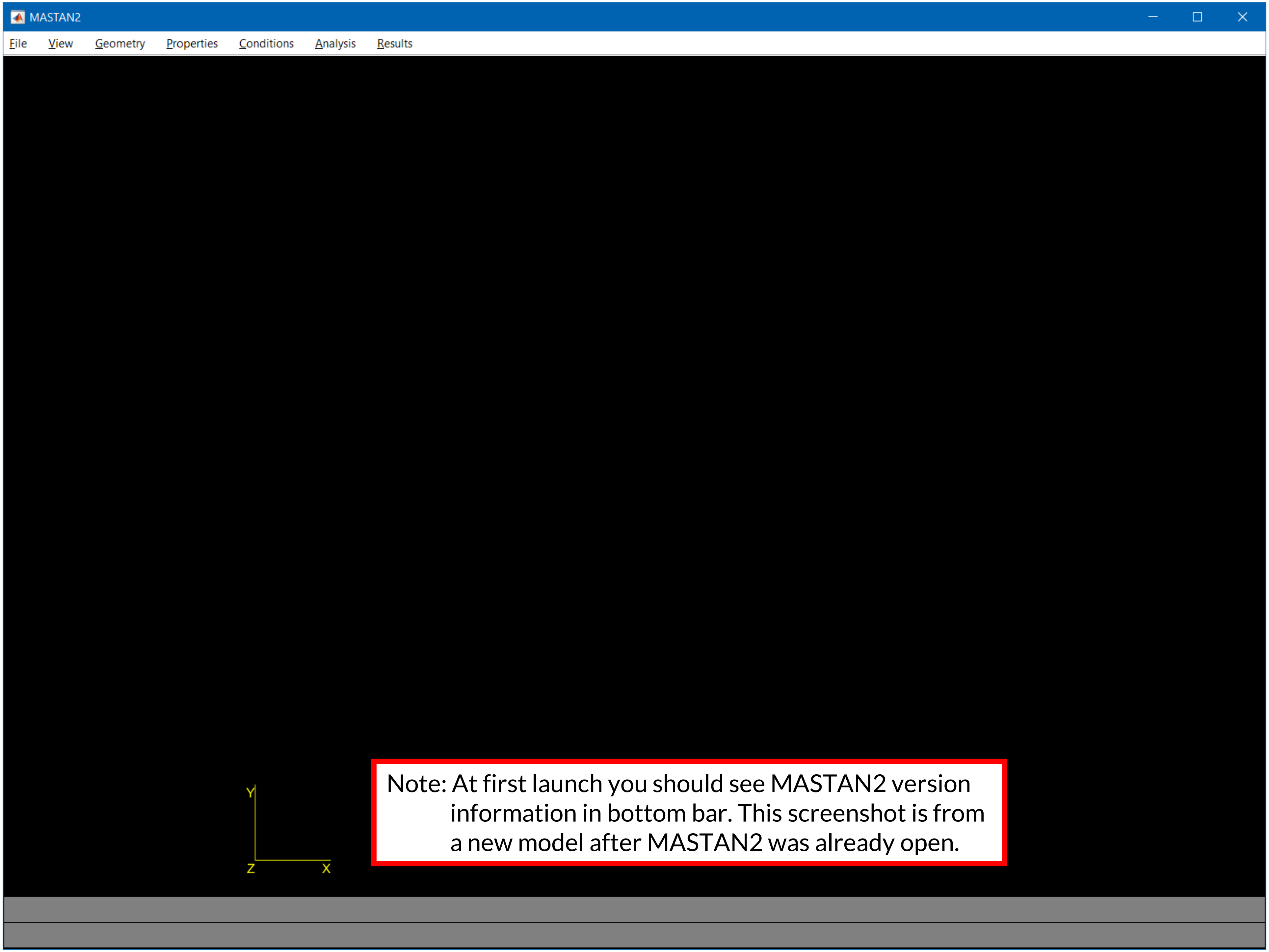
5) From the **Geometry** menu select **Extrude Element**.

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**.

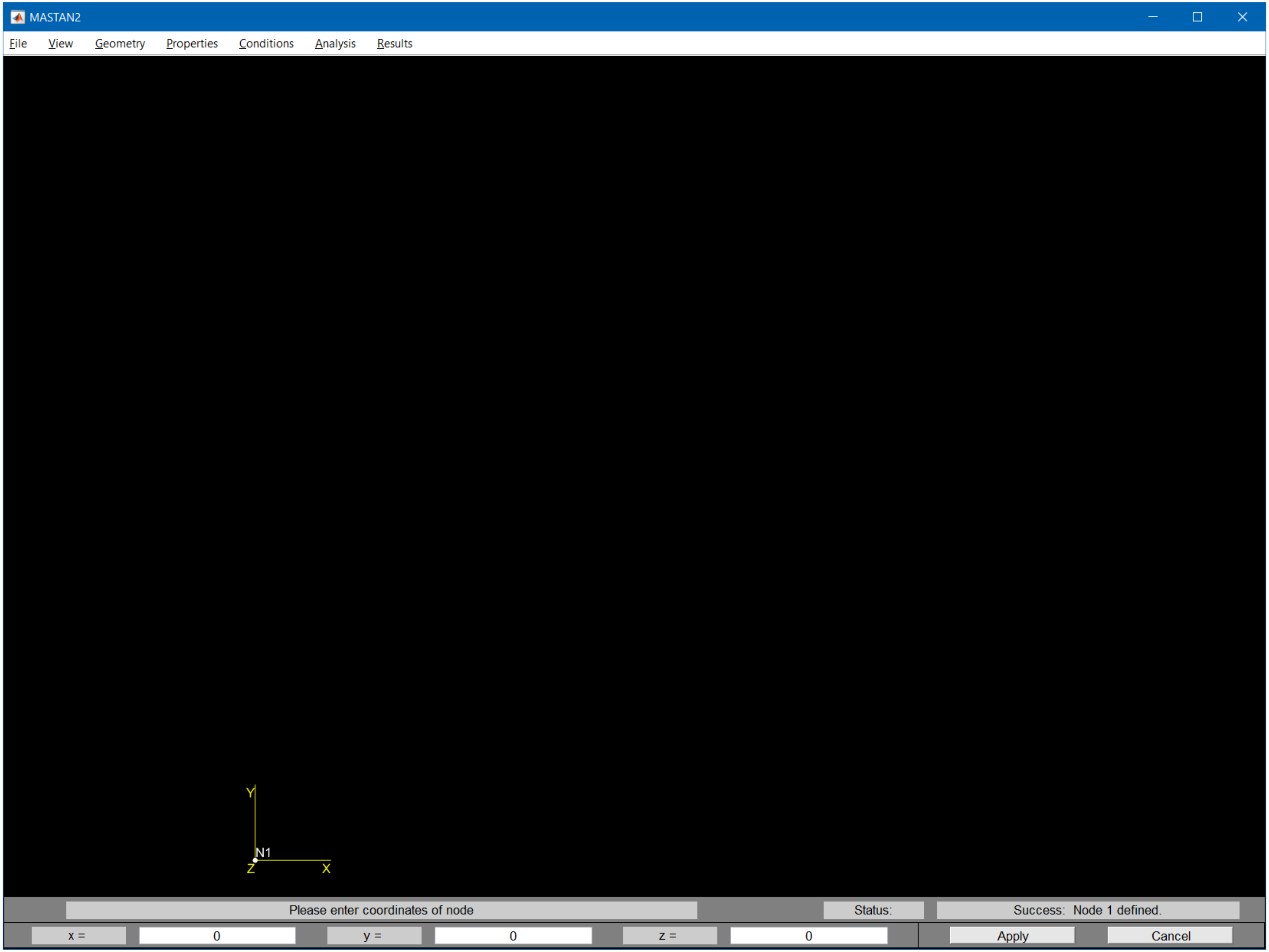
7) Repeatedly click the **>** button to the right of **Times =** to increase **1** to **24**.

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.


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
X
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



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
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**.

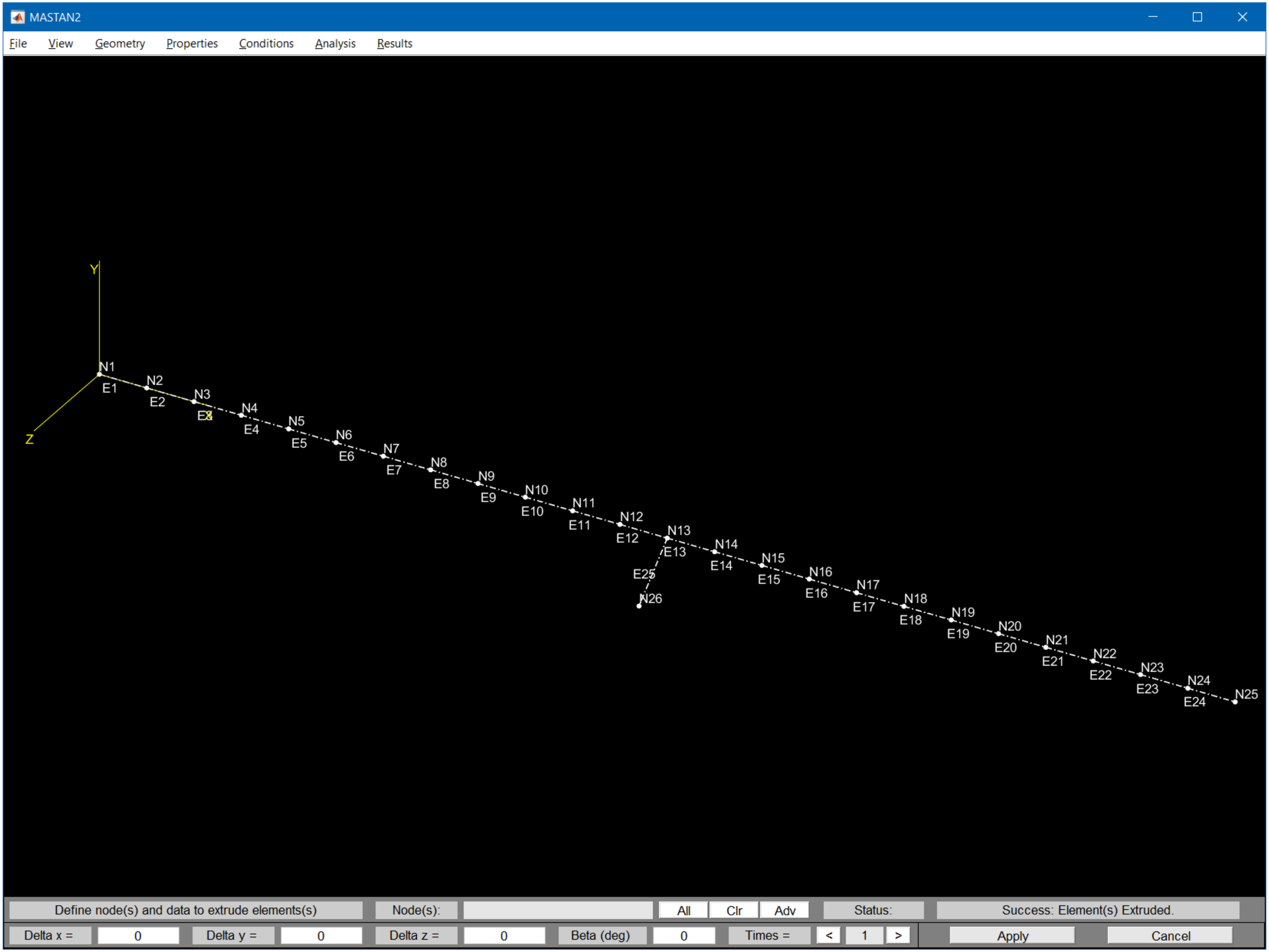
2) Now with the main member defined, continue to use **Extrude Element** to illustrate cross section.

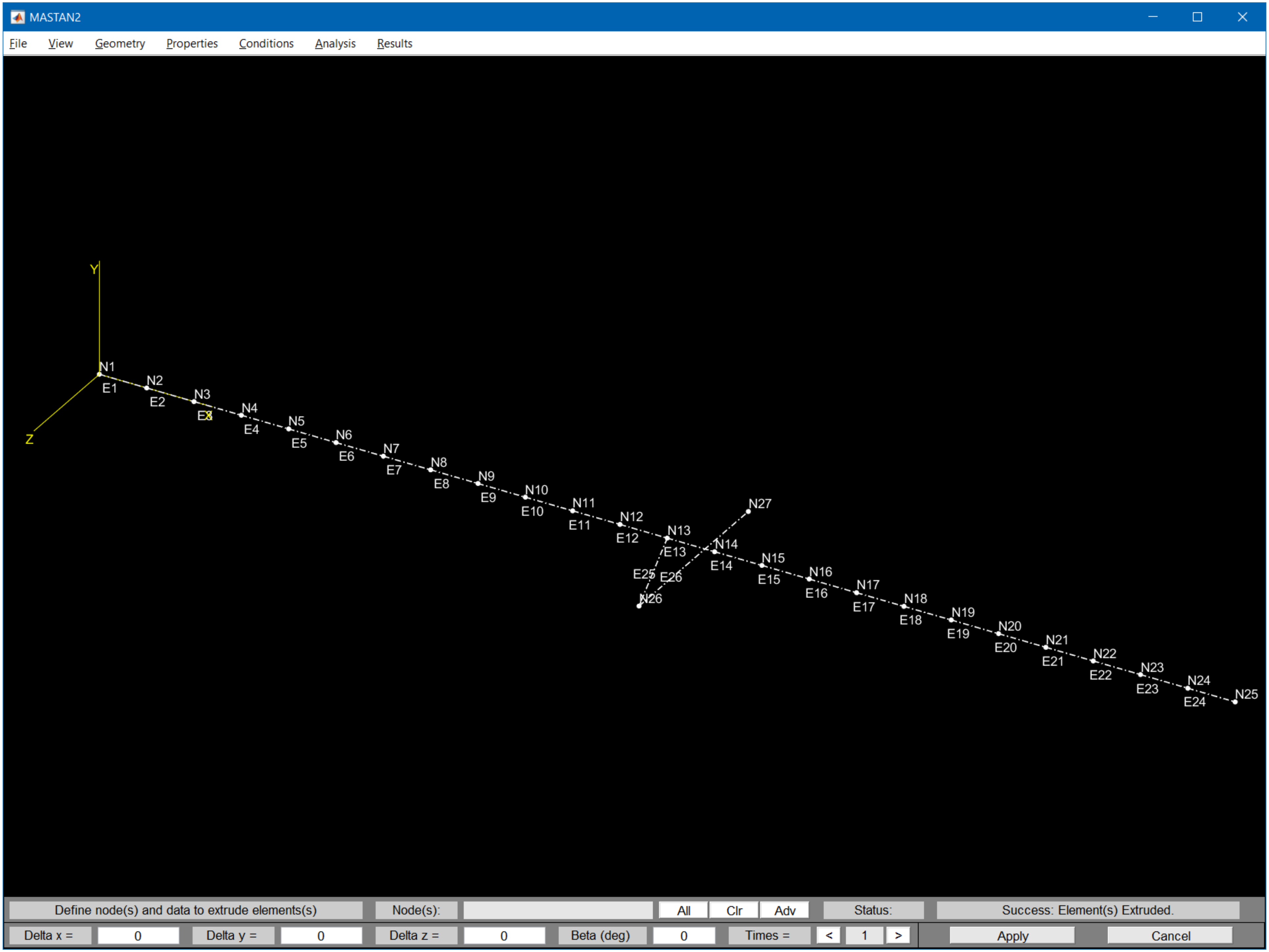
3) At the bottom menu bar, click on node **13** to populate the list of nodes. Click in the edit box to the right of **Delta y =** and change **0** to **-1.8405**. Click in the edit box to the right of **Delta z =** and change **0** to **2.0677**.

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 E3 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
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

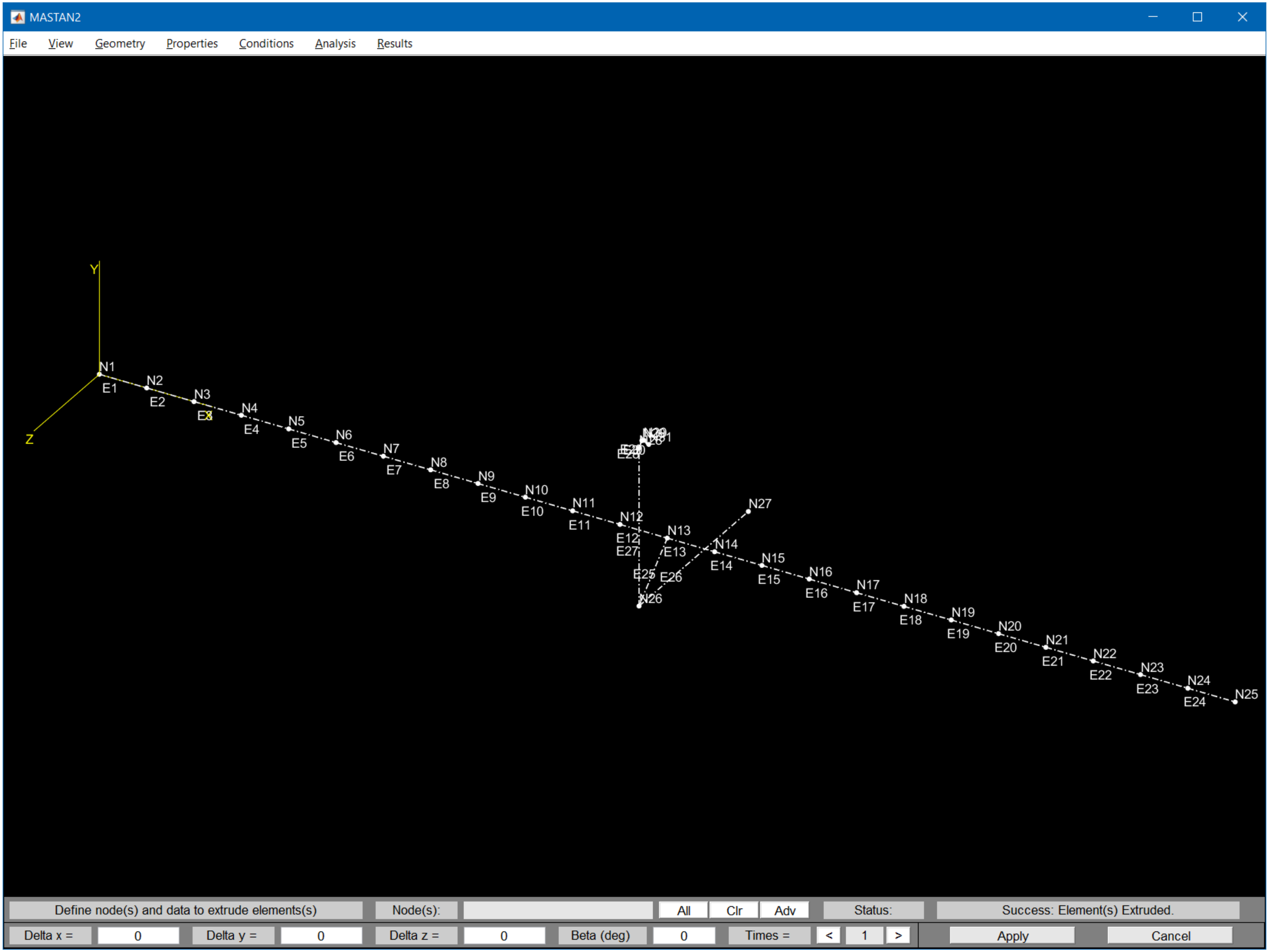
1) Continue defining the cross section by clicking on **Node 26** to populate the list of nodes. Click in the edit box to the right of **Delta y =** and change **0** to **6.6862**. Click on the **Apply** Button.

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
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**.

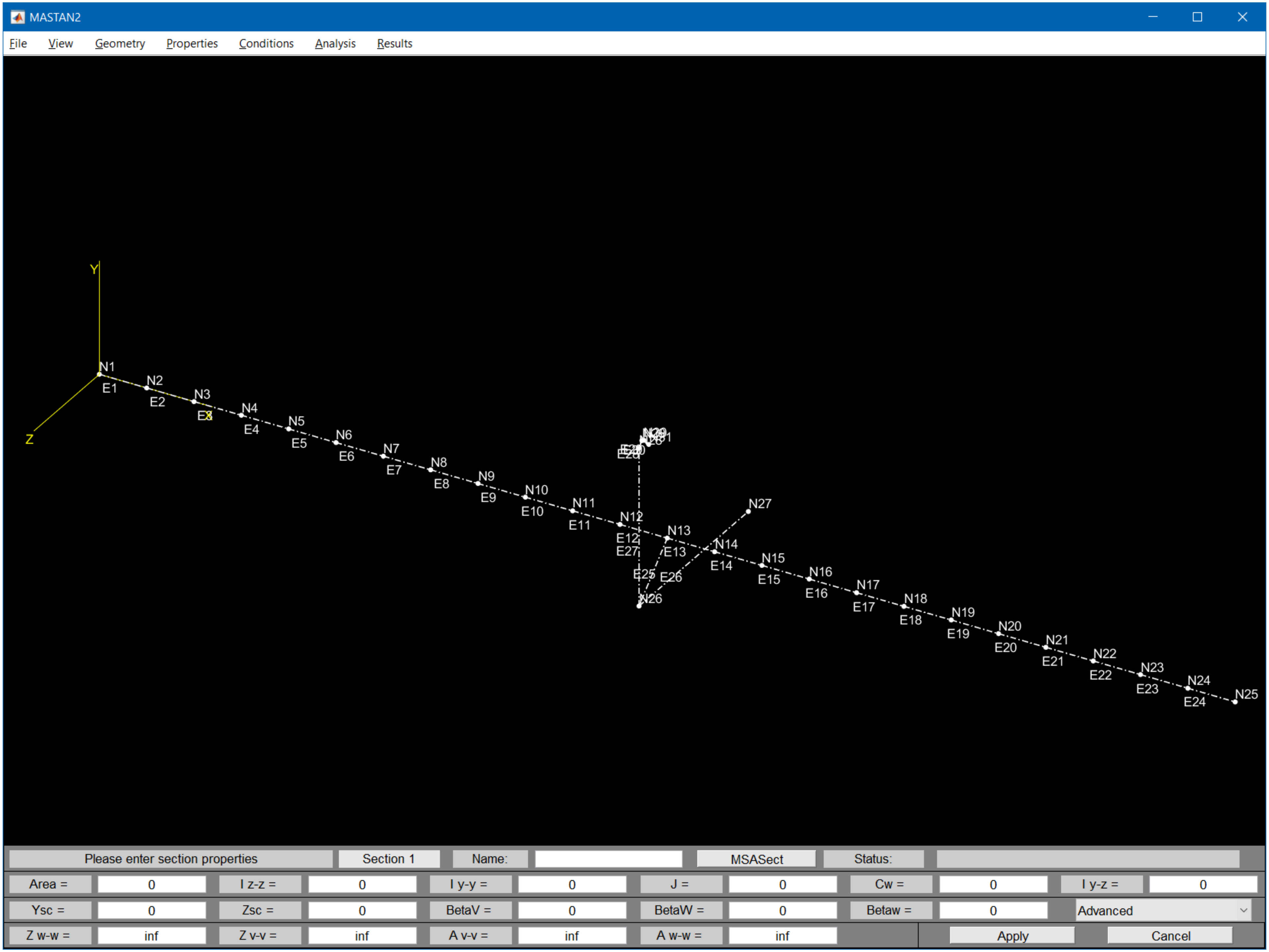
4) After the interface loads, click on **General** to select the radio button next to it.

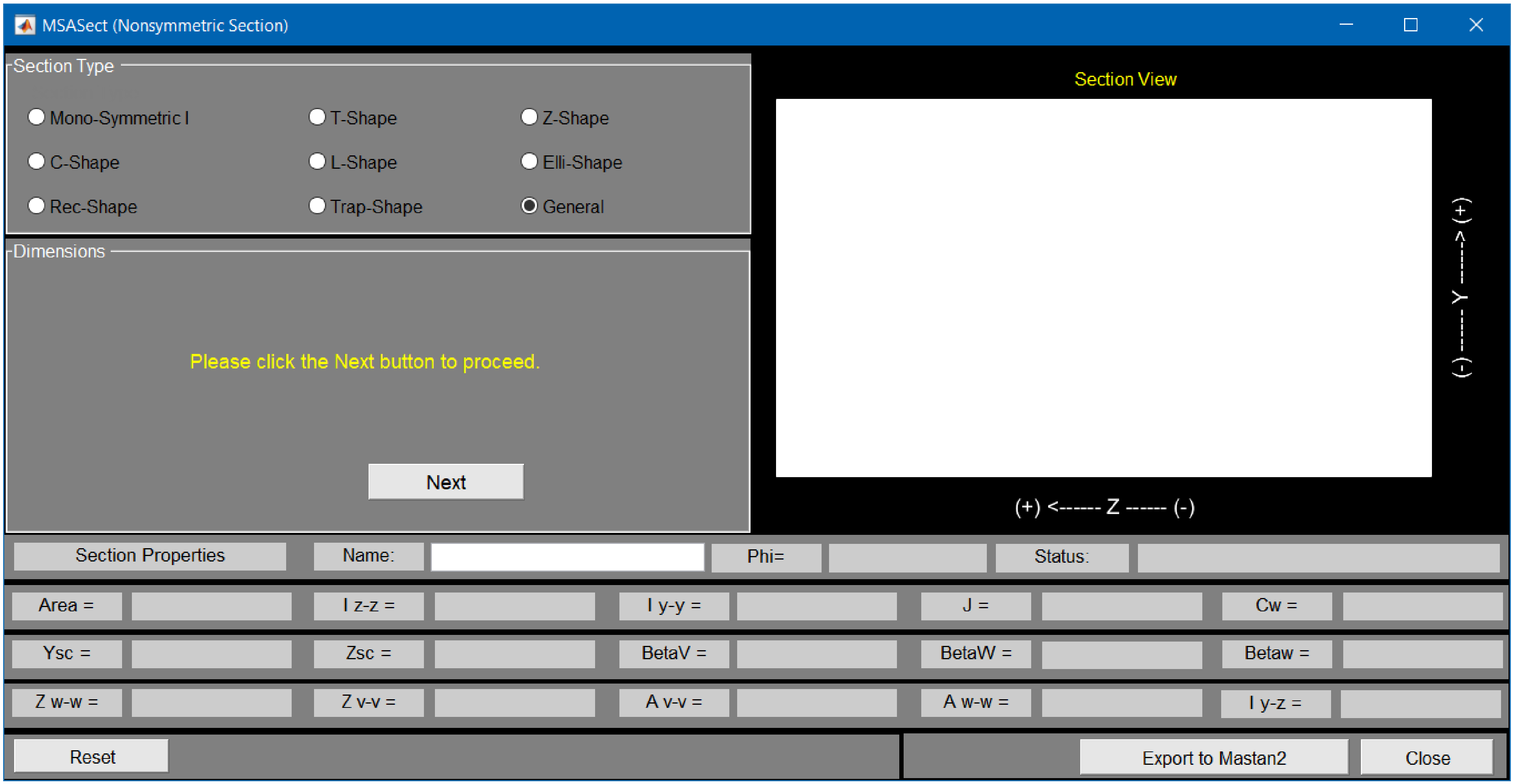
5) Click **Next** to open the editable dialog boxes.

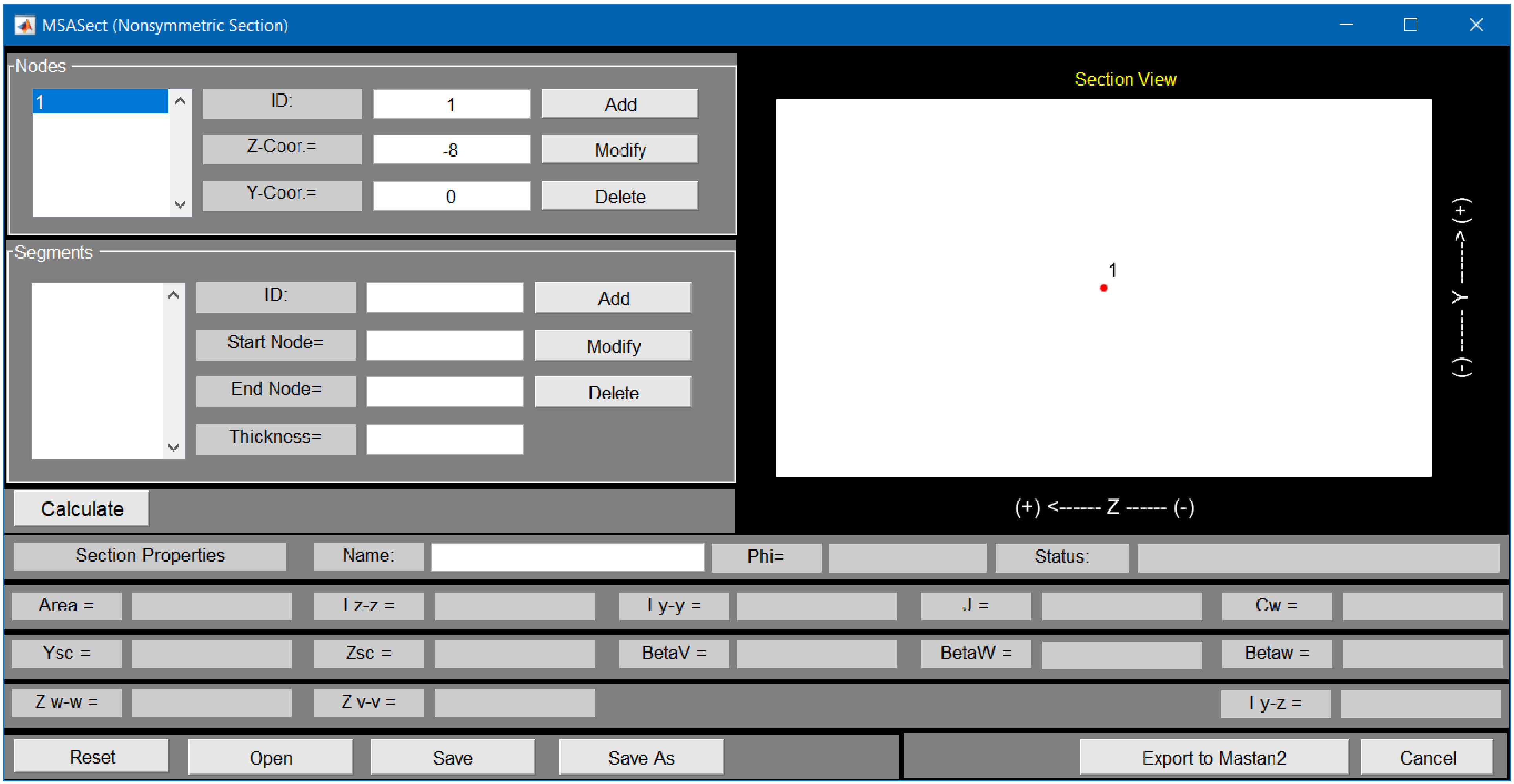
6) Click the edit box to the right of **ID:** and enter **1**. Click the edit box to the right of **Z-Coor.=** and enter **-8**. Click the edit box to the right of **Y-Coor.=** and enter **0**. Click **Add** to save the node.

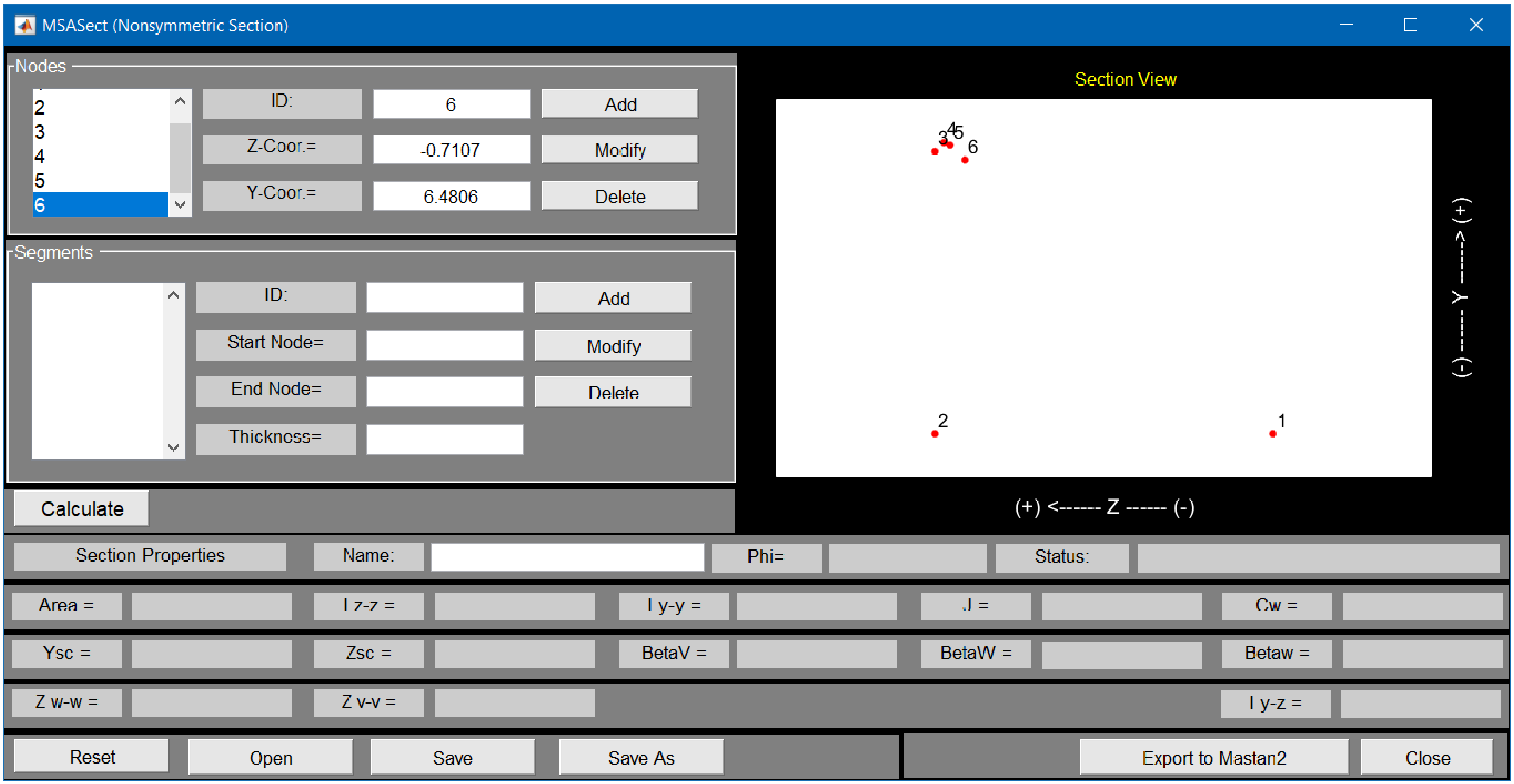
7) Repeat entering the values below for each node clicking **Add** after each one.

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
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
E27
E13
E14
E15
E16
E17
E18
E19
E20
E21
E22
E23
E24
E25
E26
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
(+) <------ 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 =
A v-v =
A w-w =
I y-z =
Reset
Export to Mastan2
Close


MSASect (Nonsymmetric Section)
Nodes
1
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.

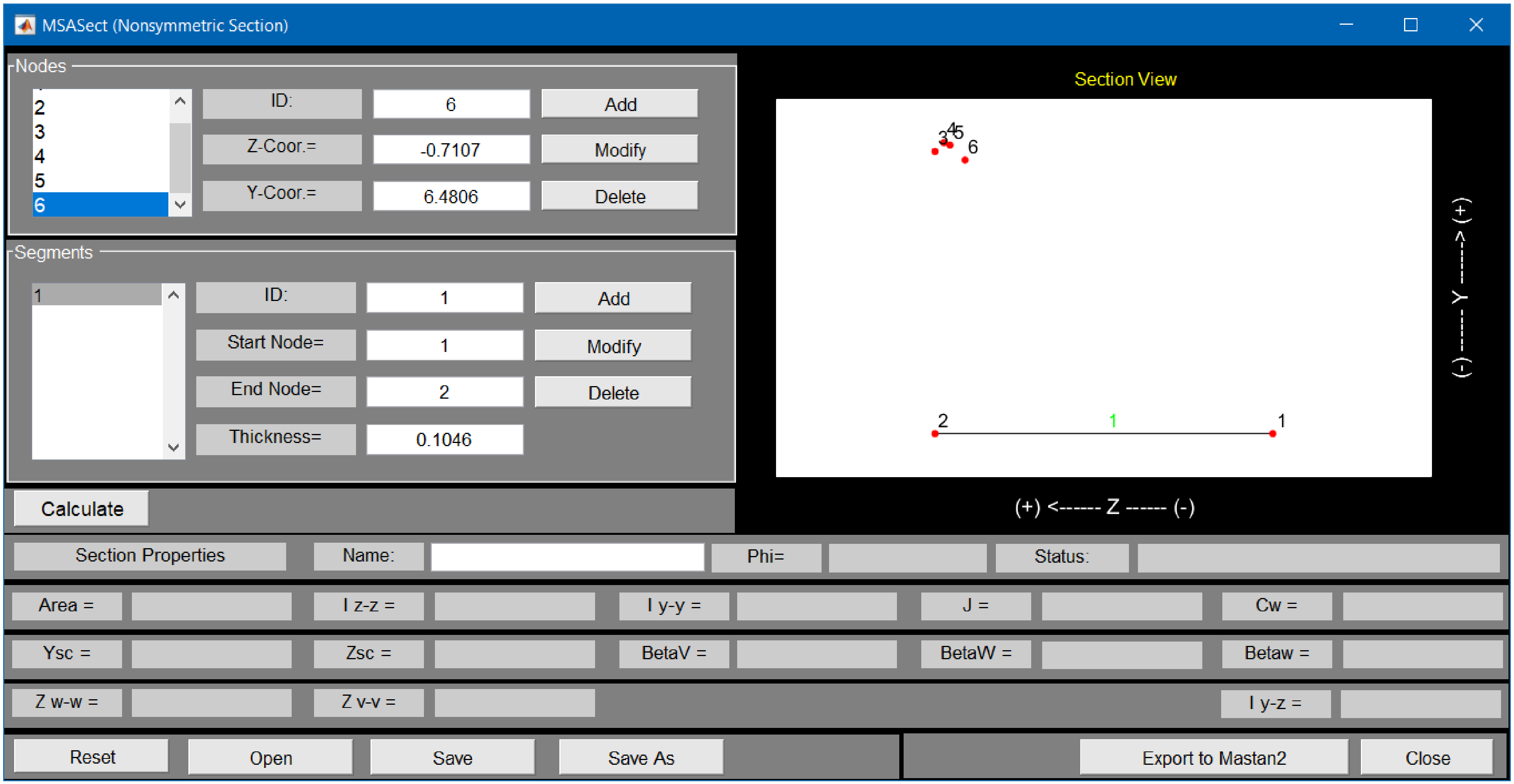
2) Repeat entering the values for each segment updating the **ID:**, **Start Node=**, and **End Node=** values by adding **1** to each number until all 5 segments are entered. Click **Add** after each to save.

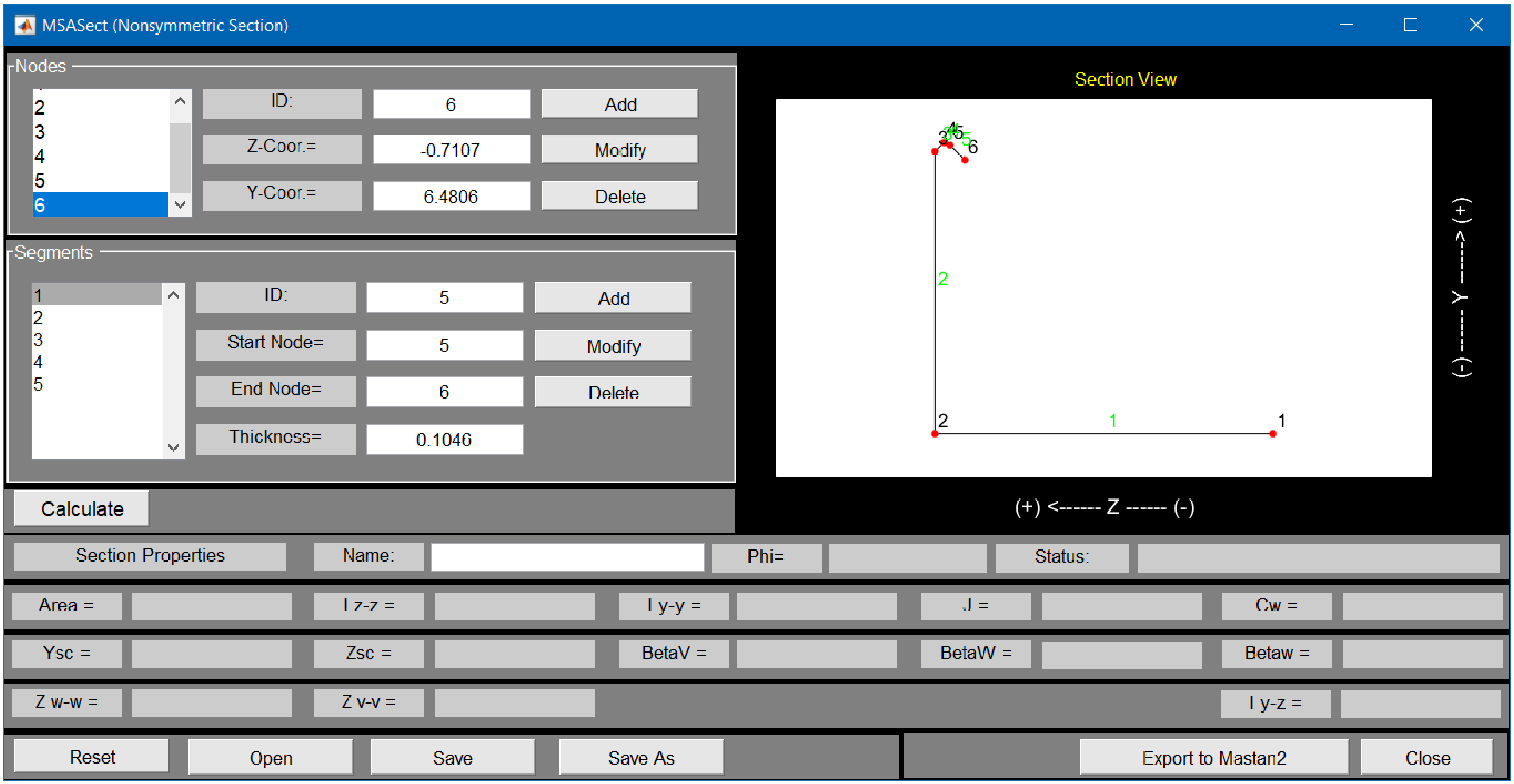
3) Click **Calculate** to determine the properties.

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

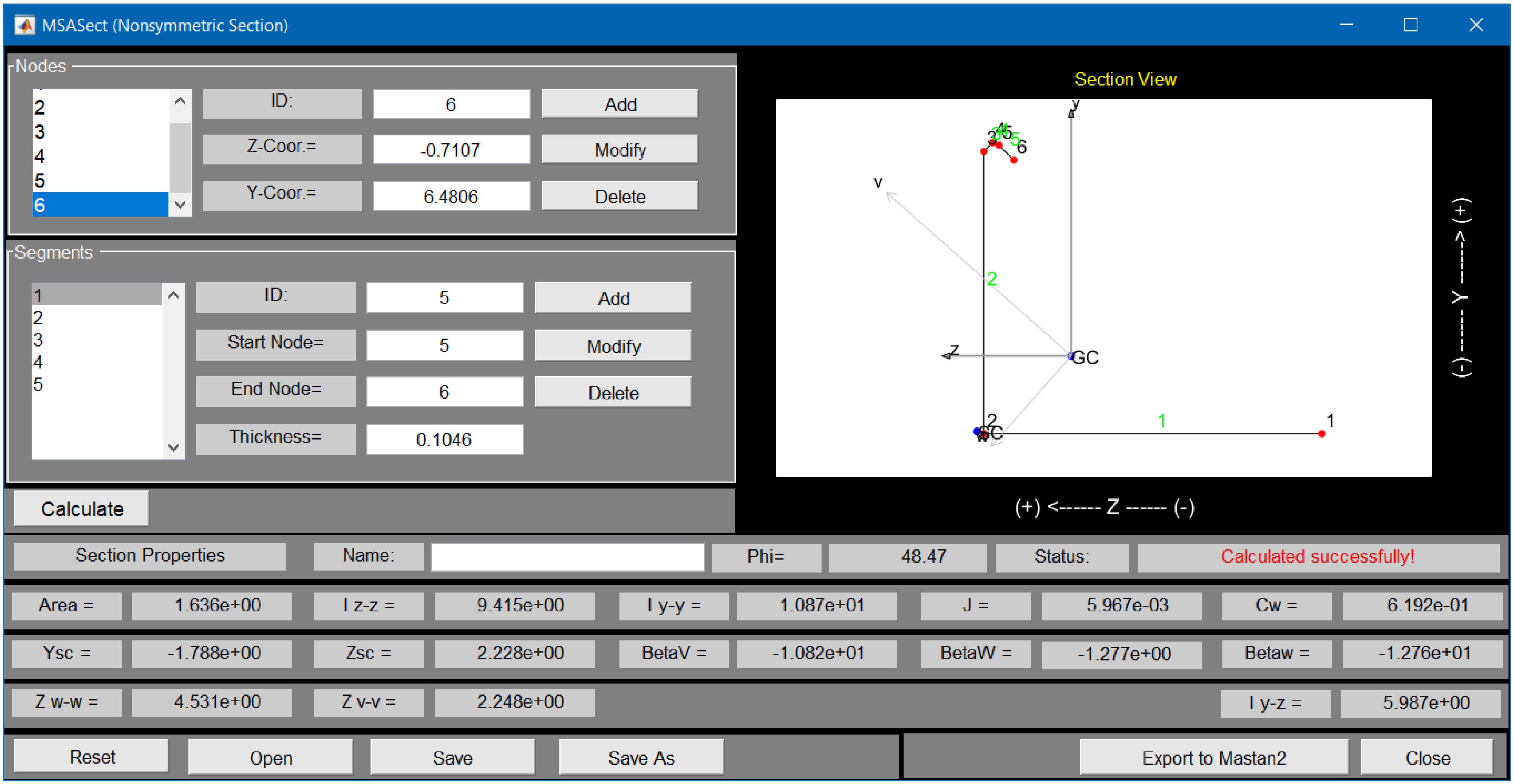
5) Click **Export to MASTAN2** to copy values to main program. Then click **Close** to return.

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

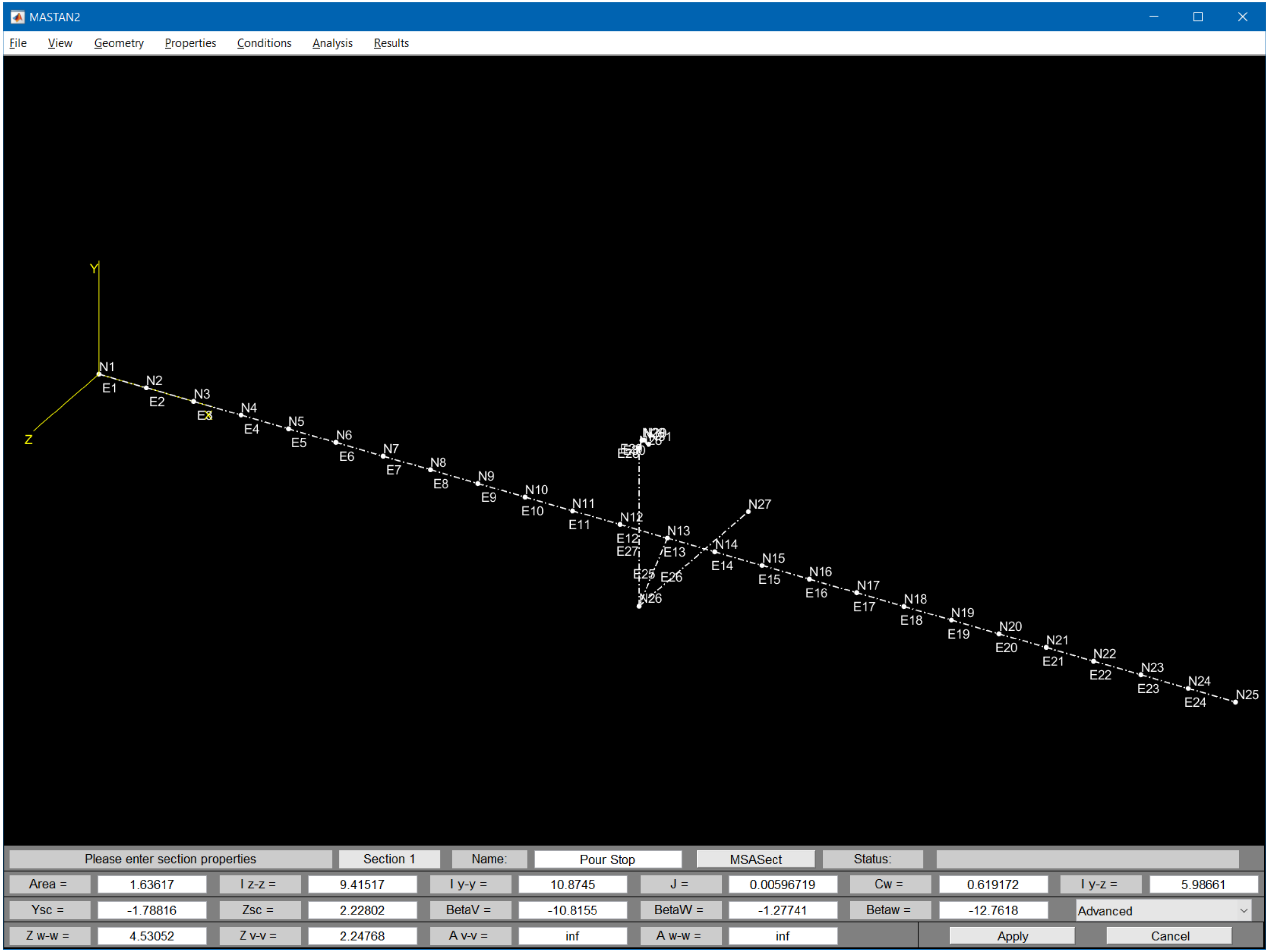

MSASect (Nonsymmetric Section)
Nodes
2
3
4
5
6
ID:
6
Add
Z-Coor.=
-0.7107
Modify
Y-Coor.=
6.4806
Delete
Segments
1
ID:
1
Add
Start Node=
1
Modify
End Node=
2
Delete
Thickness=
0.1046
Calculate
Section View
(+) <------ Z ------ (-)
(-) ------ Y ------> (+)
Section Properties
Name:
Phi=
Status:
Area =
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
ID:
6
Add
Z-Coor.=
-0.7107
Modify
Y-Coor.=
6.4806
Delete
Segments
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

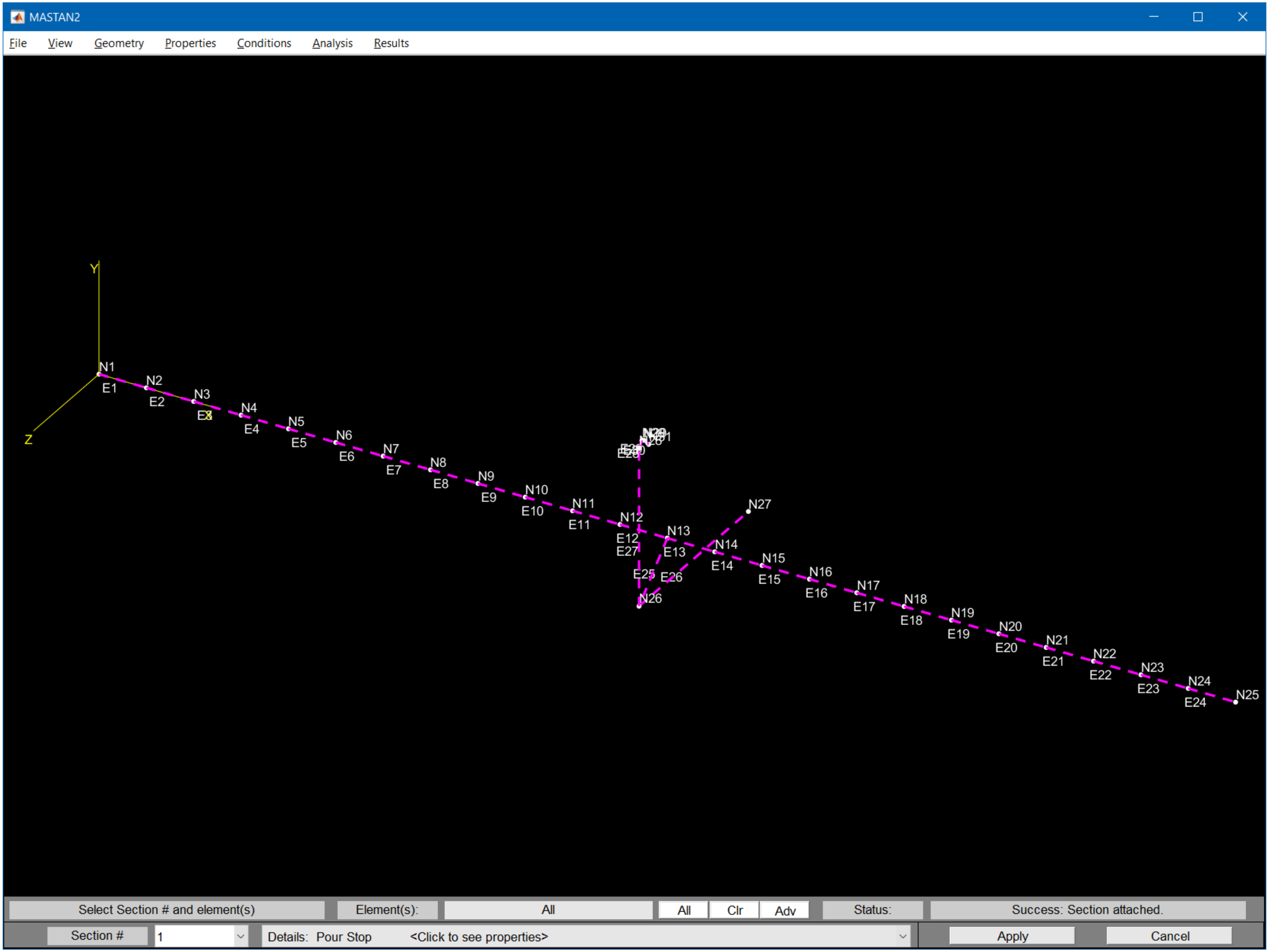
Section Properties - Assigning

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
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
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**.

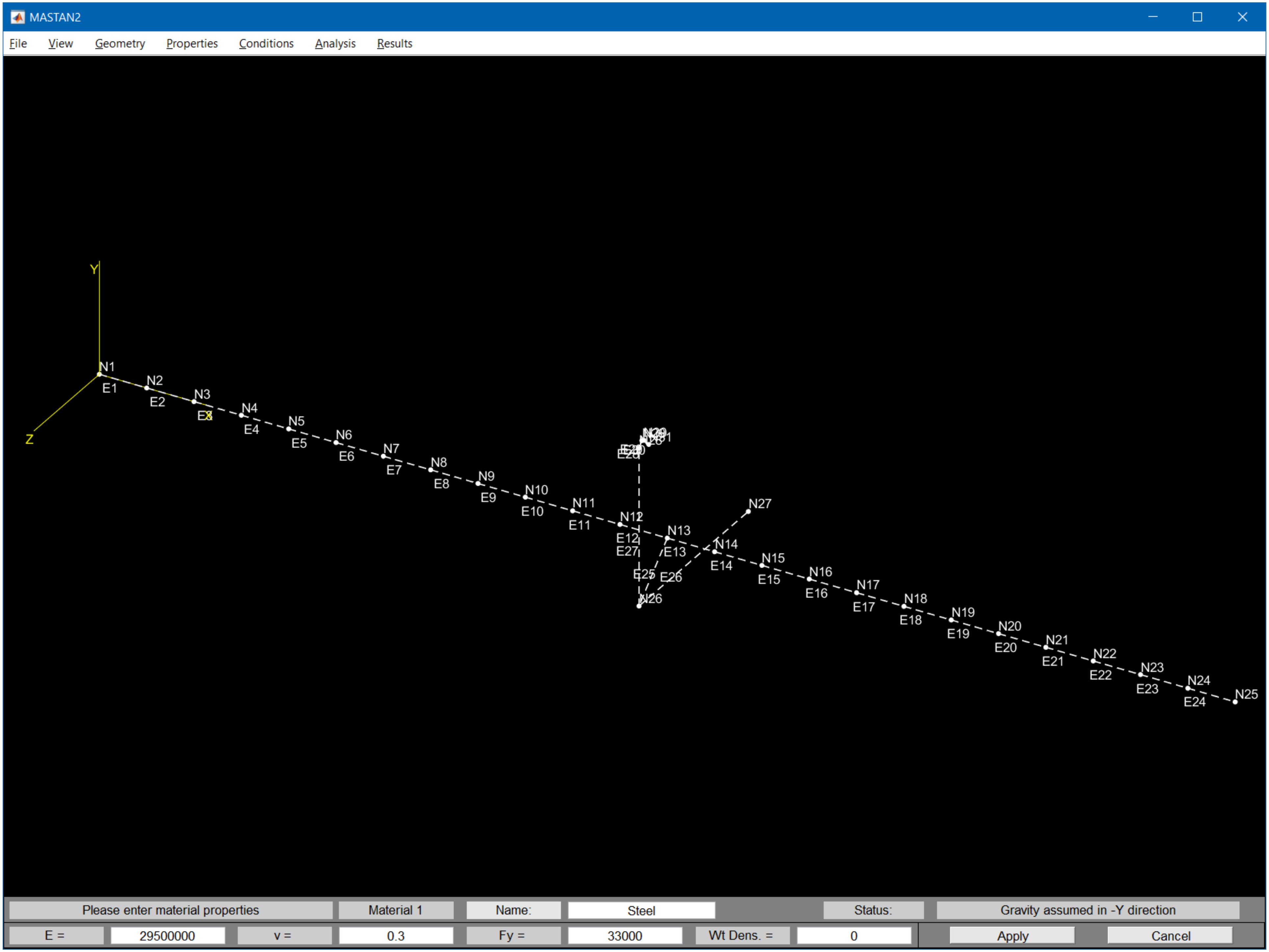
3) Click on the **Apply** button to save Material #1.

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

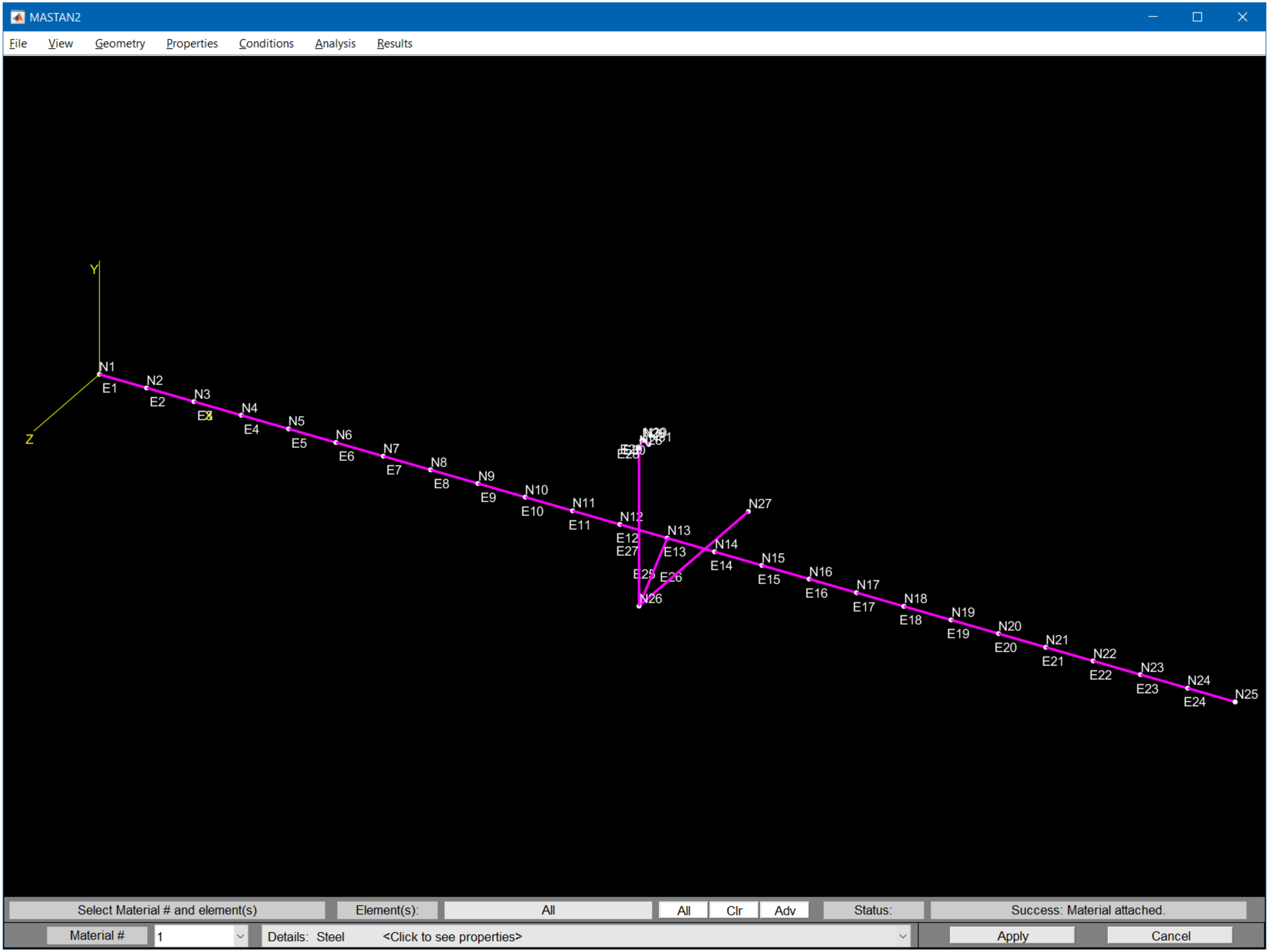
5) At the bottom menu bar, create the list of elements to be assigned the properties of Material 1 by clicking on the **All** button to the right of **Elements:**. Click on the **Apply** button.

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



MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
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
N26
N27
E1
E2
E4
E5
E6
E7
E8
E9
E10
E11
E12
E27
E13
E14
E15
E16
E17
E18
E19
E20
E21
E22
E23
E24
E25
E26
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**.

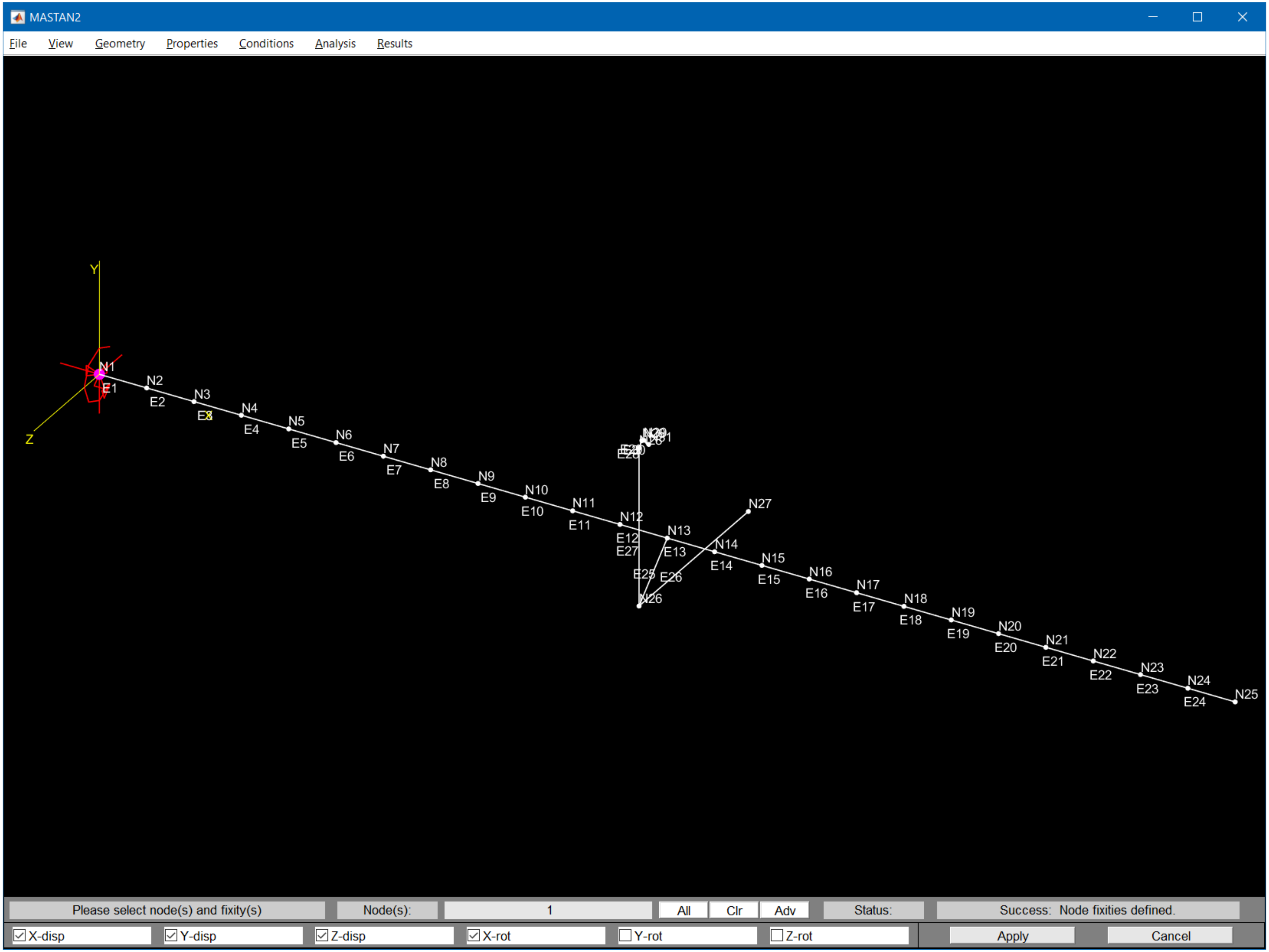
4) Click on the **Apply** button.

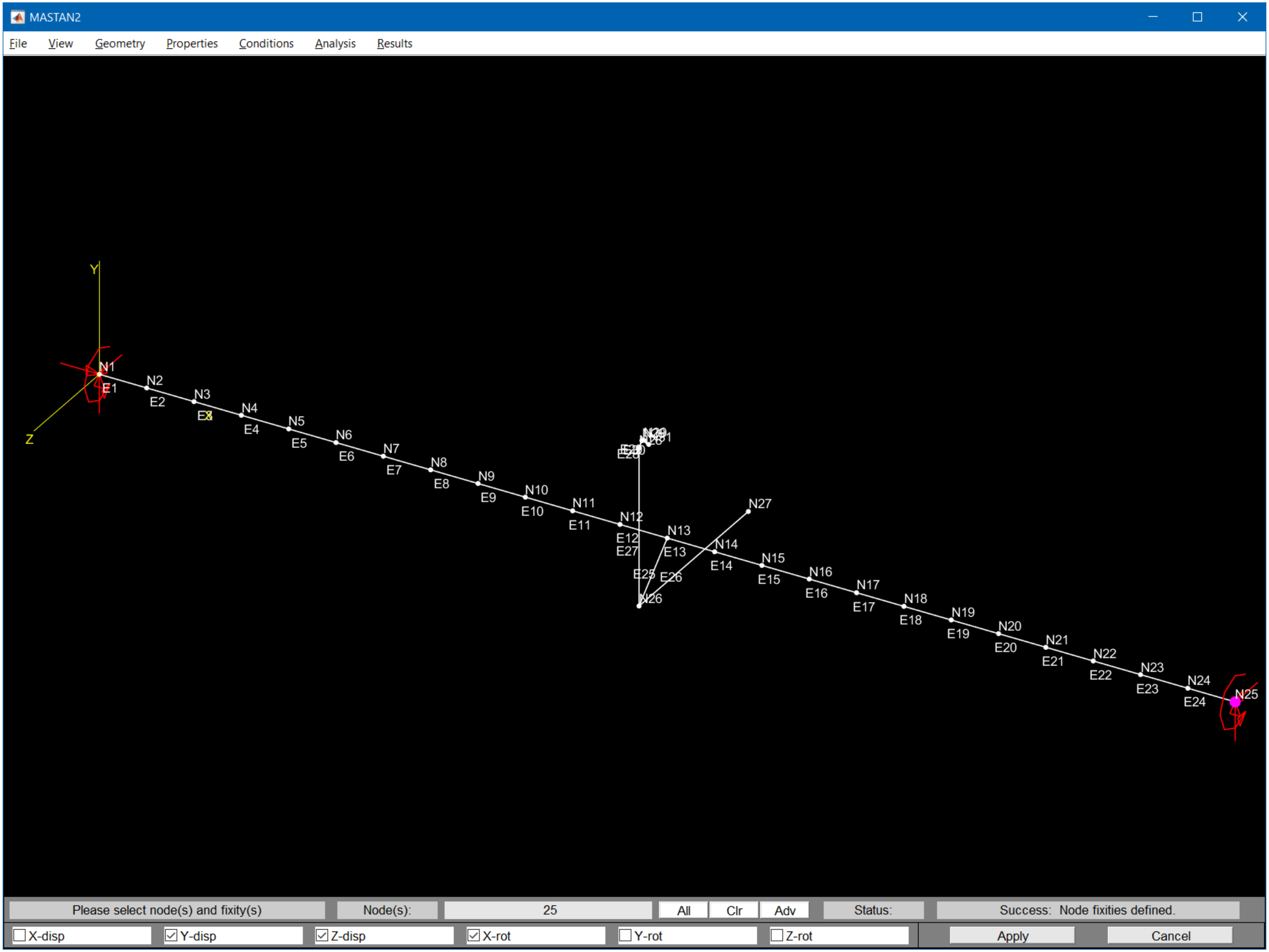
5) At the bottom menu bar, define a roller support with torsion fixed support by clicking in the **check boxes** just to the left of **X-disp** to release it and leave **Y-disp**, **Z-disp**, and **X-rot** constrained.

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
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
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
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
E27
E13
E14
E15
E16
E17
E18
E19
E20
E21
E22
E23
E24
E25
E26
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.

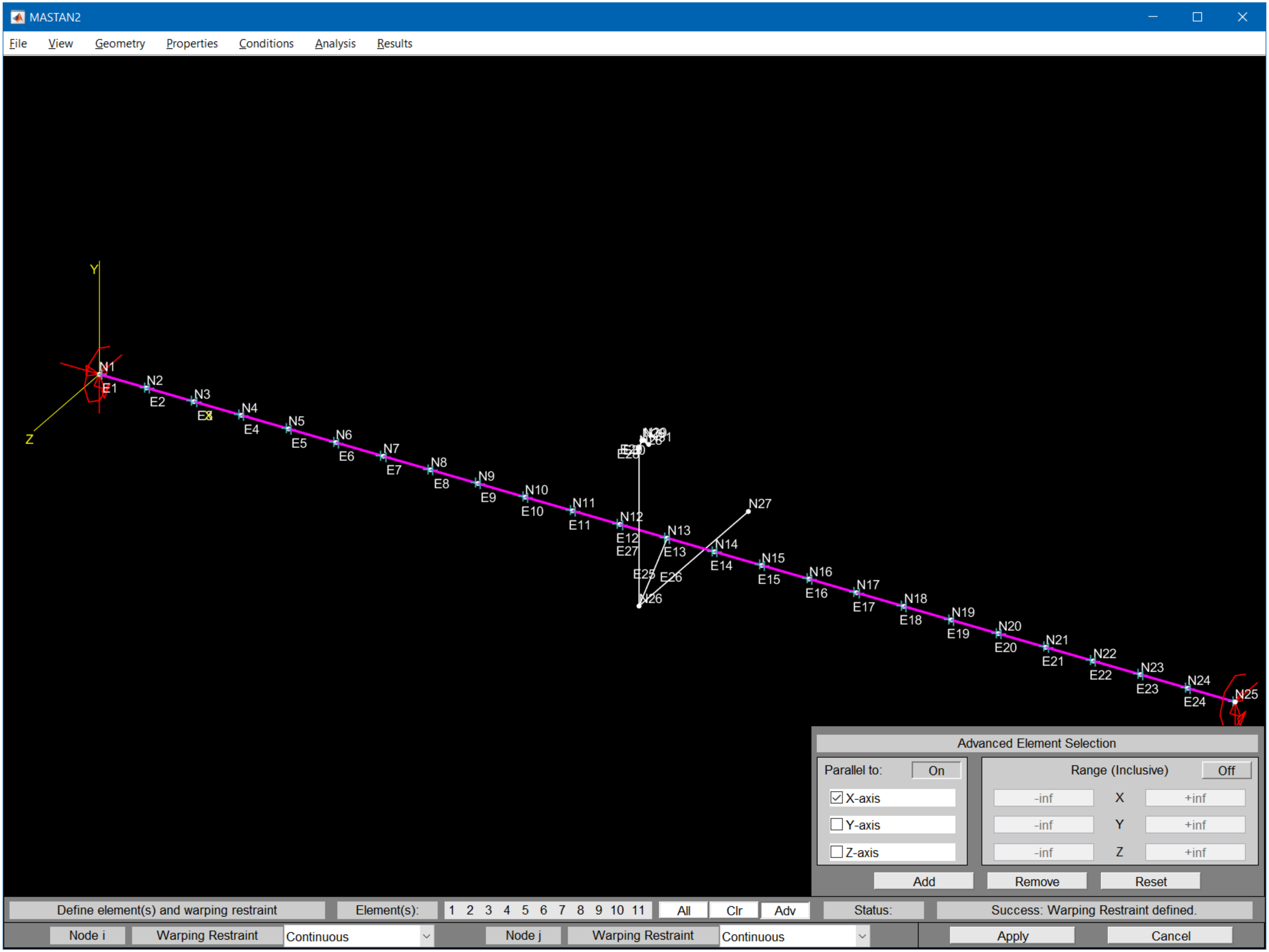
5) Click **Clr** to empty the list of elements. Click on the left most element of the beam.

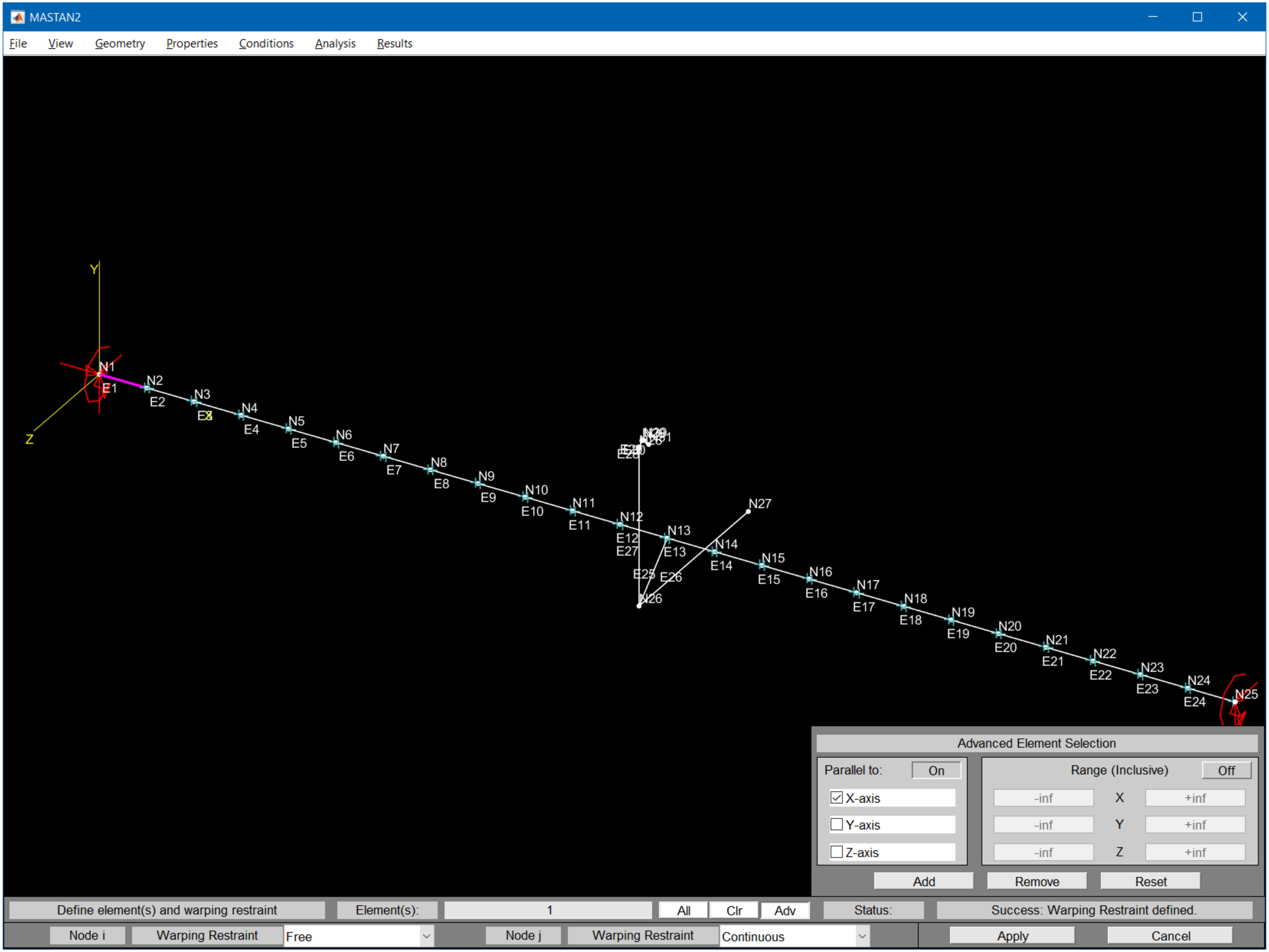
6) Click on the menu to the right of **Warping Restraint for Node i** and set the value to **Free**. Node j is set from the previous step. Click on the **Apply** button.

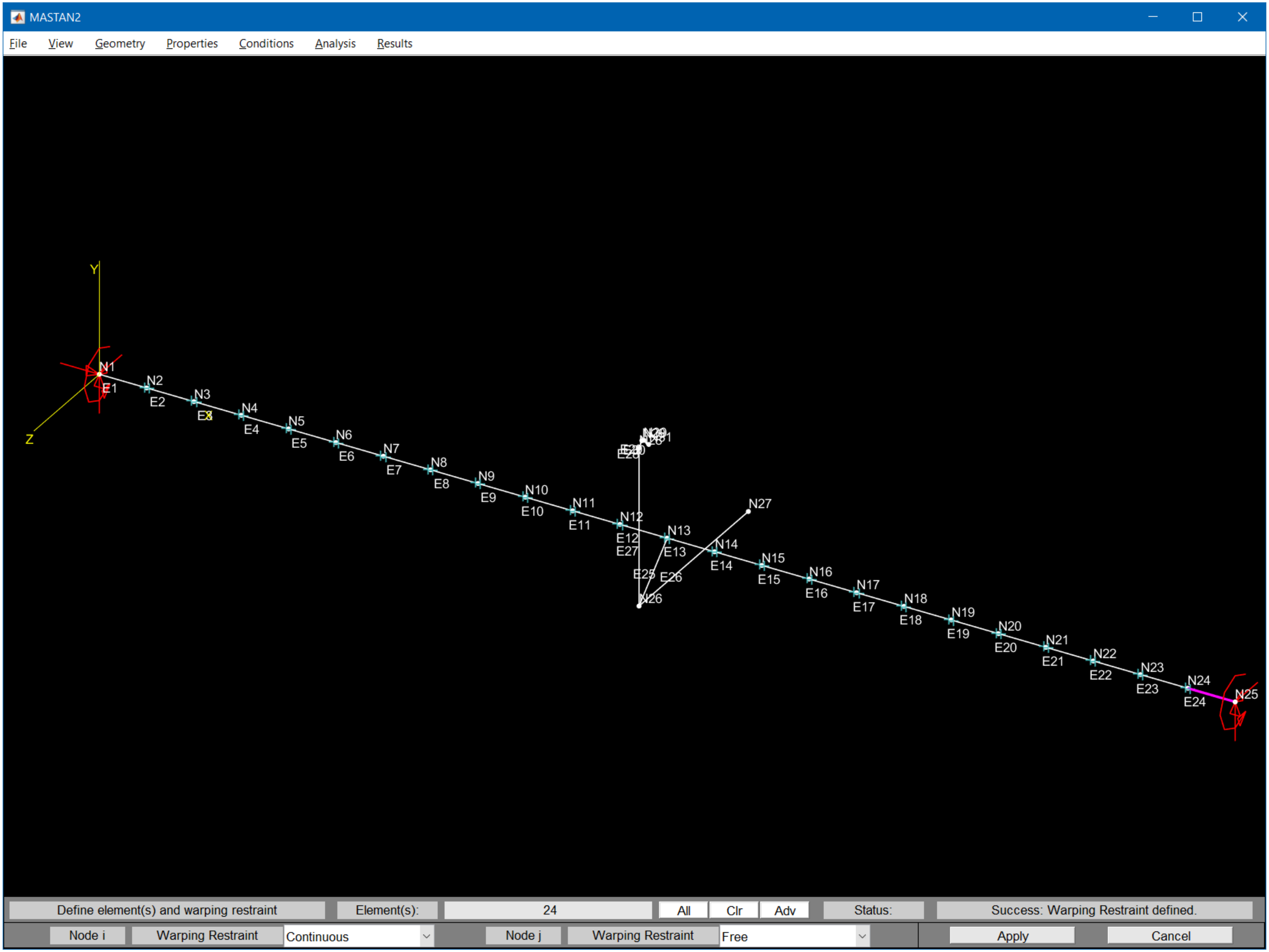
7) Click **Clr** to empty the list of elements. Click on the right most element of the beam. This might require you to click **Adv** to close the pop-up menu.

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
Advanced Element Selection
Parallel to:
On
X-axis
Y-axis
Z-axis
Range (Inclusive)
Off
-inf
X
+inf
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
Apply
Cancel

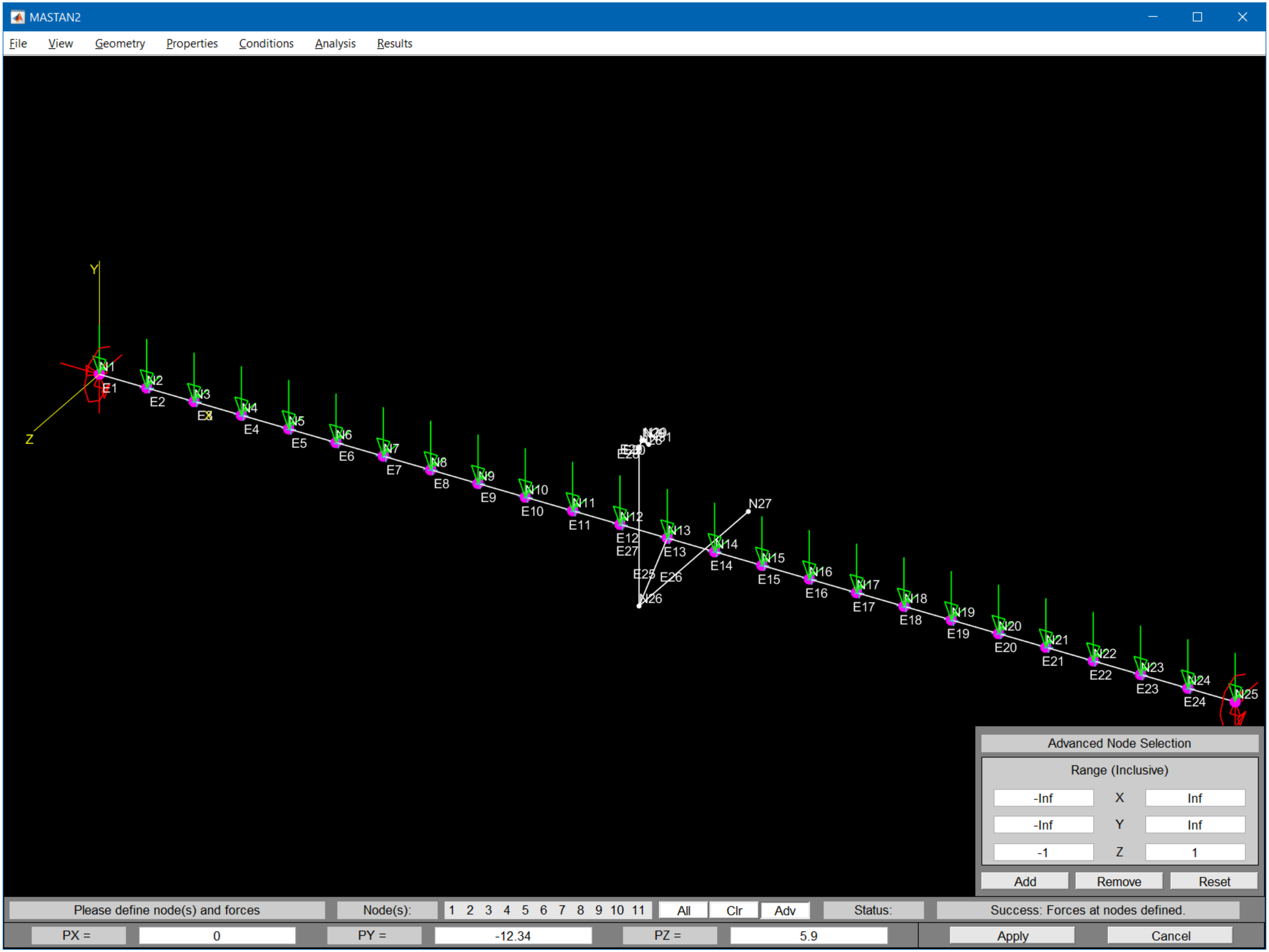

MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Advanced Element Selection
Parallel to:
On
X-axis
Y-axis
Z-axis
Range (Inclusive)
Off
-inf
X
+inf
Y
Z
Add
Remove
Reset
Define element(s) and warping restraint
Element(s):
1
All
Clr
Adv
Status:
Success: Warping Restraint defined.
Node i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

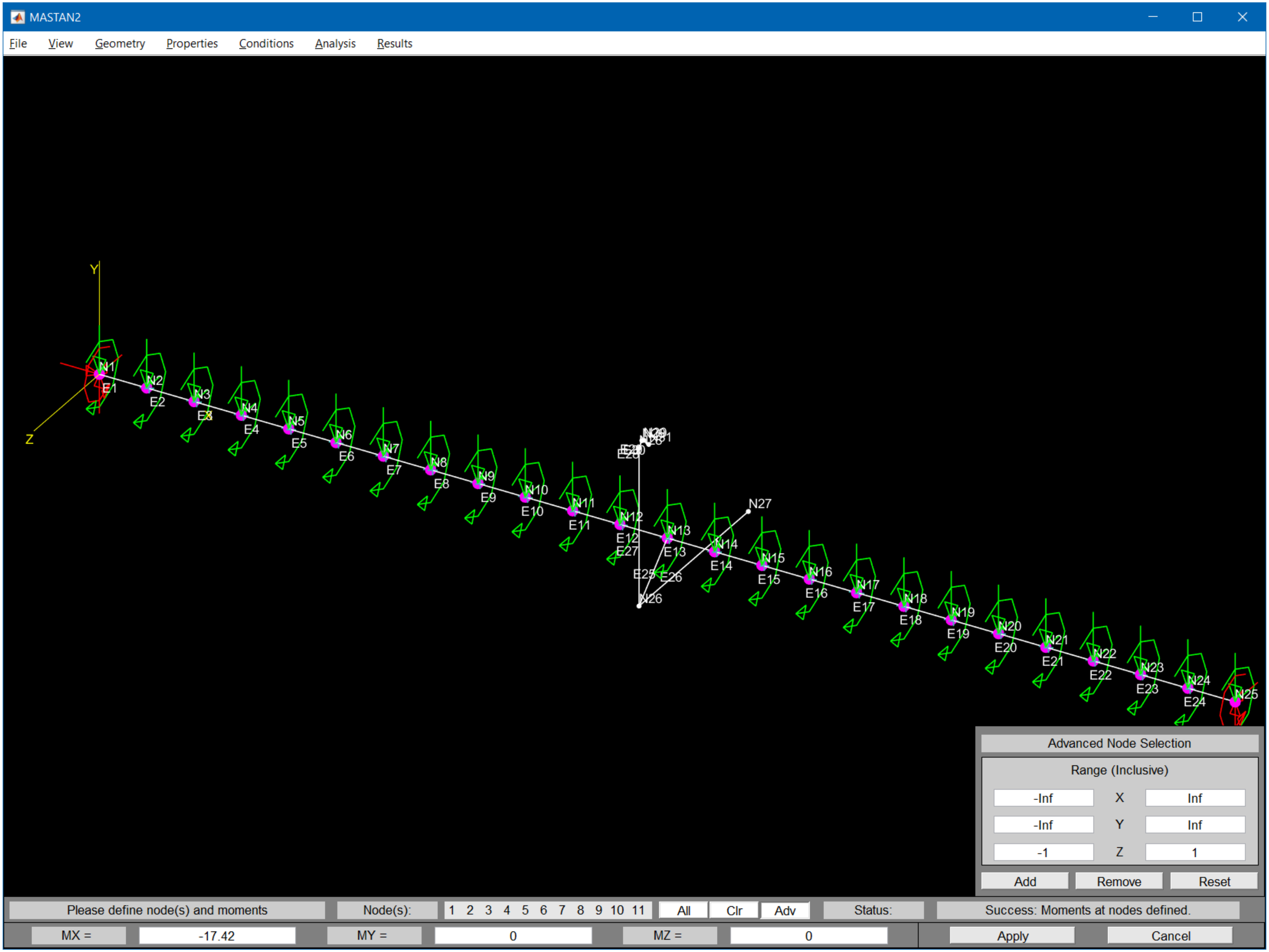

MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
Define element(s) and warping restraint
Element(s):
24
All
Clr
Adv
Status:
Success: Warping Restraint defined.
Node i
Warping Restraint
Continuous
Node j
Warping Restraint
Free
Apply
Cancel

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
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

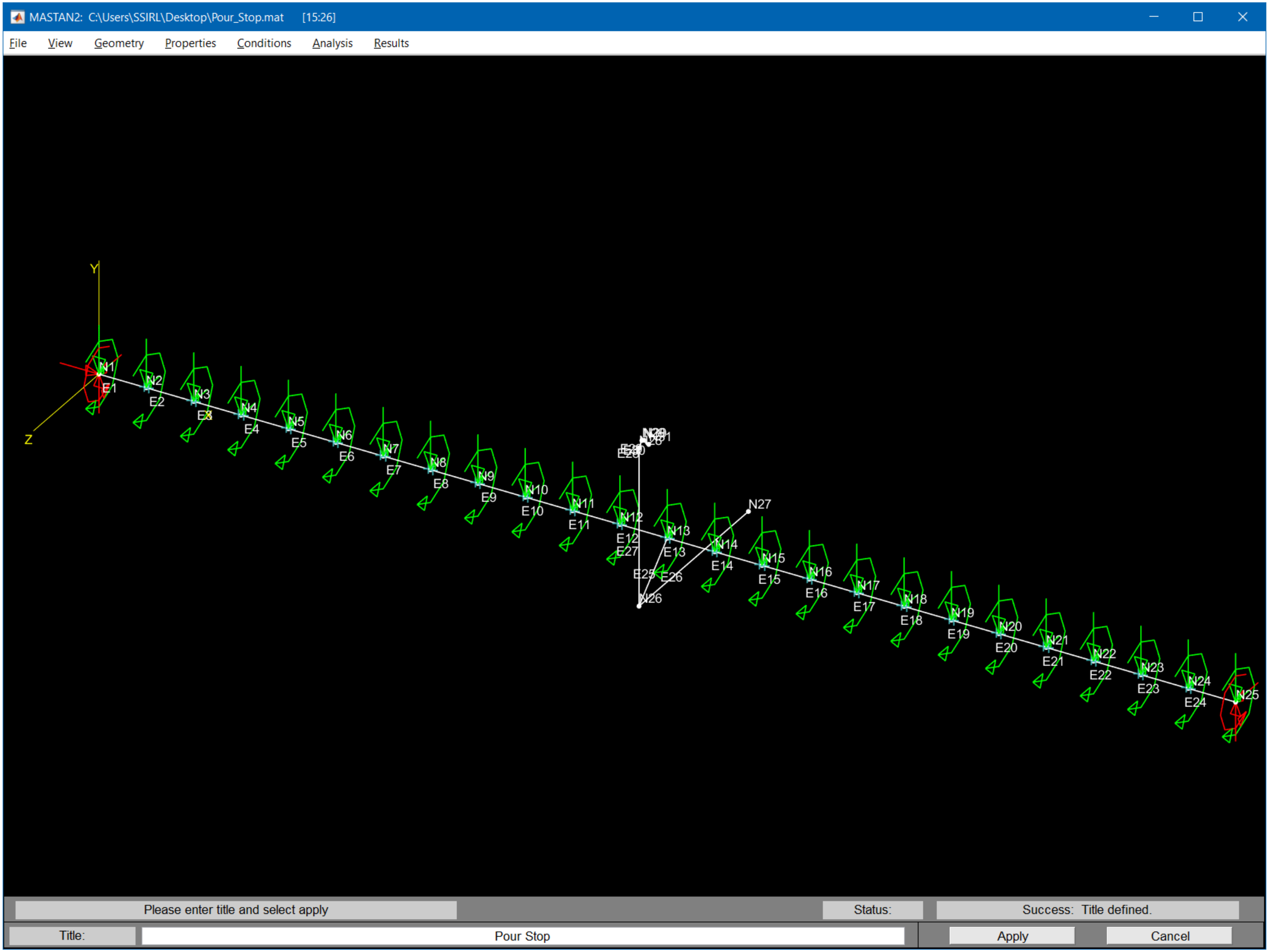
Naming and Saving

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
Y
Z
N27
N26
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**.

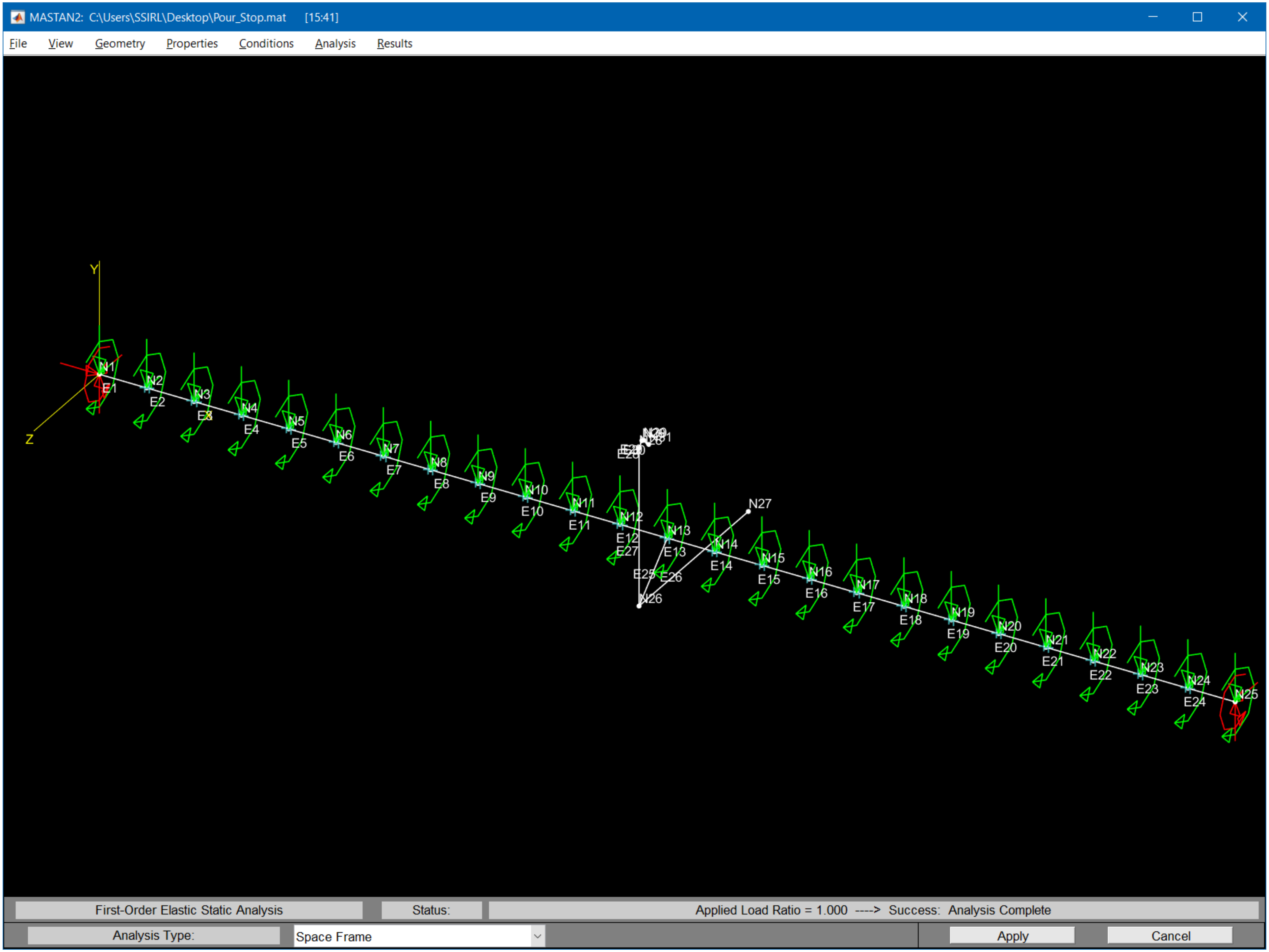
7) On the undeflected shape, click on the midspan node of interest, node **13**, and the displacements for base 6 degree of freedoms are provided in the bottom menu bar.

Results:

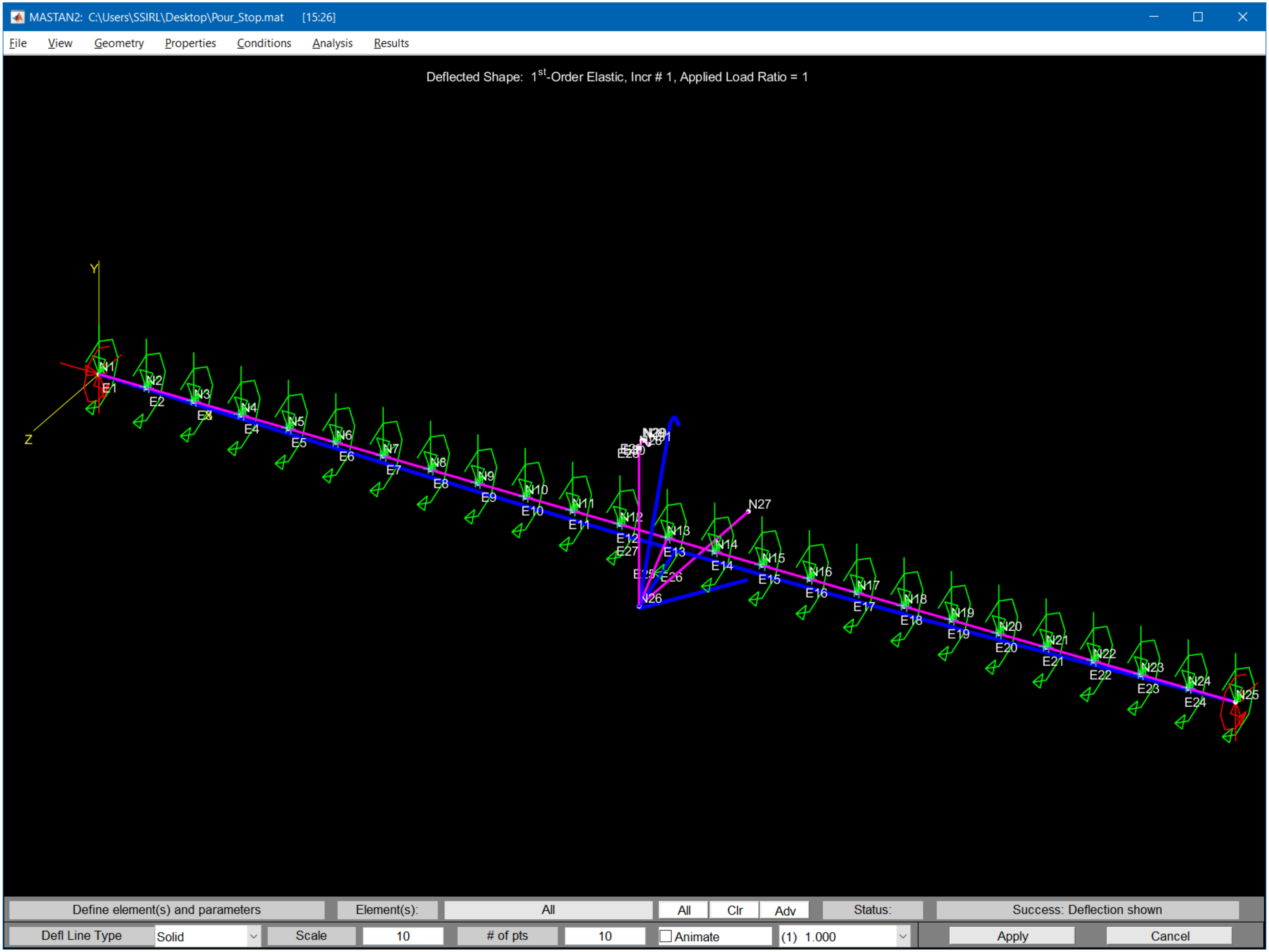
Disp X	Disp Y	Disp Z	Rot X	Rot Y	Rot Z
~0	-0.08033	-0.0598	-0.03472	~0	~0

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.

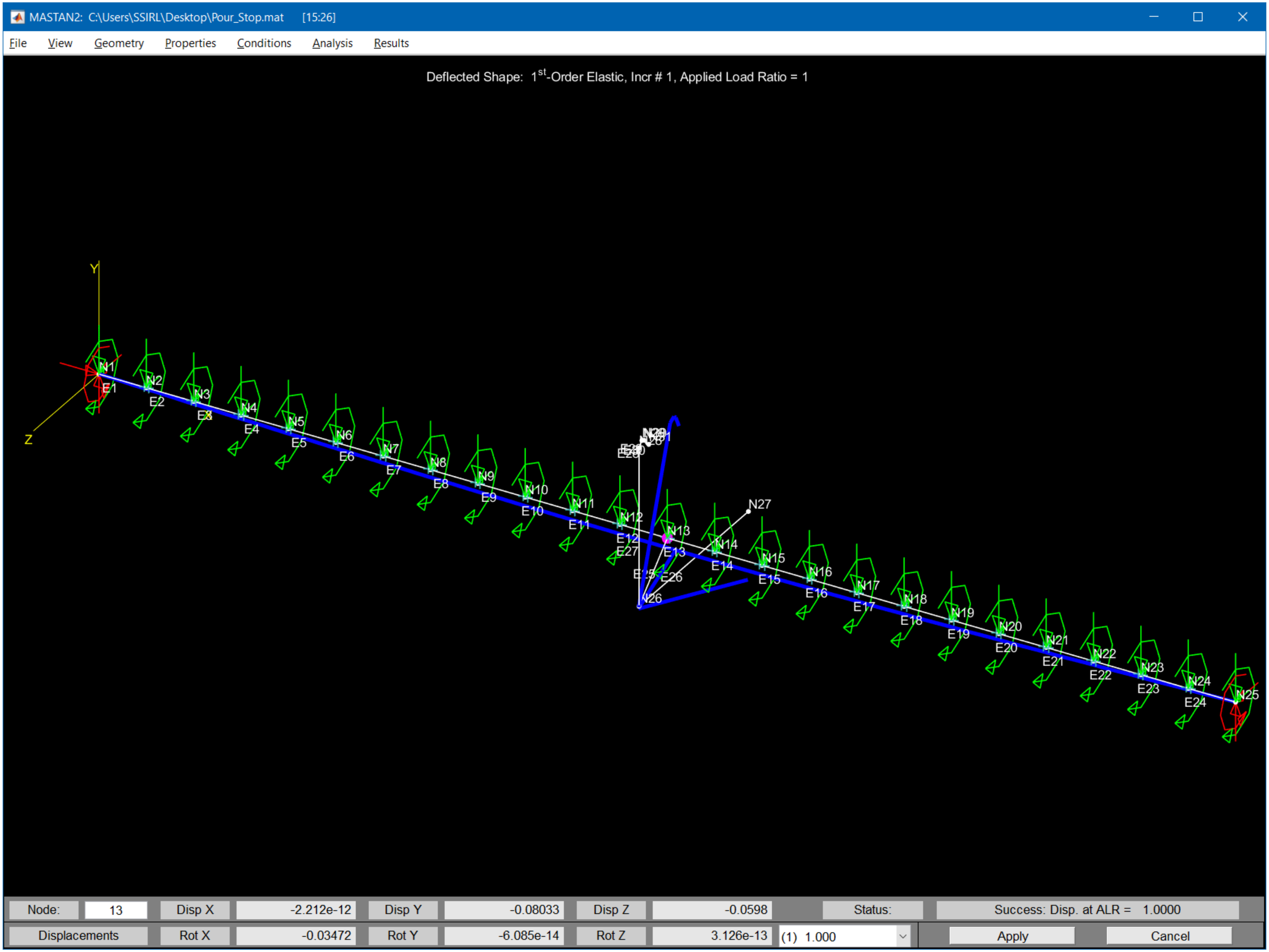
MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [15:41]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
N27
N26
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
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 E26 E27
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**.

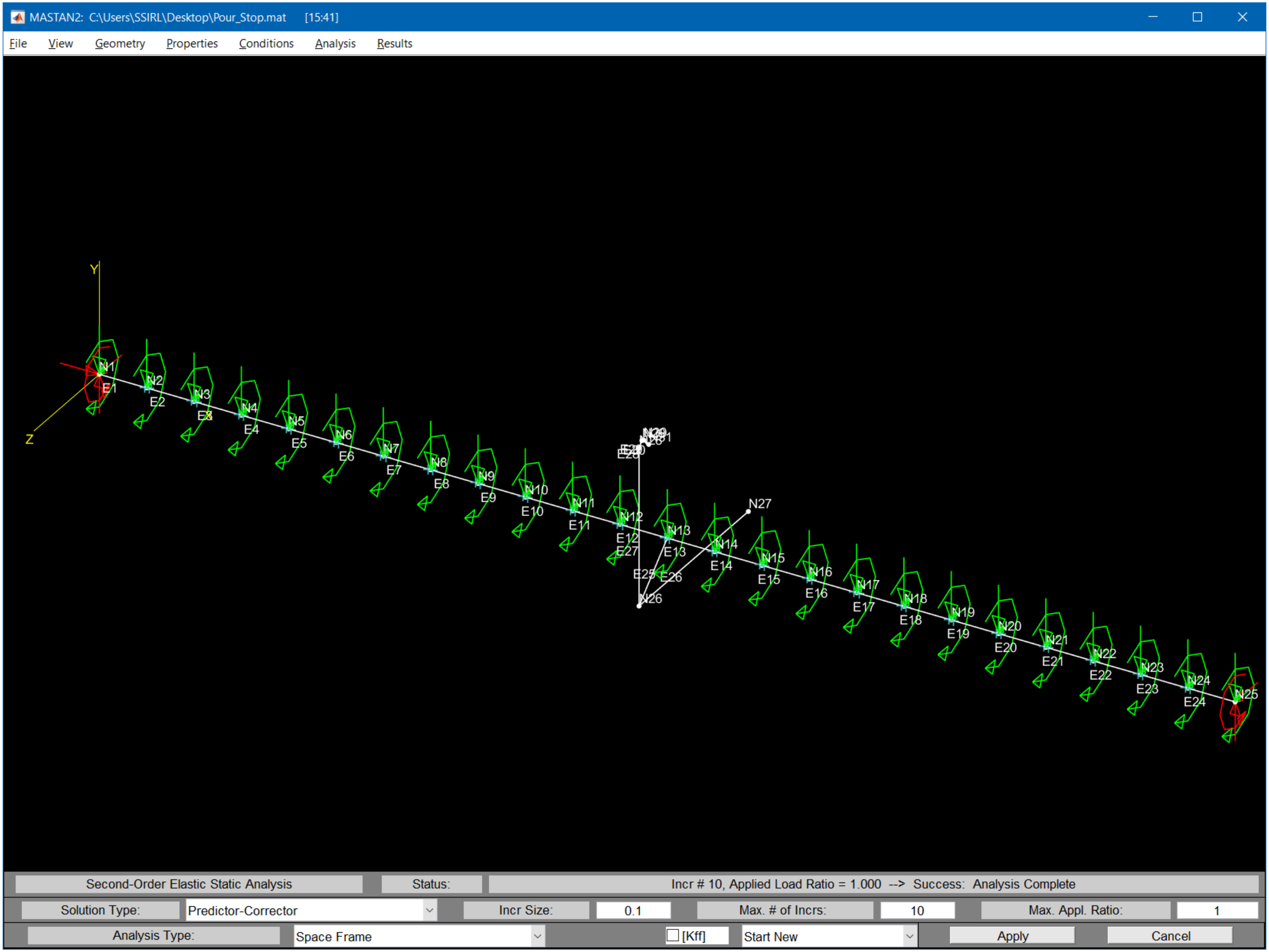
7) On the undeflected shape, click on the midspan node of interest, node **13**, and the displacements for base 6 degree of freedoms are provided in the bottom menu bar.

Results:

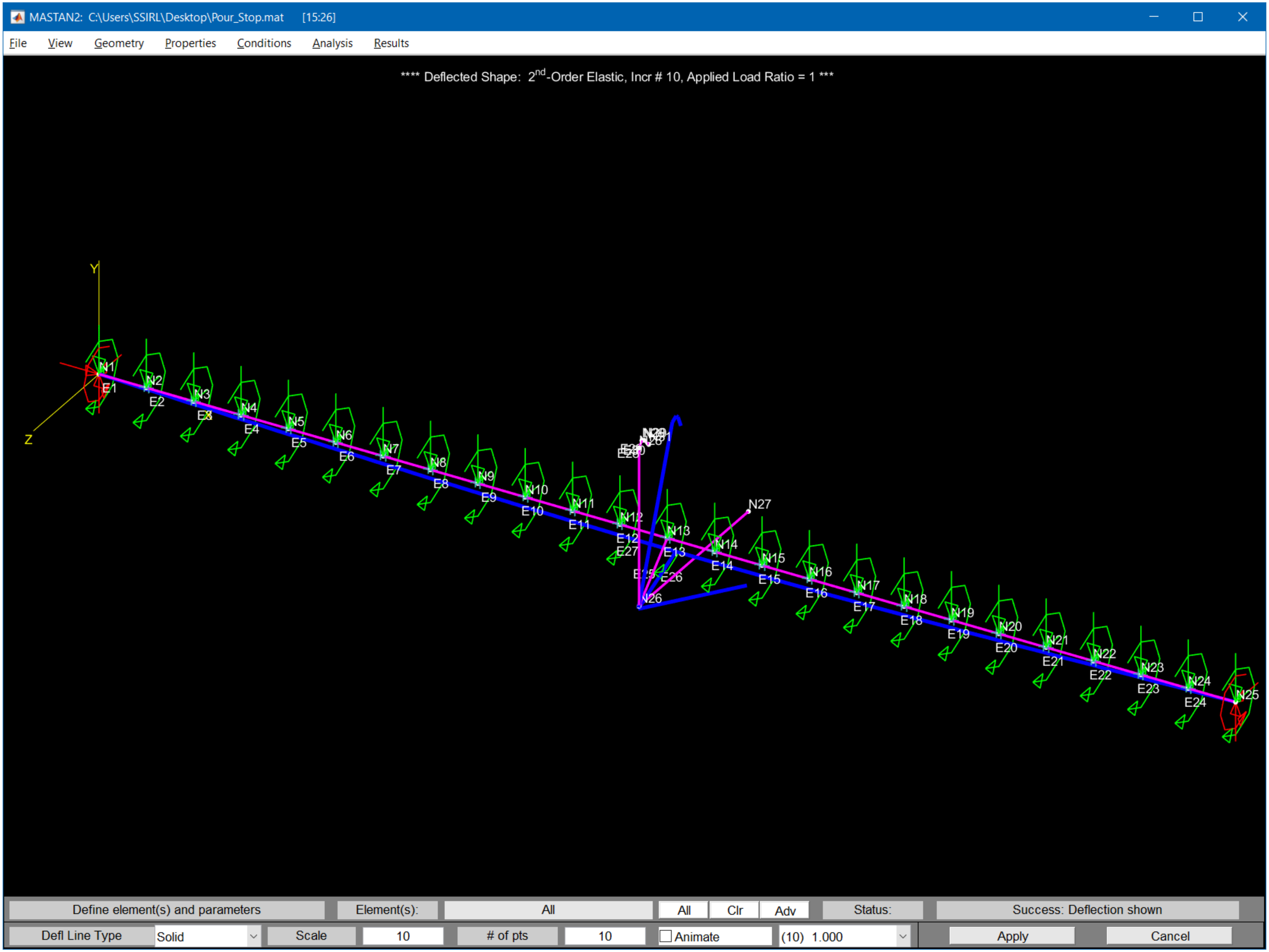
Disp X	Disp Y	Disp Z	Rot X	Rot Y	Rot Z
-2.992e-4	-0.08758	-0.06218	-0.03726	~0	~0

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.

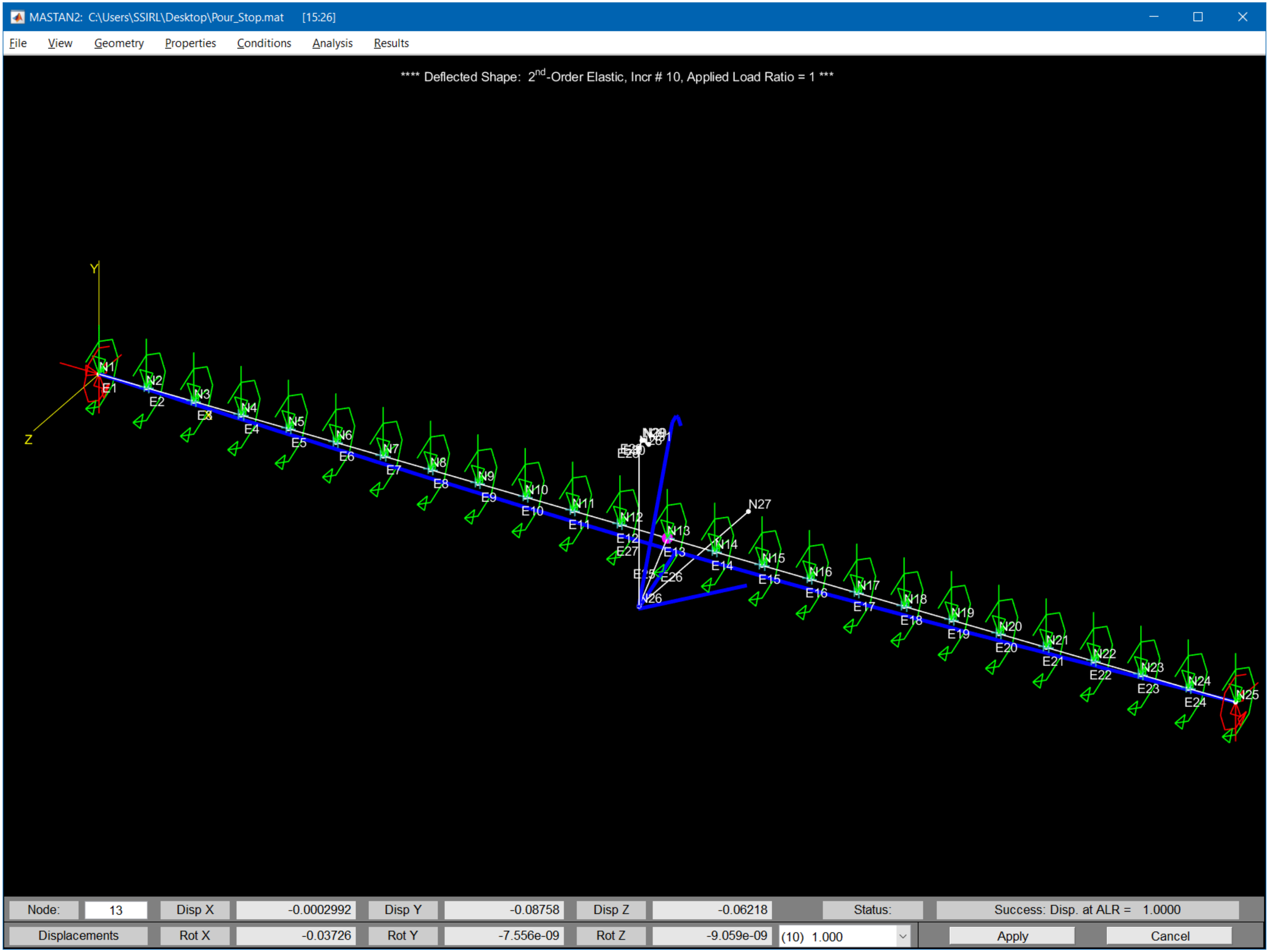
MASTAN2


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
N3
E3
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
E26
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: 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 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 E26 E27
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

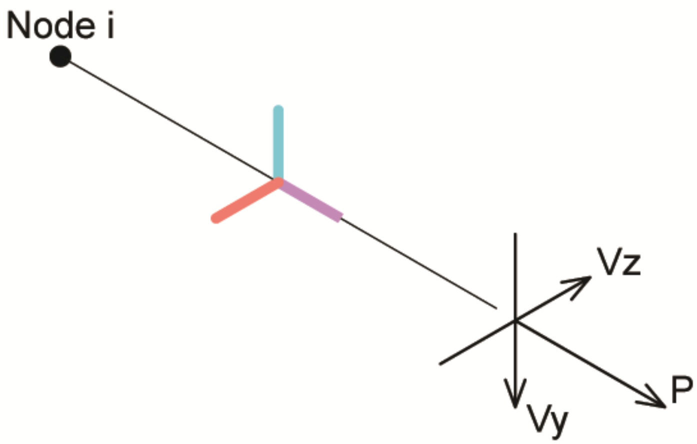
The next section of this tutorial will provide instructions on how it is possible to calculate the internal stresses from the MASTAN2 output information. After showing how to locate the necessary information from within MASTAN2, the next few pages will provide background and the sign conventions that 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 2^{nd}-order analysis internal forces.

Notes:

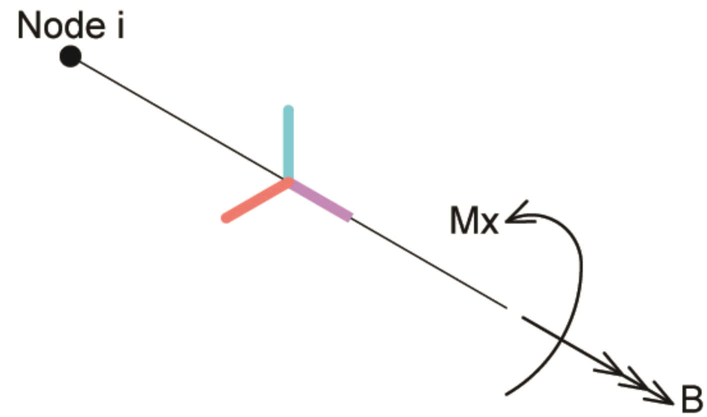
The internal forces taken from MASTAN2 are presented with the sign convention shown on the following page. Also included are the required transformations to get to a principal orientation.

The stress calculations will account for this information.

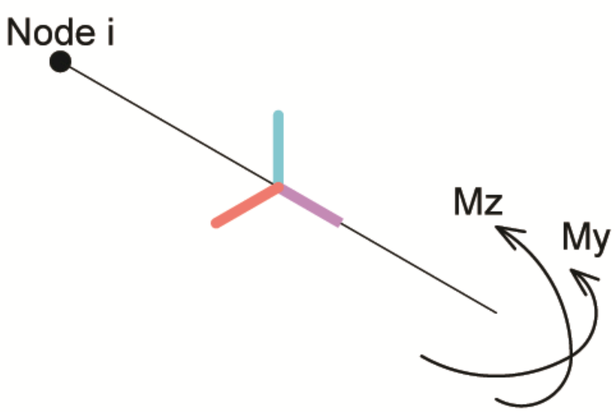
Sign Conventions



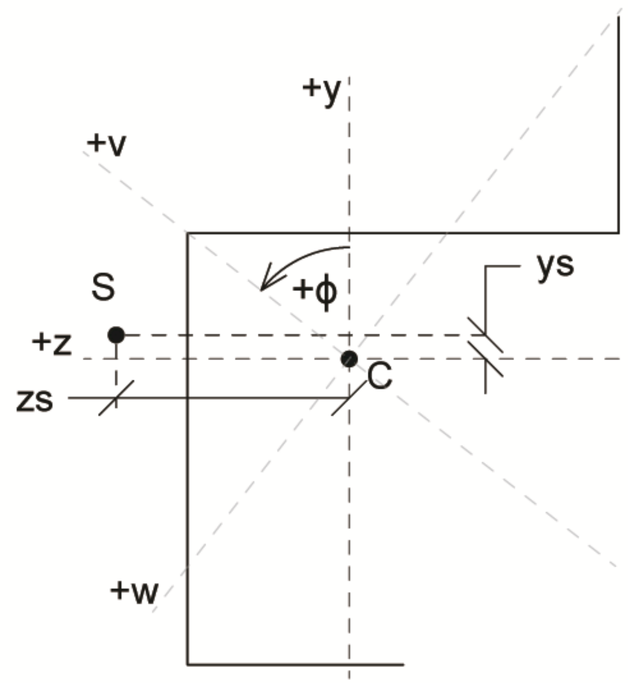
$$\begin{bmatrix} V_v \\ V_w \end{bmatrix} = \begin{bmatrix} cos\phi & sin\phi \\ -sin\phi & cos\phi \end{bmatrix} \cdot \begin{bmatrix} V_y \\ V_z \end{bmatrix}$$



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



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



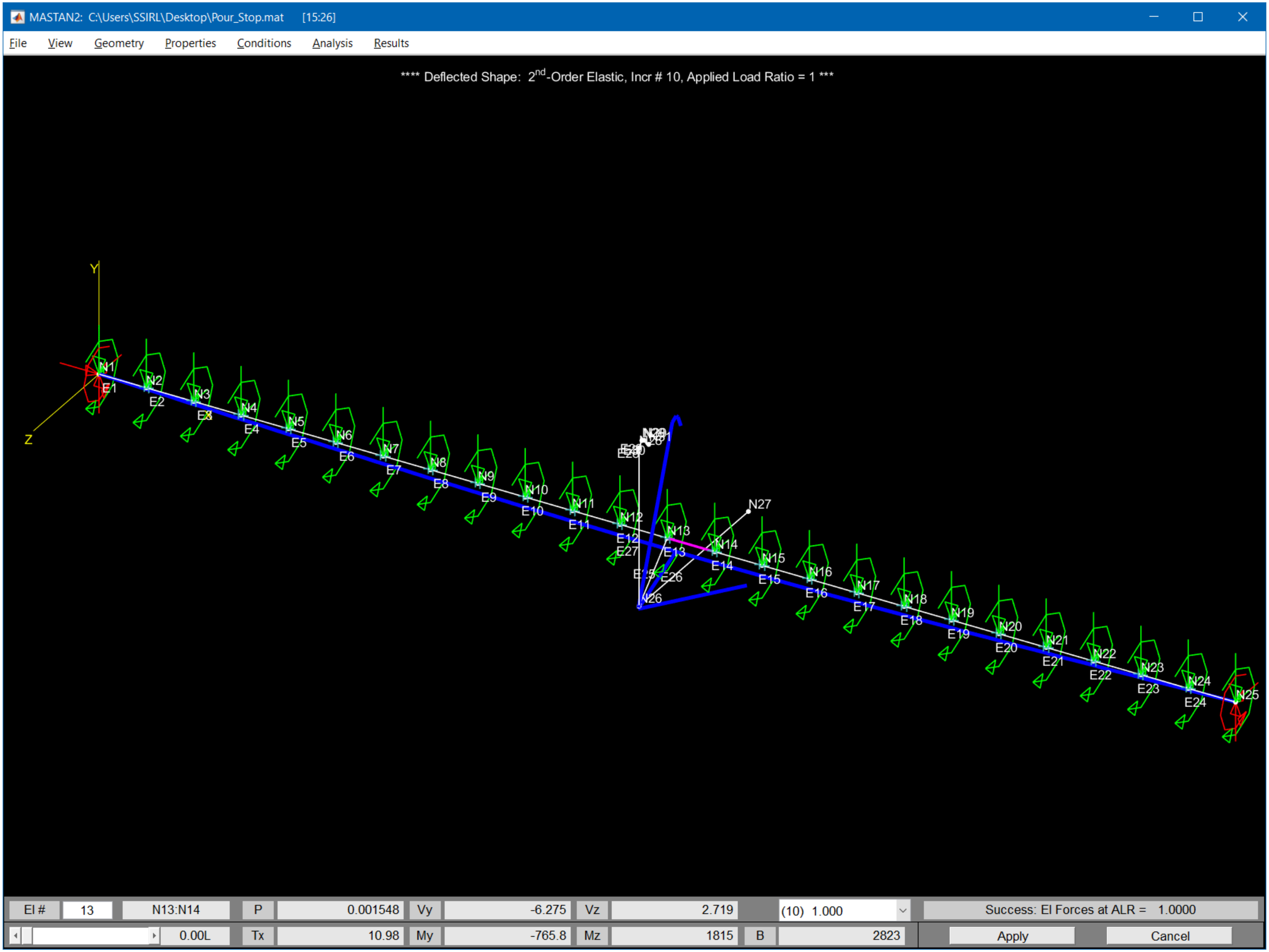
Getting Internal Forces

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.

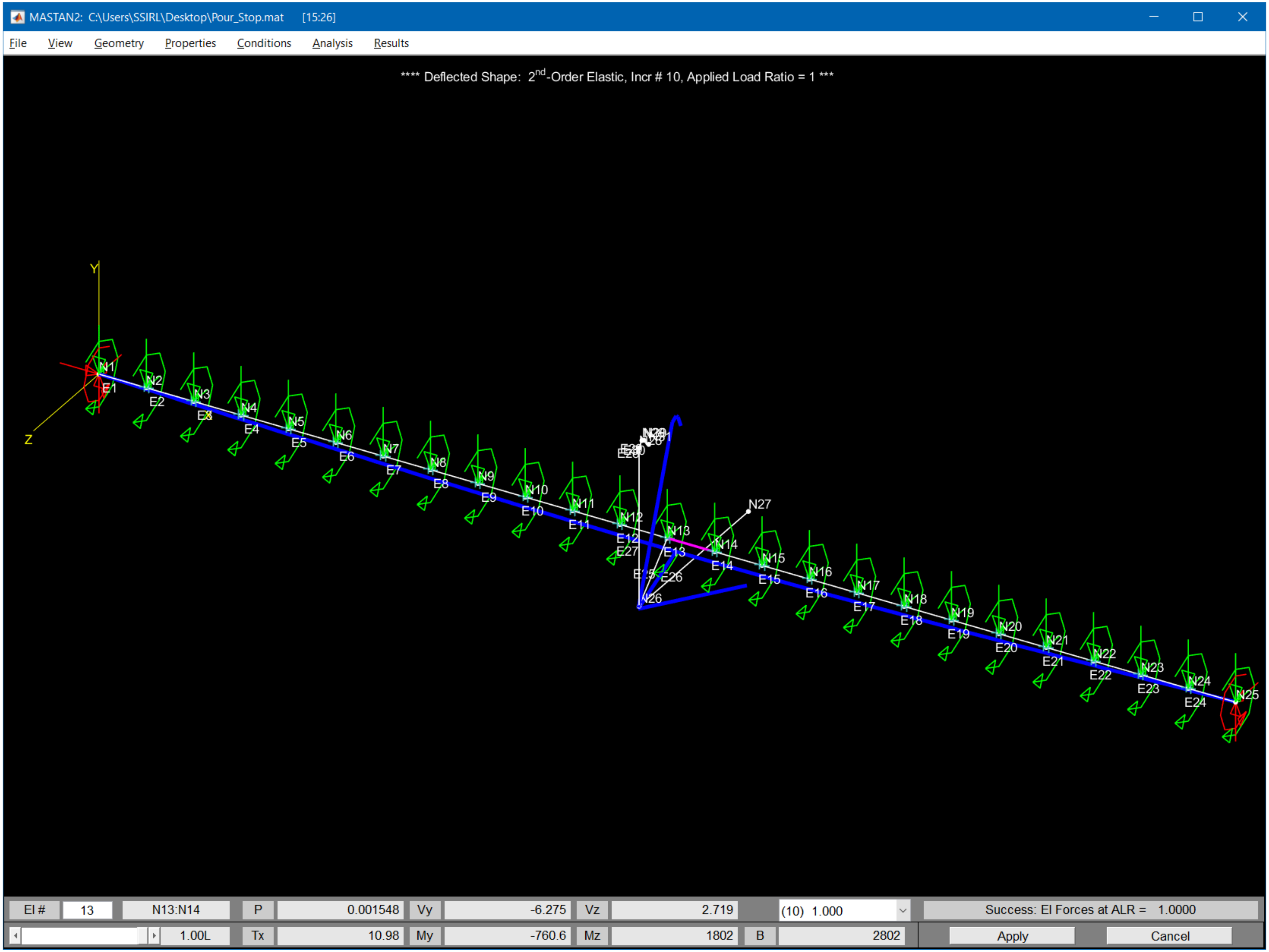
Forces at the start of element 1

Forces at the end of element 1

MASTAN2

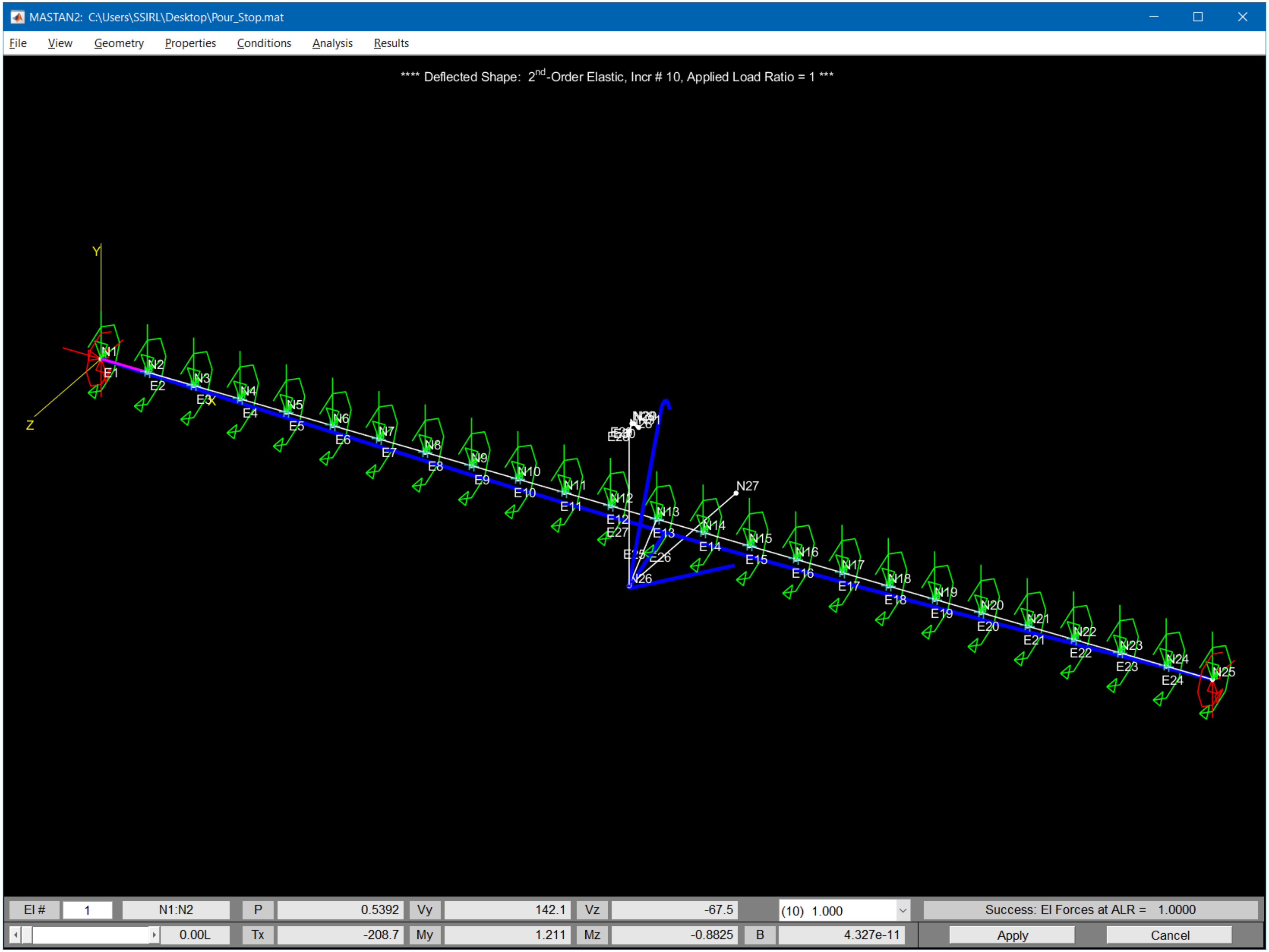

MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [15:26]
File
View
Geometry
Properties
Conditions
Analysis
Results
**** Deflected Shape: 2nd-Order Elastic, Incr # 10, Applied Load Ratio = 1 ***
El #
13
N13:N14
P
0.001548
Vy
-6.275
Vz
2.719
(10) 1.000
Success: El Forces at ALR = 1.0000
0.00L
Tx
10.98
My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

MASTAN2

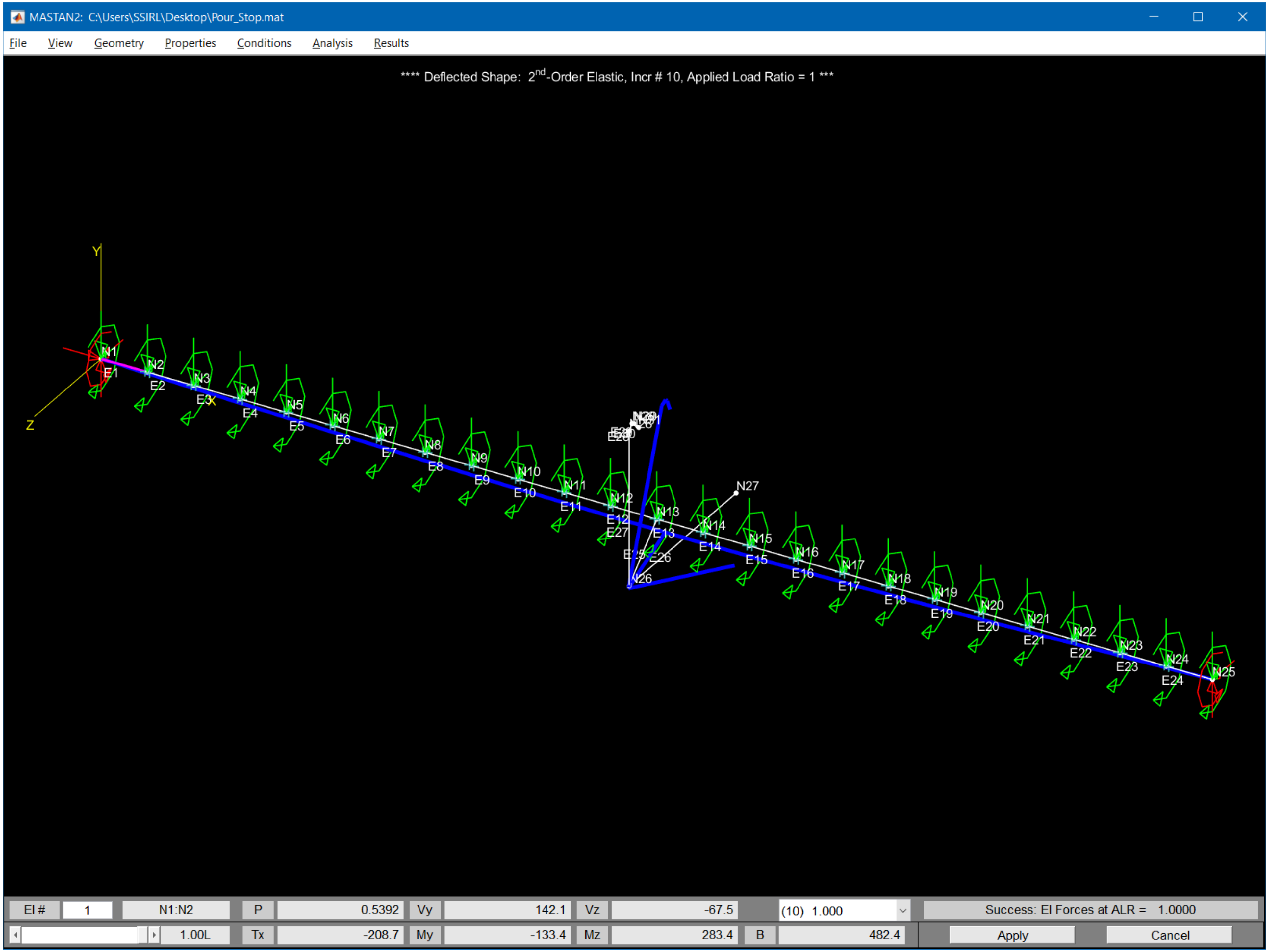


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [15:26]
File View Geometry Properties Conditions Analysis Results
**** Deflected Shape: 2nd-Order Elastic, Incr # 10, Applied Load Ratio = 1 ***
Y
Z
N1 N2 N3 N4 N5 N6 N7 N8 N9 N10 N11 N12 N13 N14 N15 N16 N17 N18 N19 N20 N21 N22 N23 N24 N25 N26 N27
E1 E2 E3 E4 E5 E6 E7 E8 E9 E10 E11 E12 E13 E14 E15 E16 E17 E18 E19 E20 E21 E22 E23 E24 E26 E27
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 ***
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

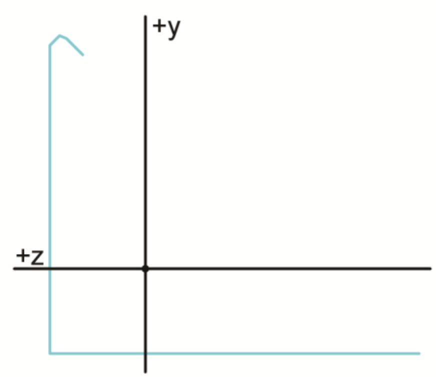
Normal Stresses

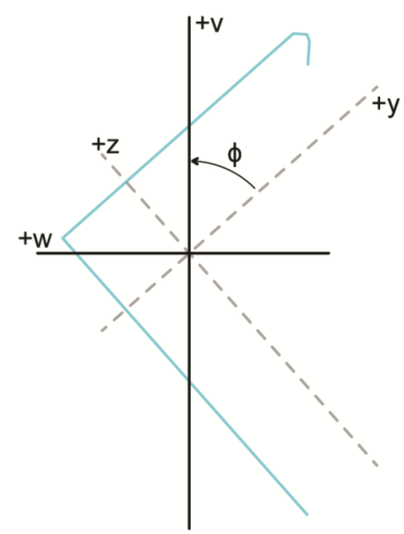
The internal normal stresses can be calculated in the geometric orientation by

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

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.

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

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

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

Calculating Warping Constants

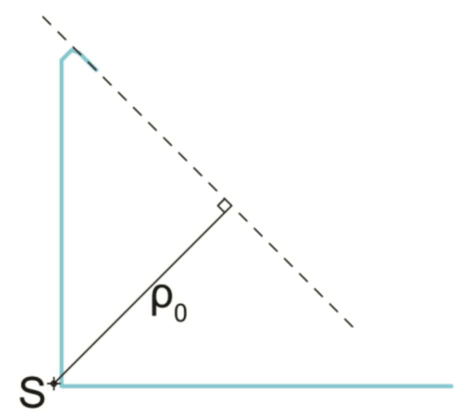
Determining the stresses from warping, both shear stresses and normal stresses, requires the use of warping coefficients. The values can be calculated based on the equations show below with additional details in the following documents. These documents also provide additional information on the stresses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

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

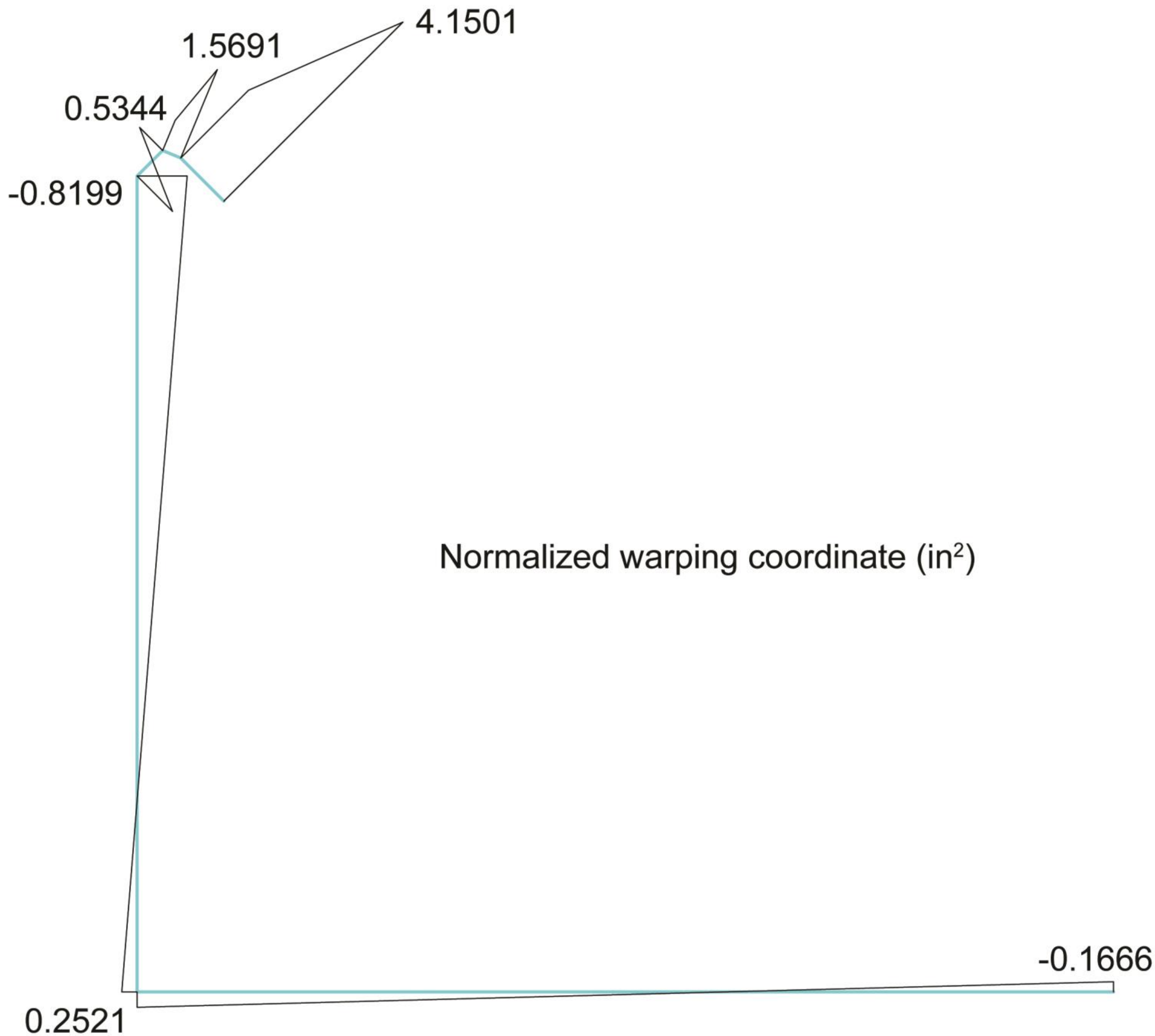
(2) SDI Technical Note 3. (Steel Deck Institute, 2018)

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

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



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.



Normalized warping coordinate (in^2)

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: **T_ω**

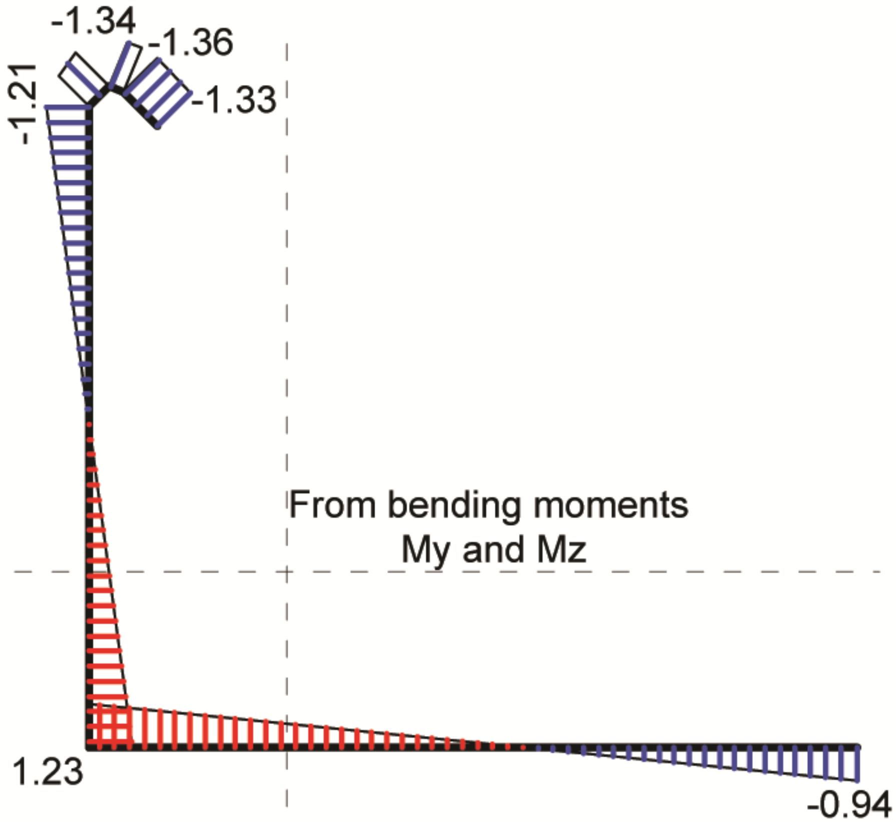
4) Combined shear stress from **Vy, Vz, & T_ω** & Shear stress from twisting: **T_T**

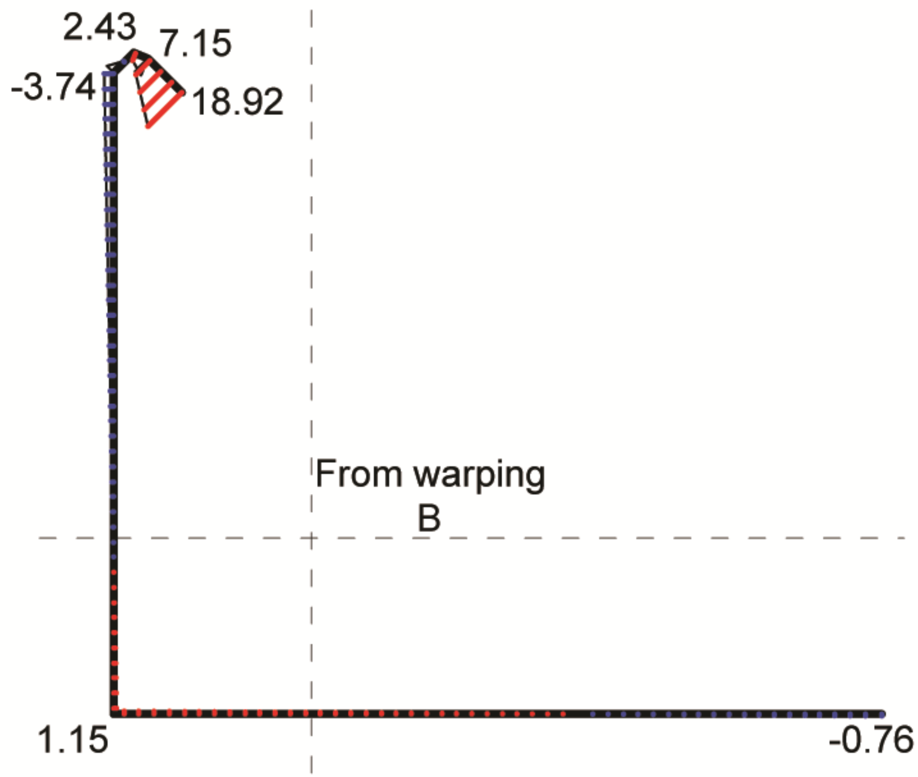
No total combined shear stress is shown as the warping and transverse shear stress components are the same across the thickness of the section while the torsional component varies linearly across the thickness of the section with 0 stress at the centerline to ± the value shown at the extremes.

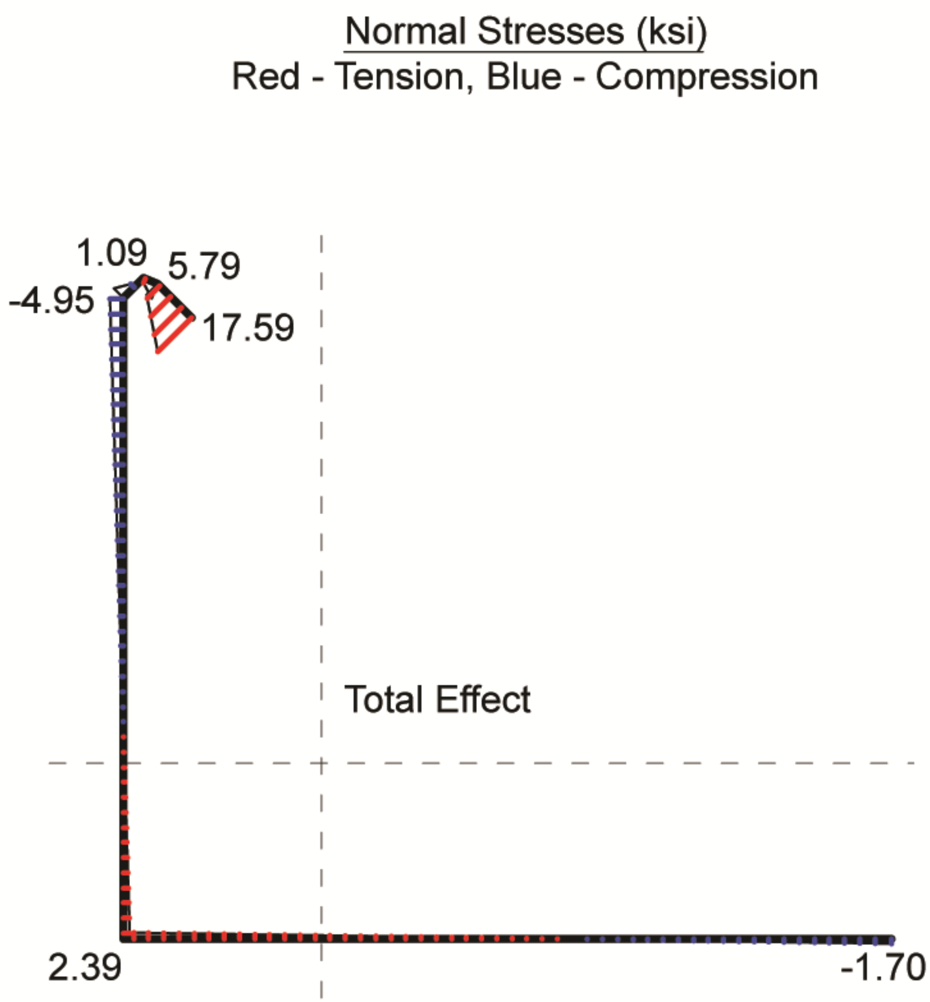
MASTAN2

<u>Normal Stresses (ksi)</u>

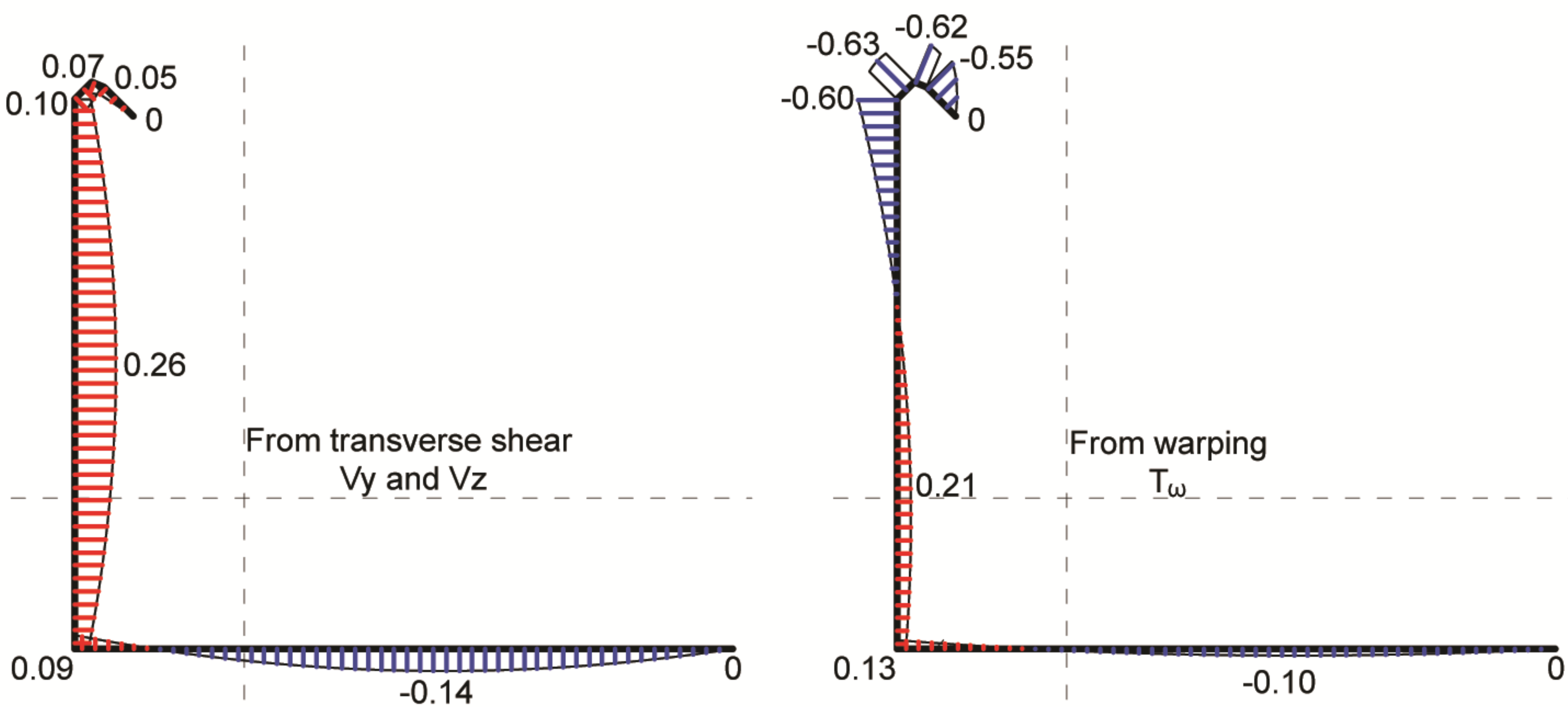
Red - Tension, Blue - Compression





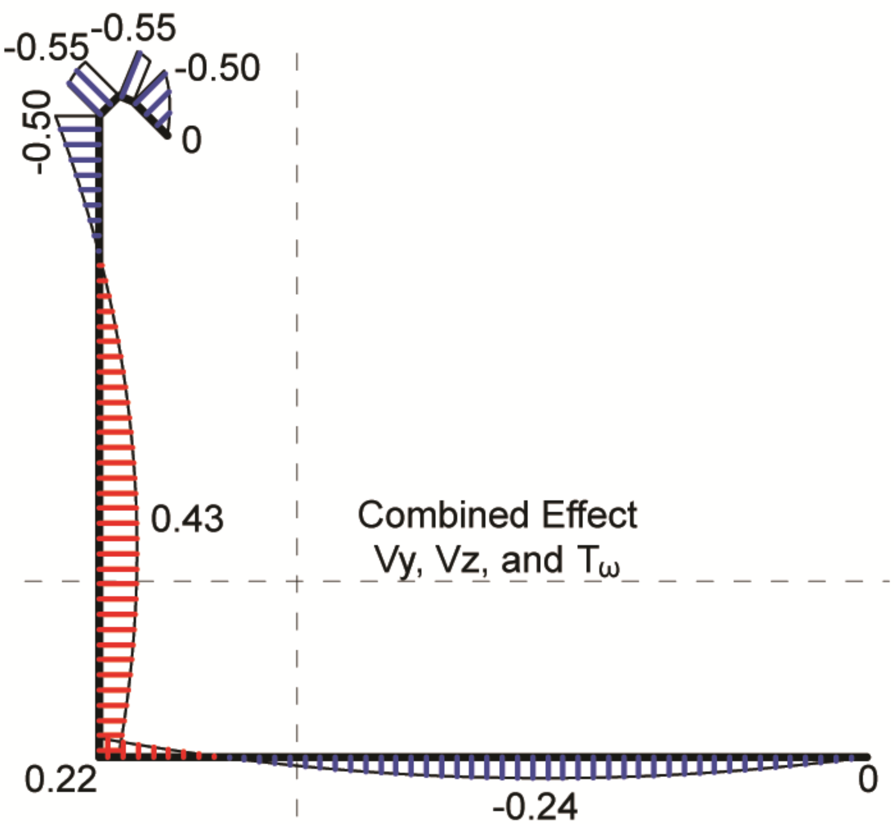

Normal Stresses (ksi)
Red - Tension, Blue - Compression
1.09
5.79
-4.95
17.59
Total Effect
2.39
-1.70

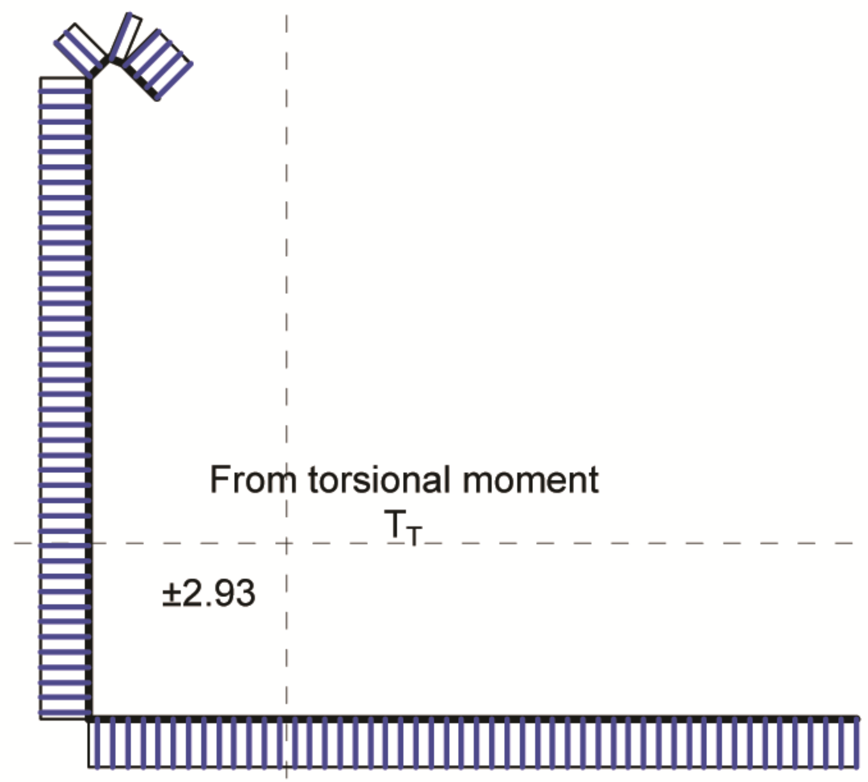
Shear Stresses (ksi)
Red - CCW, Blue - CW



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**.

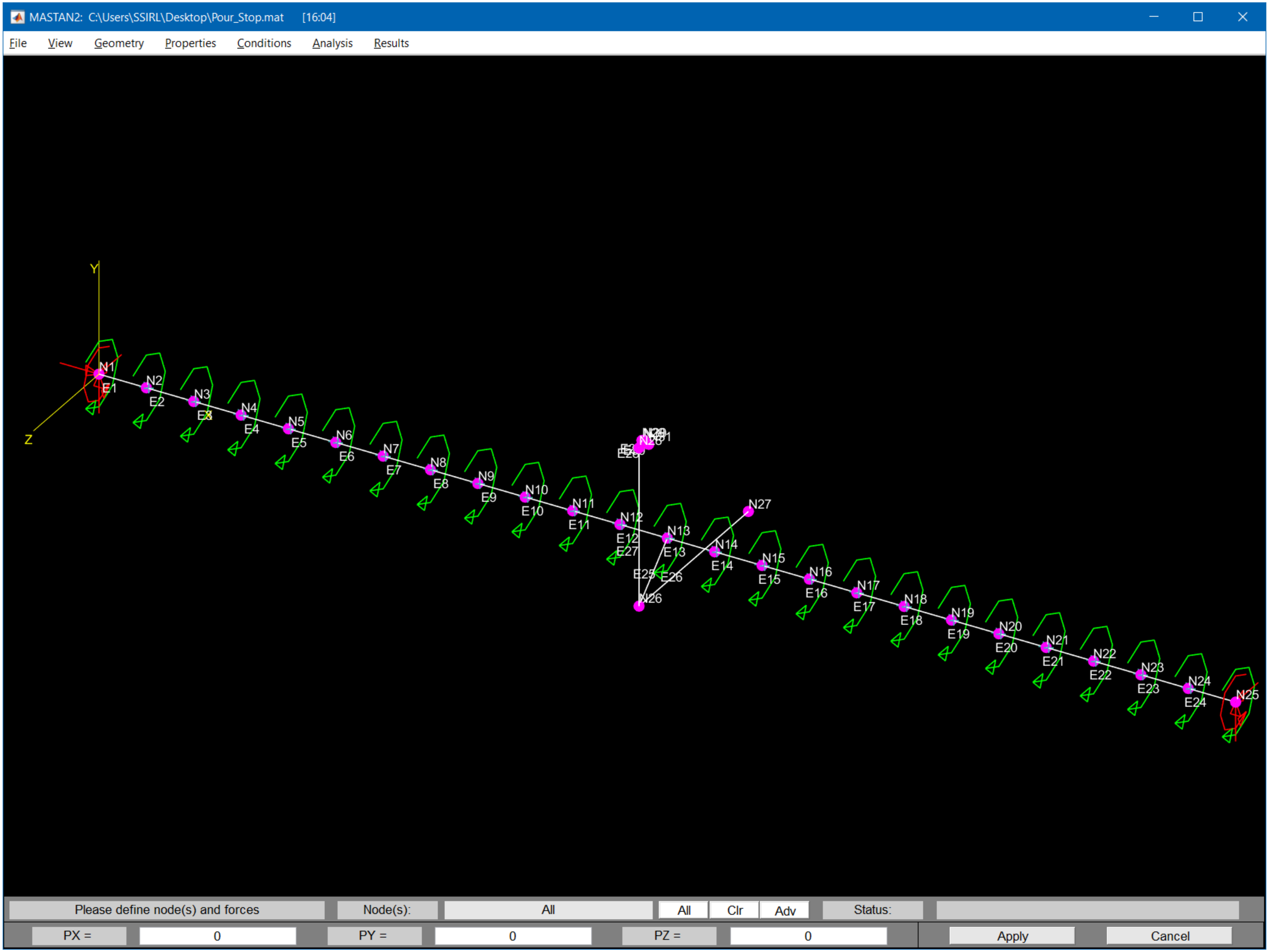
6) At the bottom menu bar, click on **Element(s) local x'-y'-z'** to open the drop-down menu. Select **System global X-Y-Z**.

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.

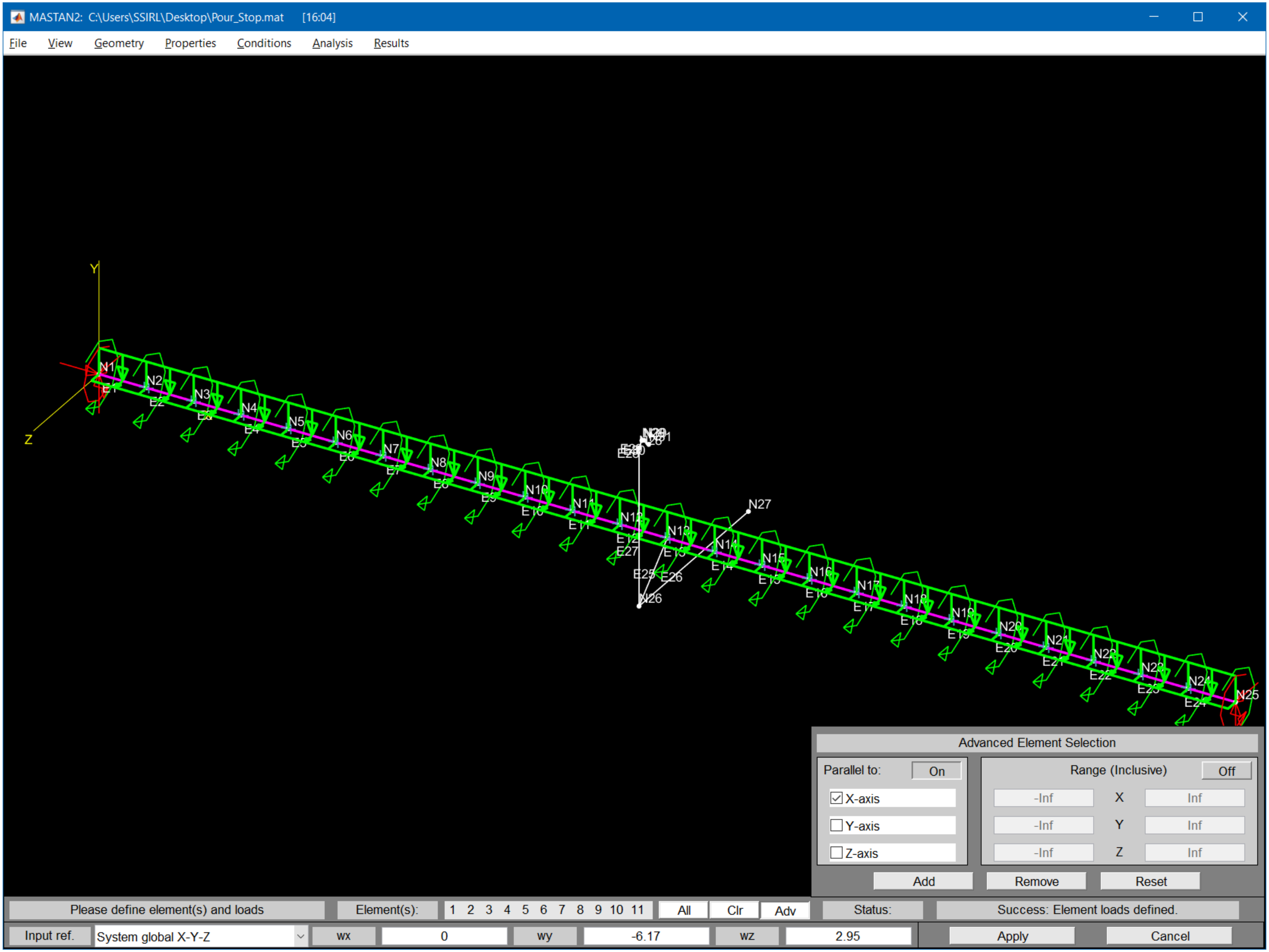
8) Click **Add** to add the main beam elements to the element list.

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

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [16:04]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
N27
N26
Please define node(s) and forces
Node(s):
All
All
Clr
Adv
Status:
PX =
0
PY =
0
PZ =
0
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat [16:04]
File View Geometry Properties Conditions Analysis Results
Y
Z
N27
N26
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
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**.

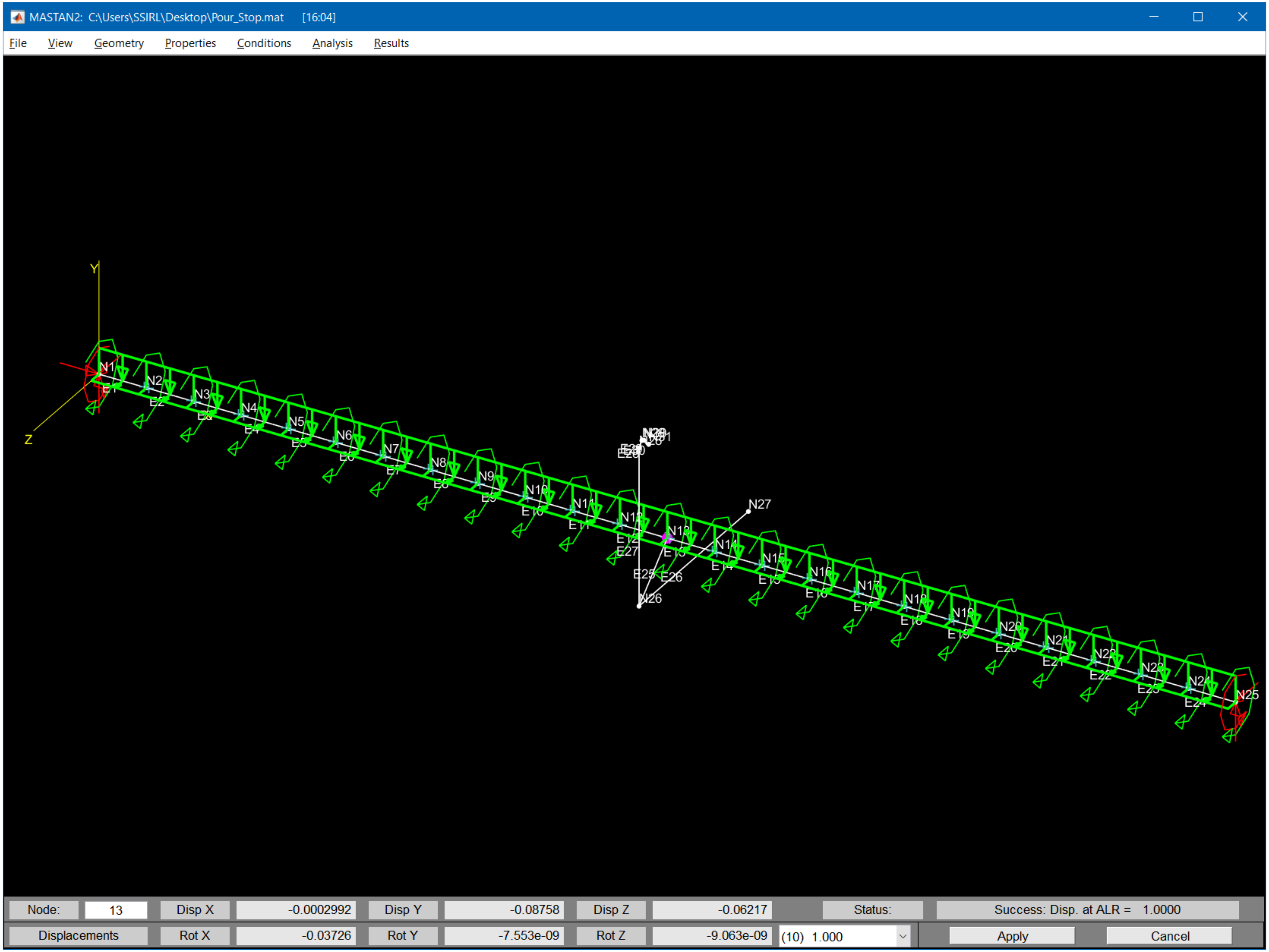
5) On the undeflected shape, click on the midspan node of interest, node **13**, and the displacements for base 6 degree of freedoms are provided in the bottom menu bar.

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
N1
N2
N3
N4
N5
N6
N7
N8
N9
N27
N26
N25
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**.

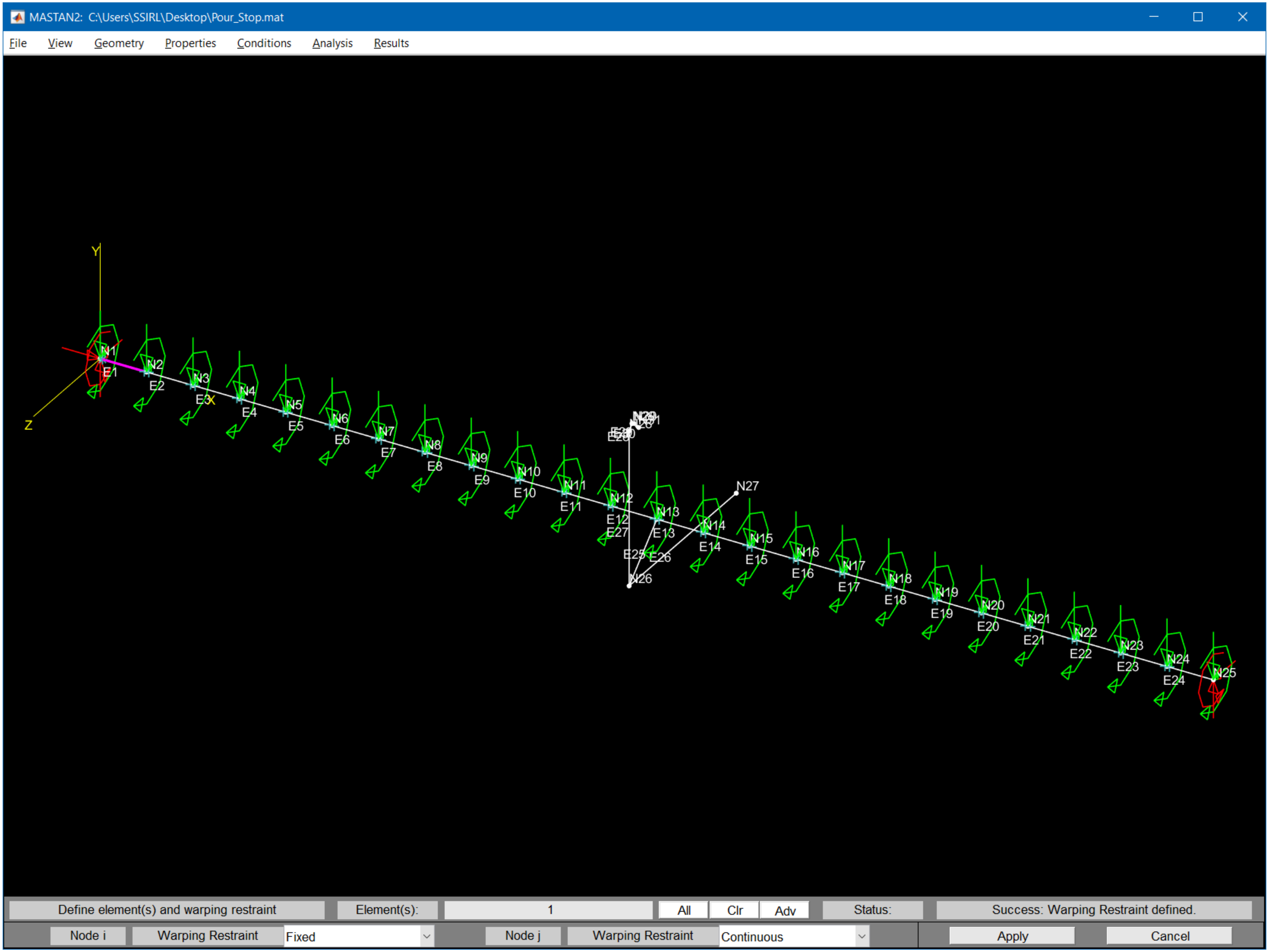
4) Click on the **Apply** button.

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.

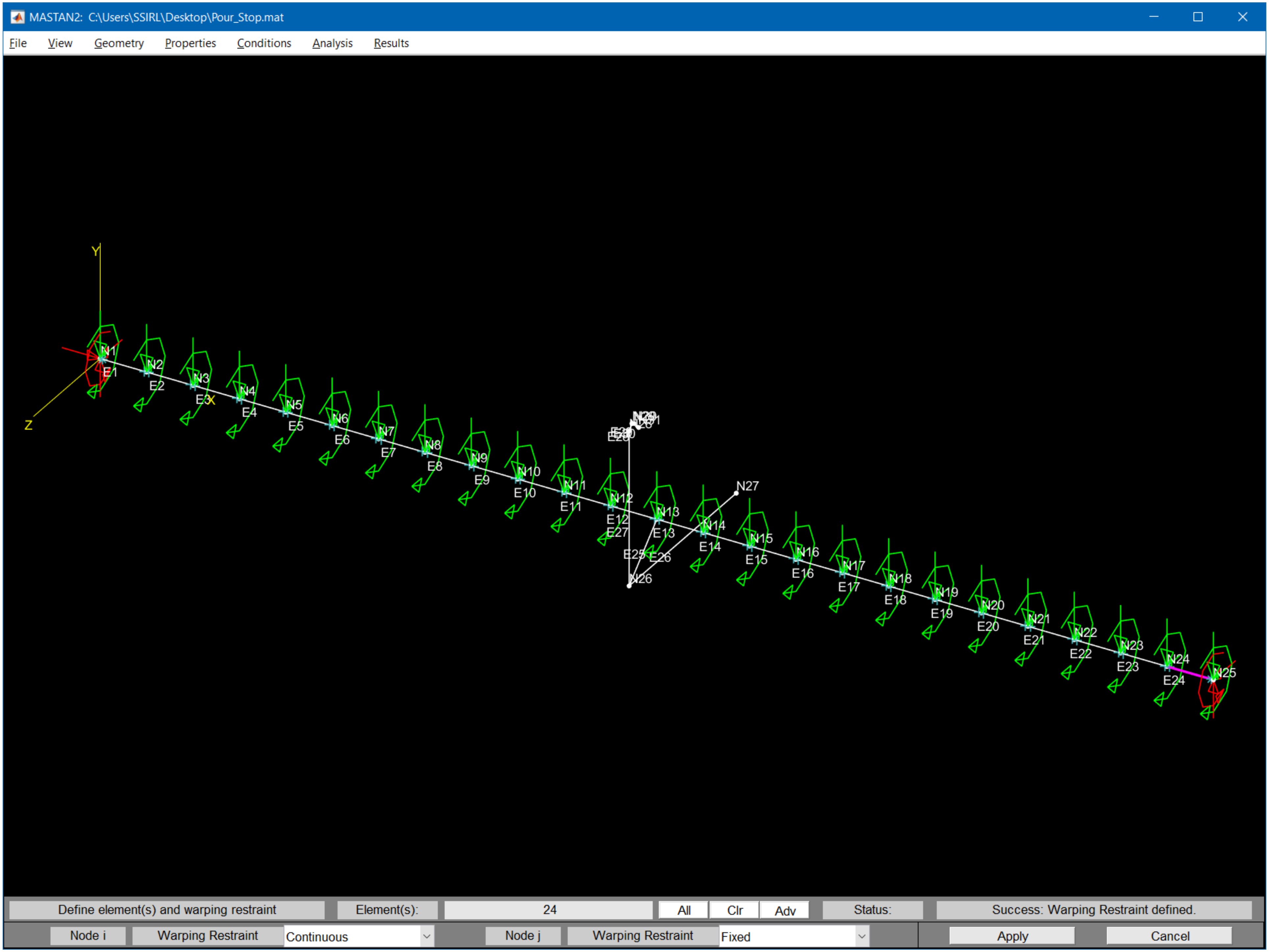
6) Click on the menu to the right of **Warping Restraint for Node i** and set the value to **Continuous**. Click on the menu to the right of **Warping Restraint for Node j** and set the value to **Fixed**.

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
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.

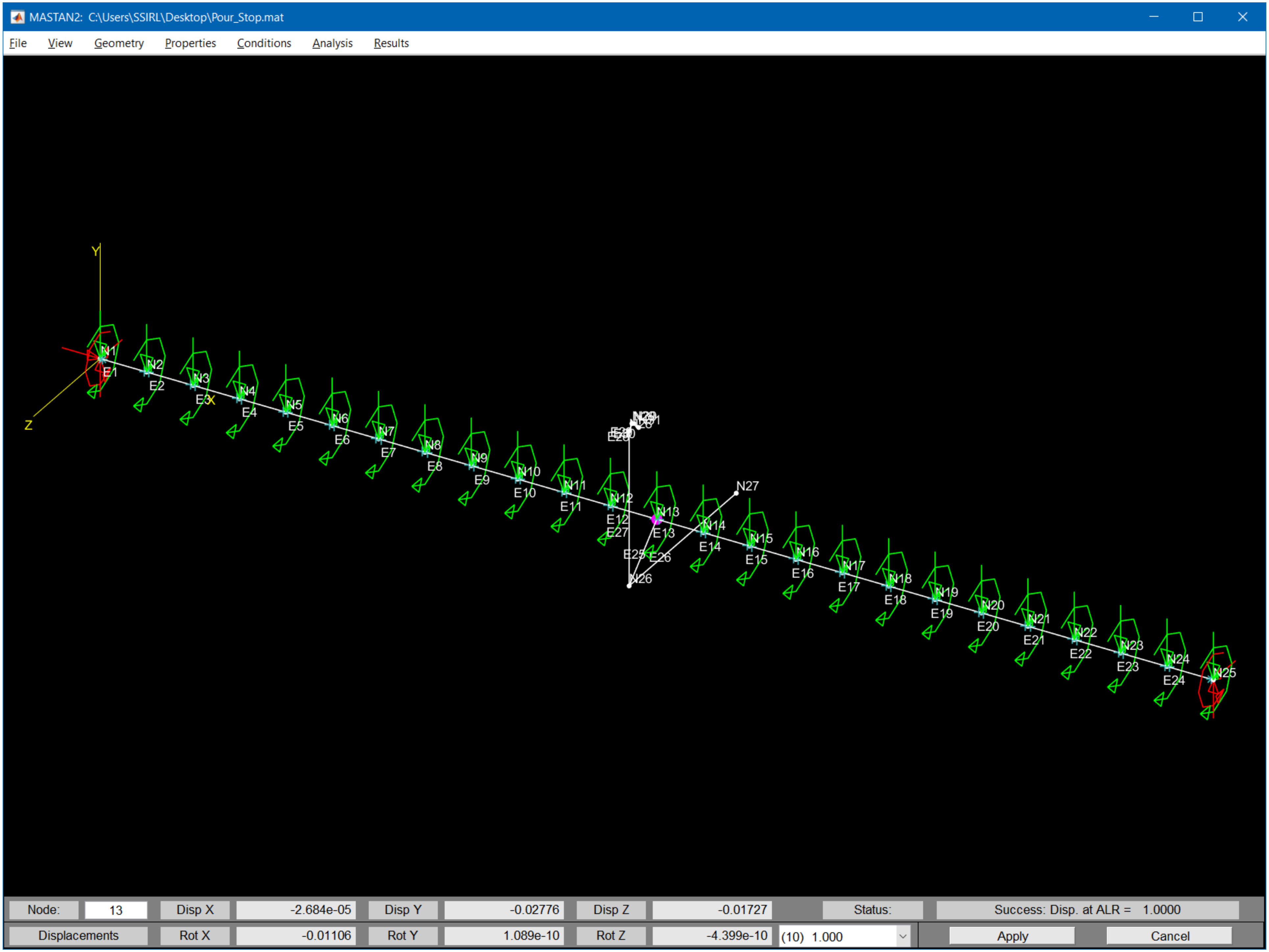
Results:

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

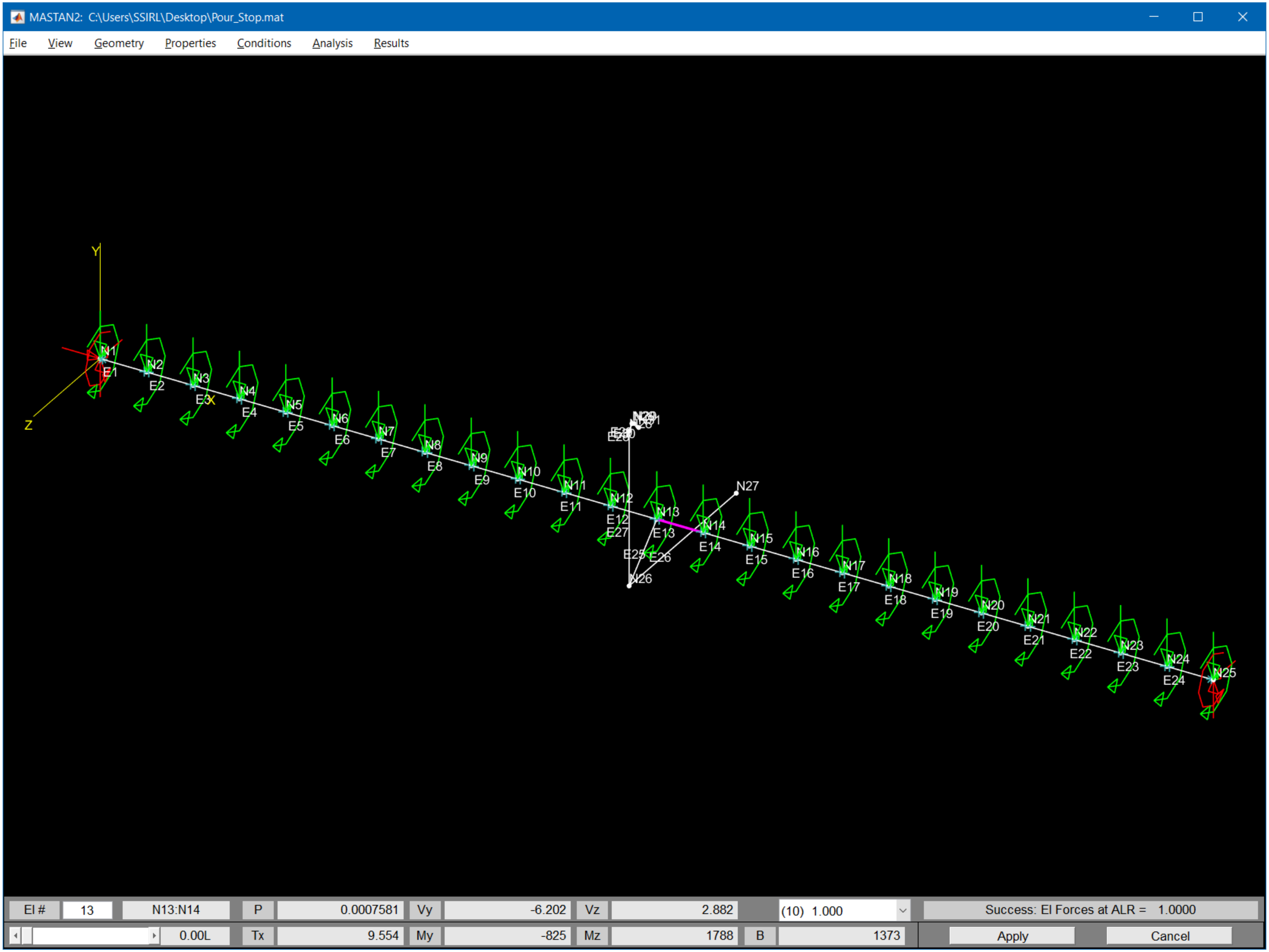
6) From the **Results** menu select **Element Forces**.

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.

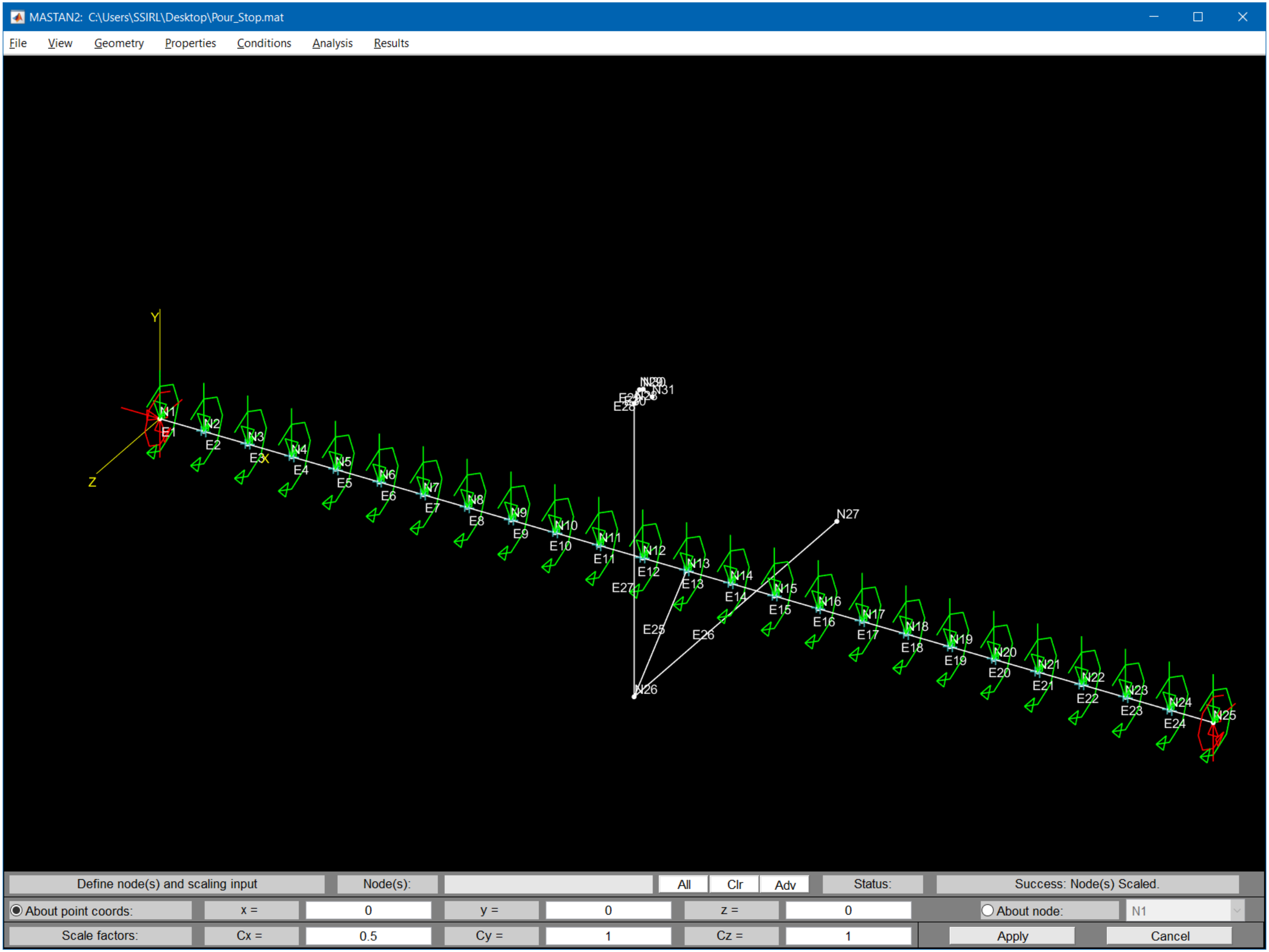
1) From the **Geometry** menu select **Scale Node(s)**.

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
Y
Z
X
N27
N26
E25
E26
E27
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.

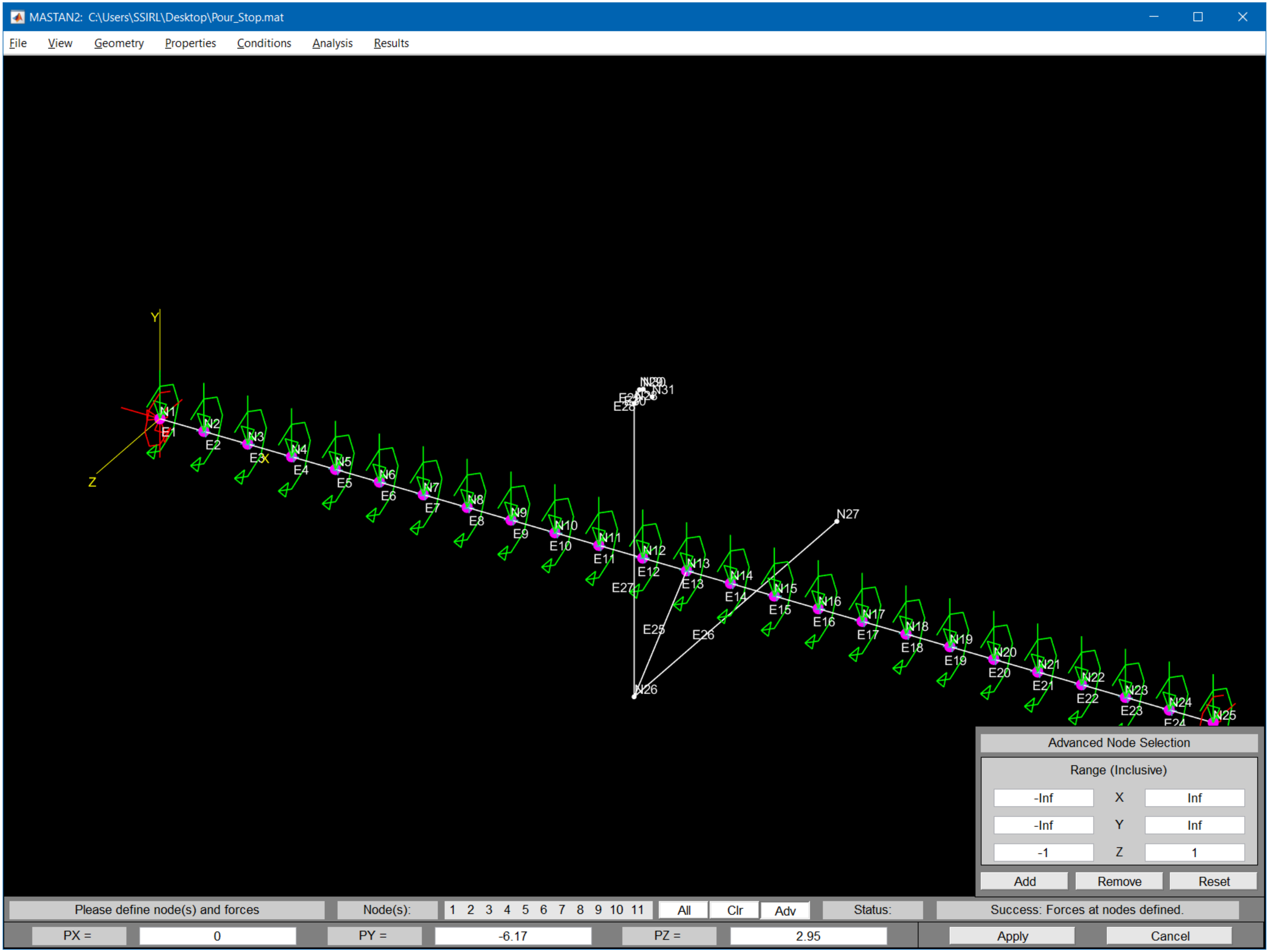
6) From the **Conditions** menu select **Define Moments**.

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

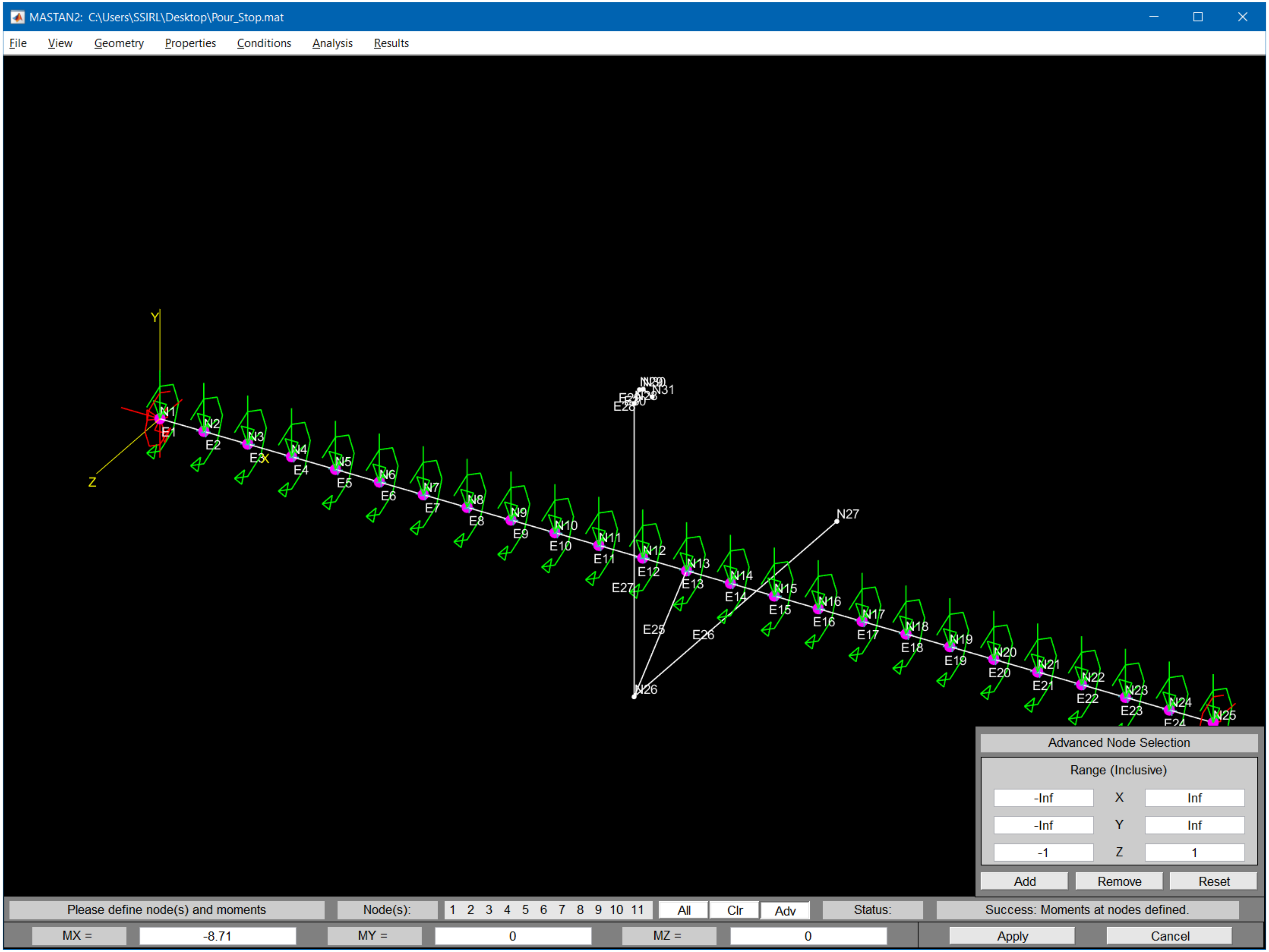
8) Click the **Adv** button to open the pop-up menu. To select the main beam nodes, change the edit box to the left of **Z** to **-1**. Change the edit box to the right of **Z** to **1**.

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
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
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


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.

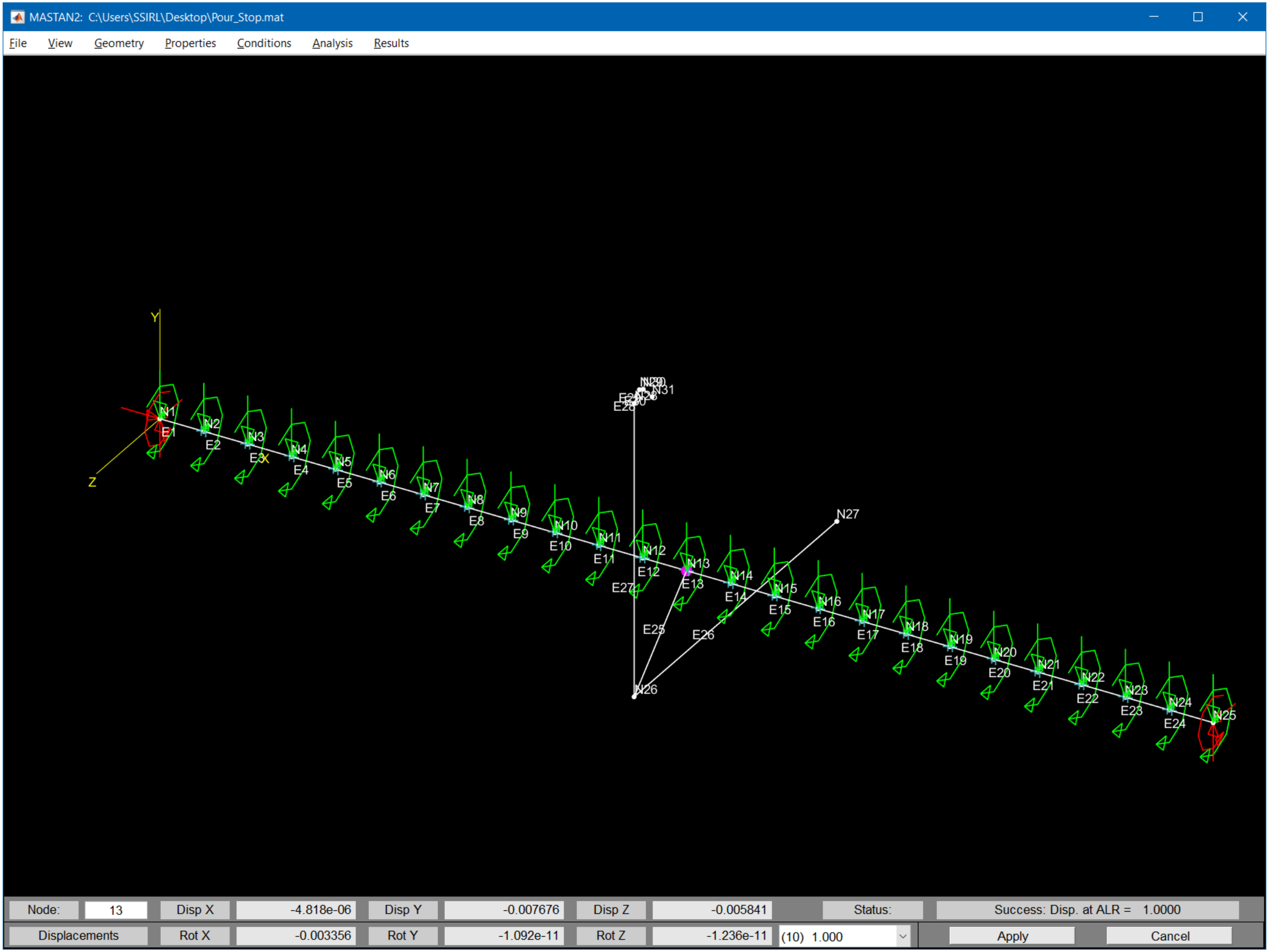
Results:

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**.

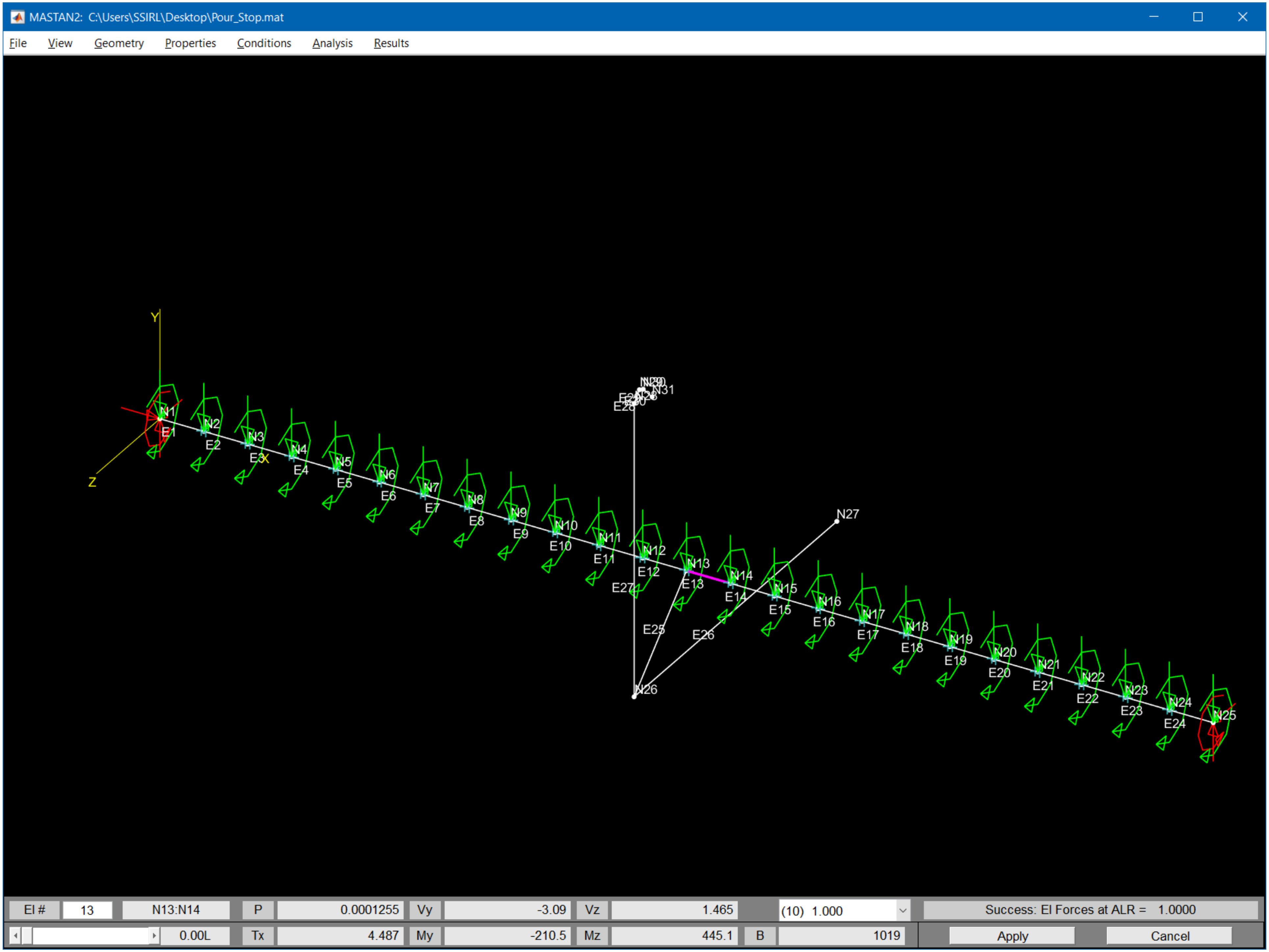
7) On the undeflected shape, click on the element of interest, element **13**, and the internal forces are provided in the bottom menu bar. These are the forces at the start of the member and the middle of the beam.

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Pour_Stop.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
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.

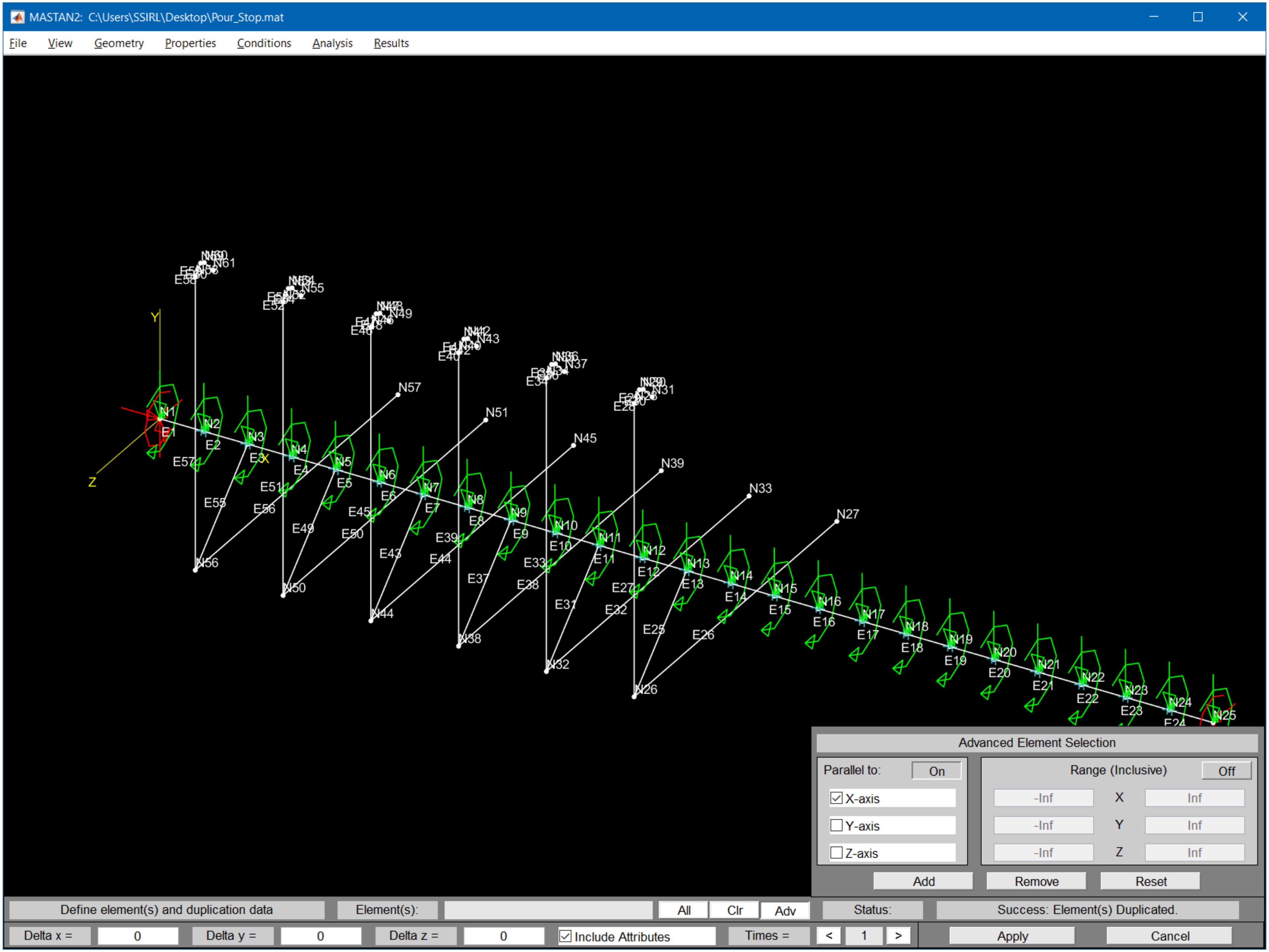
3) Click the **Adv** button to open the pop-up menu. Ensure the check box next to **X-axis** is selected.

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**.

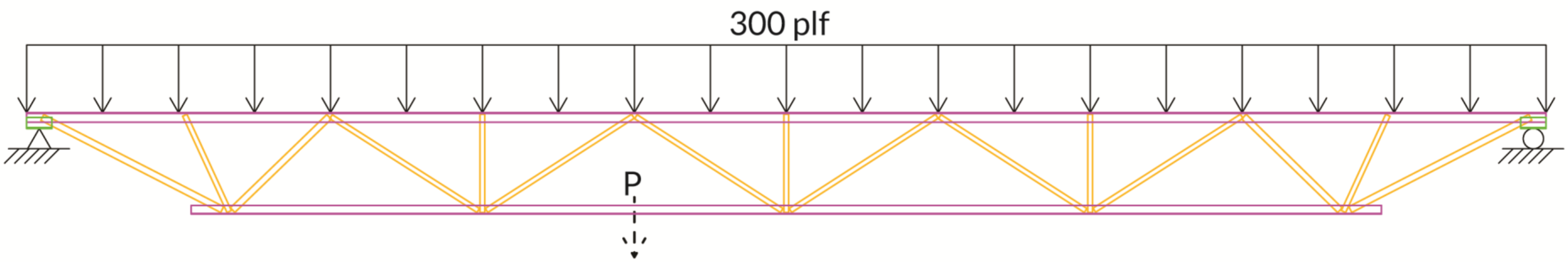
6) Click on the **Apply** button.

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

This completes the tutorial.

Tutorial for MASTAN2 v5.1 – Steel Joist



Department of Civil and Environmental Engineering
UNIVERSITY OF WISCONSIN–MADISON

Bucknell UNIVERSITY

NEW MILLENNIUM BUILDING SYSTEMS

American Iron and Steel Institute

STEEL JOIST INSTITUTE SJI

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

MASTAN2

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.

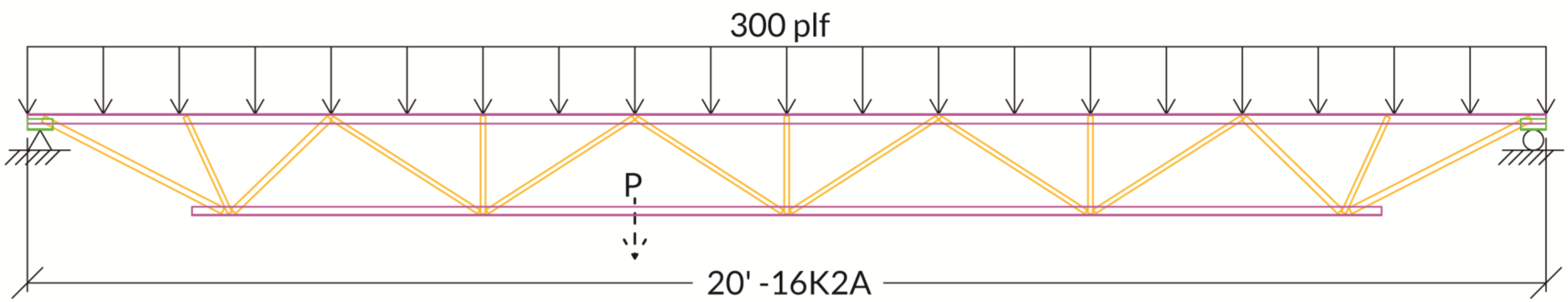
Section 1: Overview

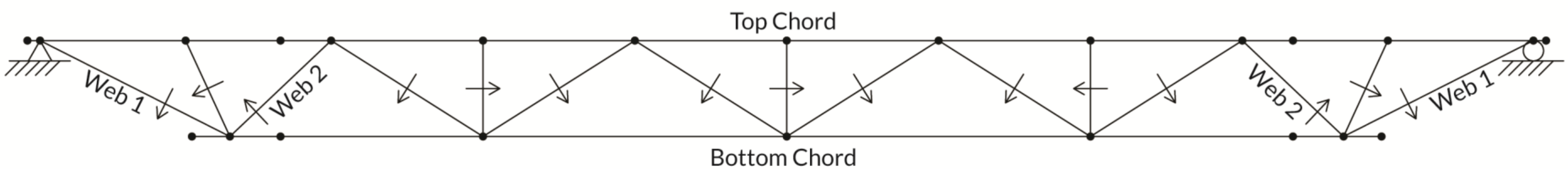
Overview

This tutorial provides step-by-step guidance for the sample joist structure. 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 works with a single open-web steel joist. The model will be created to show how to analyze a joist for a uniform distributed load on the top chord accounting for the non-doubly symmetric section properties. This model will then be adjusted to allow for the application of an eccentric point load on the bottom chord. Further details of each model will be provided in the corresponding section.





Arrows indicate the open side of the web channels. Web members not otherwise labeled are Web 3

Section 2: Getting Started

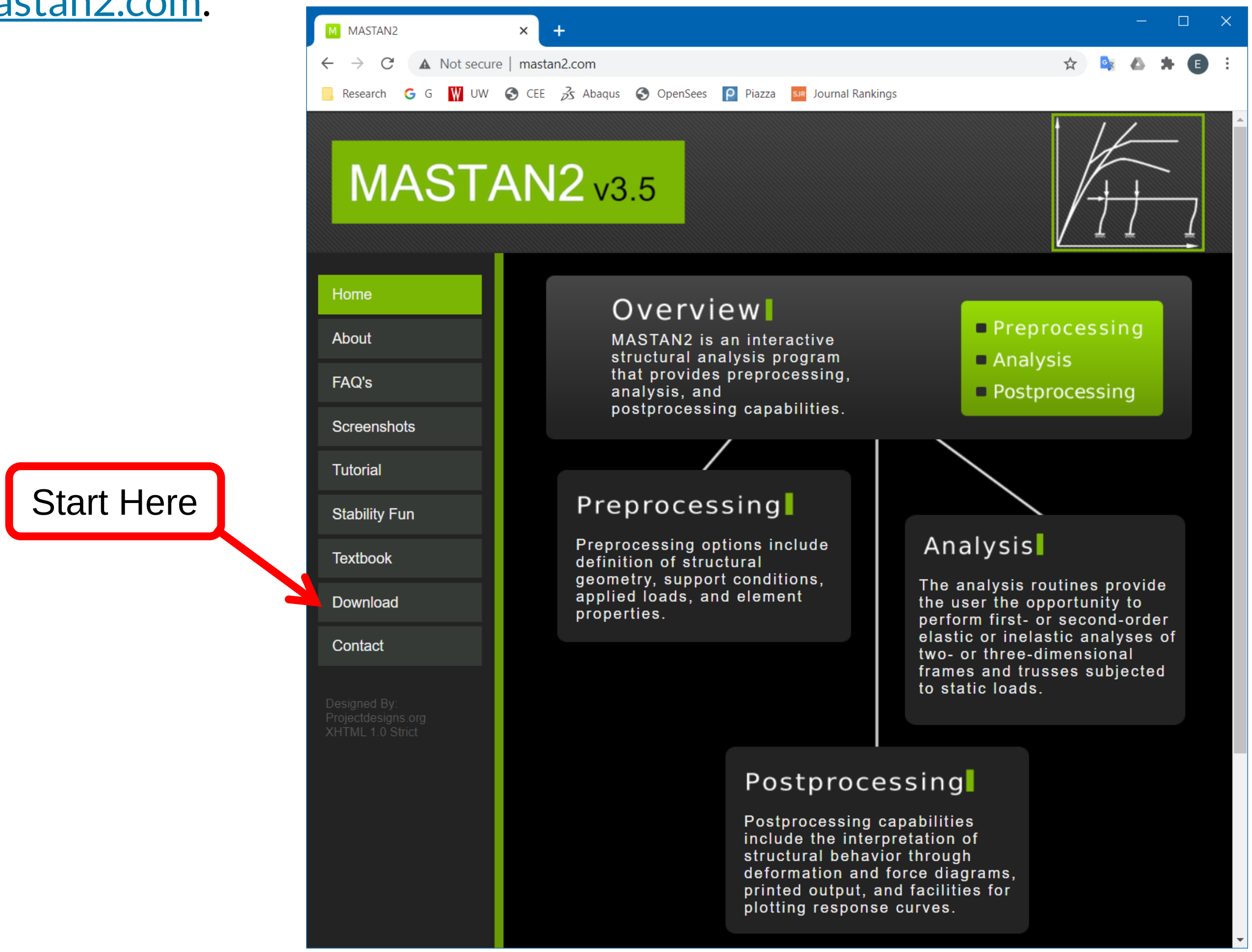
MASTAN2 General Information

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.

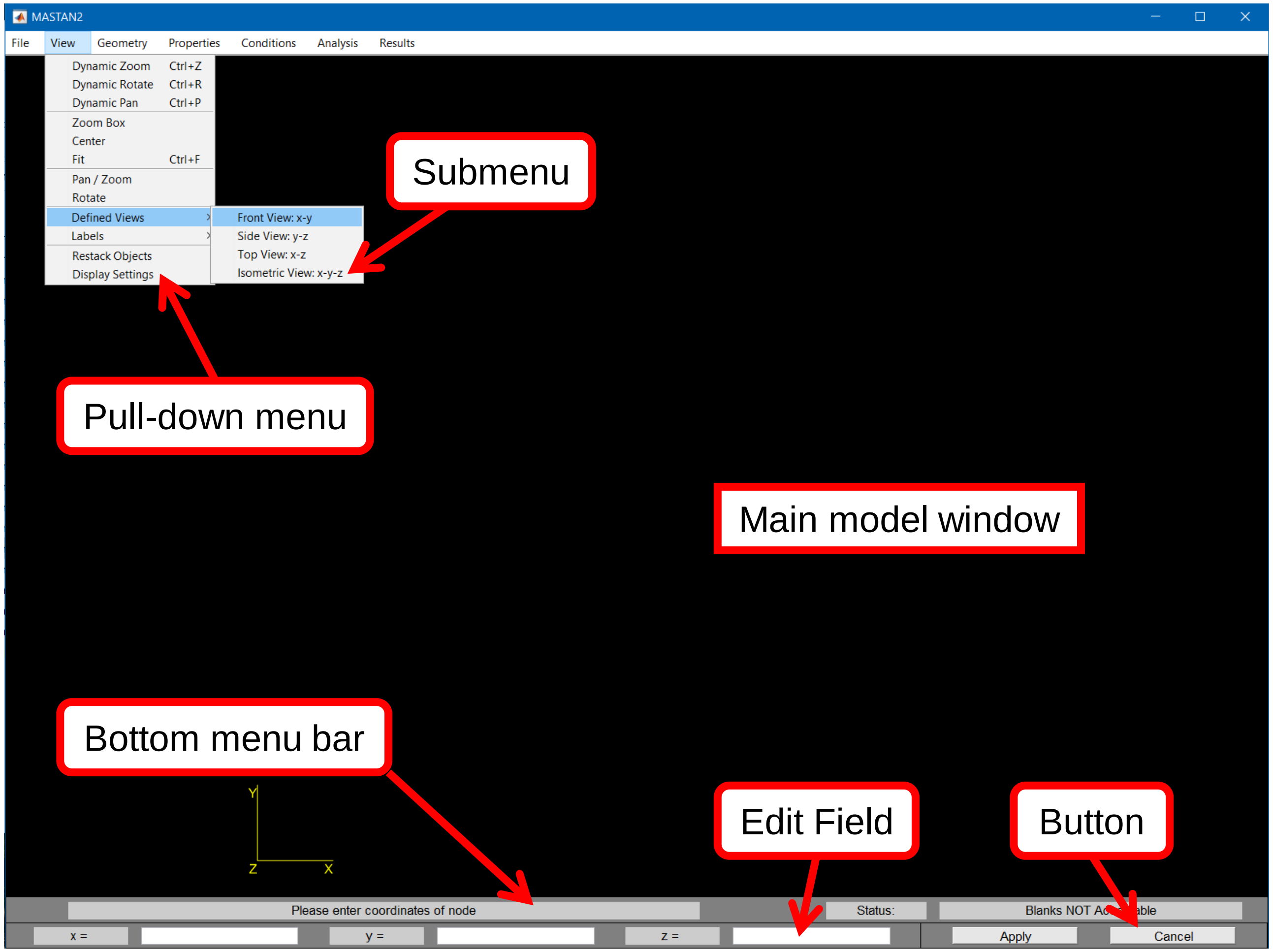
Launching MASTAN2

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: Base Joist Geometry

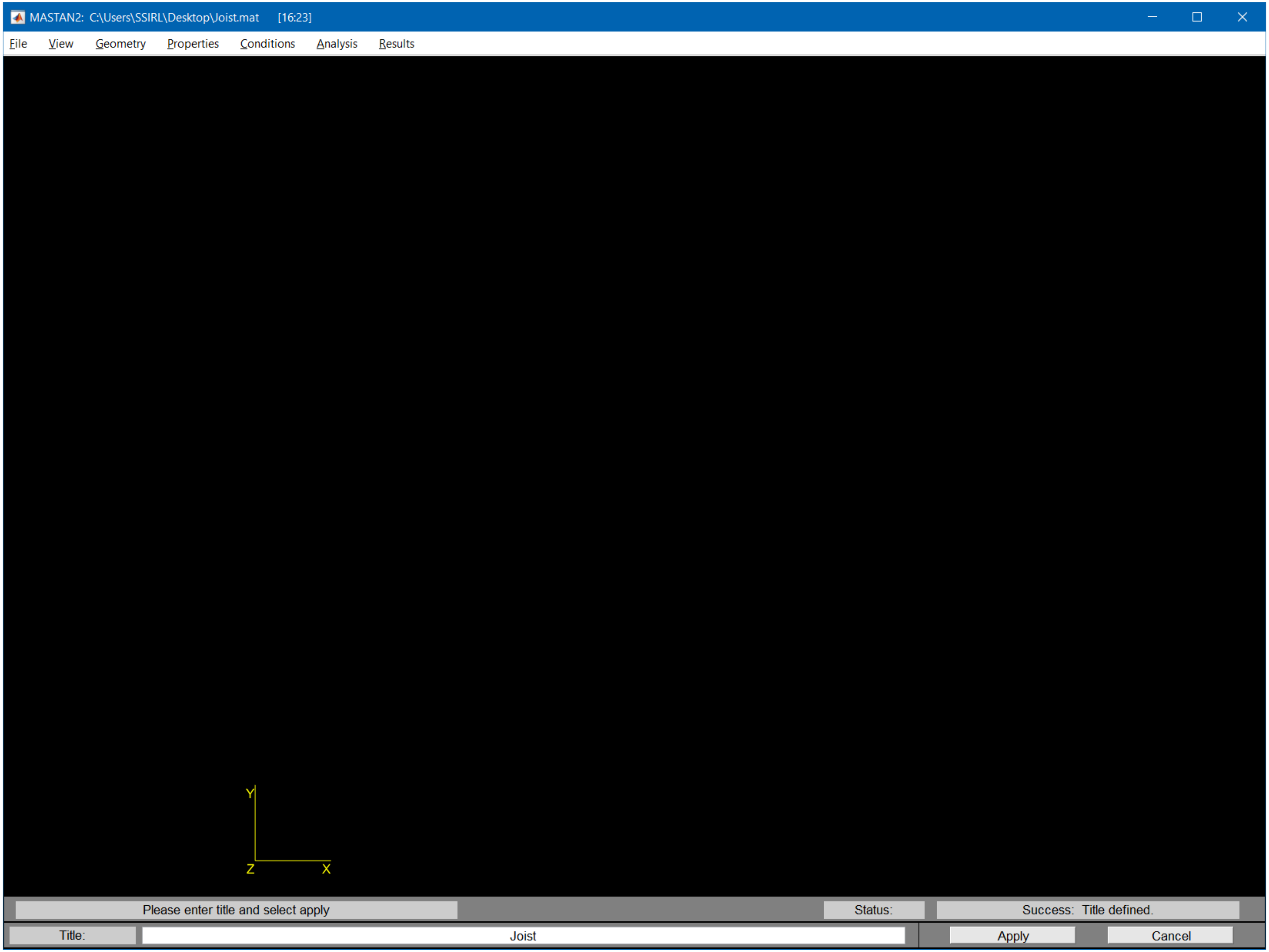
Naming and Saving

While you can build the model and complete the analysis without saving or applying a title, due to the complexity of the model we will create a save file immediately. For the remainder of this tutorial, there will not be a reminder to save. However, it can be useful to save the file as you go along, particularly before any action that is not easily reversed as there is no undo feature. A file can be reopened while still working in that file without saving it to revert to the previous save version of the model that is unaffected by your last steps.

1) Start with a new, empty model.

2) 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.

3) From the **File** menu select **Save As …** . After selecting your destination folder, type in the filename **Joist** 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: C:\Users\SSIRL\Desktop\Joist.mat [16:23]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
Z
X
Please enter title and select apply
Status:
Success: Title defined.
Title:
Joist
Apply
Cancel

Defining the Joist

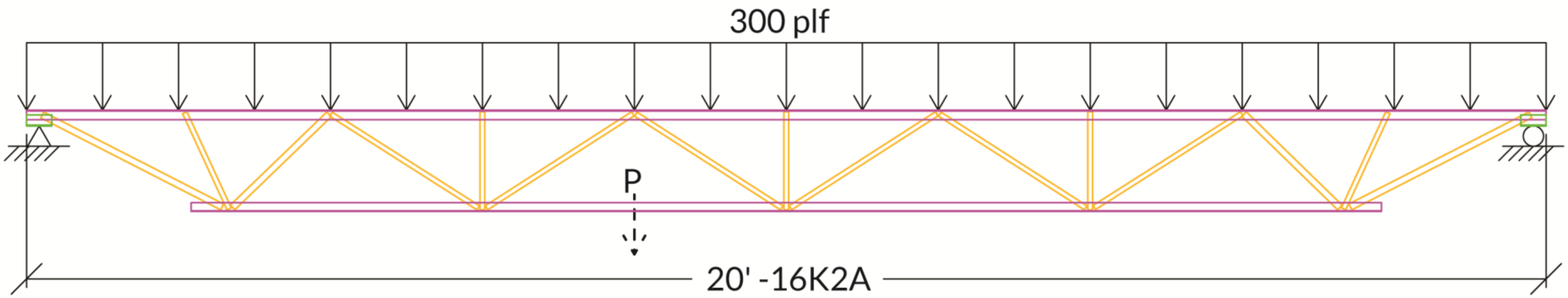
The joist will be modeled with the chords as two separate members to allow them to twist independently. The geometry could be input by defining individual nodes and then individual elements or making use of the extrude element tool extensively. Instead, the **Input Geometry** tool will be used to define most of the joist geometry. As a large part of creating the frame is the prep work, an explanation of what values were used is provided on this page and the next pages explain how to use the values and links to what the values are.

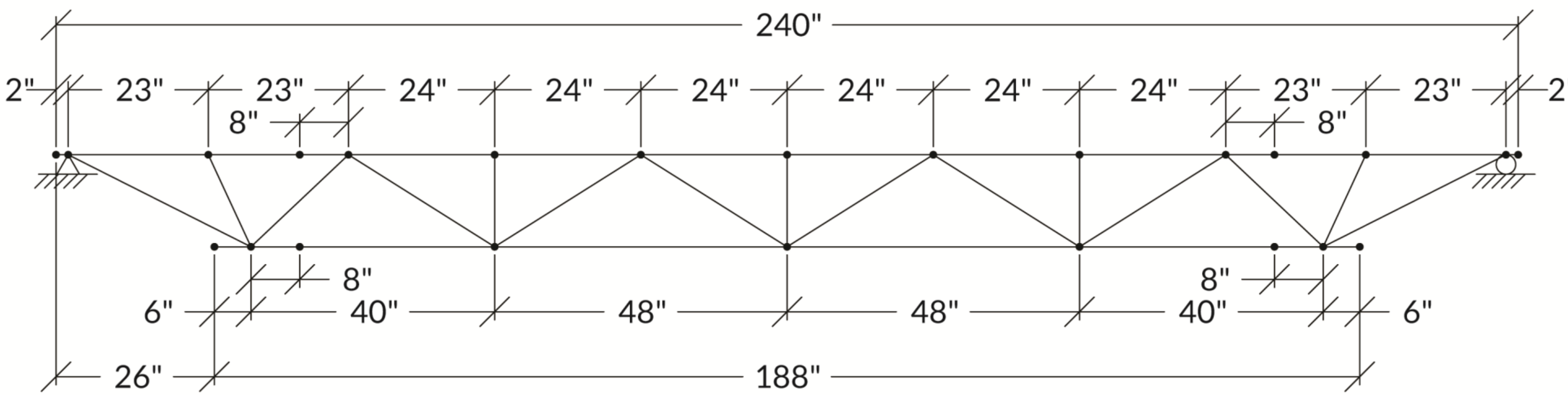
The joist was defined as a series of nodes along the top chord and then the bottom chord. The first set is for the chords set back in the negative z-direction. The second set is for the chords in the positive z-direction. The third set is on the x-y plane for member to member connections and the webs. The x position was defined based on the simple joist geometry.

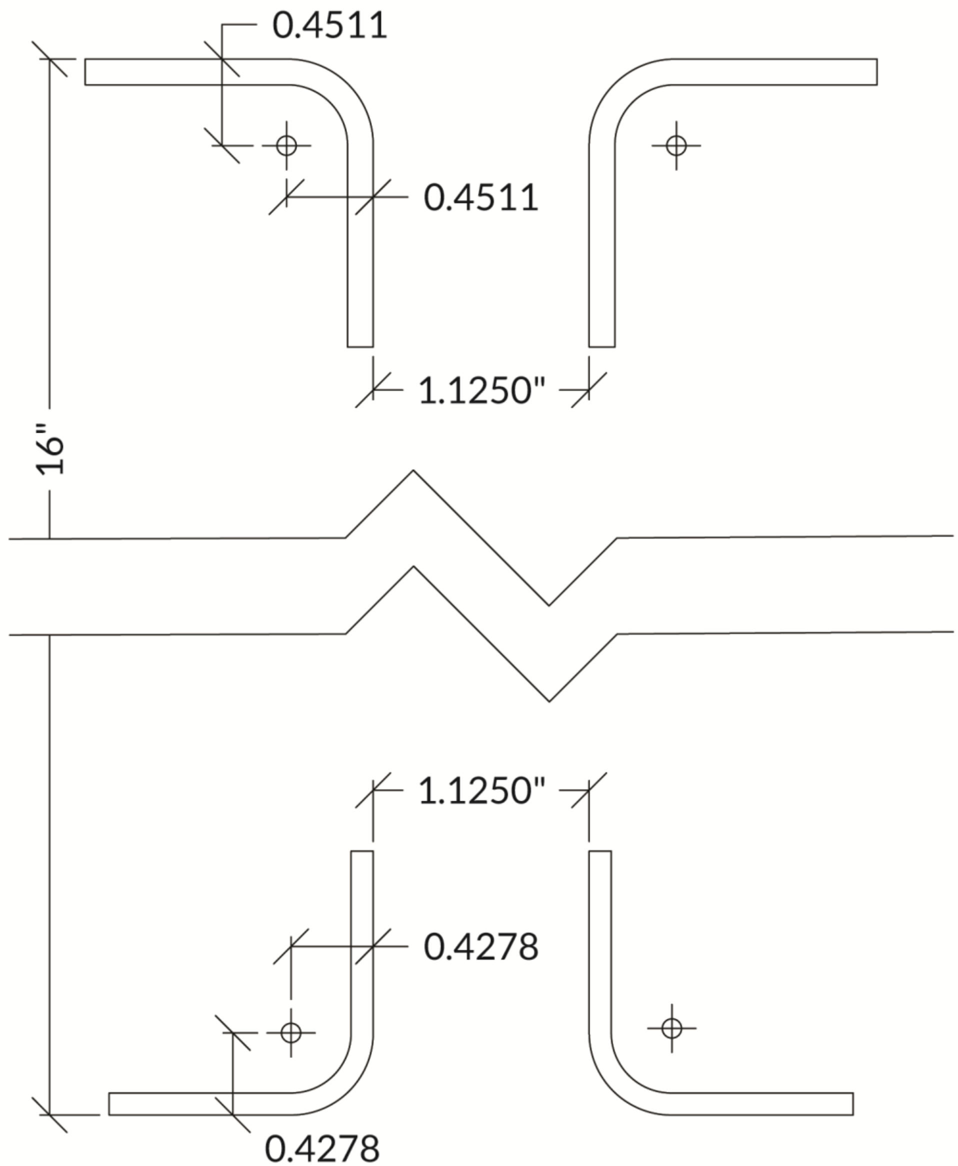
A node is defined for the end of the members, each web intersection, and where the joist bracing connects. The y position of the top chord nodes is set as 16" minus the centroid of the top chord angle while the bottom chord is 0" plus the centroid of the bottom chord angle. The z position similarly accounts for the centroid position plus a 1-1/8" gap between the chords.

The elements are defined in order to connect all back chords, all front chords, all webs, and then the connections in between.

MASTAN2


300 plf
P
20' -16K2A


240"
2"
23"
23"
24"
24"
24"
24"
24"
24"
23"
23"
2"
8"
8"
8"
8"
6"
40"
48"
48"
40"
6"
26"
188"


0.4511
0.4511
1.1250"
16"
1.1250"
0.4278
0.4278

Defining Geometry

For entering the node coordinates and element information below, the values are provided in two different formats for your use. Node and element information needs to be copied only once.

A - Lists all the node information in separate readable segments to be copied in order.

B - Lists all the element information in separate readable segments to be copied in order.

C - Lists all the node and then all element information in small font lists to be copied all at once.

1) From the **Geometry** menu select **Input Geometry** .

2) At the bottom menu bar, **Nodes** should be already selected. In the edit box to the right of **X_coord Y_coord Z_coord**, enter manually or copy and paste the coordinate values.

3) Click on **Nodes** to open a pop-up menu. Click on **Elements** to update what information is imported.

4) In the edit box to the right of **Node_i Node_j Beta(deg)**, enter manually or copy and paste the element start and end nodes. When only 2 values are provided, beta is assumed to be 0.

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

Note: **Apply** button could be clicked before defining the **Element** list to just input the **Nodes** first. The **Element** list will then use the existing node information on the second use of **Apply** button.

MASTAN2

Copy all 3 tables in order

Nodes 1

0	15.54894	-1.01356
2	15.54894	-1.01356
25	15.54894	-1.01356
40	15.54894	-1.01356
48	15.54894	-1.01356
72	15.54894	-1.01356
96	15.54894	-1.01356
120	15.54894	-1.01356
144	15.54894	-1.01356
168	15.54894	-1.01356
192	15.54894	-1.01356
200	15.54894	-1.01356
215	15.54894	-1.01356
238	15.54894	-1.01356
240	15.54894	-1.01356
26	0.42775	-0.99025
32	0.42775	-0.99025
40	0.42775	-0.99025
72	0.42775	-0.99025
120	0.42775	-0.99025
168	0.42775	-0.99025
200	0.42775	-0.99025
208	0.42775	-0.99025
214	0.42775	-0.99025

Nodes 2

0	15.54894	1.01356
2	15.54894	1.01356
25	15.54894	1.01356
40	15.54894	1.01356
48	15.54894	1.01356
72	15.54894	1.01356
96	15.54894	1.01356
120	15.54894	1.01356
144	15.54894	1.01356
168	15.54894	1.01356
192	15.54894	1.01356
200	15.54894	1.01356
215	15.54894	1.01356
238	15.54894	1.01356
240	15.54894	1.01356
26	0.42775	0.99025
32	0.42775	0.99025
40	0.42775	0.99025
72	0.42775	0.99025
120	0.42775	0.99025
168	0.42775	0.99025
200	0.42775	0.99025
208	0.42775	0.99025
214	0.42775	0.99025

Nodes 3

0	15.54894	0
2	15.54894	0
25	15.54894	0
40	15.54894	0
48	15.54894	0
72	15.54894	0
96	15.54894	0
120	15.54894	0
144	15.54894	0
168	15.54894	0
192	15.54894	0
200	15.54894	0
215	15.54894	0
238	15.54894	0
240	15.54894	0
26	0.42775	0
32	0.42775	0
40	0.42775	0
72	0.42775	0
120	0.42775	0
168	0.42775	0
200	0.42775	0
208	0.42775	0
214	0.42775	0

MASTAN2

Copy all 5 tables in order

1 B. Chord

1	2
2	3
3	4
4	5
5	6
6	7
7	8
8	9
9	10
10	11
11	12
12	13
13	14
14	15
16	17
17	18
18	19
19	20
20	21
21	22
22	23
23	24

2 F. Chord

25	26
26	27
27	28
28	29
29	30
30	31
31	32
32	33
33	34
34	35
35	36
36	37
37	38
38	39
40	41
41	42
42	43
43	44
44	45
45	46
46	47
47	48

3 Webs

50	65
51	65
53	65
53	67
54	67
55	67
55	68
56	68
57	68
57	69
58	69
59	69
59	71
61	71
62	71

4 B. Conn.

50	2
51	3
53	5
54	6
55	7
56	8
57	9
58	10
59	11
61	13
62	14
65	17
67	19
68	20
69	21
71	23

5 F. Conn.

50	26
51	27
53	29
54	30
55	31
56	32
57	33
58	34
59	35
61	37
62	38
65	41
67	43
68	44
69	45
71	47

Copy all values to the **Node** section

Nodes

0	15.54894	-1.01356
2	15.54894	-1.01356
25	15.54894	-1.01356
40	15.54894	-1.01356
48	15.54894	-1.01356
72	15.54894	-1.01356
96	15.54894	-1.01356
120	15.54894	-1.01356
144	15.54894	-1.01356
168	15.54894	-1.01356
192	15.54894	-1.01356
200	15.54894	-1.01356
215	15.54894	-1.01356
238	15.54894	-1.01356
240	15.54894	-1.01356
26	0.42775	-0.99025
32	0.42775	-0.99025
40	0.42775	-0.99025
72	0.42775	-0.99025
120	0.42775	-0.99025
168	0.42775	-0.99025
200	0.42775	-0.99025
208	0.42775	-0.99025
214	0.42775	-0.99025
0	15.54894	1.01356
2	15.54894	1.01356
25	15.54894	1.01356
40	15.54894	1.01356
48	15.54894	1.01356
72	15.54894	1.01356
96	15.54894	1.01356
120	15.54894	1.01356
144	15.54894	1.01356
168	15.54894	1.01356
192	15.54894	1.01356
200	15.54894	1.01356
215	15.54894	1.01356
238	15.54894	1.01356
240	15.54894	1.01356
26	0.42775	0.99025
32	0.42775	0.99025
40	0.42775	0.99025
72	0.42775	0.99025
120	0.42775	0.99025
168	0.42775	0.99025
200	0.42775	0.99025
208	0.42775	0.99025
214	0.42775	0.99025
0	15.54894	0
2	15.54894	0
25	15.54894	0
40	15.54894	0
48	15.54894	0
72	15.54894	0
96	15.54894	0
120	15.54894	0
144	15.54894	0
168	15.54894	0
192	15.54894	0
200	15.54894	0
215	15.54894	0
238	15.54894	0
240	15.54894	0
26	0.42775	0
32	0.42775	0
40	0.42775	0
72	0.42775	0
120	0.42775	0
168	0.42775	0
200	0.42775	0
208	0.42775	0
214	0.42775	0

Copy all values to the **Element** section

Elements

1	2
2	3
3	4
4	5
5	6
6	7
7	8
8	9
9	10
10	11
11	12
12	13
13	14
14	15
16	17
17	18
18	19
19	20
20	21
21	22
22	23
23	24
25	26
26	27
27	28
28	29
29	30
30	31
31	32
32	33
33	34
34	35
35	36
36	37
37	38
38	39
40	41
41	42
42	43
43	44
44	45
45	46
46	47
47	48
50	65
51	65
53	65
53	67
54	67
55	67
55	68
56	68
57	68
57	69
58	69
59	69
59	71
61	71
62	71
50	2
51	3
53	5
54	6
55	7
56	8
57	9
58	10
59	11
61	13
62	14
65	17
67	19
68	20
69	21
71	23
50	26
51	27
53	29
54	30
55	31
56	32
57	33
58	34
59	35
61	37
62	38
65	41
67	43
68	44
69	45
71	47

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Elements
Enter each element's data as 3 values separated by spaces per row (perhaps using paste)
Status:
Success: All geometry added.
Node_i Node_j Beta(deg)
Apply
Cancel

Lower Bridging Support

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

2) Click on the nodes on the bottom chord closest to the joist bridging connection: Nodes **18**, **22**, **42**, and **46** to populate the nodes to be extruded.

3) Click in the edit box to the right of **Delta y =** and change **0** to **2.5**.

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

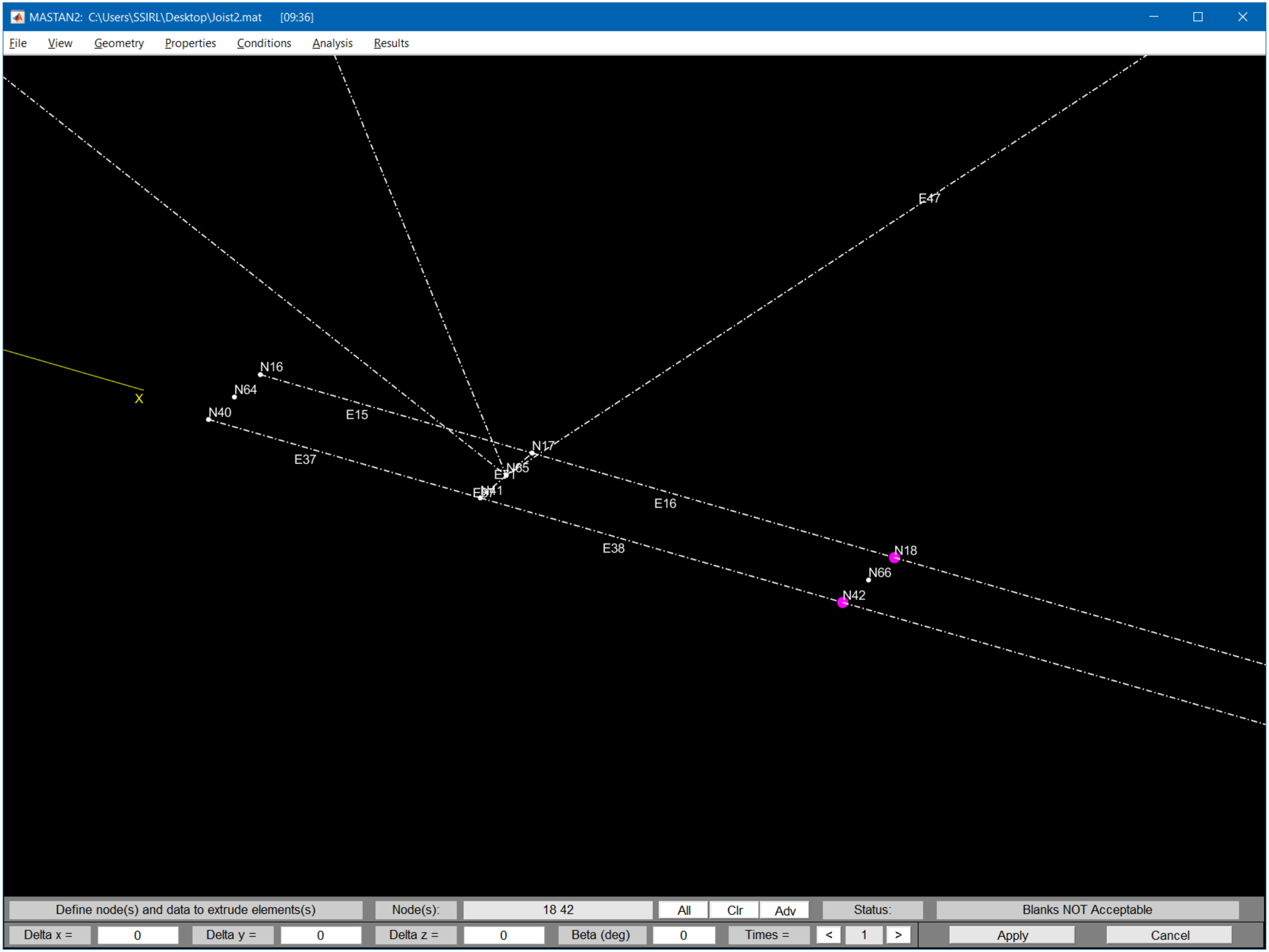
5) Click on the new nodes above the back, bottom chord to the joist bridging. Depending on the exact order you clicked the previous nodes, the node index may vary.

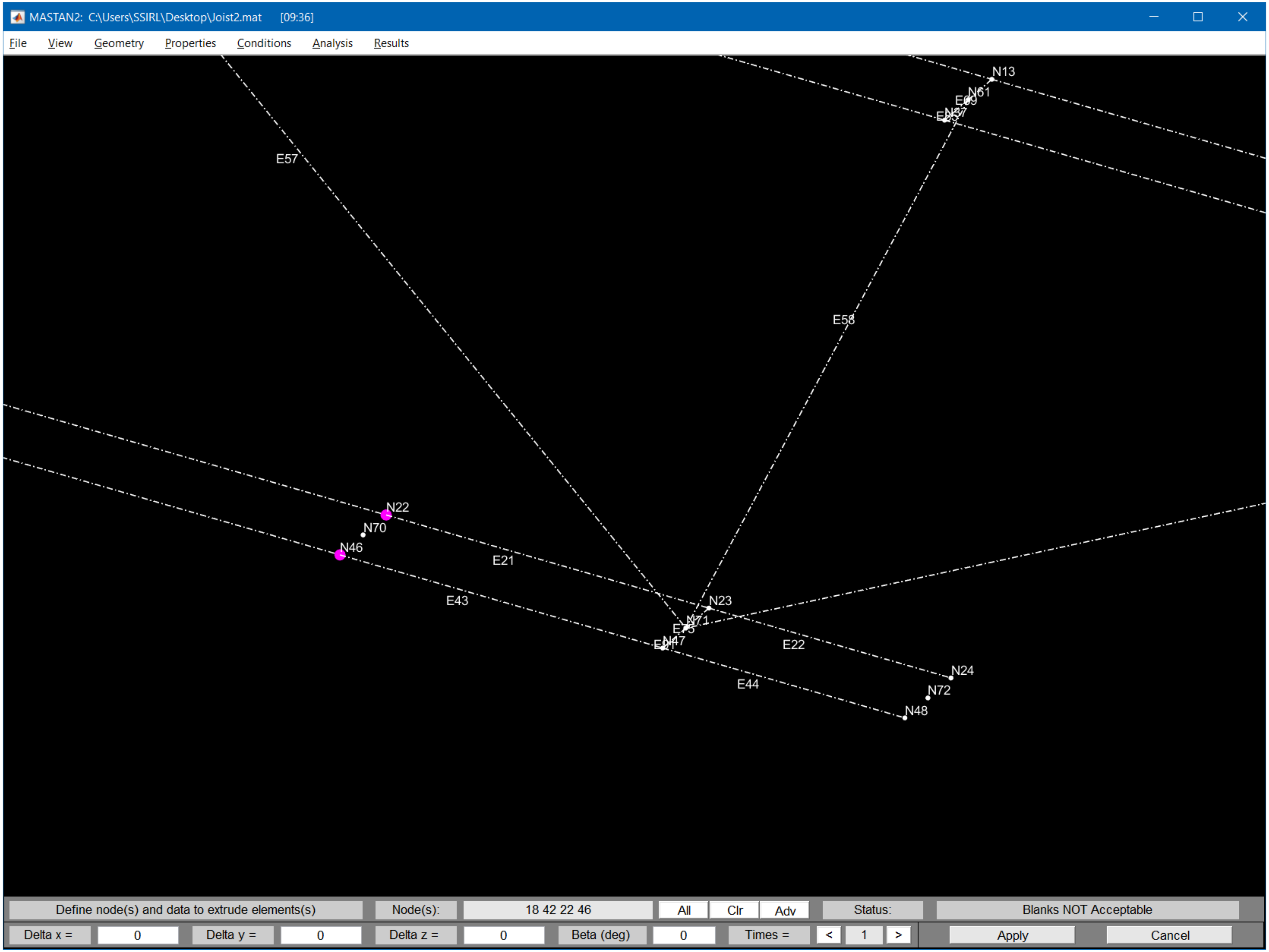
6) Click in the edit box to the right of **Delta z =** and change **0** to **0.99025**. Increase the **Times =** from **1** to **2** by clicking on the **>** button.

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

Note: The elements created in Step 2 will need to be manually selected multiple times through this project. These 4 elements will be referenced as the vertical braces after this point.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist2.mat [09:36]
File View Geometry Properties Conditions Analysis Results
E47
N16
N64
N40
X
E15
E37
N17
N65
E16
E38
N18
N66
N42
Define node(s) and data to extrude elements(s)
Node(s):
18 42
All
Clr
Adv
Status:
Blanks NOT Acceptable
Delta x =
0
Delta y =
0
Delta z =
0
Beta (deg)
0
Times =
<
1
>
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Joist2.mat [09:36]
File
View
Geometry
Properties
Conditions
Analysis
Results
N13
N61
E57
E58
N22
N70
N46
E21
E43
N23
N71
N47
E22
E44
N24
N72
N48
Define node(s) and data to extrude elements(s)
Node(s):
18 42 22 46
All
Clr
Adv
Status:
Blanks NOT Acceptable
Delta x =
0
Delta y =
0
Delta z =
0
Beta (deg)
0
Times =
<
1
>
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
E45
E46
E47
E48
E49
E50
E51
E52
E53
E54
E55
E56
E57
E58
E59
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: C:\Users\SSIRL\Desktop\Joist.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
E45
E46
E47
E48
E49
E50
E51
E52
E53
E54
E55
E56
E57
E58
E59
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

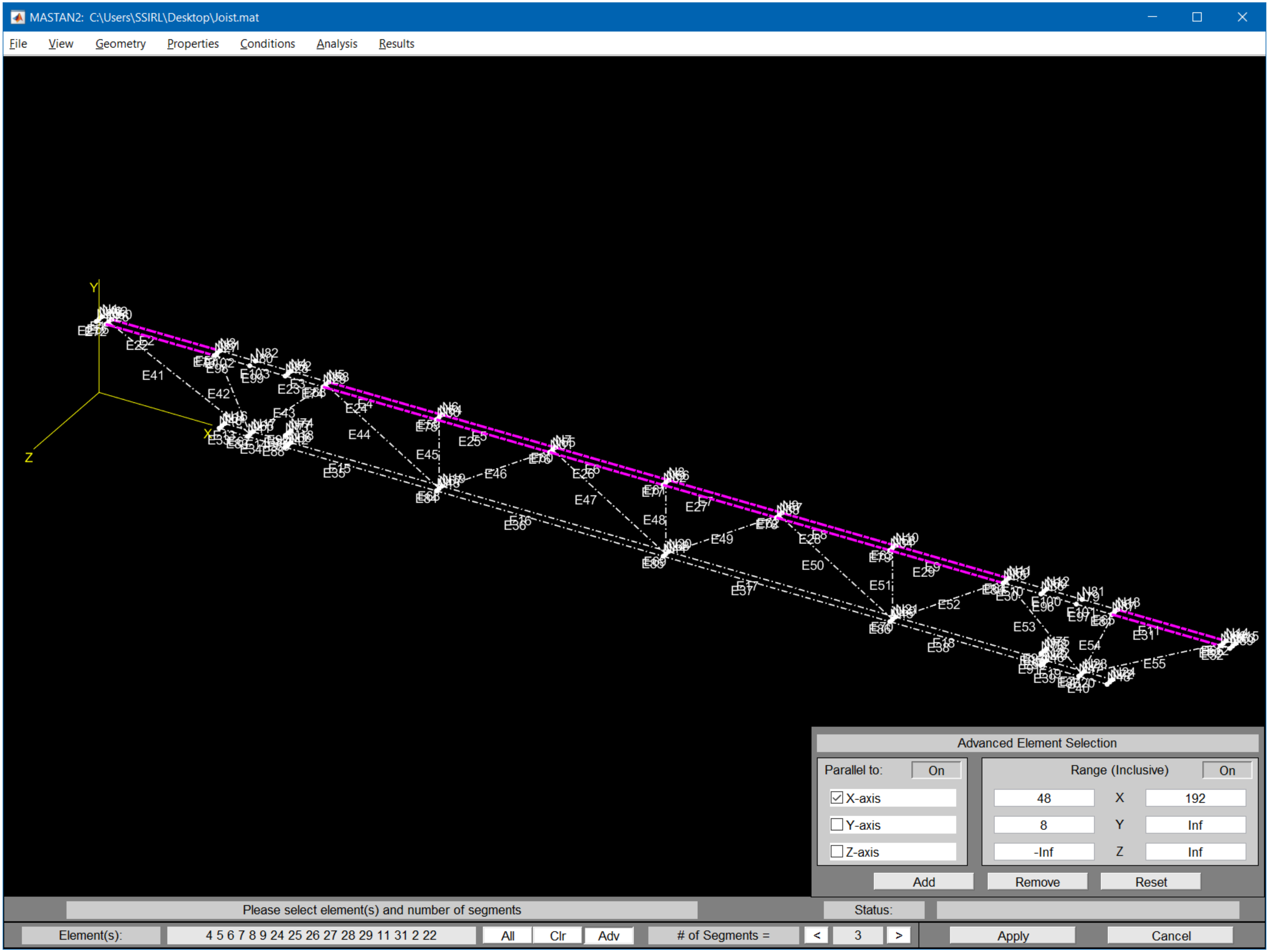
Top Chord Meshing

1) From the **Geometry** menu select **Subdivide Element(s)**. The top chord is to be meshed into approximately 8” segments.
2) Click on the 4 top chord sections near the cross-bracing support to be subdivided 2 times.
3) Click on the **Apply** button.
4) Click the **>** box to the right of **# of Segments =** to increase **2** to **3**.
5) In the bottom menu bar, use the buttons to the right of **Element(s):** to make the list of elements to be subdivided 3 times.
6) Click the **Adv** button to open pop-up menu. To select middle of the top chords, click the check box next to the **X-axis** option. Ensure the button to the right of **Range (Inclusive)** to change **Off** to **On**. Change the edit box to the left of **X** from **–Inf** to **48**. Change the edit box to the right of **X** from **Inf** to **192**. Change the edit box to the left of **Y** from **–Inf** to **8**. Click **Add** to select.
7) Click on the 4 remaining exterior top chord sections to be subdivided 3 times.
8) Click on the **Apply** button.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Please select element(s) and number of segments
Status:
Element(s):
25 3 34 12
All
Clr
Adv
# of Segments =
2
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Advanced Element Selection
Parallel to:
On
X-axis
Y-axis
Z-axis
Range (Inclusive)
On
48
X
192
8
Y
Inf
-Inf
Z
Inf
Add
Remove
Reset
Please select element(s) and number of segments
Status:
Element(s):
4 5 6 7 8 9 24 25 26 27 28 29 11 31 2 22
All
Clr
Adv
# of Segments =
<
3
>
Apply
Cancel

Bottom Chord Meshing

1) The bottom chord is to be meshed into approximately 4” segments.

2) Click on the 4-32” long bottom chord elements.

3) Click the **>** box to the right of **# of Segments =** to increase **3** to **4**.

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

5) Click on the middle 4 bottom chord elements.

6) Click the **>** box to the right of **# of Segments =** to increase **4** to **6**.

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

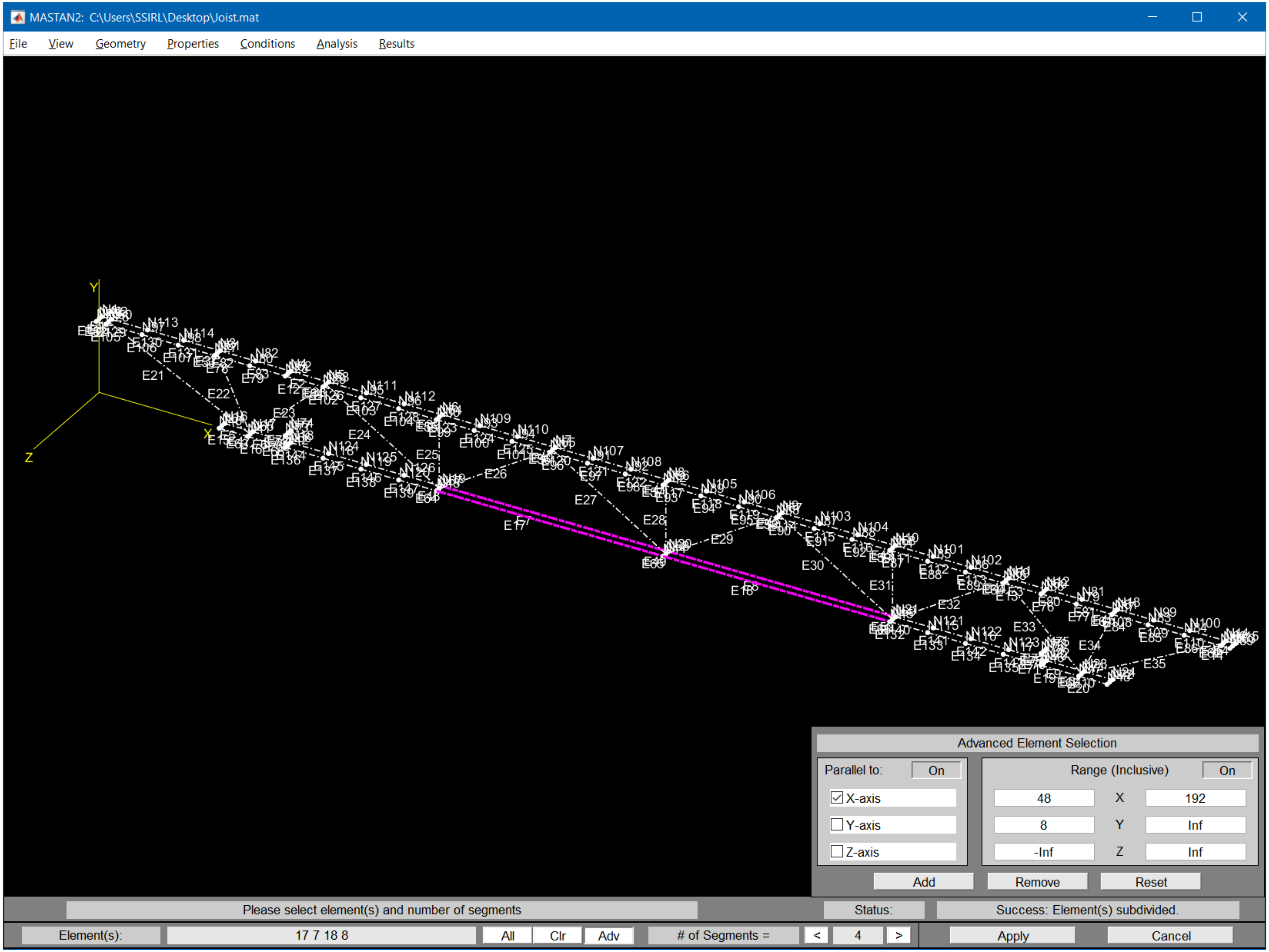
8) On the **Advanced Element Selection** pop-up, click the **Reset** button. Click the check box next to the **X-axis** option. Click the button to the right of **Range (Inclusive)** to change **Off** to **On**. Change the edit box to the right of **Y** from **Inf** to **8**. Click **Add** to select the entire bottom chord.

9) Click the **<** box to the left of **# of Segments =** to decrease **6** to **2**.

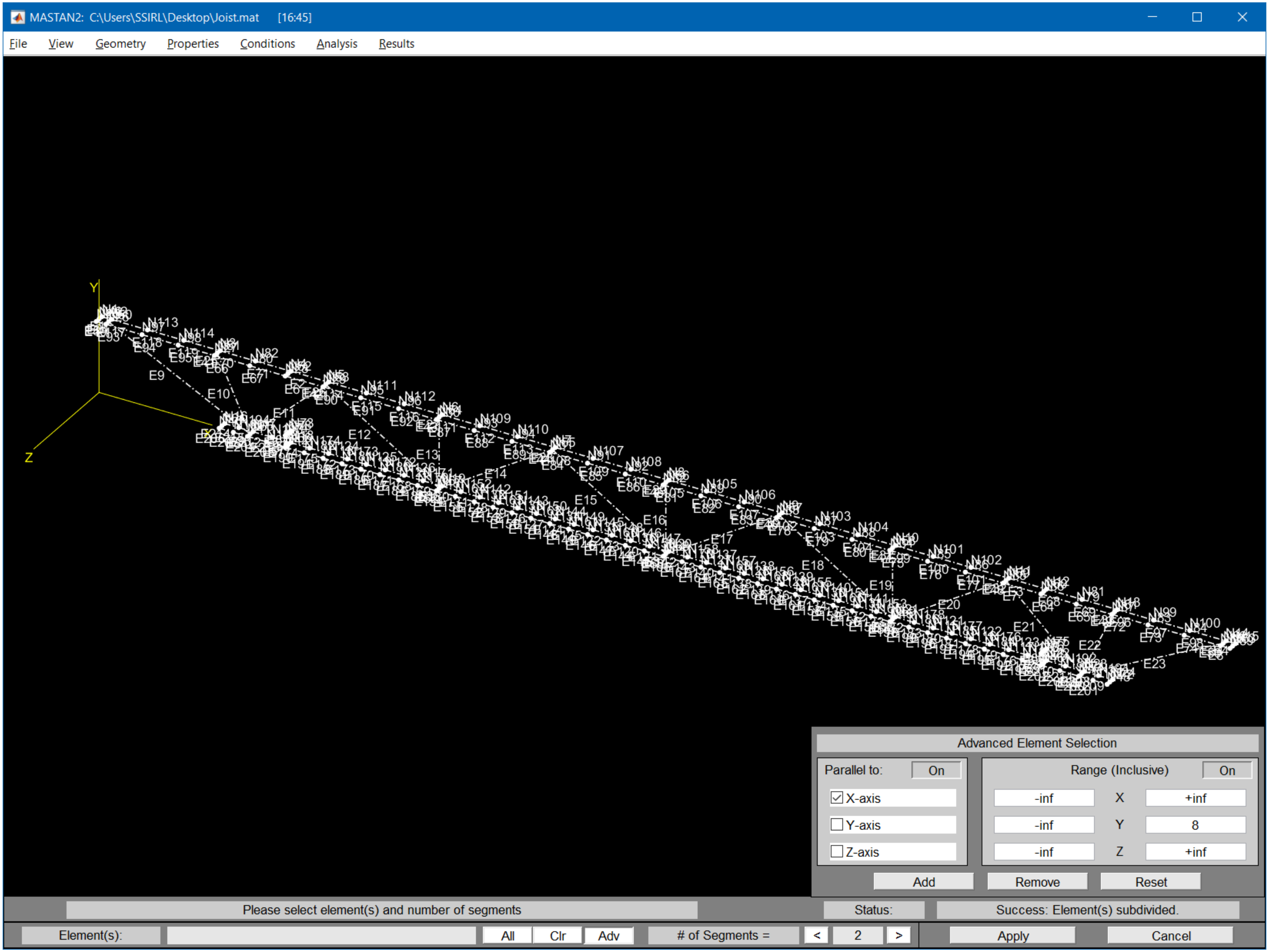
10) Click on the **Apply** button.


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Advanced Element Selection
Parallel to:
On
X-axis
Y-axis
Z-axis
Range (Inclusive)
On
48
X
192
8
Y
Inf
-Inf
Z
Inf
Add
Remove
Reset
Please select element(s) and number of segments
Status:
Success: Element(s) subdivided.
Element(s):
19 7 22 10
All
Clr
Adv
# of Segments =
<
3
>
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Advanced Element Selection
Parallel to:
On
X-axis
Y-axis
Z-axis
Range (Inclusive)
On
48
X
192
8
Y
Inf
-Inf
Z
Inf
Add
Remove
Reset
Please select element(s) and number of segments
Status:
Success: Element(s) subdivided.
Element(s):
17 7 18 8
All
Clr
Adv
# of Segments =
<
4
>
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [16:45]
File
View
Geometry
Properties
Conditions
Analysis
Results
Advanced Element Selection
Parallel to:
On
X-axis
Y-axis
Z-axis
Range (Inclusive)
On
-inf
X
+inf
-inf
Y
8
-inf
Z
+inf
Add
Remove
Reset
Please select element(s) and number of segments
Status:
Success: Element(s) subdivided.
Element(s):
All
Clr
Adv
# of Segments =
<
2
>
Apply
Cancel

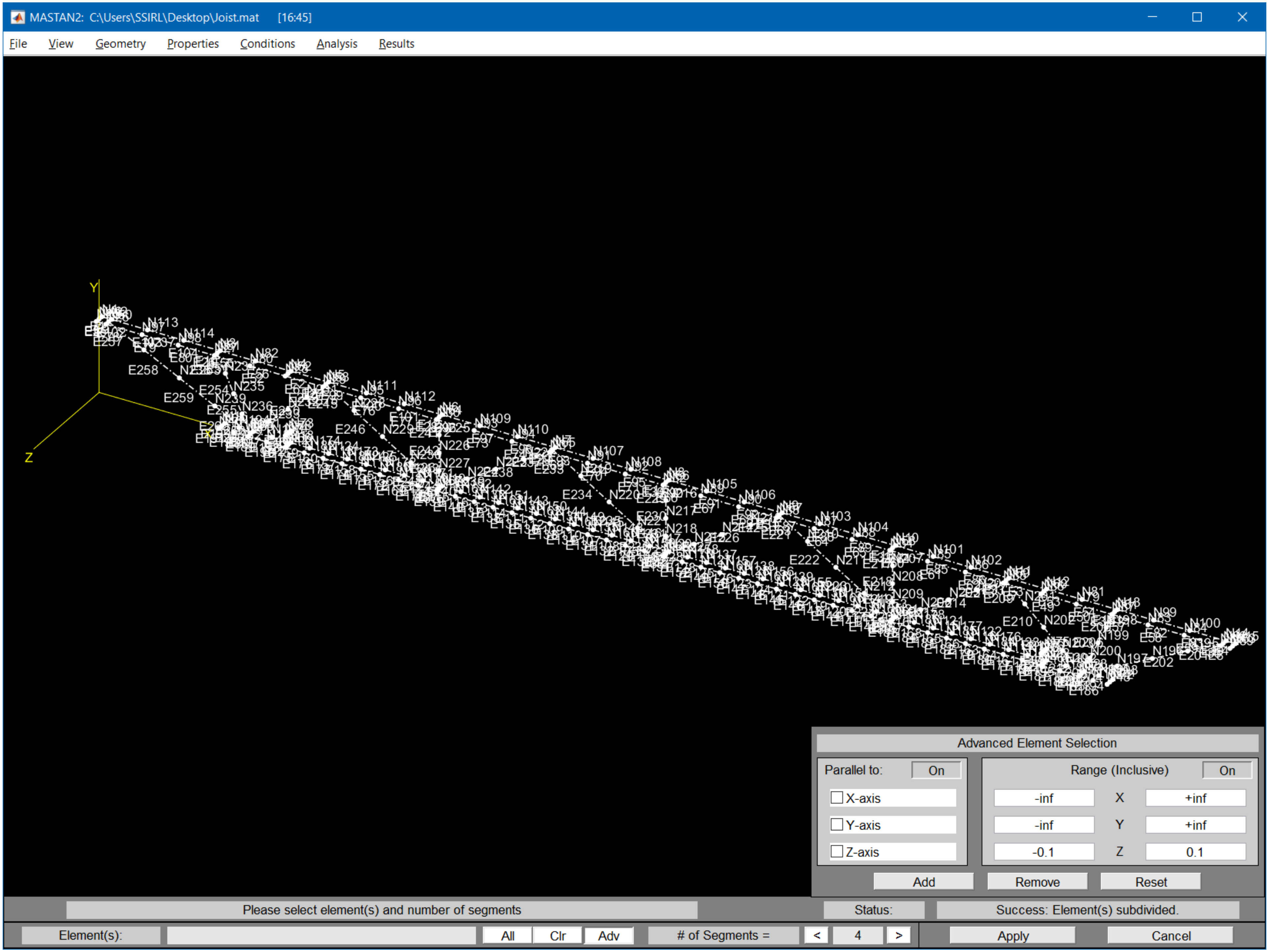
Web Meshing

1) The web is to be meshed into 4 equal segments.

2) Click the **>** box to the right of **# of Segments =** to increase **2** to **4**.

3) On the **Advanced Element Selection** pop-up, click the **Reset** button. Click the button to the right of **Range (Inclusive)** to change **Off** to **On**. Change the edit box to the left of **Z** from -**Inf** to **-0.1** and to the right of **Z** from **Inf** to **0.1**. Click **Add** to select all webs.

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


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [16:45]
File View Geometry Properties Conditions Analysis Results
Y
X
Z
Advanced Element Selection
Parallel to: On
X-axis
Y-axis
Z-axis
Range (Inclusive) On
-inf X +inf
-inf Y +inf
-0.1 Z 0.1
Add Remove Reset
Please select element(s) and number of segments
Status:
Success: Element(s) subdivided.
Element(s):
All Clr Adv
# of Segments =
< 4 >
Apply Cancel

Model Cleanup

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

2) Click on **All Unattached** to select all unconnected nodes that were included for simplicity in the initial model construction.

3) Click on the **Apply** button to remove.

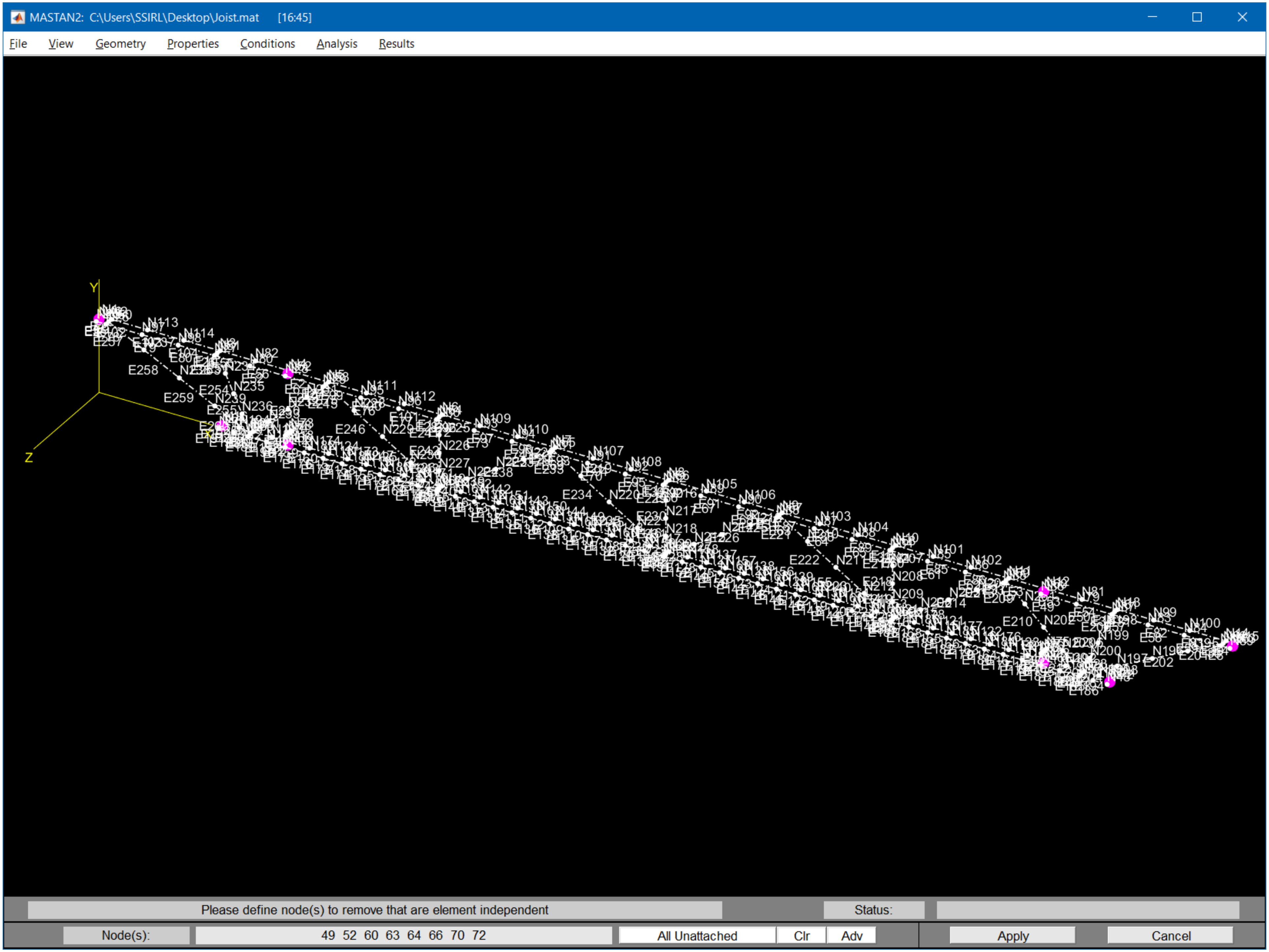
The next steps are not required; however, it will help make it easier to find results in the model.

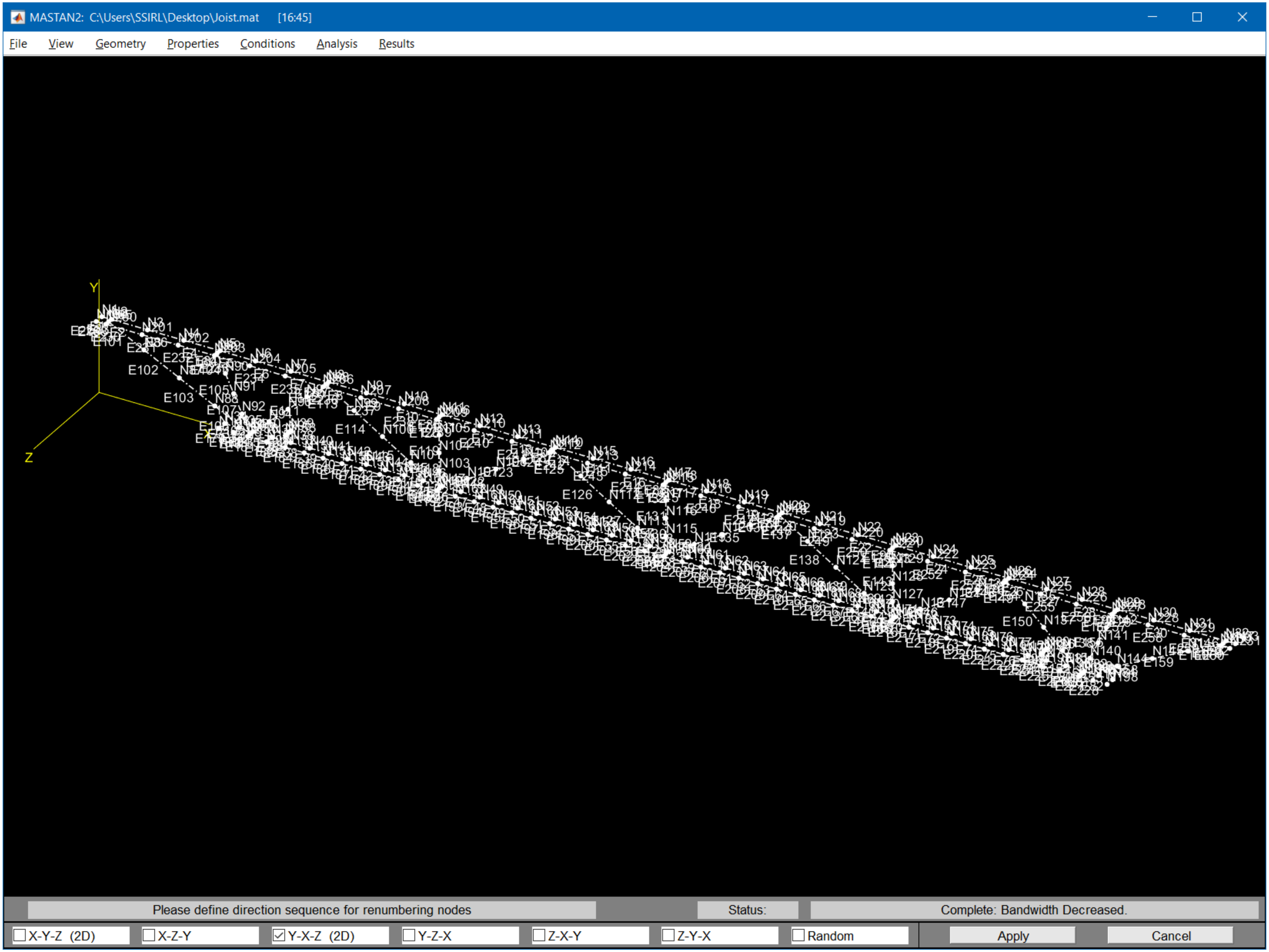
4) From the **Geometry** menu select **Renumber Elements**.

5) Click the checkbox to the left of **Y-X-Z (2D)**. Click on the **Apply** button.

6) From the **Geometry** menu select **Renumber Nodes**.

7) Click the checkbox to the left of **Y-X-Z (2D)**. Click on the **Apply** button.


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [16:45]
File View Geometry Properties Conditions Analysis Results
Please define node(s) to remove that are element independent
Status:
Node(s):
49 52 60 63 64 66 70 72
All Unattached
Clr
Adv
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [16:45]
File
View
Geometry
Properties
Conditions
Analysis
Results
Please define direction sequence for renumbering nodes
Status:
Complete: Bandwidth Decreased.
X-Y-Z (2D)
X-Z-Y
Y-X-Z (2D)
Y-Z-X
Z-X-Y
Z-Y-X
Random
Apply
Cancel

Eccentric Loading Location

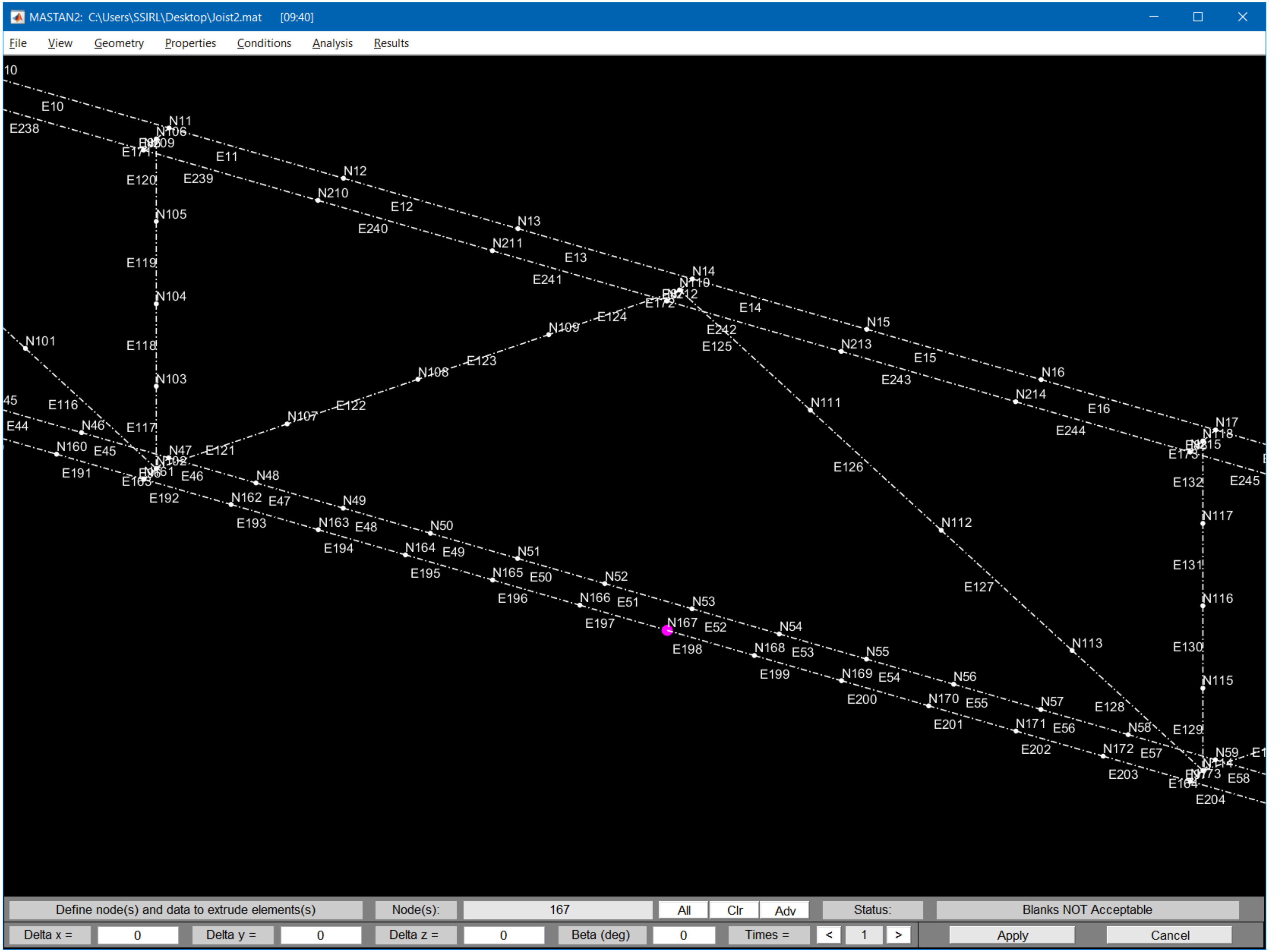
1) From the **Geometry** menu select **Extrude Element**.

2) Click on the node at the middle of the bottom chord where the loading is to be applied. If you renumbered the model, this should be **Node 167**.

3) Click in the edit box to the right of **Delta z =** and change **0** to **1.5** to define the offset loading location.

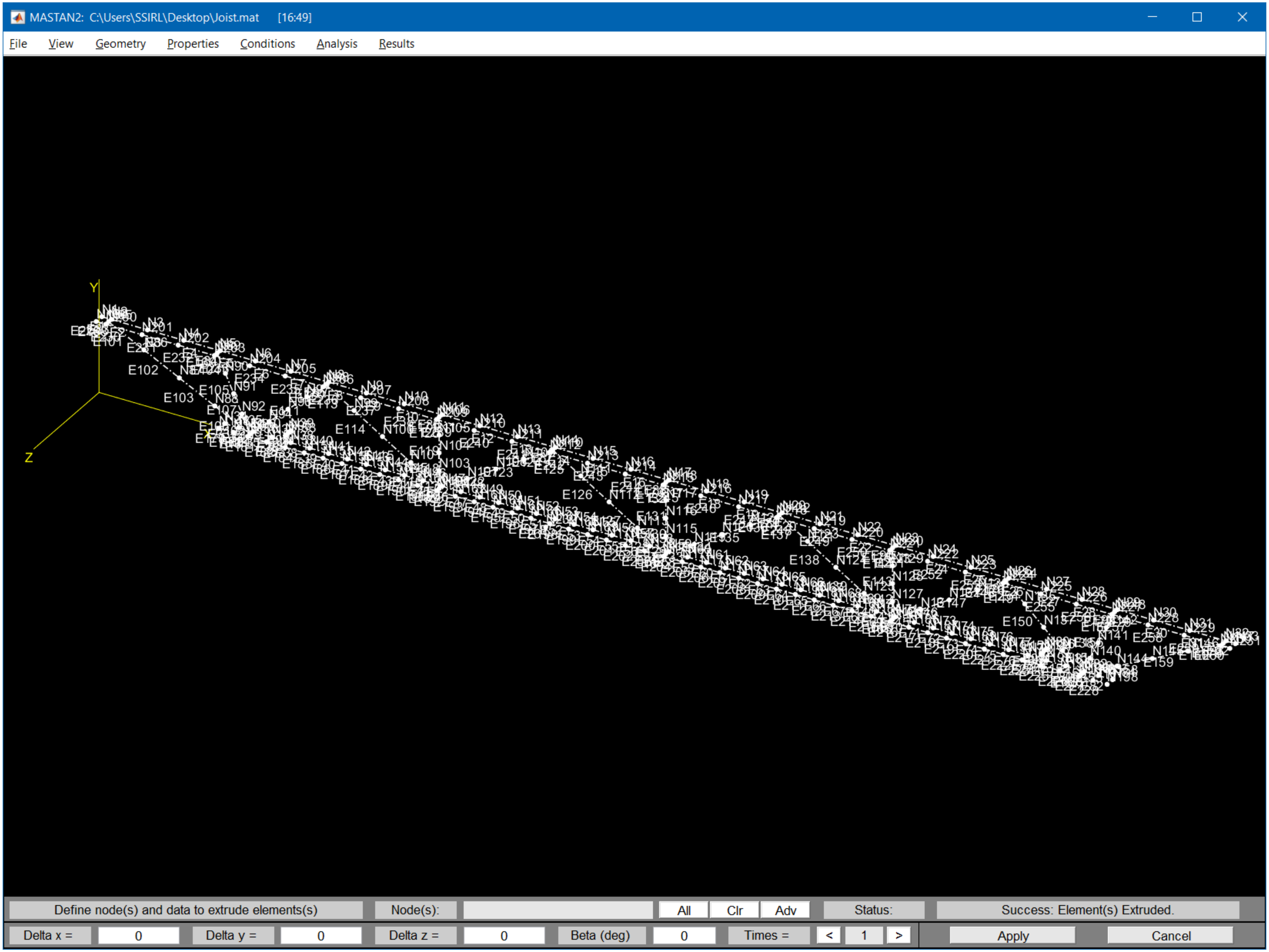
4) Click on the **Apply** Button.

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Joist2.mat [09:40]
File
View
Geometry
Properties
Conditions
Analysis
Results
Define node(s) and data to extrude elements(s)
Node(s):
167
All
Clr
Adv
Status:
Blanks NOT Acceptable
Delta x =
0
Delta y =
0
Delta z =
0
Beta (deg)
0
Times =
1
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [16:49]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
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

Section 4: Member Properties and Connections

Section Properties

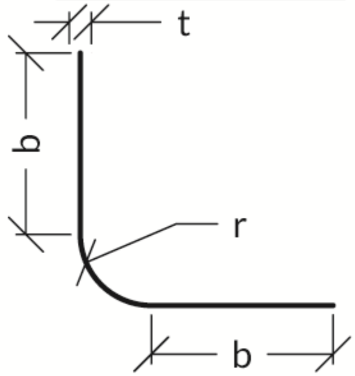
The steel joist model uses 6 sections. There is a separate entry for the top chord angle, the bottom chord angle, the three different web members, and a rigid link connector.

The information can be input by 3 methods. If the section properties are all previously calculated, values can be entered directly. The information can be input individually via the **Define Section** command after switching to the **Advanced** section properties interface or can be imported as a group via the **Input Properties** command. If the section properties need to be calculated, **MSASect** can be used calculate the information.

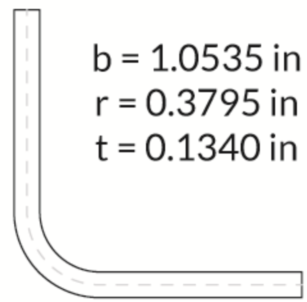
This tutorial will first import all the section property information via the **Input Properties** command. Then it will demonstrate how **MSASect** could have been used to calculate the same information. The top chord angle will be calculated and saved, but not used as part of this analysis.

Cross-Section Geometry

Chord Centerline

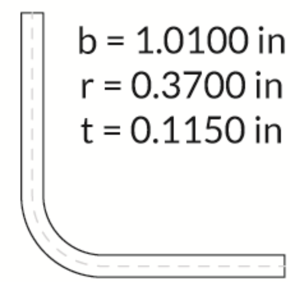


Top Chord

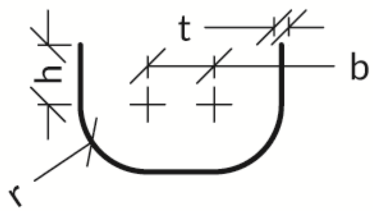


b = 1.0535 in
r = 0.3795 in
t = 0.1340 in

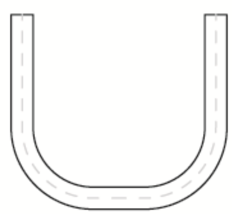
Bottom Chord



b = 1.0100 in
r = 0.3700 in
t = 0.1150 in

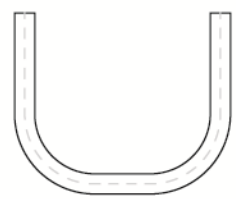


Web Centerline



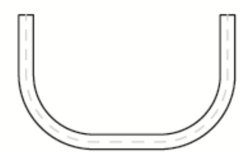
Web 1

h = 0.5844 in
b = 0.2700 in
r = 0.3700 in
t = 0.1150 in



Web 2

h = 0.5258 in
b = 0.2900 in
r = 0.3650 in
t = 0.1050 in



Web 3

h = 0.3102 in
b = 0.3466 in
r = 0.3509 in
t = 0.0767 in

Importing Section Properties

1) From the **Properties** menu select **Input Properties**.

2) At the bottom menu bar, **Sections** should already be selected. Copy and paste the values below into the edit box below **|Name| Area Izz Iyy J Cw Zzz Zyy Ayy Azz Ysc Zsc BetaV BetaW Betaw Iyz**.

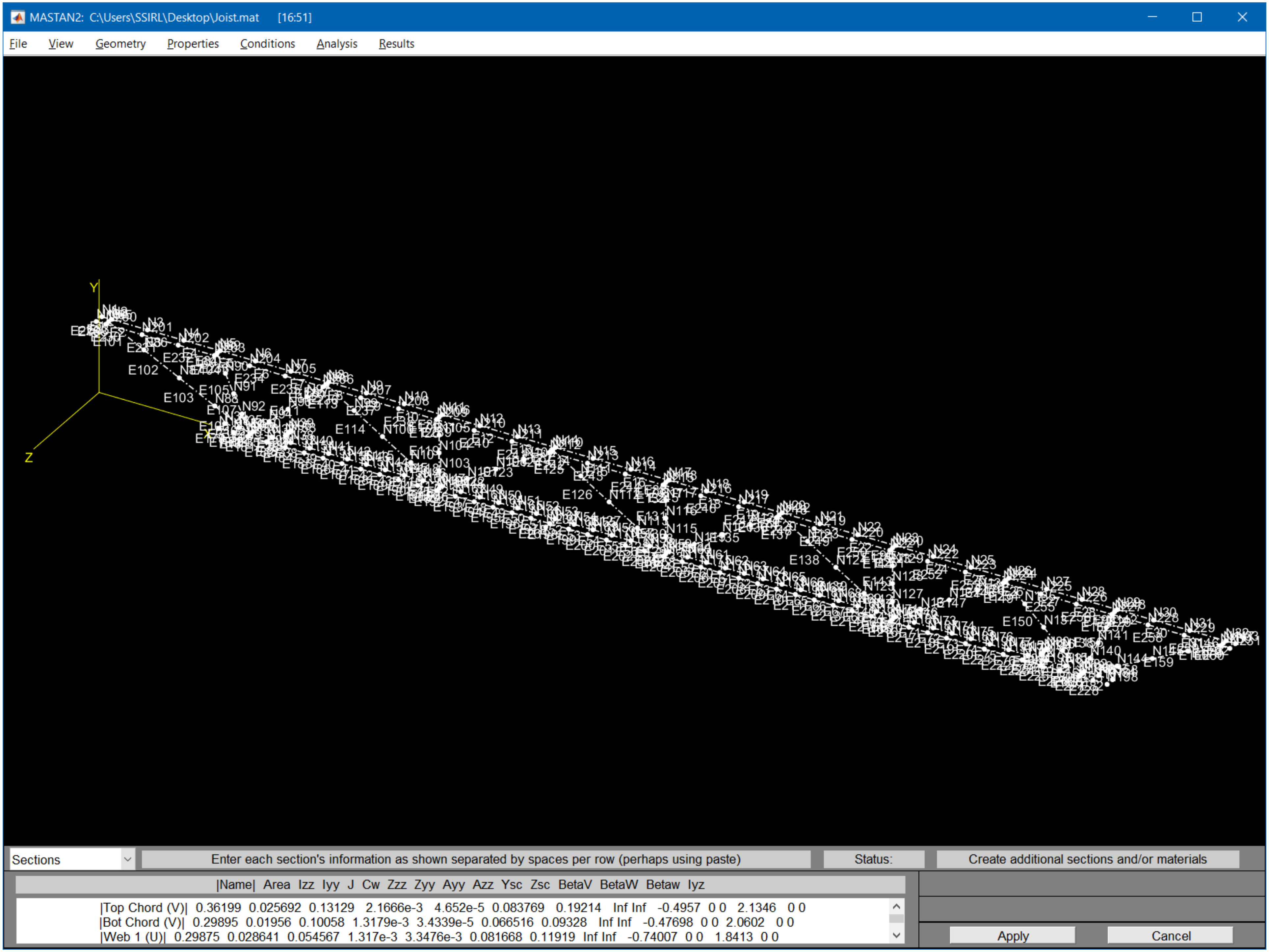
All members are entered in a principal orientation. The letter at the end of the name is to help remember the orientation with the corner of the angle members down and the opening of the channels up.

|Top Chord (V)|	0.36199	0.025692	0.13129	2.1666e-3	4.652e-5	0.083769	0.19214	Inf Inf	-0.4957	0 0	2.1346	0 0
|Bot Chord (V)|	0.29895	0.01956	0.10058	1.3179e-3	3.4339e-5	0.066516	0.09328	Inf Inf	-0.47698	0 0	2.0602	0 0
|Web 1 (U)|	0.29875	0.028641	0.054567	1.317e-3	3.3476e-3	0.081668	0.11919	Inf Inf	-0.74007	0 0	1.8413	0 0
|Web 2 (U)|	0.26093	0.021852	0.047564	9.589e-4	2.5185e-3	0.066672	0.10367	Inf Inf	-0.68065	0 0	1.7533	0 0
|Web 3 (U)|	0.15847	0.0072663	0.027533	3.1075e-4	7.4428e-4	0.029823	0.060575	Inf Inf	-0.46984	0 0	1.4813	0 0
|RIGID|	4	1.33	1.33	2.25	64	4	4	Inf Inf	0	0 0	0	0 0

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

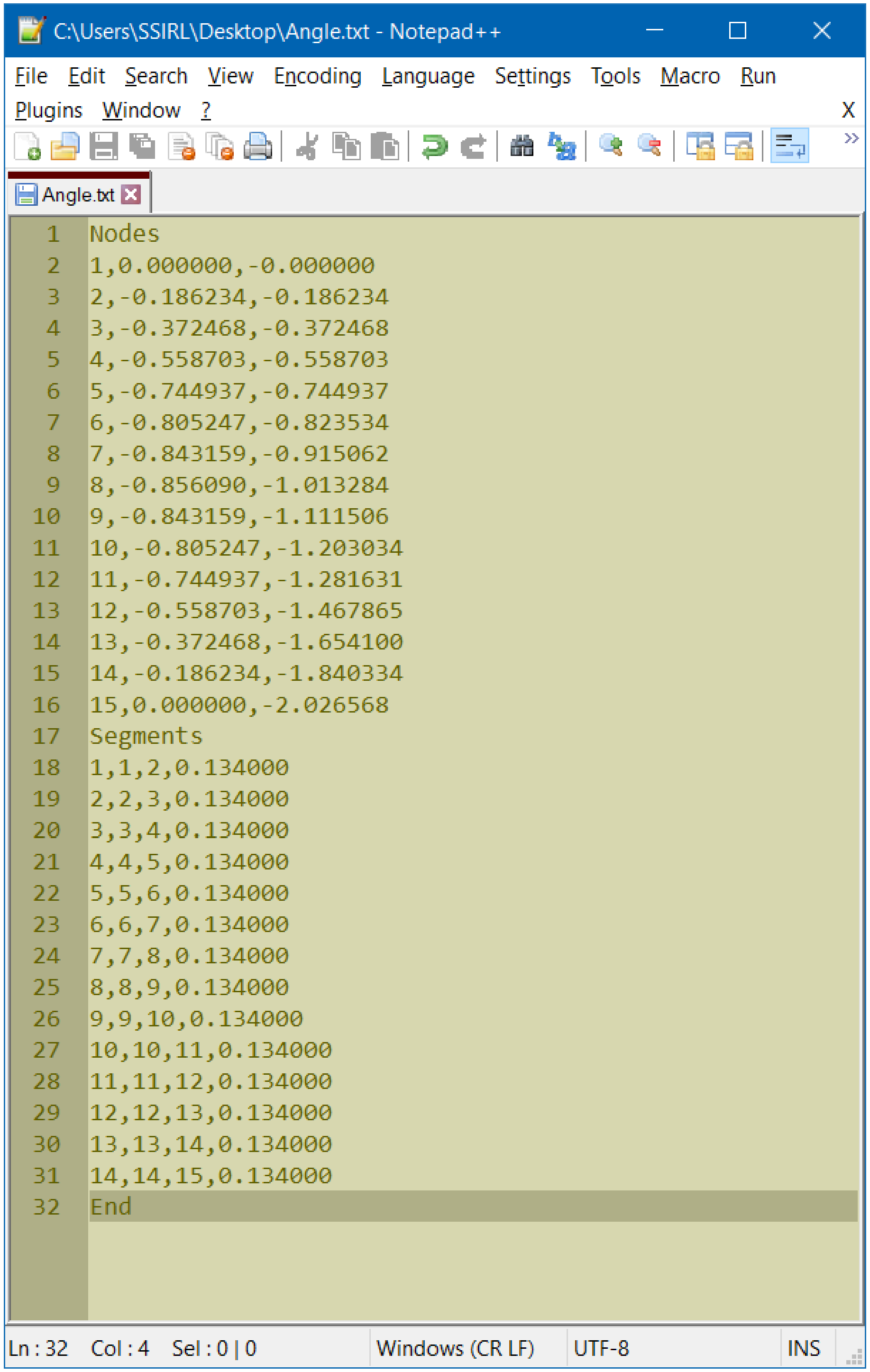
The section properties of the rigid element are approximated based on a 2” x 2” solid square as it has larger section properties than the majority of the joist elements. Combined with the higher modulus of elasticity will provide the effective rigid link. Care must be taken as too stiff of a link can cause issues with the solver and too soft will add unintended deformations.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [16:51]
File View Geometry Properties Conditions Analysis Results
Y
Z
X
Sections
Enter each section's information as shown separated by spaces per row (perhaps using paste)
Status:
Create additional sections and/or materials
|Name| Area Izz Iyy J Cw Zzz Zyy Ayy Azz Ysc Zsc BetaV BetaW Betaw Iyz
|Top Chord (V)| 0.36199 0.025692 0.13129 2.1666e-3 4.652e-5 0.083769 0.19214 Inf Inf -0.4957 0 0 2.1346 0 0
|Bot Chord (V)| 0.29895 0.01956 0.10058 1.3179e-3 3.4339e-5 0.066516 0.09328 Inf Inf -0.47698 0 0 2.0602 0 0
|Web 1 (U)| 0.29875 0.028641 0.054567 1.317e-3 3.3476e-3 0.081668 0.11919 Inf Inf -0.74007 0 0 1.8413 0 0
Apply
Cancel

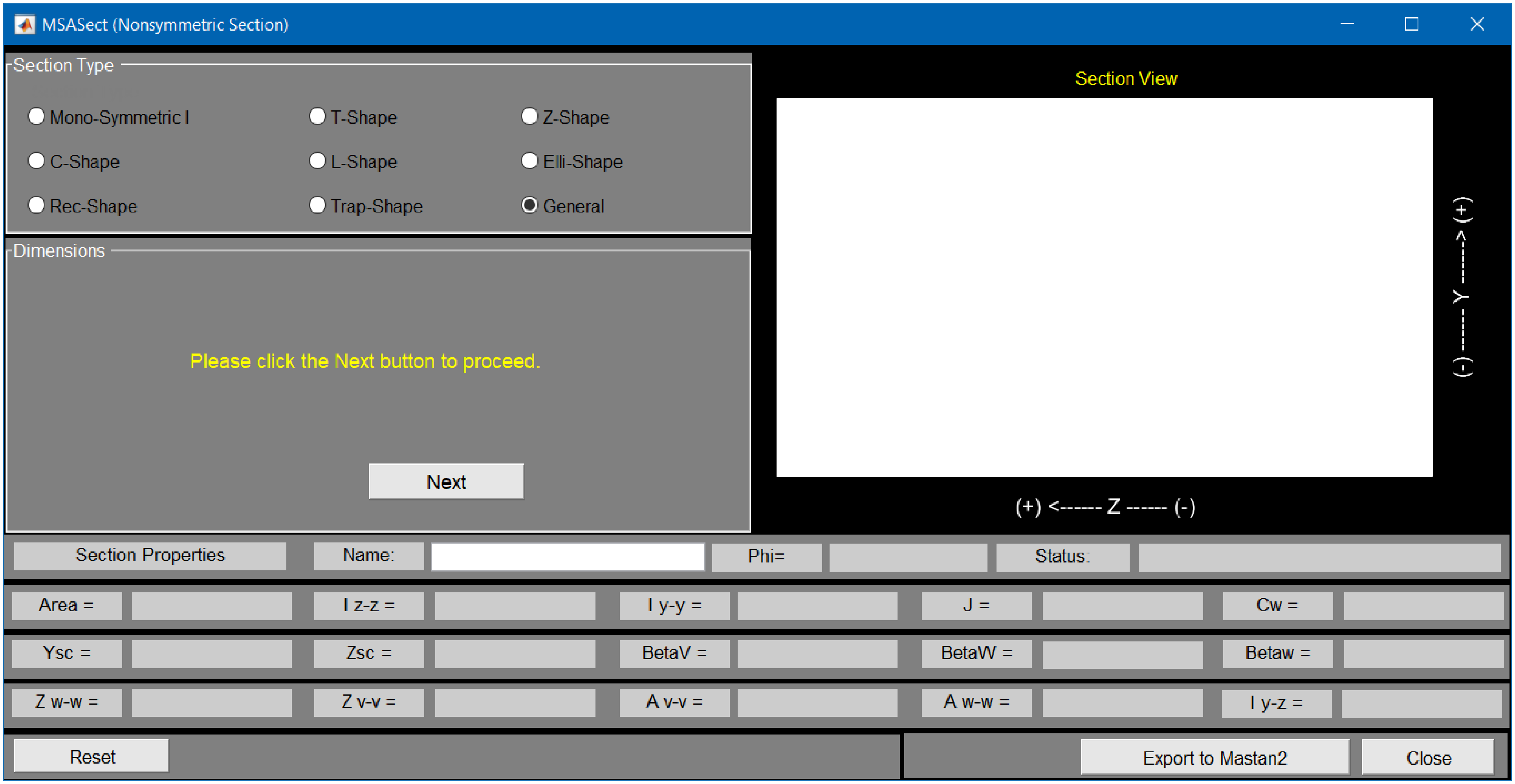
Calculating Section Properties

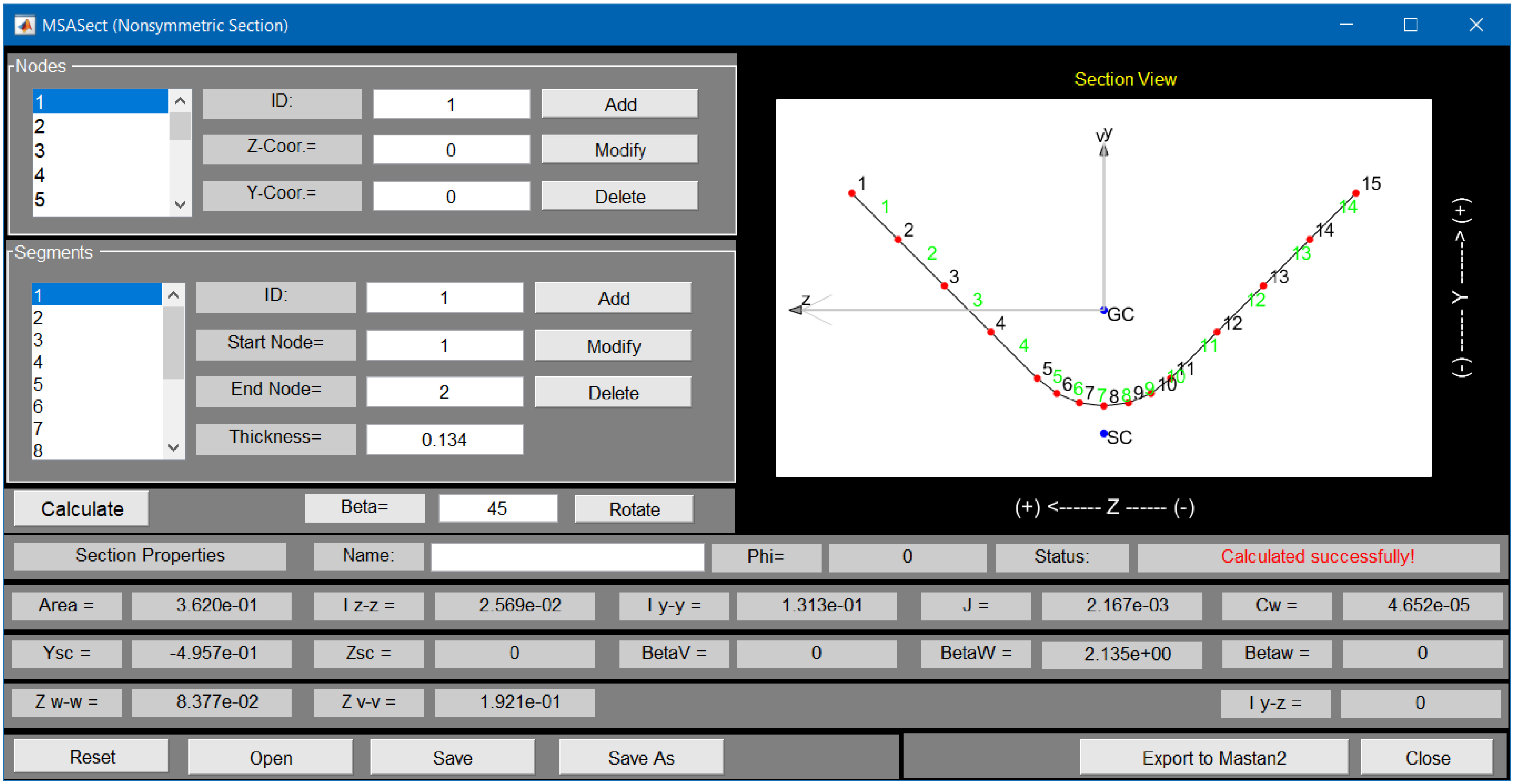
1) Outside of MASTAN2, create a text file that summarizes the node and segment data similar to the one shown here.
2) From the **Properties** menu select **Define Section**.
3) At the bottom menu bar, click on the pop-up menu on the far right that currently displays **Basic**. Click on **Advanced**.
4) Click on **MSASect**.
5) After the interface loads, click the radio button next to **General**.
6) Click **Next** to open the editable general section interface.
7) Click **Open** at the bottom of the screen.
8) Navigate to the location of the text file. After selecting it, click **Open**.
9) Click **Calculate** to determine the properties.
10) Click edit box to right of **Name:** and enter **Top Chord Alt**.
11) Click **Export to MASTAN2** to copy values to main program. Click **Close** to return to main program.
12) Click **Apply** to save Section 7.



Version to Copy for own Text File

Nodes
1,0.000000,-0.000000
2,-0.186234,-0.186234
3,-0.372468,-0.372468
4,-0.558703,-0.558703
5,-0.744937,-0.744937
6,-0.805247,-0.823534
7,-0.843159,-0.915062
8,-0.856090,-1.013284
9,-0.843159,-1.111506
10,-0.805247,-1.203034
11,-0.744937,-1.281631
12,-0.558703,-1.467865
13,-0.372468,-1.654100
14,-0.186234,-1.840334
15,0.000000,-2.026568
Segments
1,1,2,0.134000
2,2,3,0.134000
3,3,4,0.134000
4,4,5,0.134000
5,5,6,0.134000
6,6,7,0.134000
7,7,8,0.134000
8,8,9,0.134000
9,9,10,0.134000
10,10,11,0.134000
11,11,12,0.134000
12,12,13,0.134000
13,13,14,0.134000
14,14,15,0.134000
End


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
(+) <------ 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 =
A v-v =
A w-w =
I y-z =
Reset
Export to Mastan2
Close


MSASect (Nonsymmetric Section)
Nodes
ID: 1
Z-Coor.= 0
Y-Coor.= 0
Add
Modify
Delete
Segments
ID: 1
Start Node= 1
End Node= 2
Thickness= 0.134
Add
Modify
Delete
Calculate
Beta= 45
Rotate
Section View
GC
SC
(+) <------ Z ------ (-)
(-) ------- Y -------> (+)
Section Properties
Name:
Phi= 0
Status: Calculated successfully!
Area = 3.620e-01
I z-z = 2.569e-02
I y-y = 1.313e-01
J = 2.167e-03
Cw = 4.652e-05
Ysc = -4.957e-01
Zsc = 0
BetaV = 0
BetaW = 2.135e+00
Betaw = 0
Z w-w = 8.377e-02
Z v-v = 1.921e-01
I y-z = 0
Reset
Open
Save
Save As
Export to Mastan2
Close

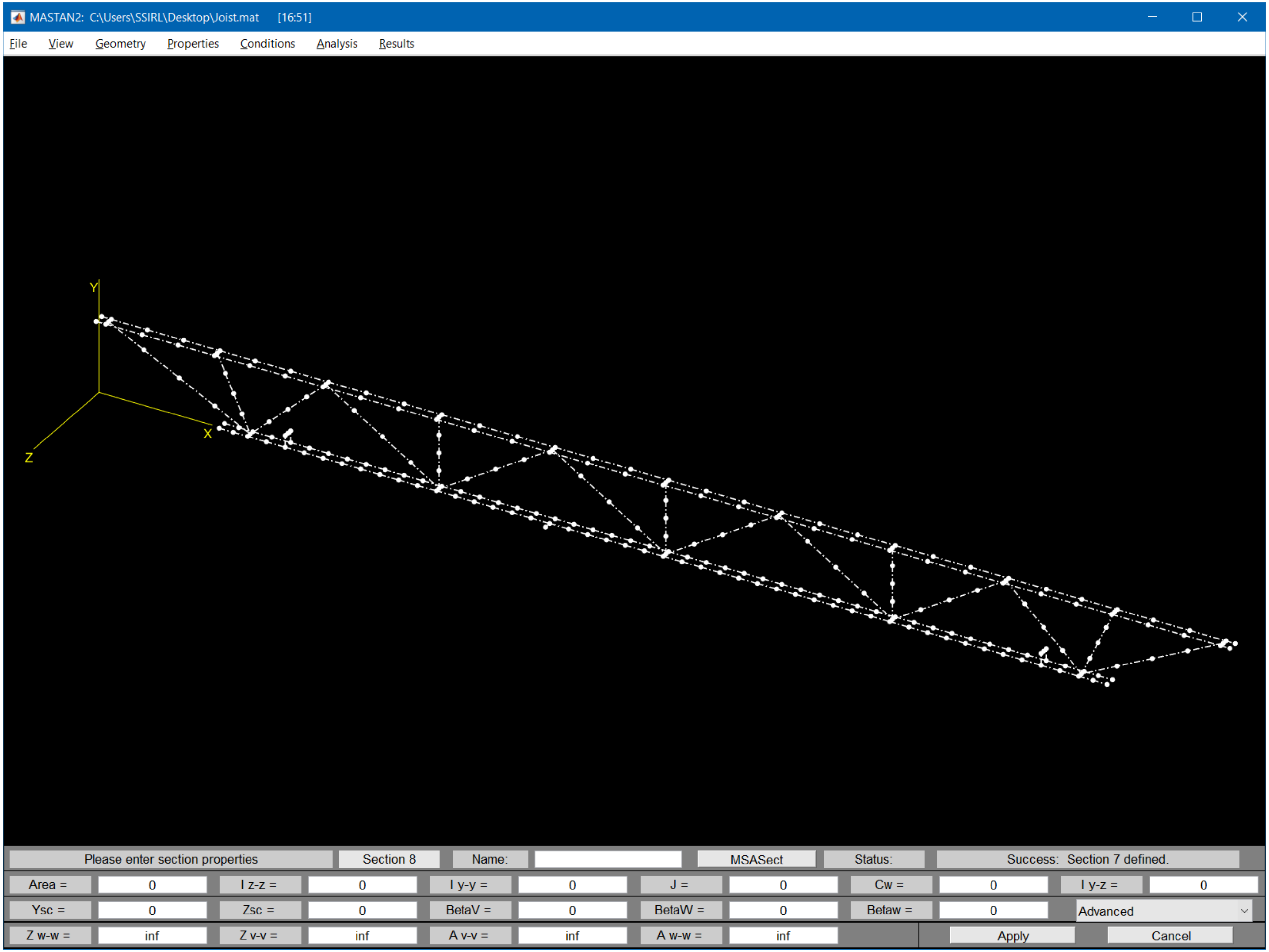
Improving Visual

Currently there are many elements and node labels that make it difficult to understand what is going on. While not necessary, this tutorial will turn off the labels. Following these steps again would allow the user to put the labels back into the model.

1) From the **View** menu select **Labels** and submenu option **Node #s**.

2) From the **View** menu select **Labels** and submenu option **Element #s**.

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [16:51]
File View Geometry Properties Conditions Analysis Results
Y
X
Z
Please enter section properties
Section 8
Name:
MSASect
Status:
Success: Section 7 defined.
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

Section Properties – Assigning – 1

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) Click the **Adv** button to open pop-up menu. Click the **Reset** button. To select both top chords, click the check box next to the **X-axis** option. Click the button to the right of **Range (Inclusive)** from **Off** to **On**. Change the edit box to the left of **Y** to **8**.

4) Click **Add** to add the top chord elements to the element list.

5) Click on the **Apply** button to assign Section 1 to the top chord elements.

6) Change the **Section #** by clicking on the current section number just to the right to open a pop-up menu with all section numbers. Click on **2** to select the "Bot Chord (V)" section.

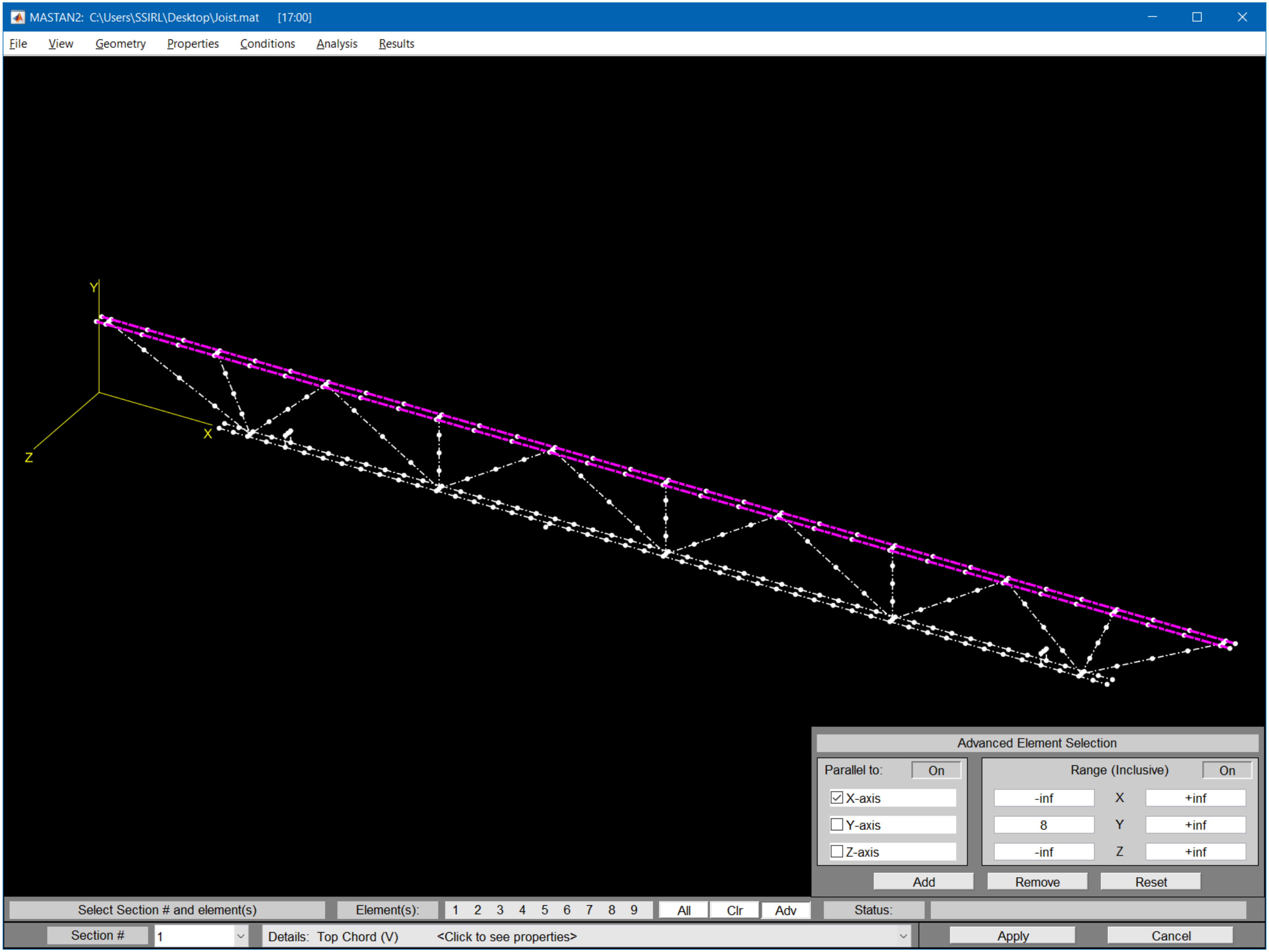
7) Select the **Clr** button located to the right of **Elements:** to clear the list of elements.

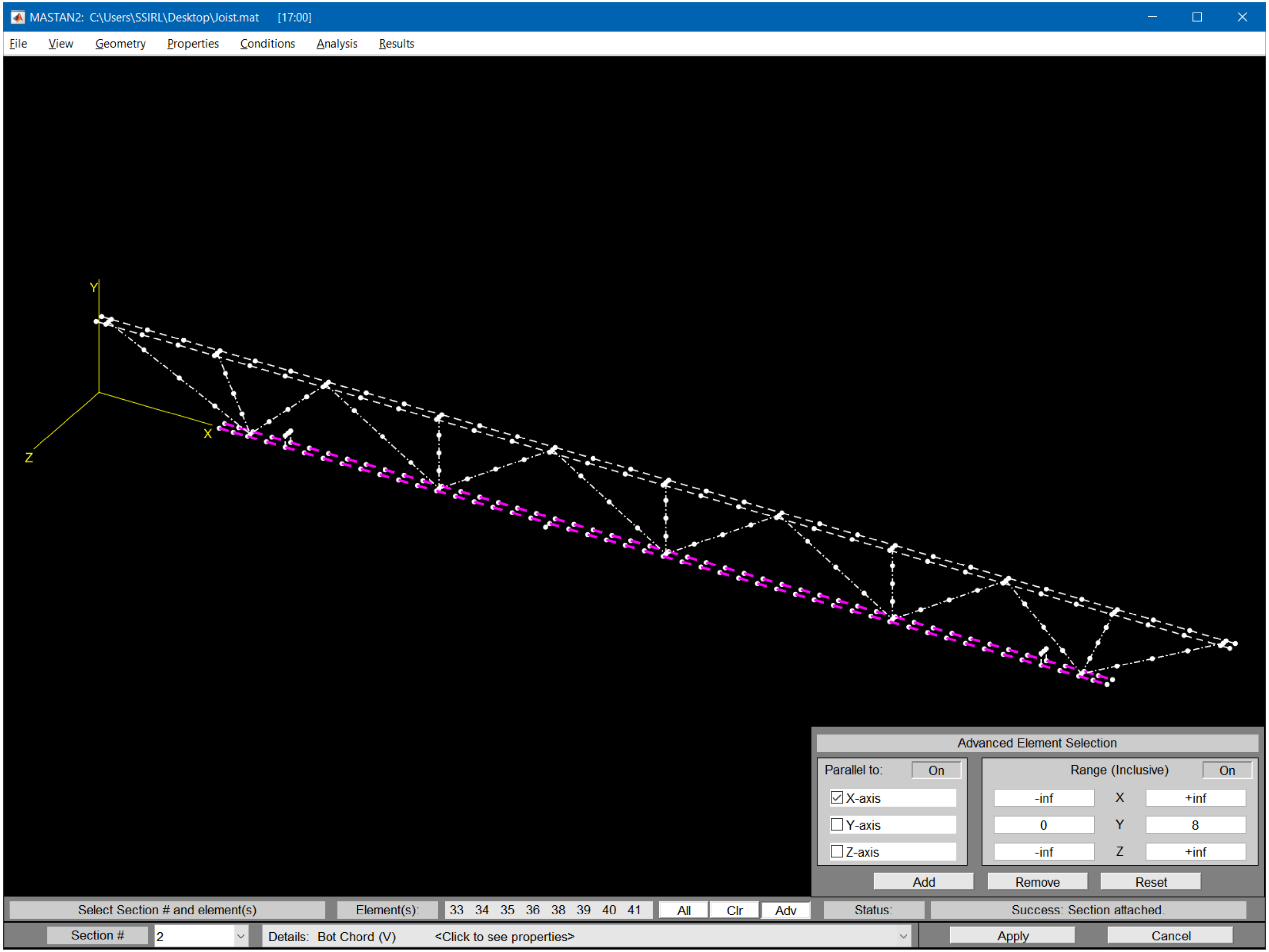
8) Change the edit box to the left of **Y** to **0**. Change the edit box to the right of **Y** to **8**.

9) Click **Add** to add the bottom chord elements to the element list.

10) Click on the **Apply** button to assign Section 2 to the bottom chord.

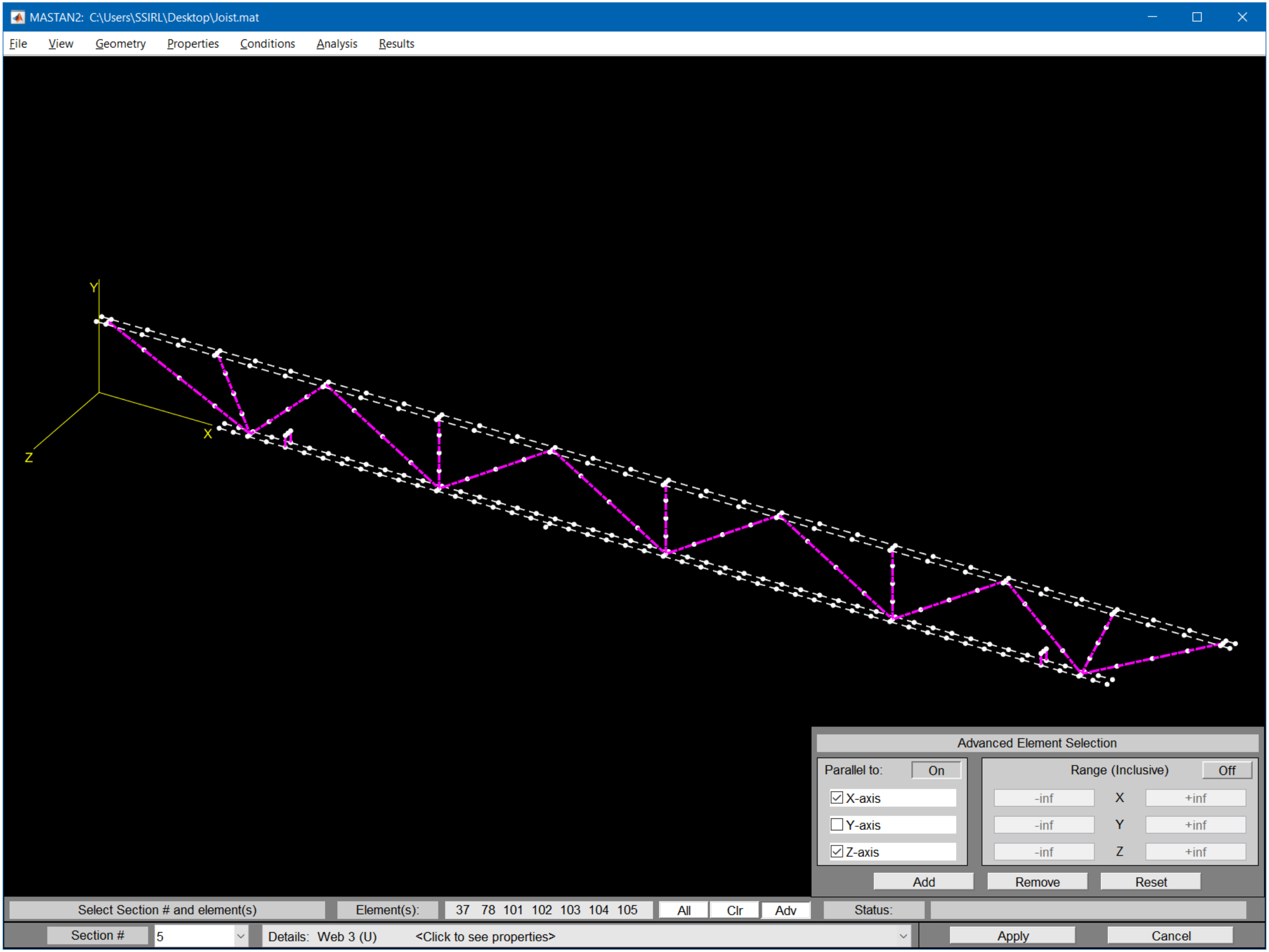
MASTAN2

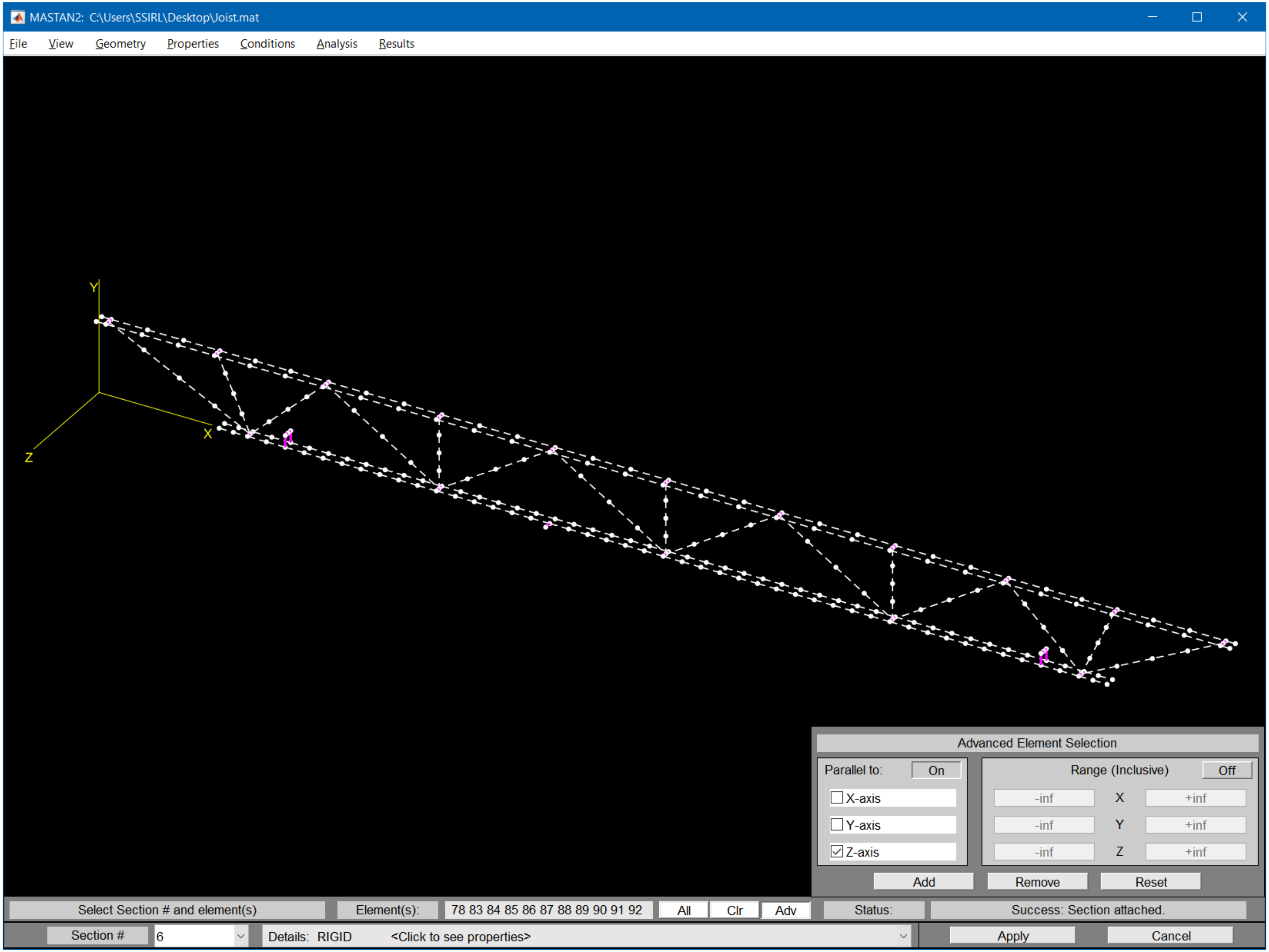

MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:00]
File View Geometry Properties Conditions Analysis Results
Y
X
Z
Advanced Element Selection
Parallel to: On
X-axis
Y-axis
Z-axis
Range (Inclusive) On
-inf X +inf
8 Y +inf
-inf Z +inf
Add Remove Reset
Select Section # and element(s)
Element(s): 1 2 3 4 5 6 7 8 9
All Clr Adv
Status:
Section # 1
Details: Top Chord (V) <Click to see properties>
Apply Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:00]
File View Geometry Properties Conditions Analysis Results
Y
X
Z
Advanced Element Selection
Parallel to: On
X-axis
Y-axis
Z-axis
Range (Inclusive) On
-inf X +inf
0 Y 8
-inf Z +inf
Add Remove Reset
Select Section # and element(s)
Element(s): 33 34 35 36 38 39 40 41
All Clr Adv
Status: Success: Section attached.
Section # 2
Details: Bot Chord (V) <Click to see properties>
Apply Cancel

Section Properties - Assigning – 2

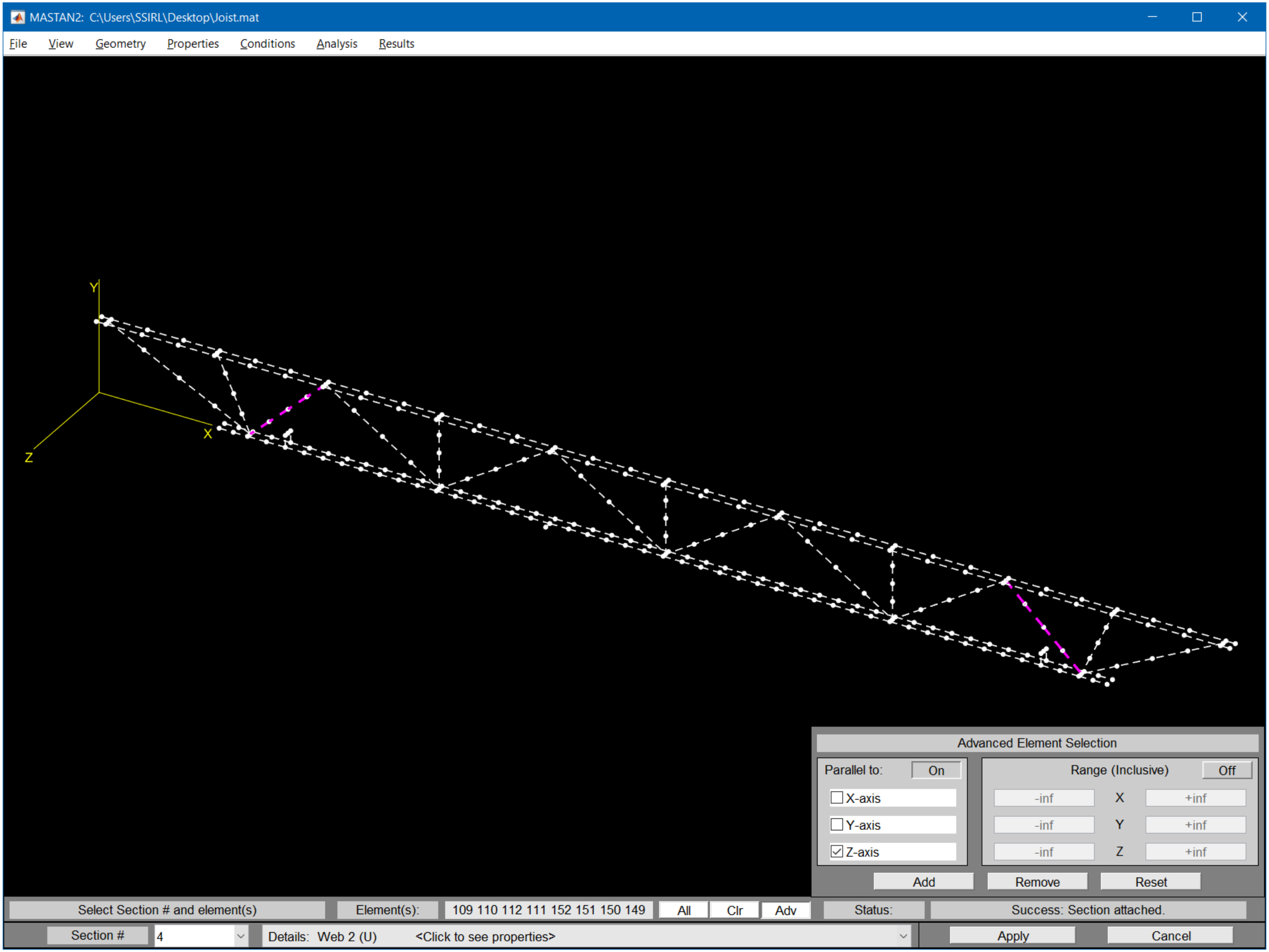
1) Change the **Section #** by clicking on the current section number just to the right to open a pop-up menu with all section numbers. Click on **5** to select the “Web 3 (U)” section.
2) Select the **Clr** button located to the right of **Elements:** to clear the list of elements. Click **Reset** at the bottom of the Advanced Element Section.
3) Click the check box next to **X-axis** and the check box next to **Z-axis**.
4) Click **All** to the right of **Element(s):** and then **Remove** to select all the webs.
5) Click on the **Apply** button to assign Section 5 to all webs temporarily.
6) Change the **Section #** by clicking on the current section number just to the right to open a pop-up menu with all section numbers. Click on **6** to select the “RIGID” section
7) Select the **Clr** button located to the right of **Elements:** to clear the list of elements.
8) Click the check box next to **X-axis**. **Z-axis** should still be selected.
9) Click **Add** to select all the rigid connectors. Additionally click on the 4 vertical braces.
10) Click on the **Apply** button to assign Section 6 to the rigid connectors.


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat
File View Geometry Properties Conditions Analysis Results
Y
X
Z
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
Select Section # and element(s)
Element(s): 37 78 101 102 103 104 105
All Clr Adv
Status:
Section # 5
Details: Web 3 (U) <Click to see properties>
Apply Cancel

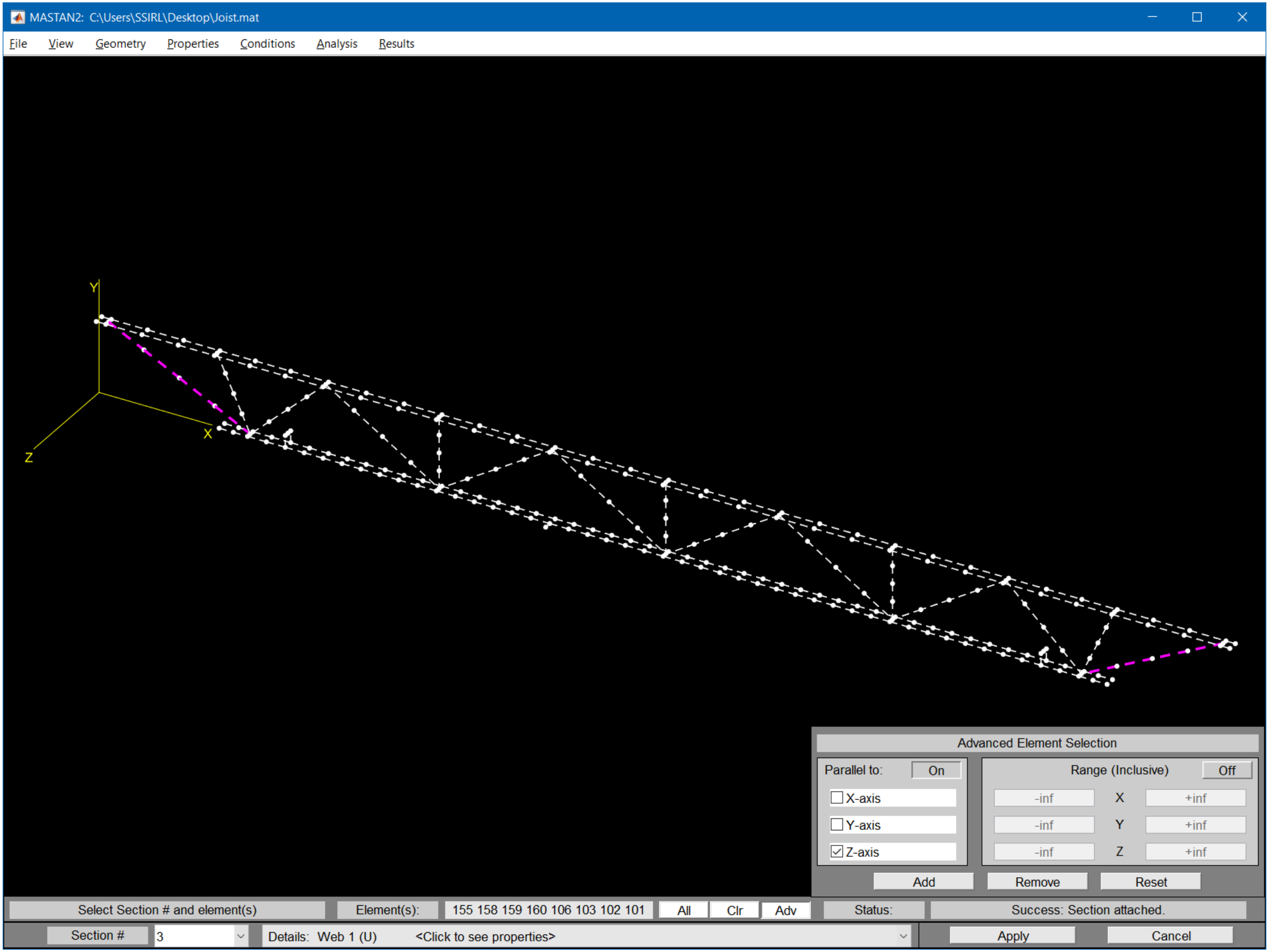

MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
Advanced Element Selection
Parallel to:
On
X-axis
Y-axis
Z-axis
Range (Inclusive)
Off
-inf
X
+inf
-inf
Y
+inf
-inf
Z
+inf
Add
Remove
Reset
Select Section # and element(s)
Element(s):
78 83 84 85 86 87 88 89 90 91 92
All
Clr
Adv
Status:
Success: Section attached.
Section #
6
Details: RIGID
<Click to see properties>
Apply
Cancel

Section Properties - Assigning – 3

1) Change the **Section #** by clicking on the current section number just to the right to open a pop-up menu with all section numbers. Click on **4** to select the "Web 2 (U)" section.
2) Select the **Clr** button located to the right of **Elements:** to clear the list of elements.
3) Click on all the elements to be assigned Web 2 section properties.
4) Assign Section 4 properties by clicking the **Apply** button.
5) Change the **Section #** by clicking on the current section number just to the right to open a pop-up menu with all section numbers. Click on **3** to select the "Web 1 (U)" section.
6) Select the **Clr** button located to the right of **Elements:** to clear the list of elements.
7) Click on all the elements to be assigned Web 1 section properties.
8) Assign Section 3 properties by clicking the **Apply** button.


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
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
Select Section # and element(s)
Element(s):
109 110 112 111 152 151 150 149
All
Clr
Adv
Status:
Success: Section attached.
Section #
4
Details: Web 2 (U)
<Click to see properties>
Apply
Cancel

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat
File View Geometry Properties Conditions Analysis Results
Y
X
Z
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
Select Section # and element(s)
Element(s): 155 158 159 160 106 103 102 101
All Clr Adv
Status: Success: Section attached.
Section # 3
Details: Web 1 (U) <Click to see properties>
Apply Cancel

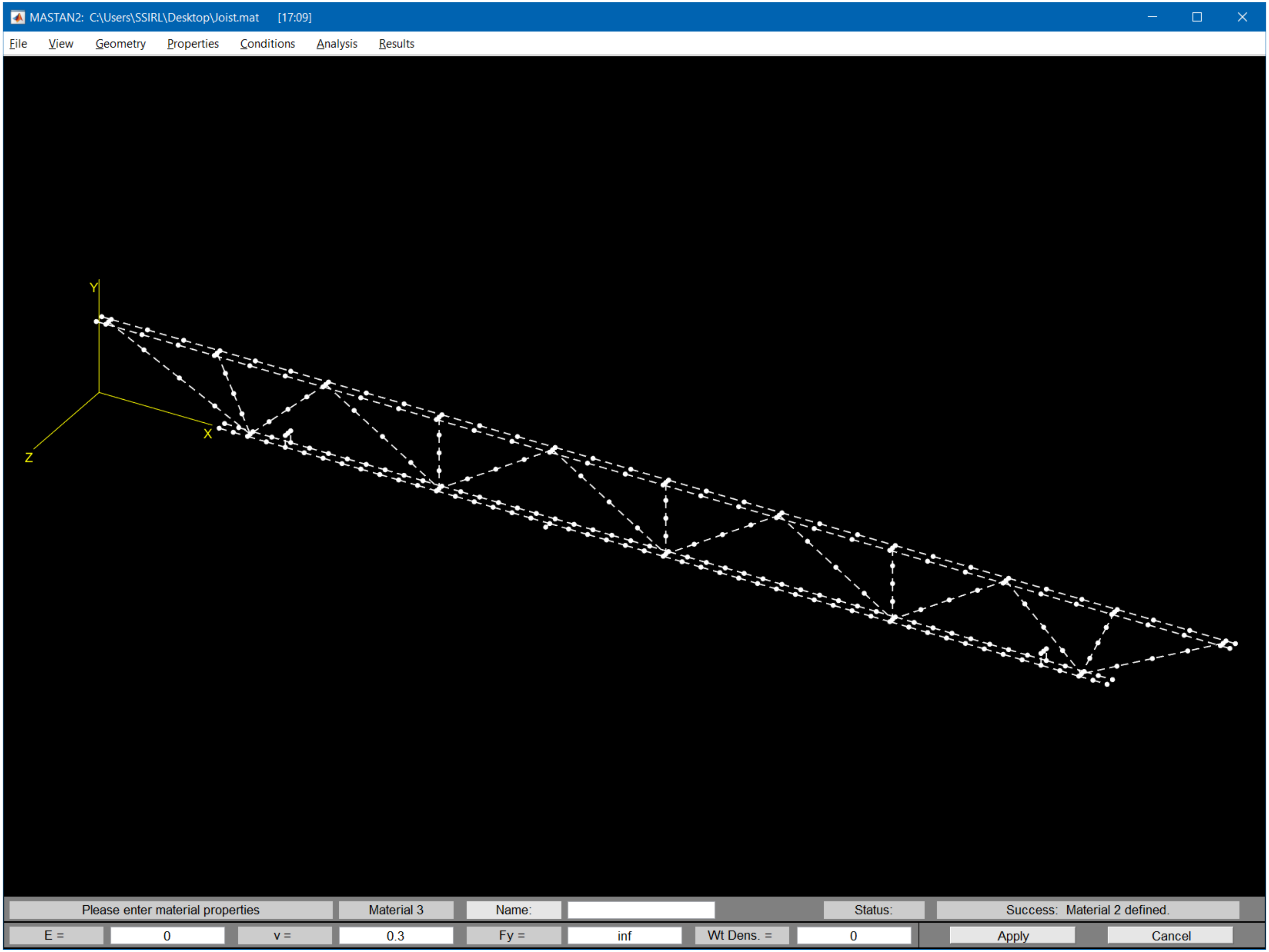
Material Properties

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 **29000000** (not 29,000,000). Similarly, click in the edit box just to the right of **Fy=** and change the **inf** to **50000**. Next, click in the edit box to the right of **Name:** and type **Steel**. Click on the **Apply** button (Material #1 is now defined with the properties of steel).

3) At the bottom menu bar, click in the edit box just to the right of **E=** and change the **0** to **2900000000** (not 2,900,000,000). Next, click in the edit box to the right of **Name:** and type **Rigid**. Click on the **Apply** button. (Material #2 is now defined 100x stiffer and cannot yield.)

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:09]
File View Geometry Properties Conditions Analysis Results
Y
X
Z
Please enter material properties
Material 3
Name:
Status:
Success: Material 2 defined.
E =
0
v =
0.3
Fy =
inf
Wt Dens. =
0
Apply
Cancel

Material Properties - Assigning

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

2) 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. (Note that elements with assigned section and material properties turn solid.)

3) Change the **Material #** by clicking on the current material number just to the right to open a pop-up menu with all section numbers. Click on **2** to select the Rigid material.

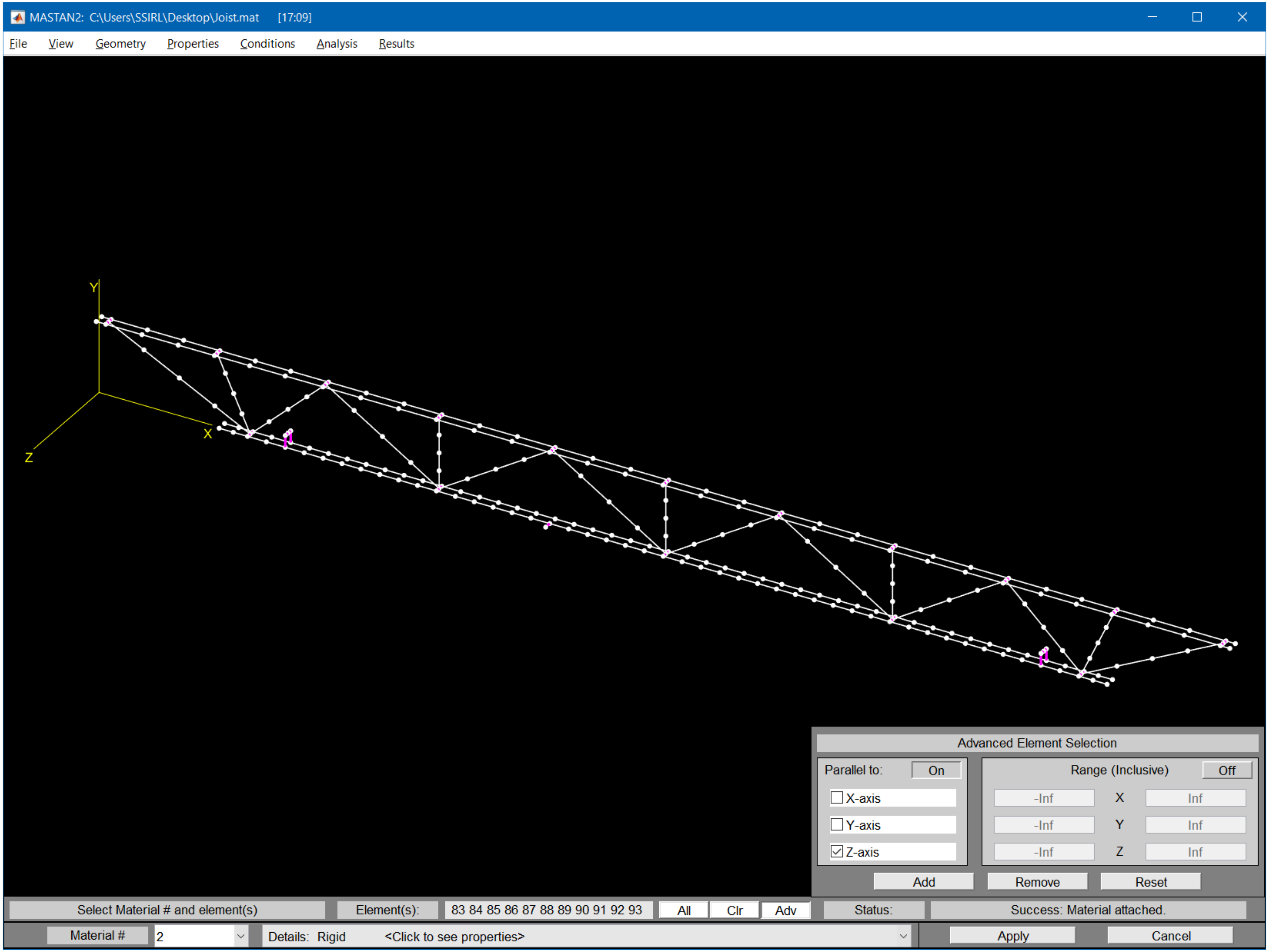
4) Select the **Clr** button located to the right of **Elements:** to clear the list of elements.

5) Click the **Adv** button to open pop-up menu. **Z-axis** check box should be selected.

6) Click **Add** to select all the rigid connectors. Additionally click on the 4 vertical braces. Clicking **Adv** will close the pop-up menu making it easier to click all members.

7) Click on the **Apply** button to assign Material 2.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:09]
File View Geometry Properties Conditions Analysis Results
Y
X
Z
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
Select Material # and element(s)
Element(s): 83 84 85 86 87 88 89 90 91 92 93
All Clr Adv
Status: Success: Material attached.
Material # 2
Details: Rigid <Click to see properties>
Apply Cancel

Support Conditions - Bracing

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

2) At the bottom menu bar, define the lateral support by clicking in the **check box** just to the left of **Z-disp.**

3) Click the **Adv** menu. Change the edit box to the left of **Y** from **-Inf** to **15**. Change the edit box to the left of **Z** from **-Inf** to **-0.1** and the edit box to the right of **Z** from **Inf** to **0.1**. Click the **Add** button.

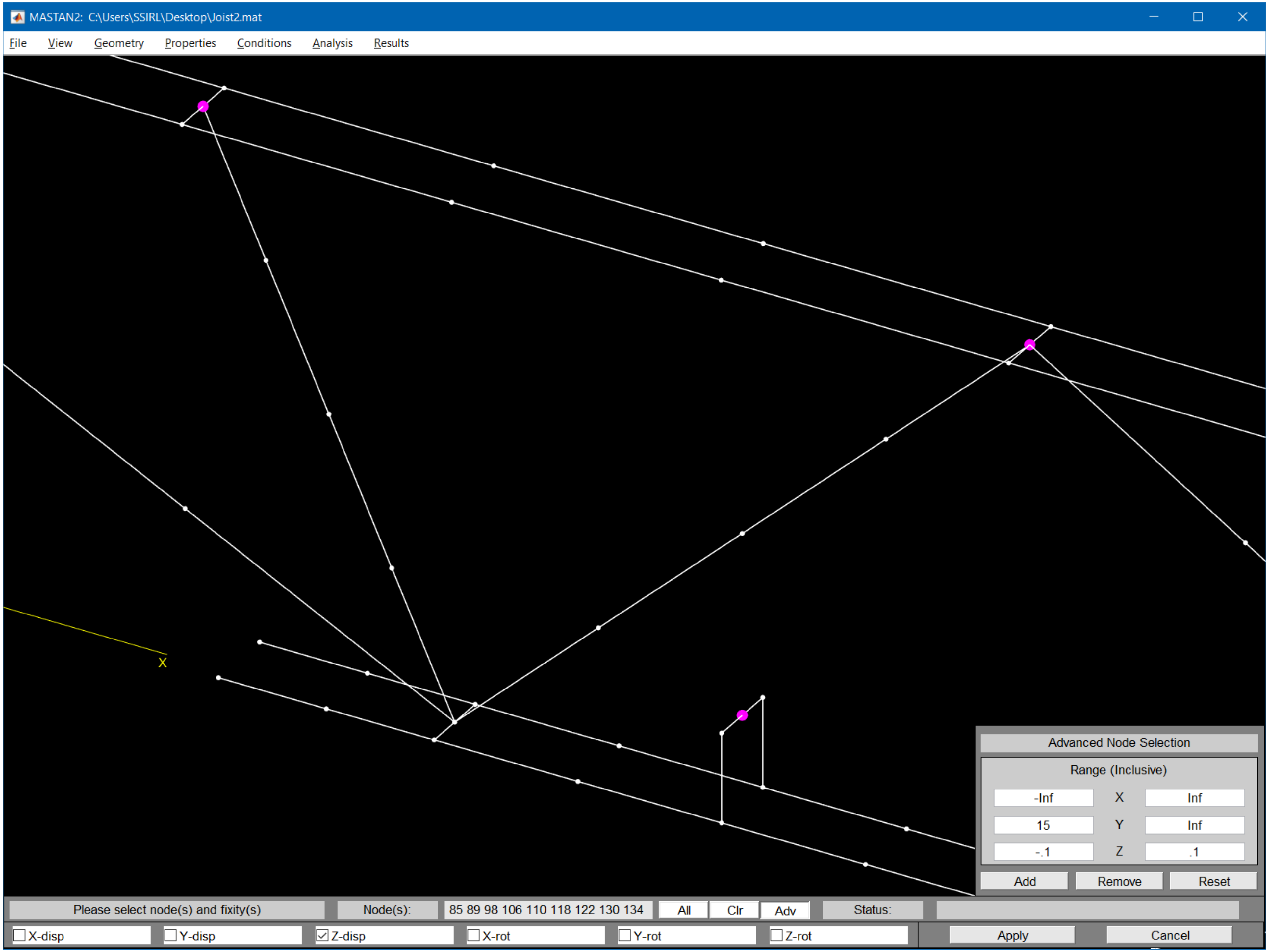
4) From the **View** menu select **Zoom Box**. Click to draw a small box around the brace connection on the bottom chord at one end. Click on the middle node to add to the list of nodes.

5) From the **View** menu select **Fit**. From the **View** menu select **Zoom Box**. Click to draw a small box around the brace connection at the other end. Click on the middle node to add to the list of nodes.

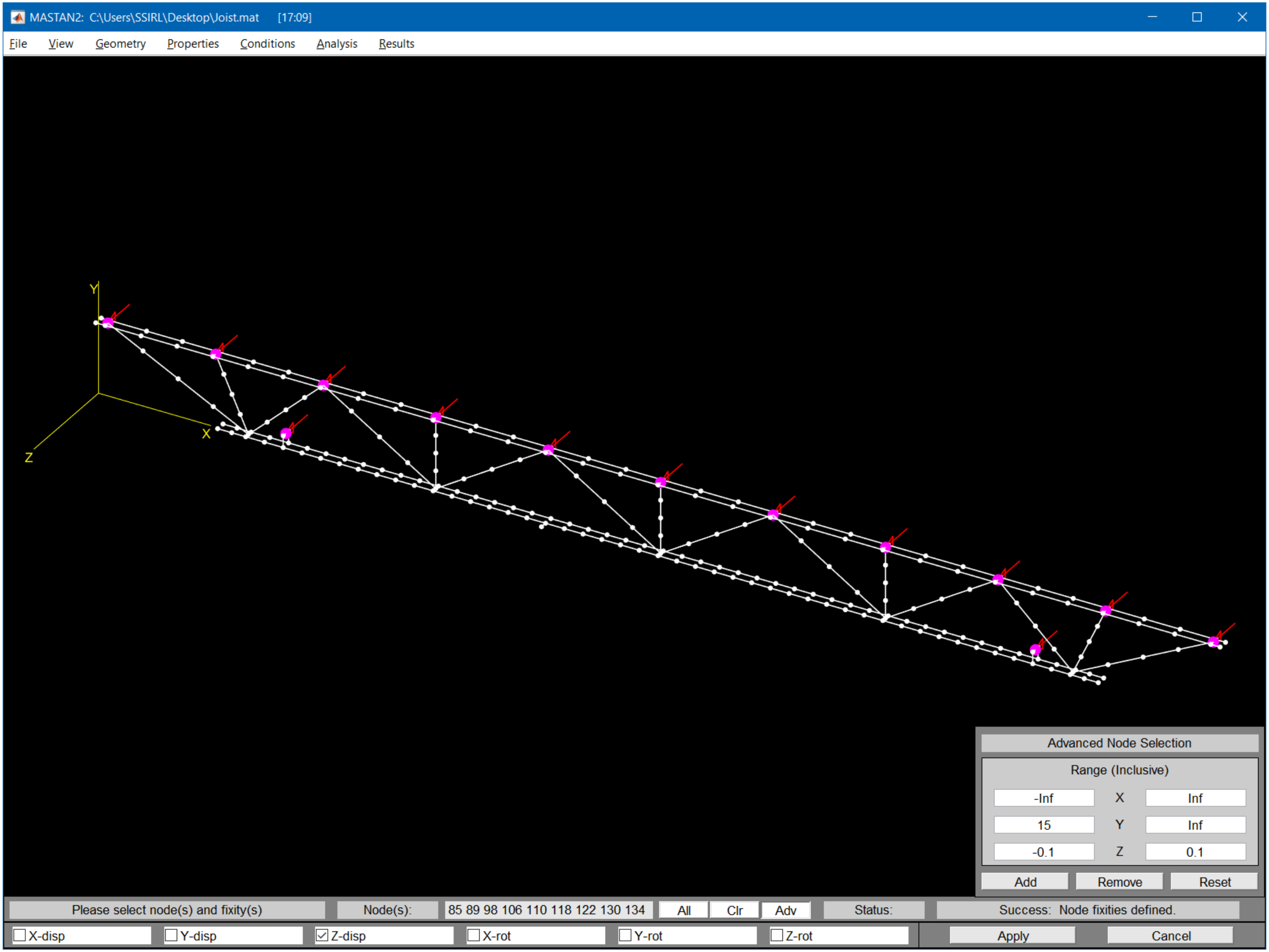
6) Click on the **Apply** button.

7) From the **View** menu select **Fit**.

MASTAN2

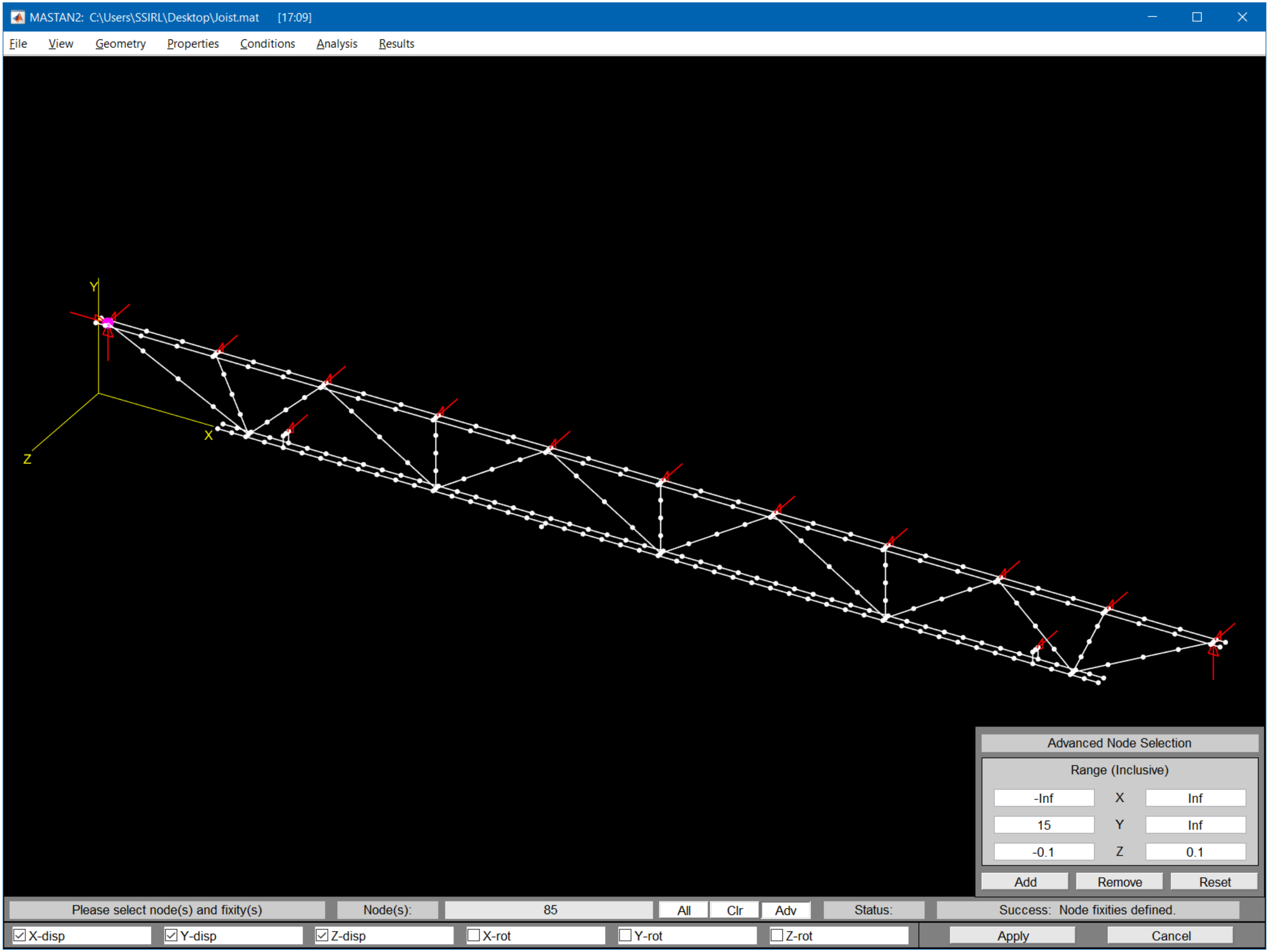

MASTAN2: C:\Users\SSIRL\Desktop\Joist2.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
X
Advanced Node Selection
Range (Inclusive)
-Inf
X
Inf
15
Y
Inf
-.1
Z
.1
Add
Remove
Reset
Please select node(s) and fixity(s)
Node(s):
85 89 98 106 110 118 122 130 134
All
Clr
Adv
Status:
X-disp
Y-disp
Z-disp
X-rot
Y-rot
Z-rot
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:09]
File View Geometry Properties Conditions Analysis Results
Y
X
Z
Advanced Node Selection
Range (Inclusive)
-Inf X Inf
15 Y Inf
-0.1 Z 0.1
Add Remove Reset
Please select node(s) and fixity(s)
Node(s): 85 89 98 106 110 118 122 130 134
All Clr Adv
Status:
Success: Node fixities defined.
X-disp Y-disp Z-disp X-rot Y-rot Z-rot
Apply Cancel

Support Conditions – Pin and Roller

1) To update the end nodes to provide pin supports, start by clicking **Clr** to empty the list of nodes.
2) From the **View** menu select **Zoom Box**. Click to draw a small box around the right roller node. It should currently just have a lateral support. Click on the node to add to the list of nodes.
3) Define a roller support by clicking in the **check box** just to the left of **Y-disp**. **Z-disp** should still be selected from before.
4) From the **View** menu select **Fit**.
5) Click on the **Apply** button.
6) From the **View** menu select **Zoom Box**. Click to draw a small box around the left pin node.
7) Click on the **Clr** button to empty the list of nodes. Click on the left pin.
8) Define a pin support by clicking in the **check box** just to the left of **X-disp**. **Y-disp** and **Z-disp** should still be selected from before.
9) From the **View** menu select **Fit**.
10) Click on the **Apply** button.


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:09]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
Advanced Node Selection
Range (Inclusive)
-Inf
X
Inf
15
Y
Inf
-0.1
Z
0.1
Add
Remove
Reset
Please select node(s) and fixity(s)
Node(s):
85
All
Clr
Adv
Status:
Success: Node fixities defined.
X-disp
Y-disp
Z-disp
X-rot
Y-rot
Z-rot
Apply
Cancel

Chord Member Orientation

1) From the **View** menu select **Labels** and submenu option **Element local (x'-y'-z') axes**. The purple line shows the positive x axis. The blue line shows the positive geometric y axis. The red line shows the positive geometric z axis. Based on the orientation of the angle sections as input, the y axis represents the direction from the centroid to the corner of the angle.

2) From the **Geometry** menu select **Re-orient Element(s)**.

3) At the bottom menu bar, click in the edit box to the right of **Beta (Deg)** and change **0.0** to **135**.

4) Click the **Adv** button to open pop-up menu. Click the **Reset** button. To select the +z top chord angle, click the check box next to the **X-axis** option. Click the button to the right of **Range (Inclusive)** to **On**. Change the edit box to the left of **Y** to **8**. Change the edit box to the left of **Z** to **0**.

5) Click **Add** to add all these elements to the element list. Click on the **Apply** button to re-orient the elements.

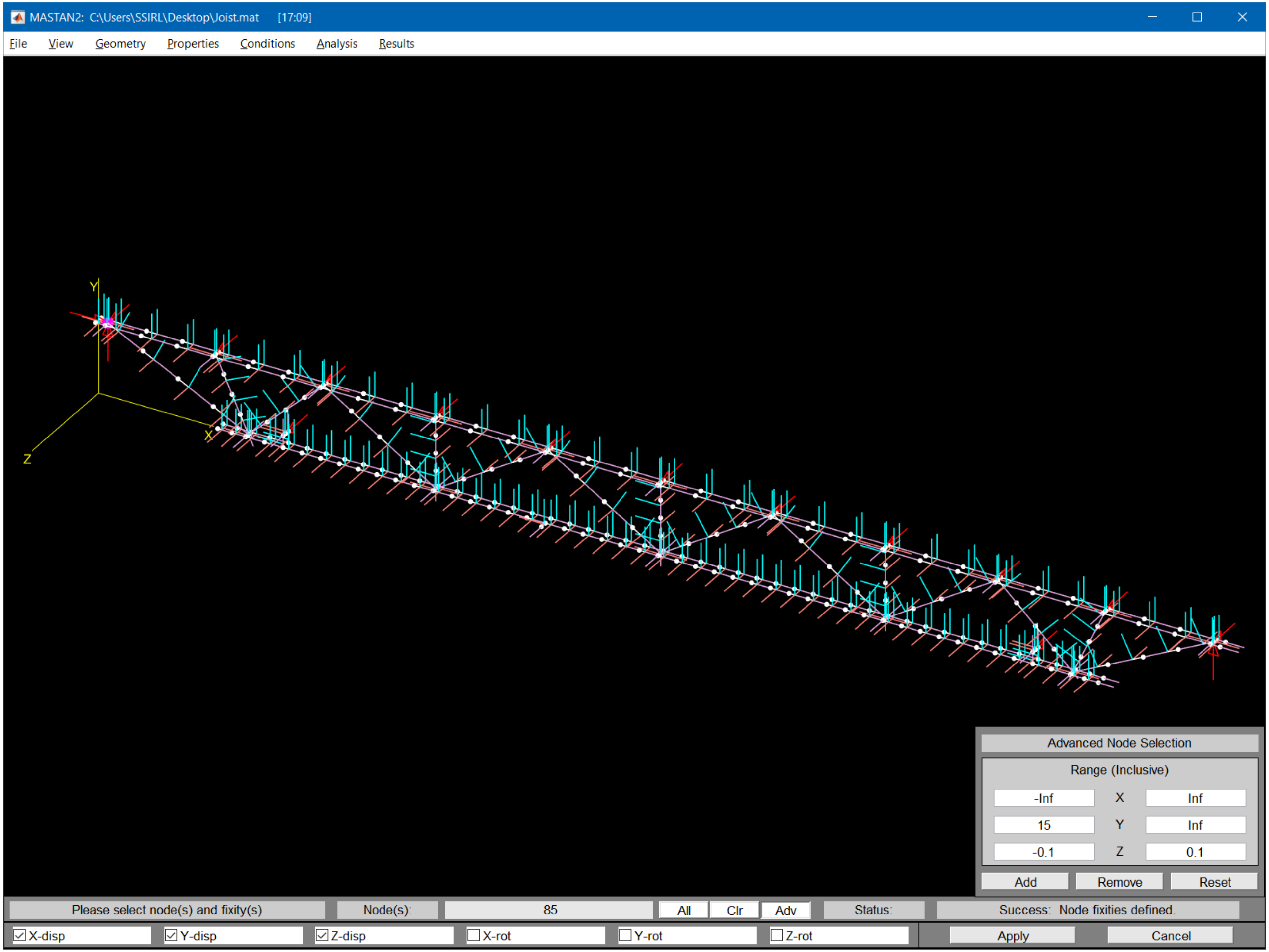
6) Repeat this for the remaining 3 angle members orienting in the correct direction with the values shown in the table:

Member	-Z Top	+Z Bottom	-Z Bottom
Beta	-135	45	-45
Y-Range	8 to inf	0 to 8	0 to 8
Z-Range	-2 to 0	0 to 2	-2 to 0

Iso. View

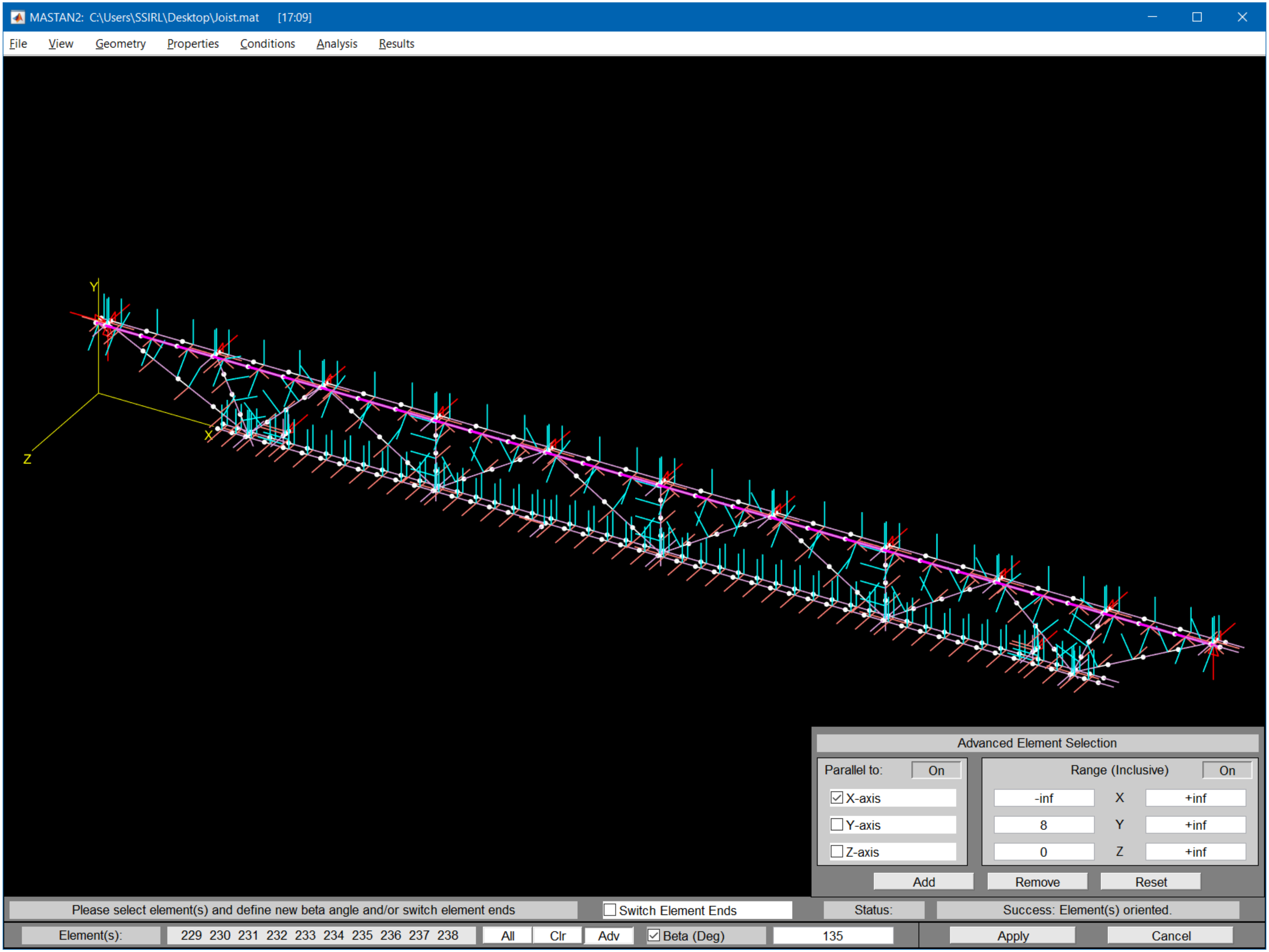
Side View

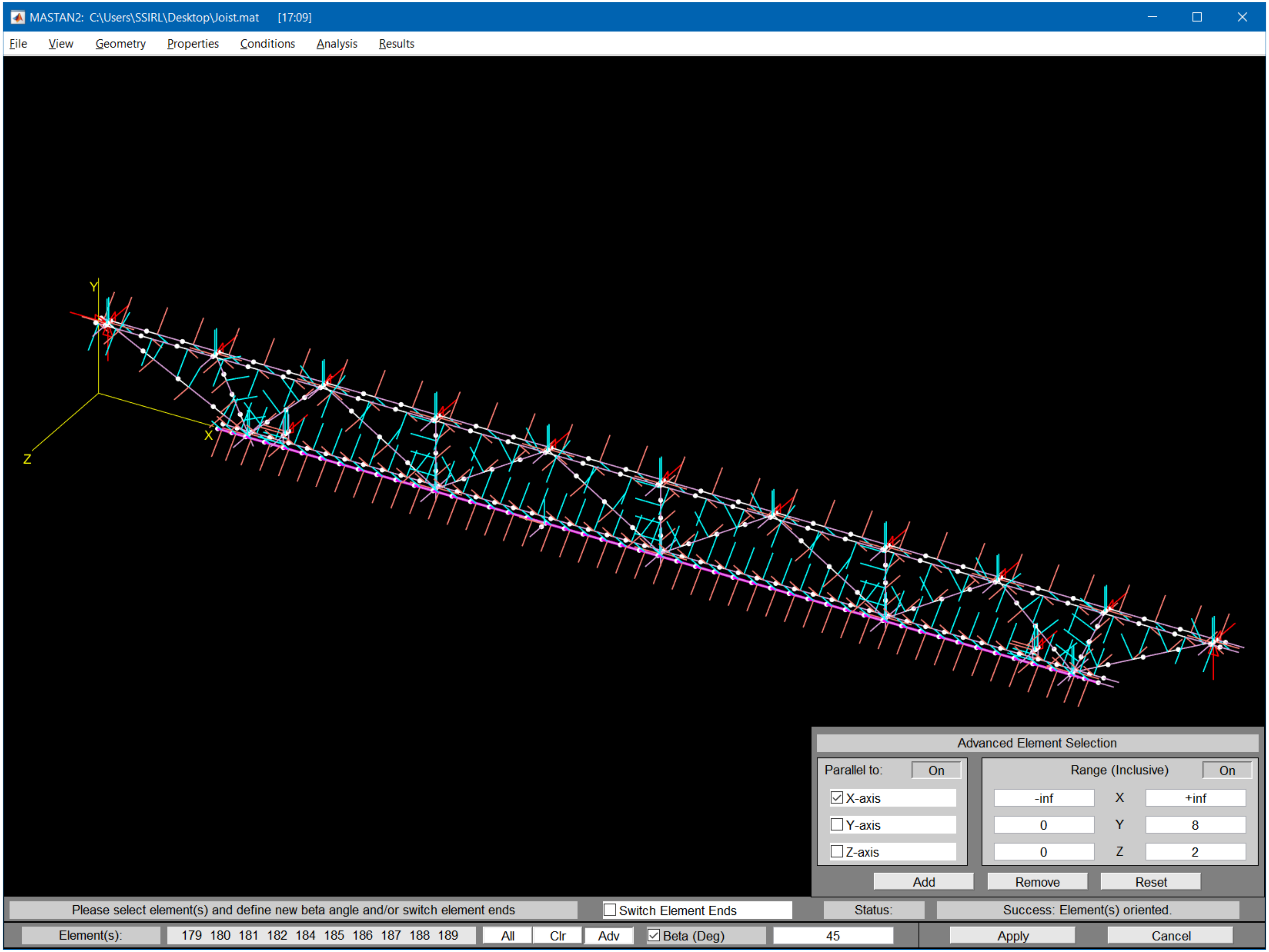
MASTAN2

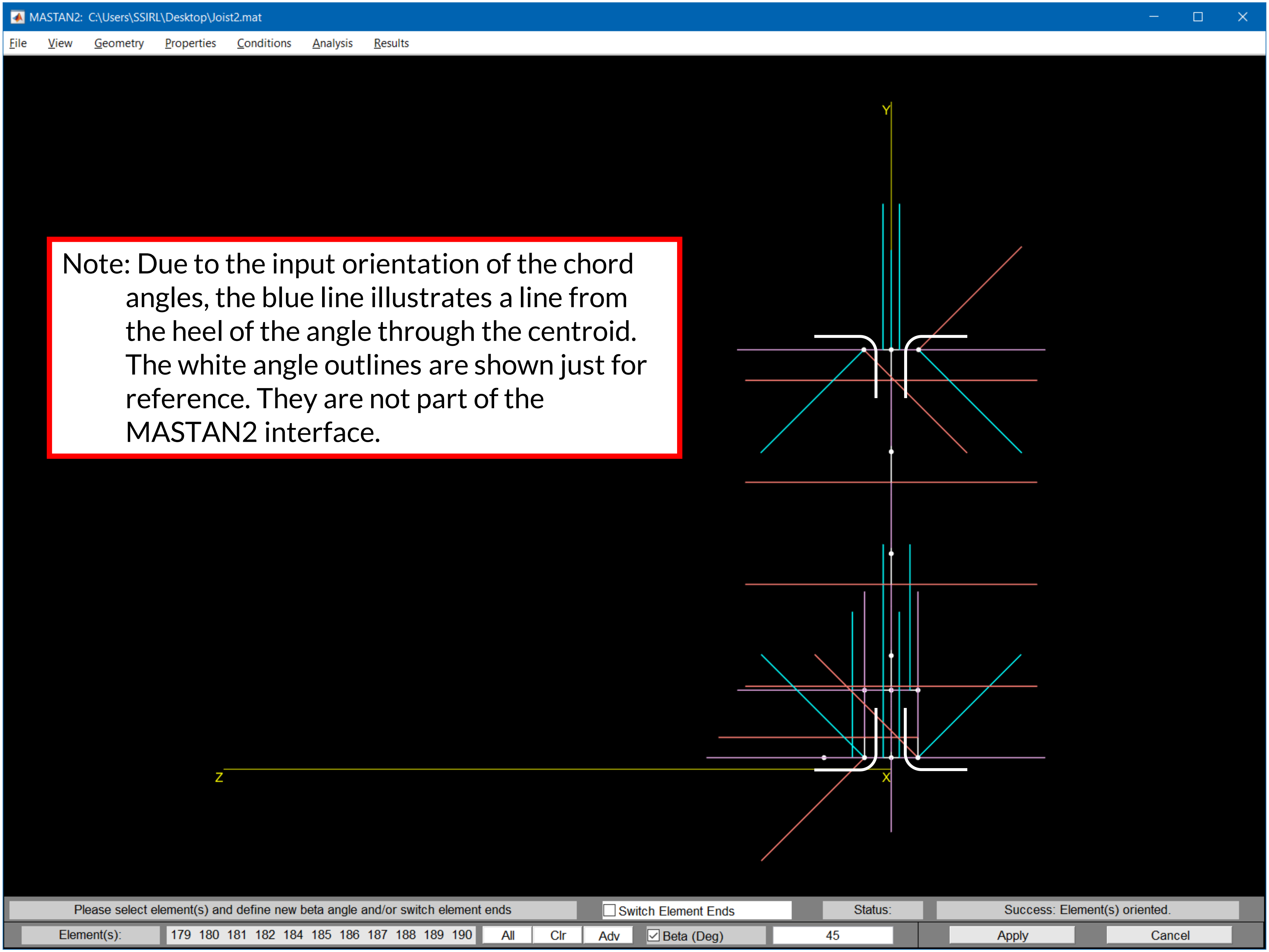


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:09]
File View Geometry Properties Conditions Analysis Results
Y
X
Z
Advanced Node Selection
Range (Inclusive)
-Inf X Inf
15 Y Inf
-0.1 Z 0.1
Add Remove Reset
Please select node(s) and fixity(s)
Node(s): 85
All Clr Adv
Status: Success: Node fixities defined.
X-disp Y-disp Z-disp X-rot Y-rot Z-rot
Apply Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:09]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
Advanced Element Selection
Parallel to:
On
X-axis
Y-axis
Z-axis
Range (Inclusive)
On
-inf
X
+inf
8
Y
+inf
0
Z
+inf
Add
Remove
Reset
Please select element(s) and define new beta angle and/or switch element ends
Switch Element Ends
Status:
Success: Element(s) oriented.
Element(s):
229 230 231 232 233 234 235 236 237 238
All
Clr
Adv
Beta (Deg)
135
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:09]
File View Geometry Properties Conditions Analysis Results
Y
X
Z
Advanced Element Selection
Parallel to: On
X-axis
Y-axis
Z-axis
Range (Inclusive) On
-inf X +inf
0 Y 8
0 Z 2
Add Remove Reset
Please select element(s) and define new beta angle and/or switch element ends
Switch Element Ends
Status:
Success: Element(s) oriented.
Element(s):
179 180 181 182 184 185 186 187 188 189
All Clr Adv
Beta (Deg)
45
Apply Cancel



Note: Due to the input orientation of the chord angles, the blue line illustrates a line from the heel of the angle through the centroid. The white angle outlines are shown just for reference. They are not part of the MASTAN2 interface.

Web Member Orientation

Based on the orientation of the web sections as input, the y axis points in the direction of the opening of the channel.

1) At the bottom menu bar, click in the edit box to the right of **Beta (Deg)** and update it to **180**.

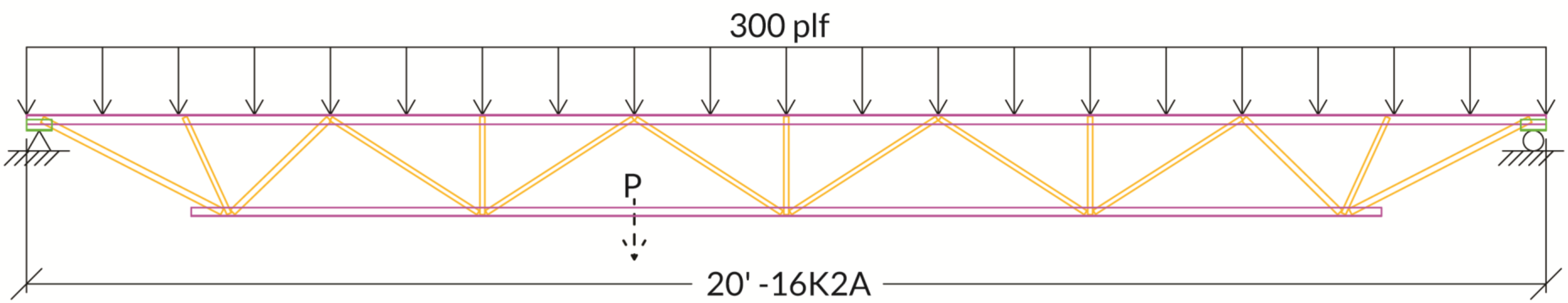
2) On the **Advanced Element Selection** pop-up, click the **Reset** button. Click the check box next to the **X-axis** and **Z-axis** option. Create the list of webs to be flipped by clicking on the **All** button to the right of **Elements:**. Then click on **Remove**.

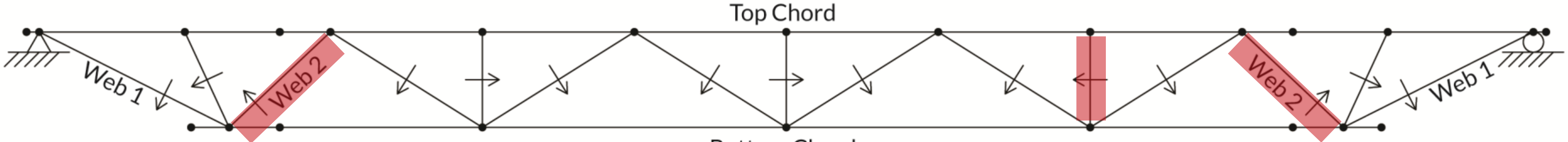
3) Manually click the 4 vertical braces, the members assigned Web 2, and the vertical web to the right of center.

   Note: If you are having trouble seeing the webs that were unselected, look at the image associated with Step 5 now. The same elements are still selected without all the local axes information on the screen. Also, available again is the diagram with web orientations.

4) Click on the **Apply** button to re-orient the elements.

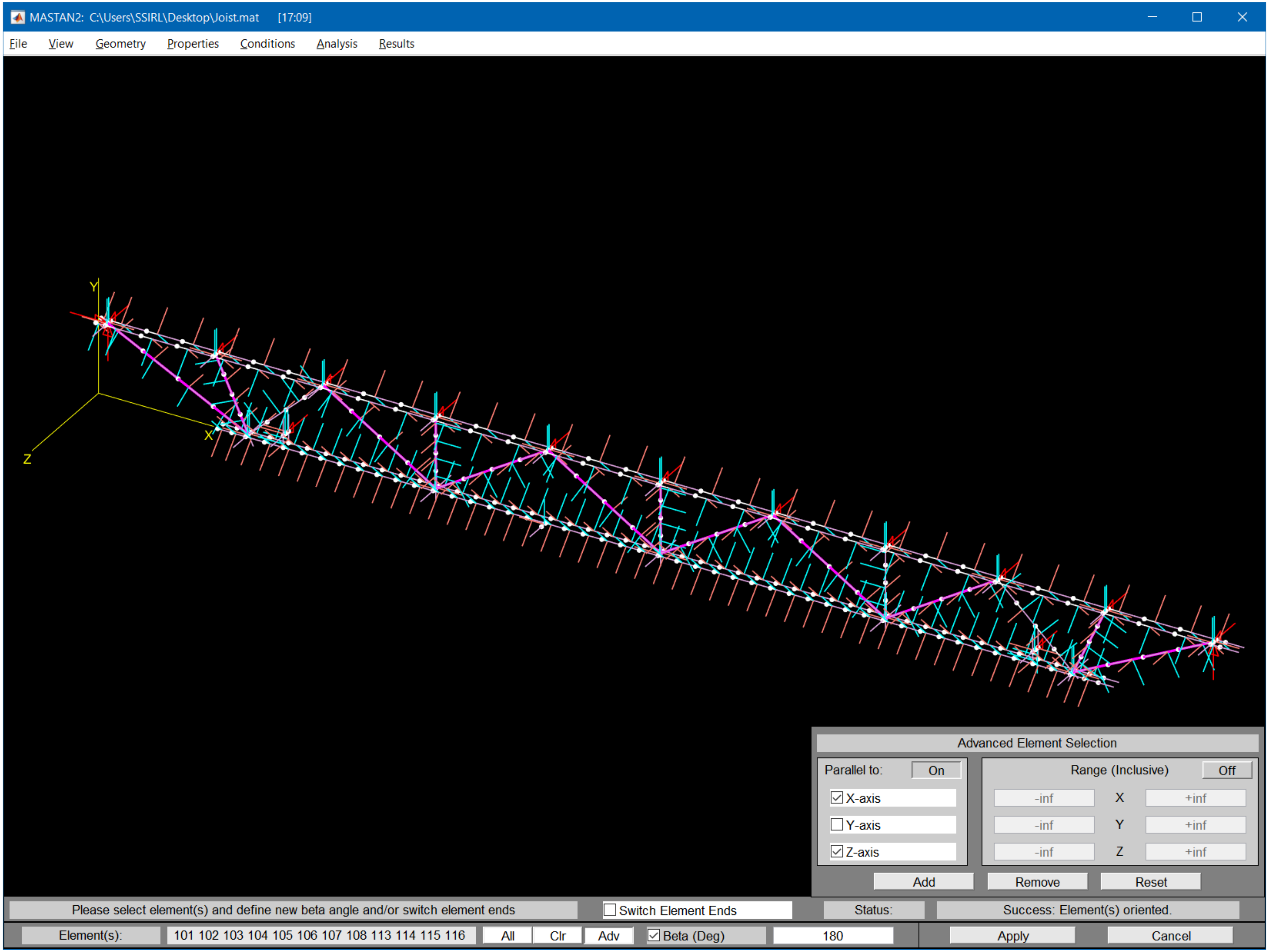
5) From the **View** menu select **Labels** and submenu option **Element local (x'-y'-z') axes**.



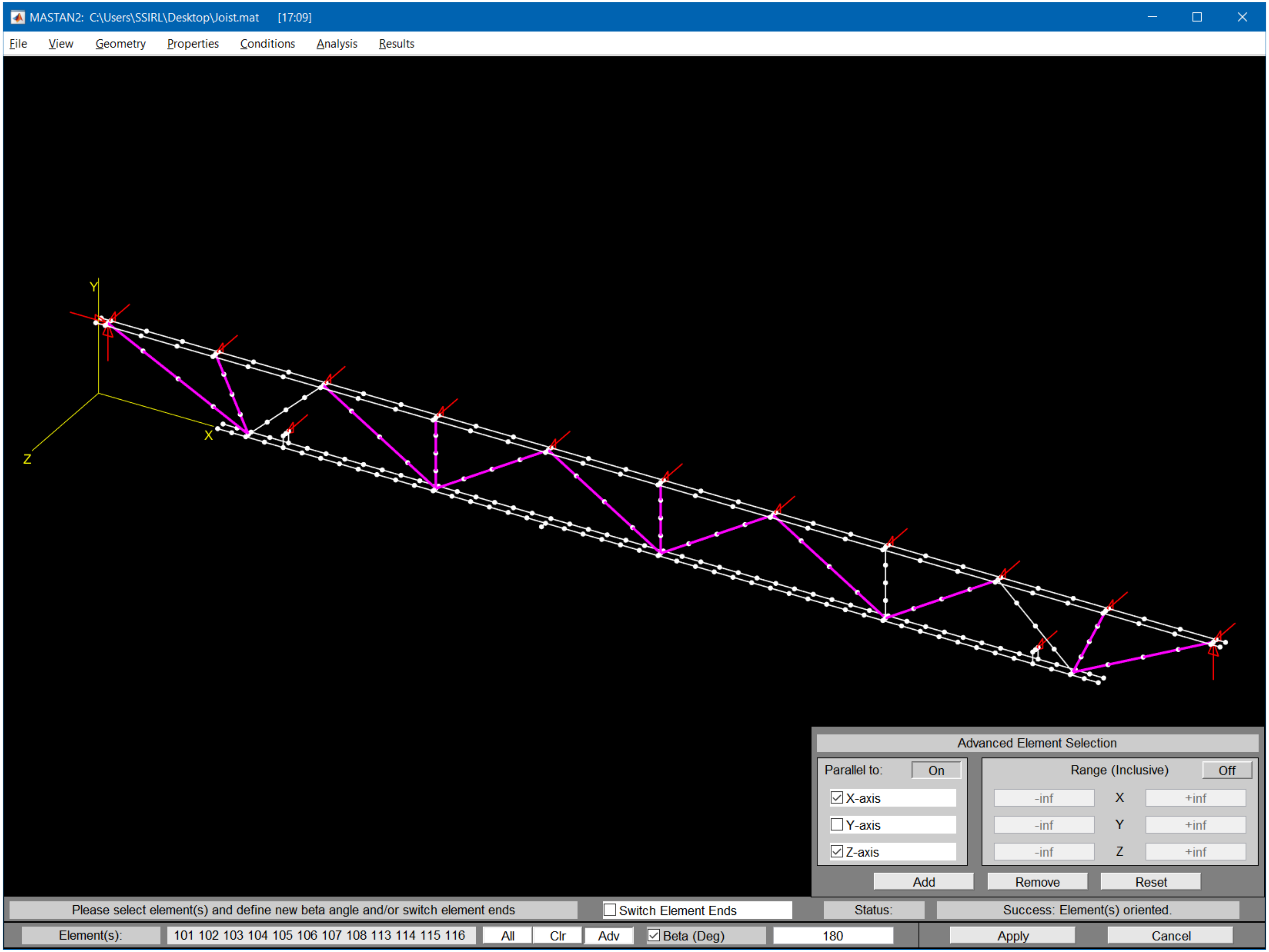


Arrows indicate the open side of the web channels. Web members not otherwise labeled are Web 3

Web members highlighted in red do not need to be flipped as the sections were defined. These are the members that are to be unselected.


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:09]
File View Geometry Properties Conditions Analysis Results
Y
X
Z
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
Please select element(s) and define new beta angle and/or switch element ends
Switch Element Ends
Status:
Success: Element(s) oriented.
Element(s):
101 102 103 104 105 106 107 108 113 114 115 116
All Clr Adv
Beta (Deg)
180
Apply Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:09]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
Advanced Element Selection
Parallel to:
On
X-axis
Y-axis
Z-axis
Range (Inclusive)
Off
-inf
+inf
X
Y
Z
Add
Remove
Reset
Please select element(s) and define new beta angle and/or switch element ends
Switch Element Ends
Status:
Success: Element(s) oriented.
Element(s):
101 102 103 104 105 106 107 108 113 114 115 116
All
Clr
Adv
Beta (Deg)
180
Apply
Cancel

Adding Warping Effects

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

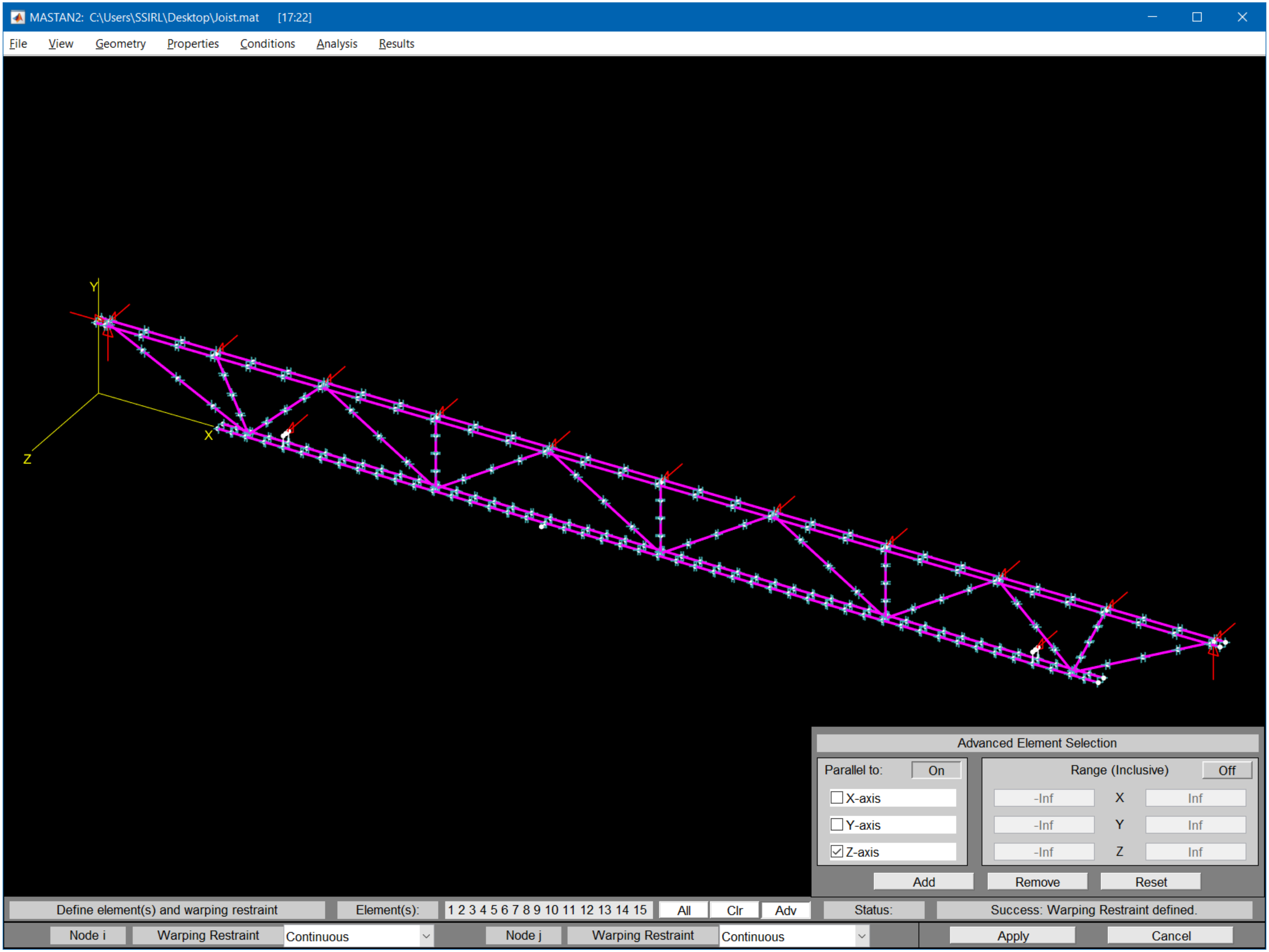
2) At the bottom menu bar, click on the menu to the right of **Warping Restraint for Node i** and set the value to **Continuous**. Repeat this for the **Warping Restraint for Node j**.

3) Click the **Adv** button to open pop-up menu. Unmark the check box next to the **X-axis** option. The check box next to the **Z-axis** option should still be selected.

4) Create the list of elements to be assigned continuous warping by clicking on the **All** button to the right of **Elements:**. Then click on **Remove**. Click on the 4 vertical braces to remove them.

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

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:22]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
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 warping restraint
Element(s):
1 2 3 4 5 6 7 8 9 10 11 12 13 14 15
All
Clr
Adv
Status:
Success: Warping Restraint defined.
Node i
Warping Restraint
Continuous
Node j
Warping Restraint
Continuous
Apply
Cancel

Adding Warping Effects – Starting Node

1) Click **Clr** to empty the list of elements.

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

3) Click on the left most element of each chord.

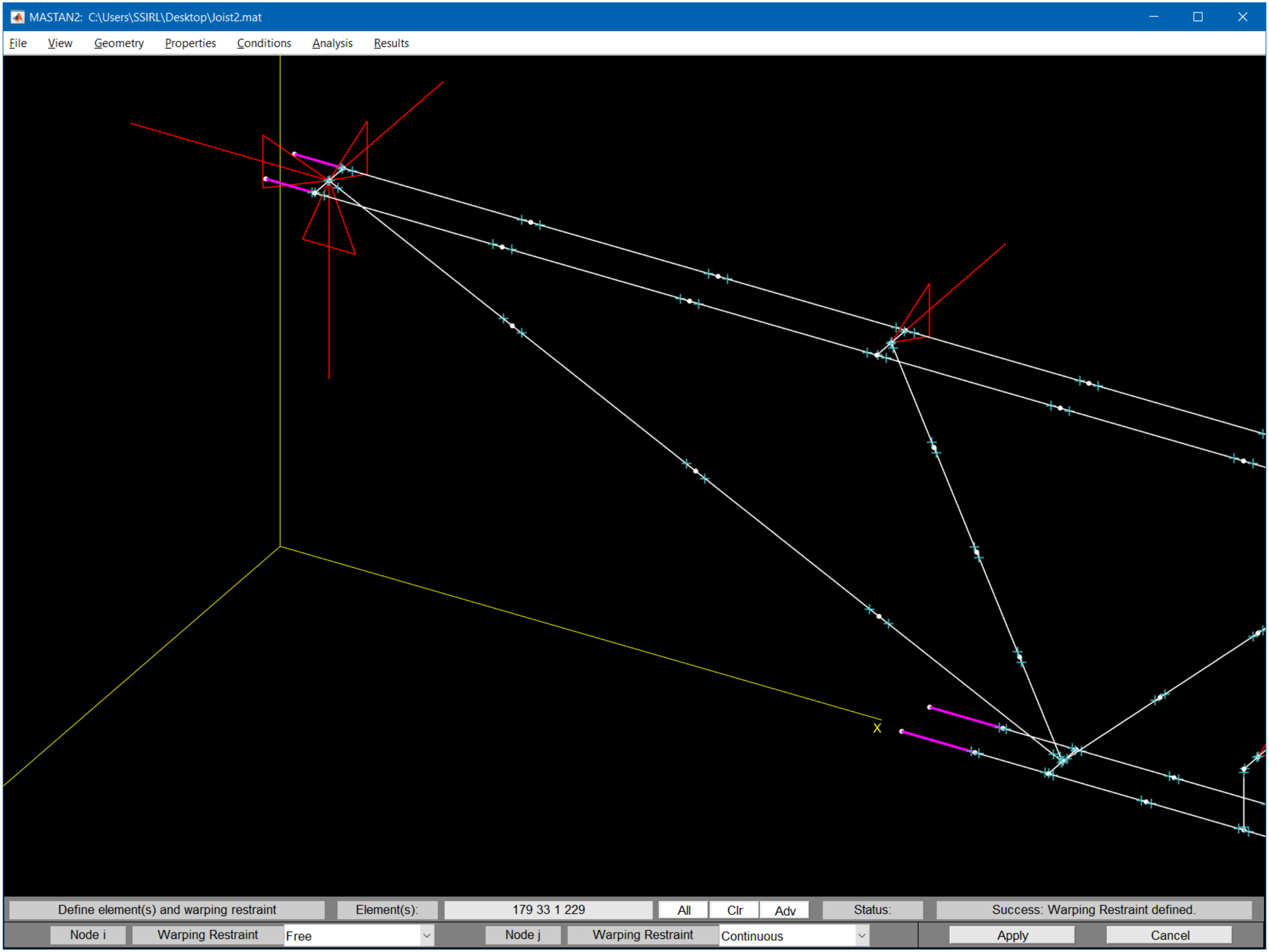
4) Click on the **Apply** button.

5) Click **Clr** to empty the list of elements.

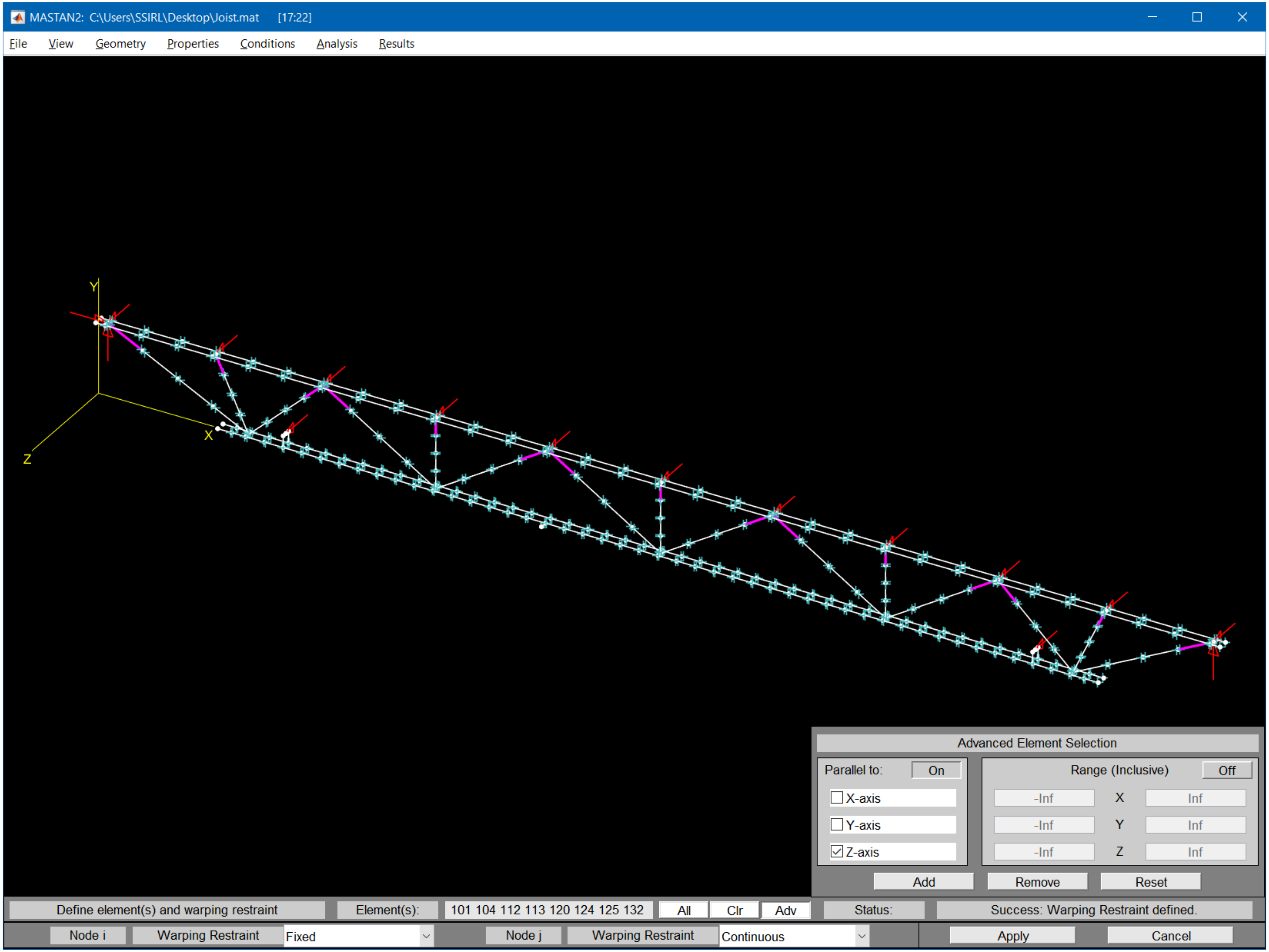
6) Click on the menu to the right of **Warping Restraint for Node i** and set the value to **Fixed**. Node j is set from a previous step.

7) Click on the top end of all 15 web members

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


MASTAN2: C:\Users\SSIRL\Desktop\Joist2.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
X
Define element(s) and warping restraint
Element(s):
179 33 1 229
All
Clr
Adv
Status:
Success: Warping Restraint defined.
Node i
Warping Restraint
Free
Node j
Warping Restraint
Continuous
Apply
Cancel

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:22]
File View Geometry Properties Conditions Analysis Results
Y
X
Z
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 warping restraint
Element(s):
101 104 112 113 120 124 125 132
All Clr Adv
Status:
Success: Warping Restraint defined.
Node i Warping Restraint Fixed
Node j Warping Restraint Continuous
Apply Cancel

Adding Warping Effects – Ending Node

1) Click **Clr** to empty the list of elements.

2) 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**.

3) Click on the right most element of each chord.

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

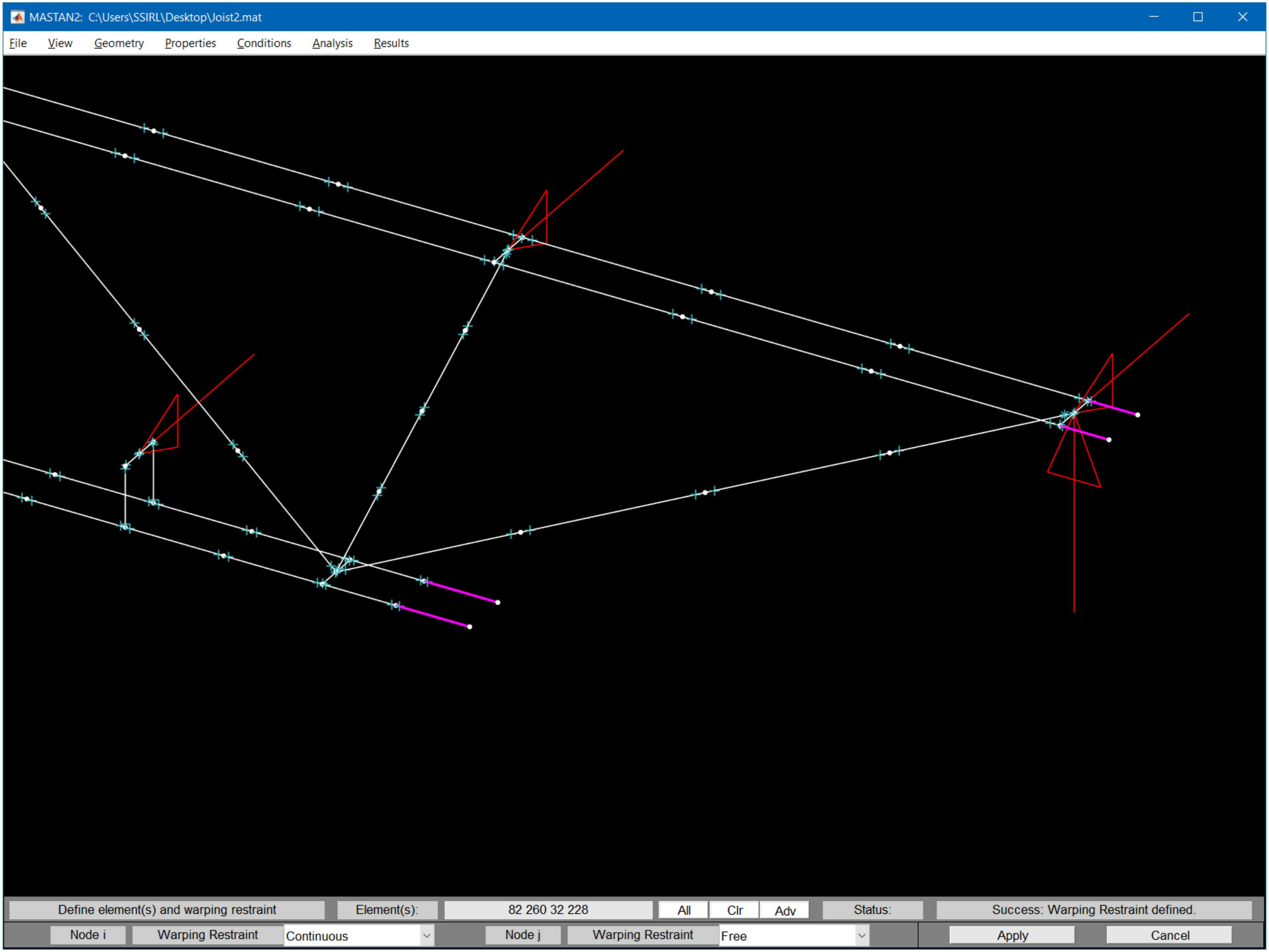
5) Click **Clr** to empty the list of elements.

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

7) Click on the bottom end of all 15 web members.

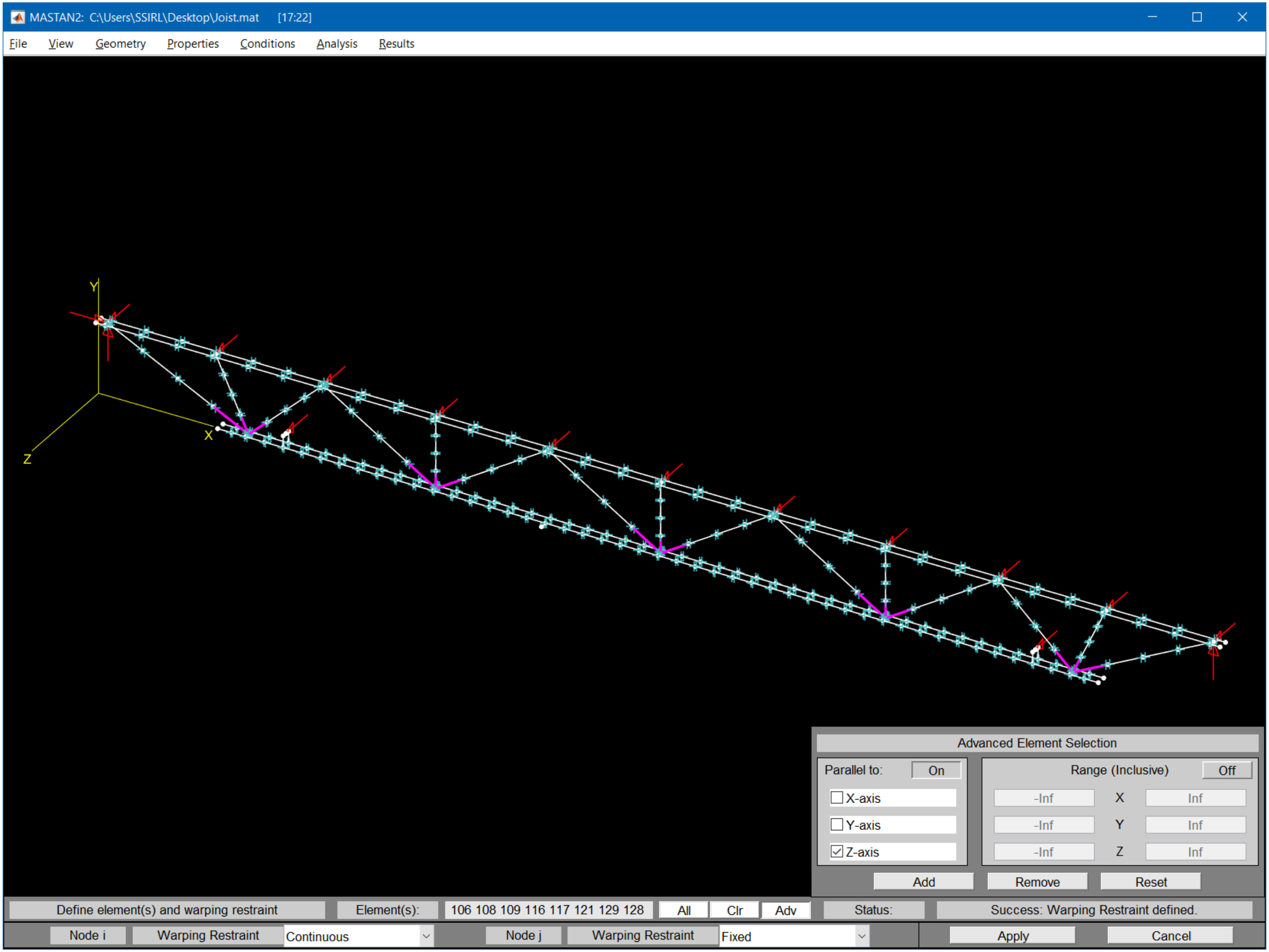
8) Click on the **Apply** button.

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Joist2.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Define element(s) and warping restraint
Element(s):
82 260 32 228
All
Clr
Adv
Status:
Success: Warping Restraint defined.
Node i
Warping Restraint
Continuous
Node j
Warping Restraint
Free
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:22]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
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
Define element(s) and warping restraint
Element(s):
106 108 109 116 117 121 129 128
All
Clr
Adv
Status:
Success: Warping Restraint defined.
Node i
Warping Restraint
Continuous
Node j
Warping Restraint
Fixed
Apply
Cancel

End Moment Release

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

2) At the bottom menu bar, click on the menu to the right of **Type** for **Node i** and set the value to **Semi-Rigid**.

3) In the edit box to the right of **kz** change **inf** to **0.001**.

4) Click on the top end of all 15 web members to create the list of elements.

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

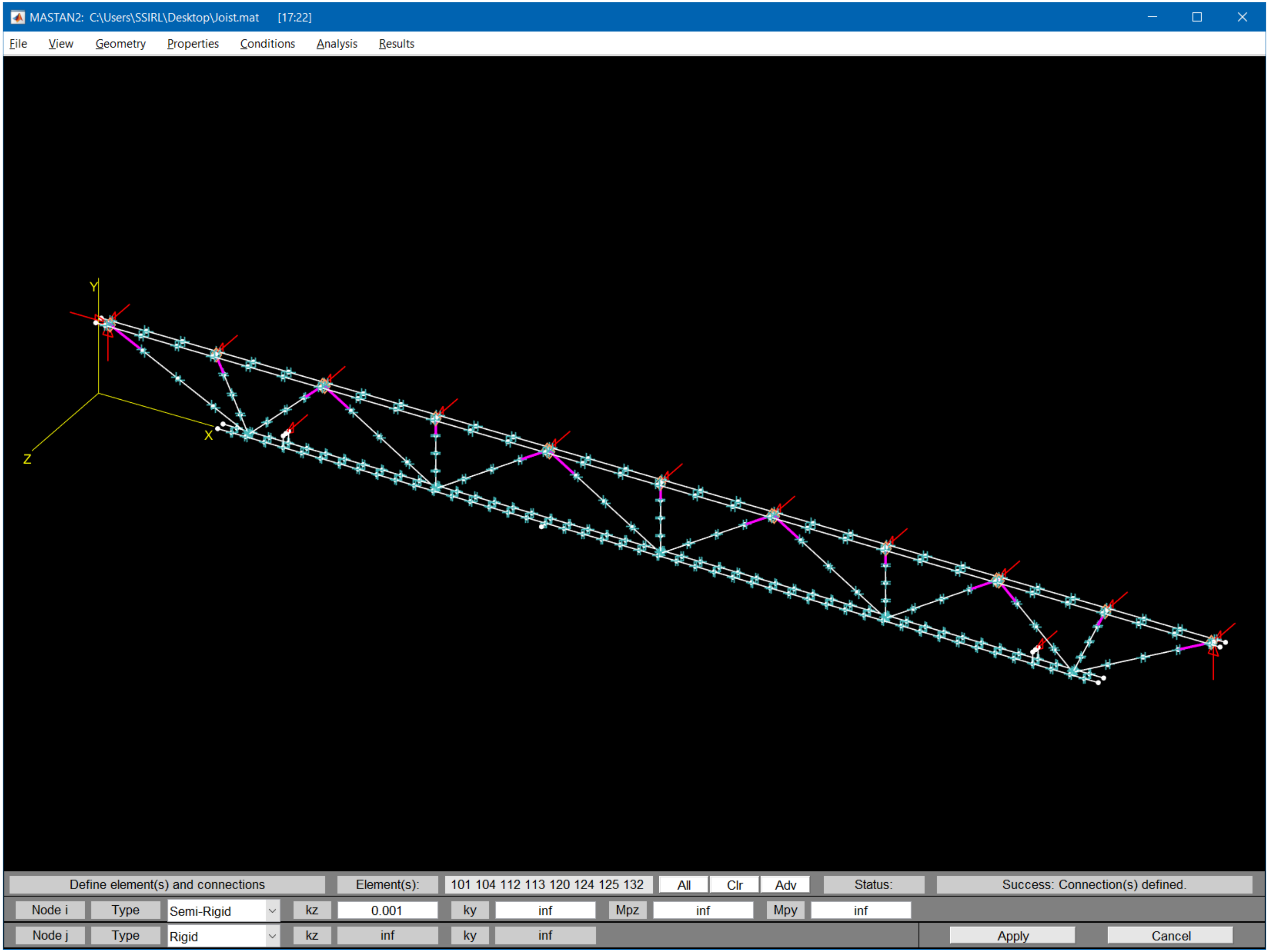
6) Click on the menu to the right of **Type** for **Node i** and set the value to **Rigid**. Click on the menu to the right of **Type** for **Node j** and set the value to **Semi-Rigid**.

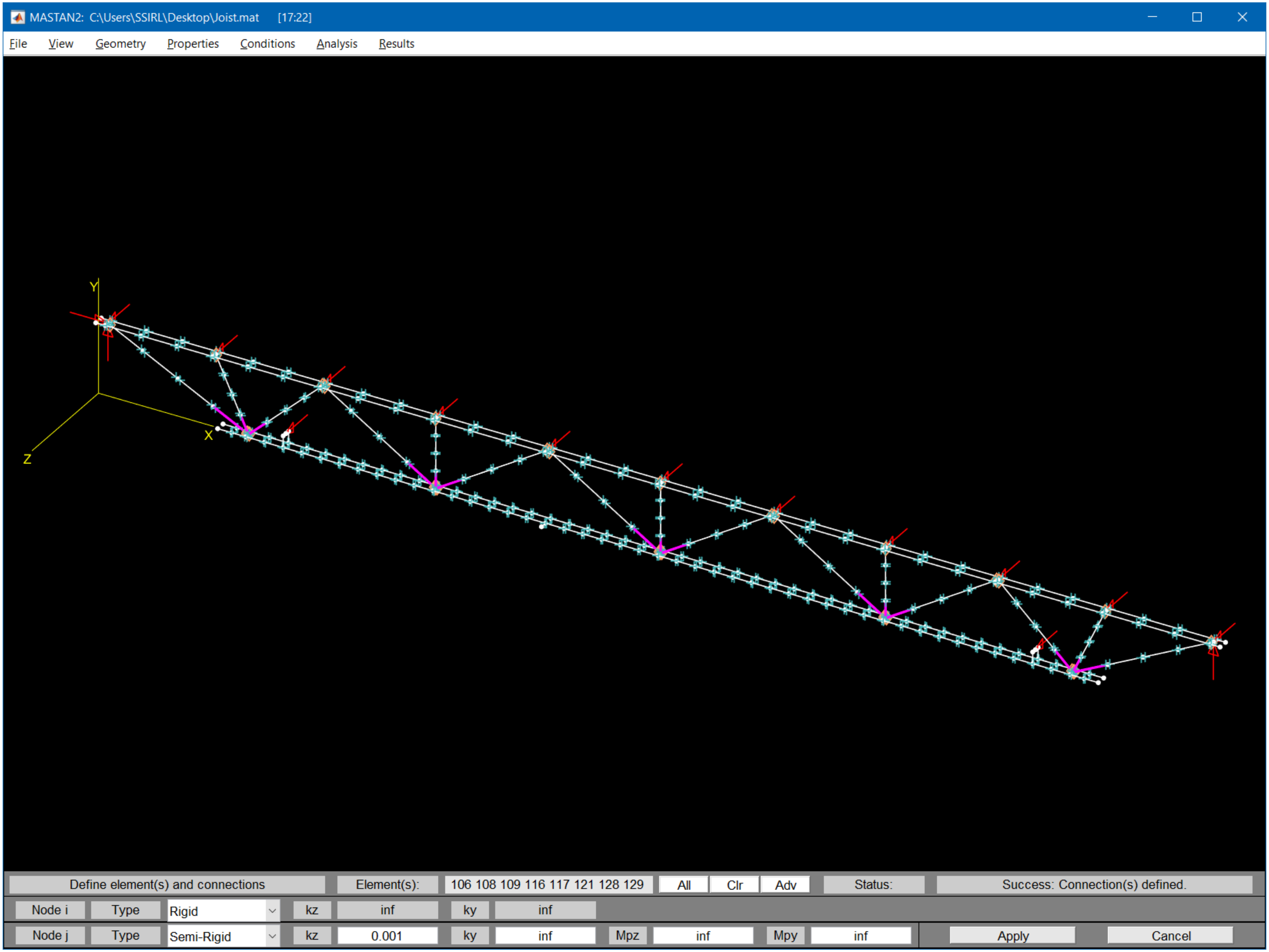
7) In the edit box to the right of **kz** change **inf** to **0.001**.

8) Click **Clr** to empty the list of elements.

9) Click on the bottom end of all 15 web members to create the list of elements.

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


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:22]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
Define element(s) and connections
Element(s):
101 104 112 113 120 124 125 132
All
Clr
Adv
Status:
Success: Connection(s) defined.
Node i
Type
Semi-Rigid
kz
0.001
ky
inf
Mpz
inf
Mpy
inf
Node j
Type
Rigid
kz
inf
ky
inf
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:22]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
Define element(s) and connections
Element(s):
106 108 109 116 117 121 128 129
All
Clr
Adv
Status:
Success: Connection(s) defined.
Node i
Type
Rigid
kz
inf
ky
inf
Node j
Type
Semi-Rigid
kz
0.001
ky
inf
Mpz
inf
Mpy
inf
Apply
Cancel

Section 5: Loading and Analysis

Distributed Loading

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

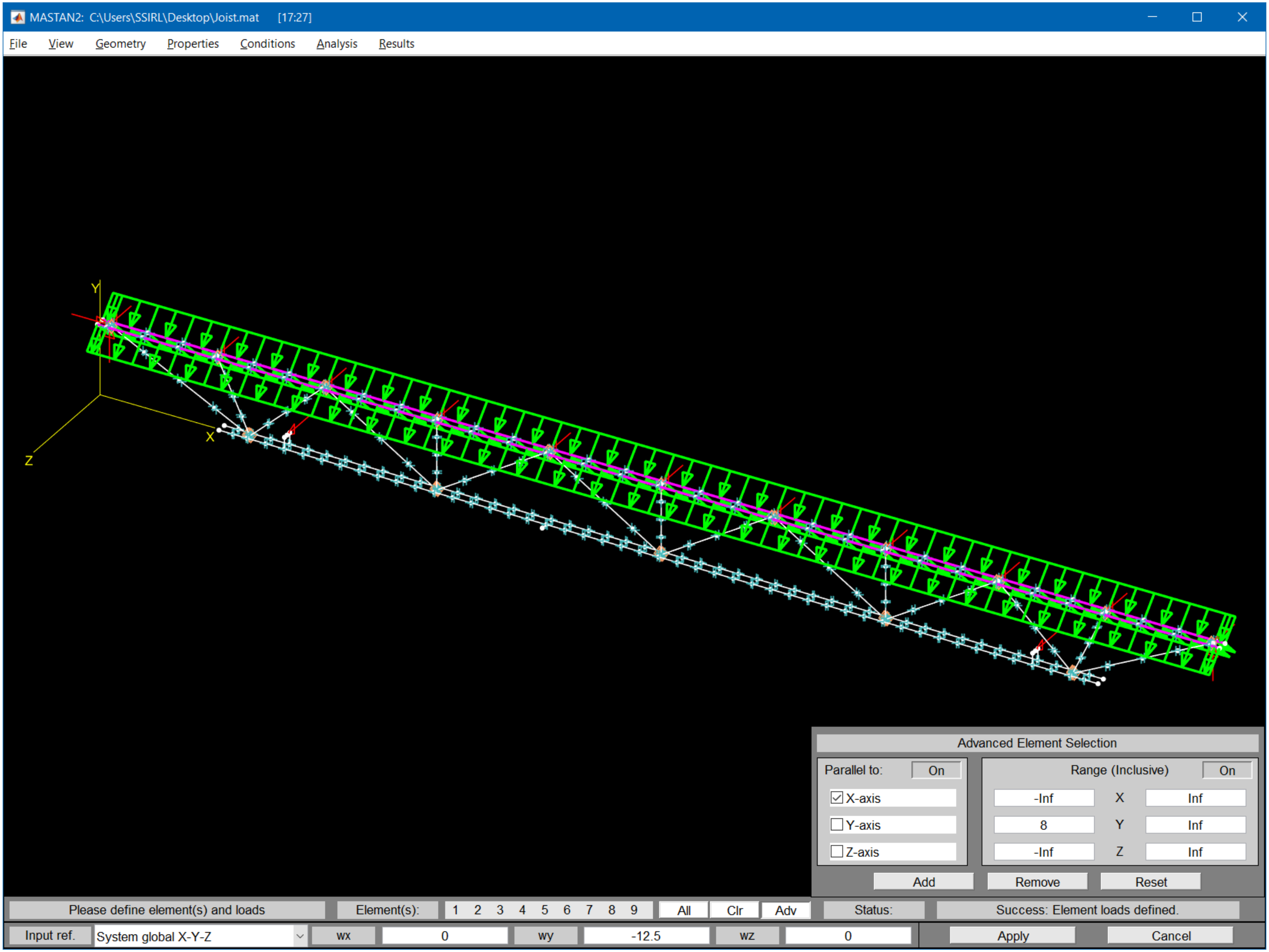
2) At the bottom menu bar, click on **Element(s) local x'-y'-z'** to open the drop down menu. Select **System global X-Y-Z**. In the edit box just to the right of **wy =** change **0** to **-12.5**.

3) Click the **Adv** button to open pop-up menu. Click the check box next to the **Z-axis** option to remove it. Click the check box next to the **X-axis** option. Click the **Off** button to the right of **Range (Inclusive)** to change it to **On**. Change the edit box to the left of **Y** to **8**.

4) Click **Add** to add the top chord elements to the element list.

5) Click on the **Apply** button. The load will be split into the local element directions.

6) From the **View** menu select **Fit**.


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:27]
File View Geometry Properties Conditions Analysis Results
Y
X
Z
Advanced Element Selection
Parallel to: On
X-axis
Y-axis
Z-axis
Range (Inclusive) On
-Inf X Inf
8 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 All Clr Adv
Status: Success: Element loads defined.
Input ref. System global X-Y-Z
wx 0 wy -12.5 wz 0
Apply Cancel

Uniform Loading 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.

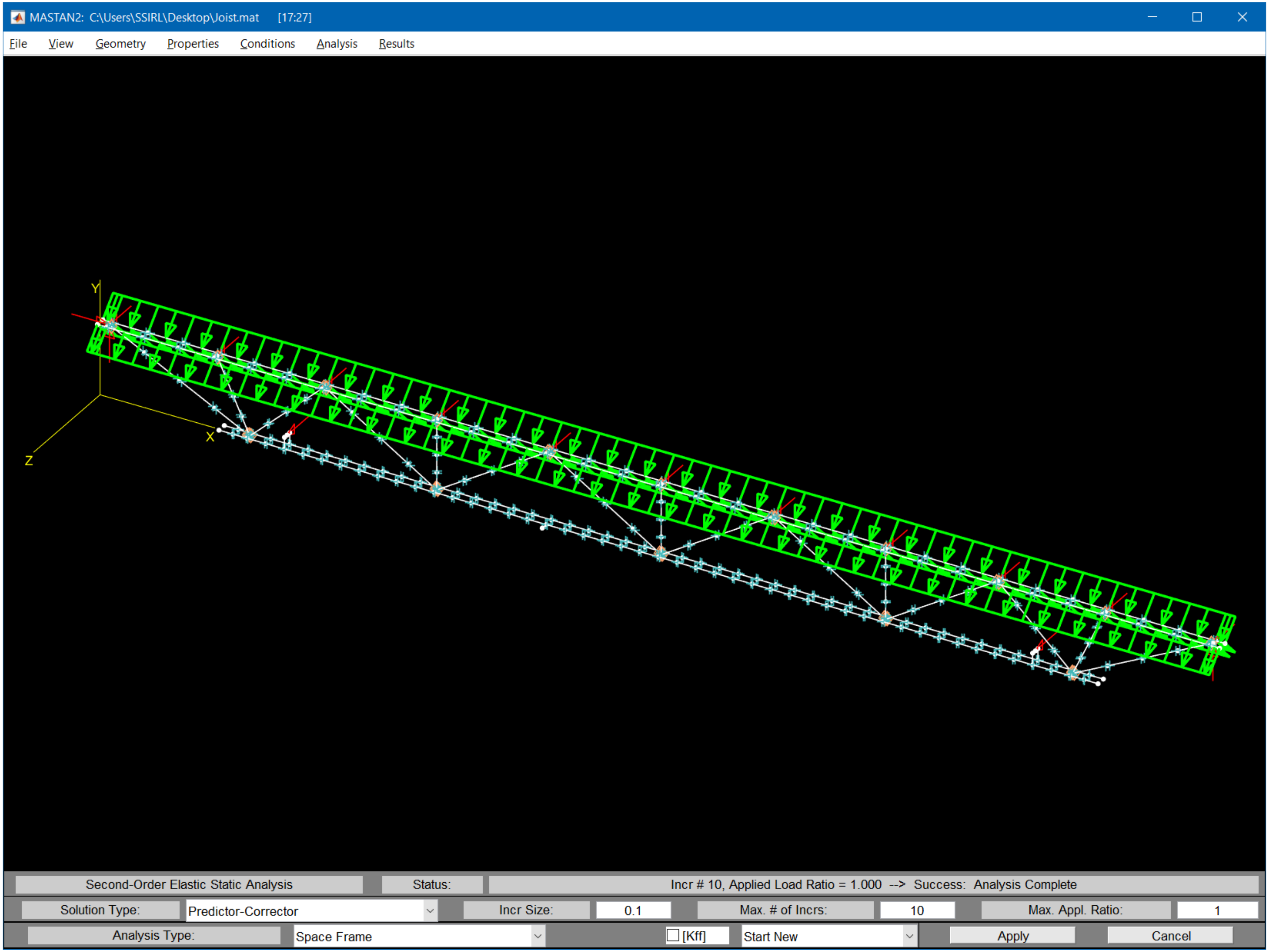
4) From the **Results** menu select **Diagrams** and submenu option **Deflected Shape**.

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

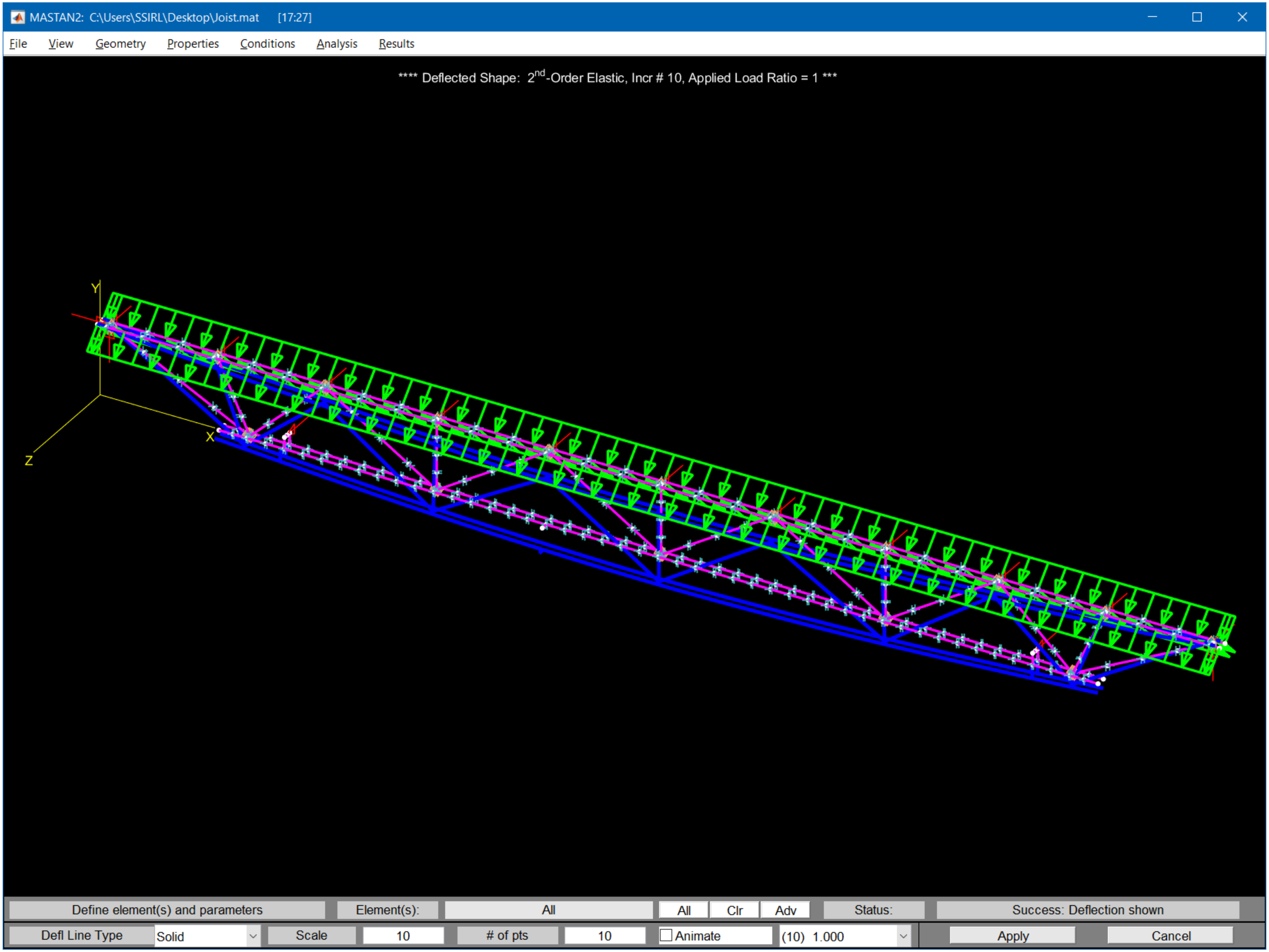
To better see what deformation is occurring, it can be useful to make use of the other defined view. From the **View** menu select **Defined Views** and make use of the submenu options for these ideas.

1) Option **Front view: x-y**.: How the vertical deflection is varying along the length
2) Option **Side view: y-z**.: How big the lateral deflection is compared to the joist
3) Option **Top view: x-z**.: How the lateral deflection varies along the length
4) Option **Isometric: x-y-z**.: Will return to the original view

It may be desired to update the deflected shape diagram with different scale factors during this process.


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:27]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
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\Joist.mat [17:27]
File View Geometry Properties Conditions Analysis Results
**** Deflected Shape: 2nd-Order Elastic, Incr # 10, Applied Load Ratio = 1 ***
Y
X
Z
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

Uniform Loading Results

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

2) On the undeflected shape, click on the midspan node of interest, node **173**, and the displacements for base 6 degree of freedoms are provided in the bottom menu bar.

Calculated vertical deflection: 0.59 in

Estimated deflection using SJI recommendations:

$$I_j = 26.767 \cdot W \cdot L^3 \cdot 10^{-6} = 26.767 \cdot 297plf \cdot (20ft - 4in)^3 \cdot 10^{-6} = 60.47in^4$$

$$\delta = 1.15 \cdot \frac{5WL^4}{384EI} = 1.15 \cdot \frac{5 \cdot 300plf(20ft - 4in)^4}{384 \cdot 29000ksi \cdot 60.47in^4} = 0.66in$$

Estimation using area of chords to calculate I:

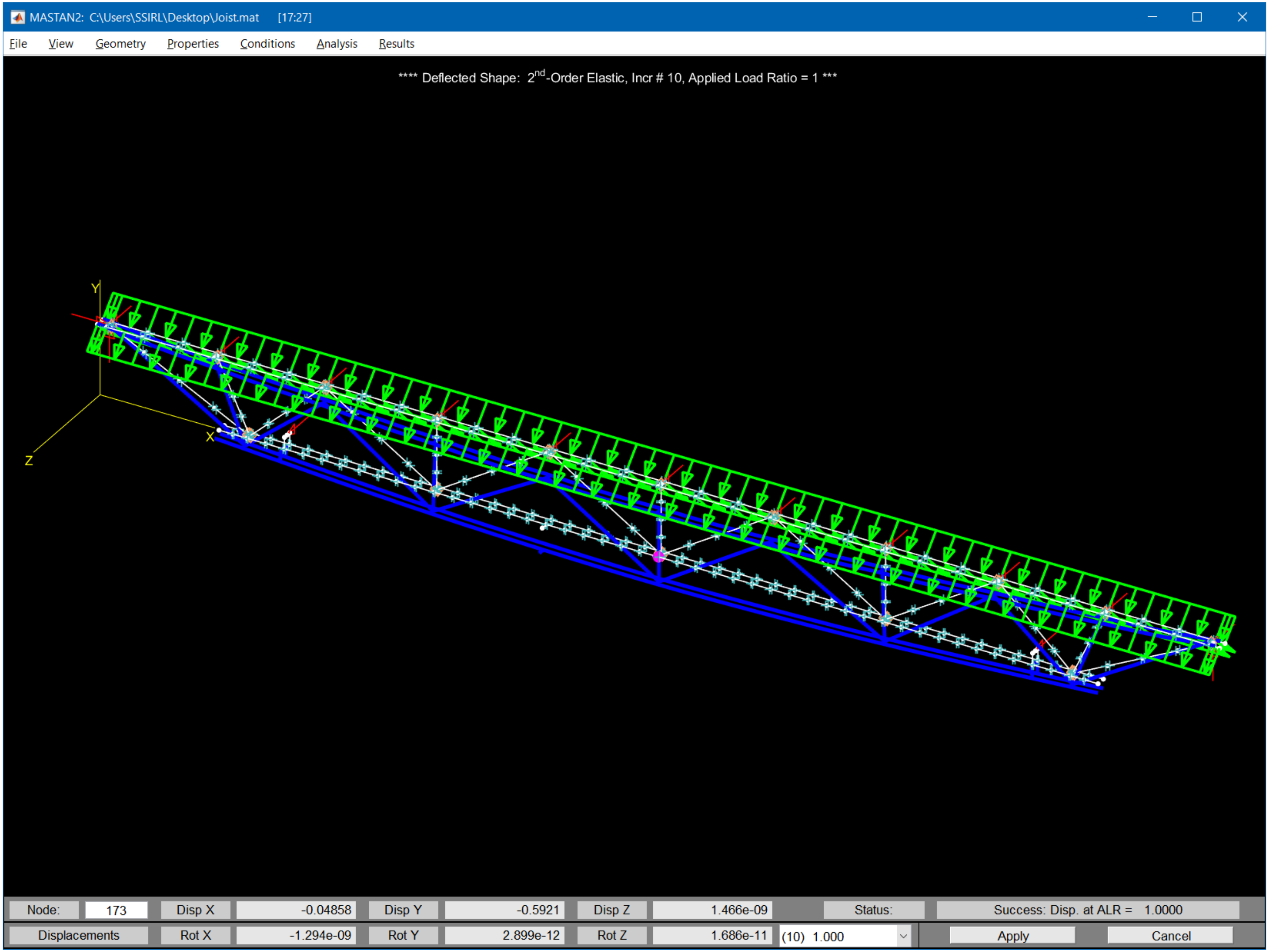
$$I_A = 2 \cdot (0.29895in^2 \cdot (8.282in)^2 + 0.36199in^2 \cdot (15.121in - 8.282in)^2) = 74.87in^4$$

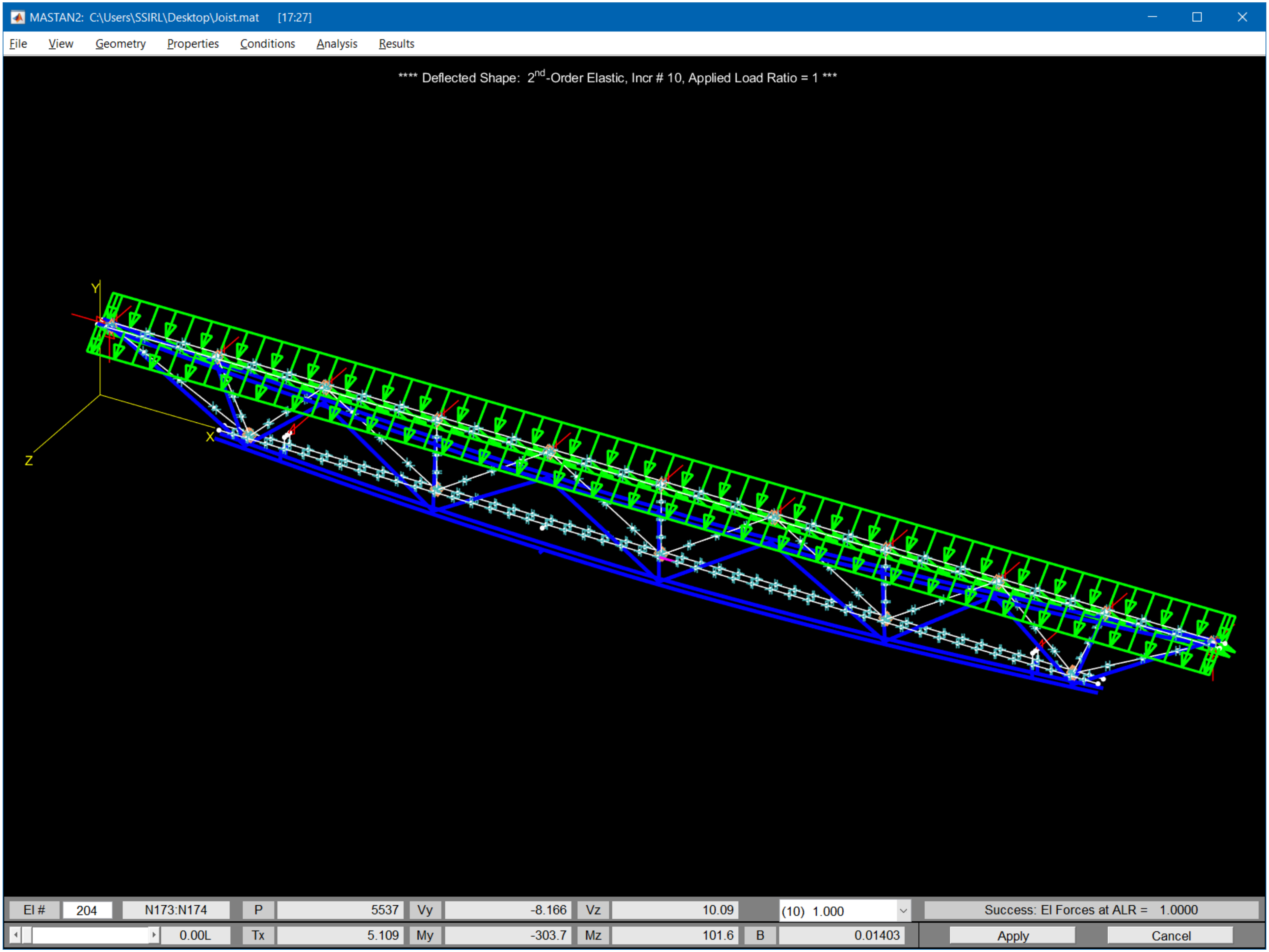
$$\delta = 1.15 \cdot \frac{5WL^4}{384EI} = 1.15 \cdot \frac{5 \cdot 300plf(20ft - 4in)^4}{384 \cdot 29000ksi \cdot 74.87in^4} = 0.53in$$

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

4) On the undeflected shape, click on the span element of interest, element **204**, and the internal forces are provided in the bottom menu bar. These are the forces at the start of the member.

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:27]
File View Geometry Properties Conditions Analysis Results
**** Deflected Shape: 2nd-Order Elastic, Incr # 10, Applied Load Ratio = 1 ***
Y
X
Z
Node: 173
Disp X -0.04858
Disp Y -0.5921
Disp Z 1.466e-09
Status:
Success: Disp. at ALR = 1.0000
Displacements
Rot X -1.294e-09
Rot Y 2.899e-12
Rot Z 1.686e-11
(10) 1.000
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:27]
File View Geometry Properties Conditions Analysis Results
**** Deflected Shape: 2nd-Order Elastic, Incr # 10, Applied Load Ratio = 1 ***
Y
X
Z
El # 204 N173:N174 P 5537 Vy -8.166 Vz 10.09 (10) 1.000 Success: El Forces at ALR = 1.0000
0.00L Tx 5.109 My -303.7 Mz 101.6 B 0.01403 Apply Cancel

Section 6: Hanging Load Analysis

Eccentric Loading

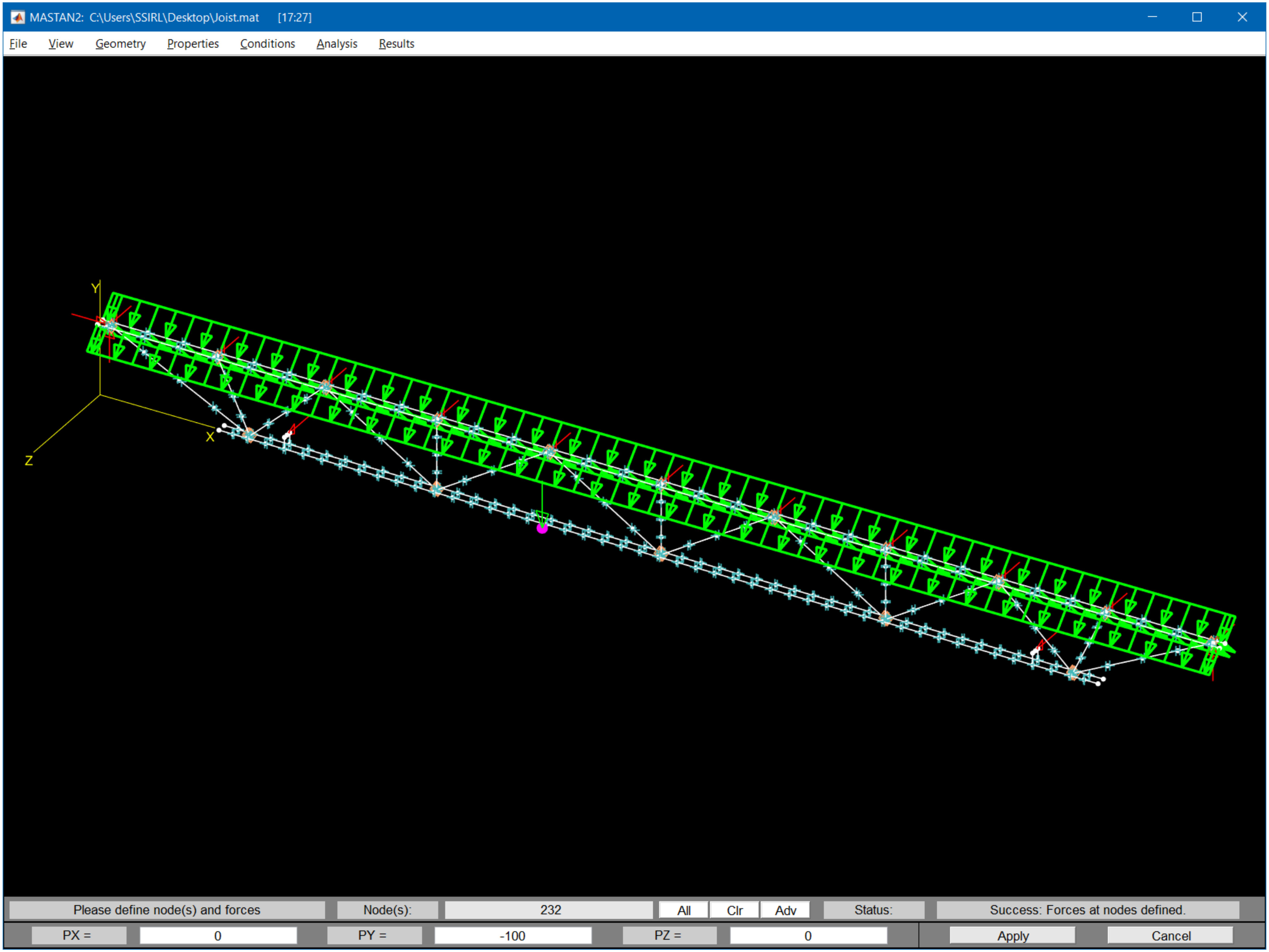
1) From the **Results** menu select **Diagrams** and submenu option **None**.

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

3) At the bottom menu bar, click in the edit box just to the right of **PY =** and change **0** to **-100** to create a handing load.

4) Click on the node at the tip of the eccentric arm created at the end of the geometry modeling, node **232**.

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


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:27]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
Please define node(s) and forces
Node(s):
232
All
Clr
Adv
Status:
Success: Forces at nodes defined.
PX =
0
PY =
-100
PZ =
0
Apply
Cancel

MASTAN2

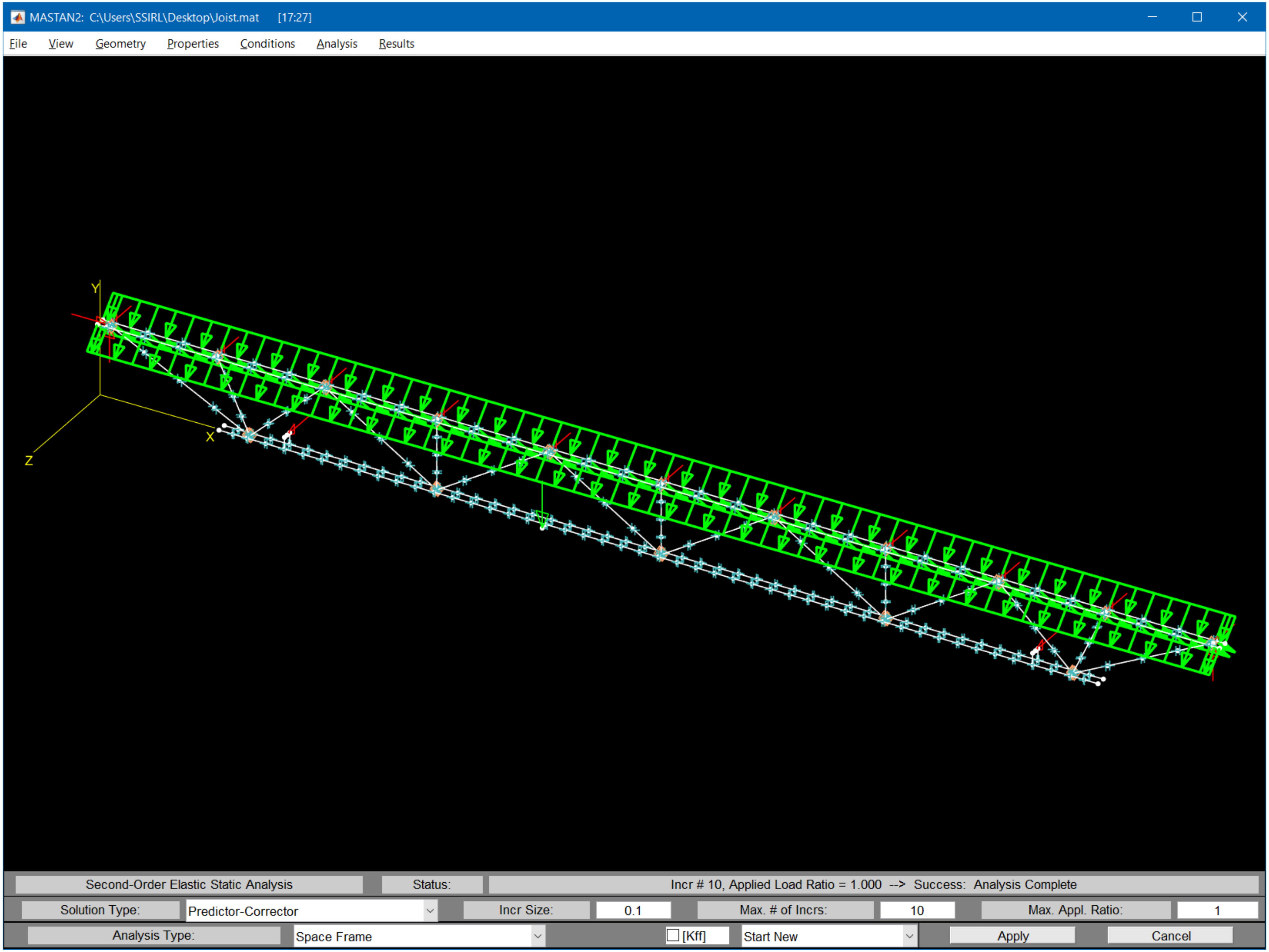
Eccentrically Loaded Joist Elastic Analysis

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.

MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:27]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
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

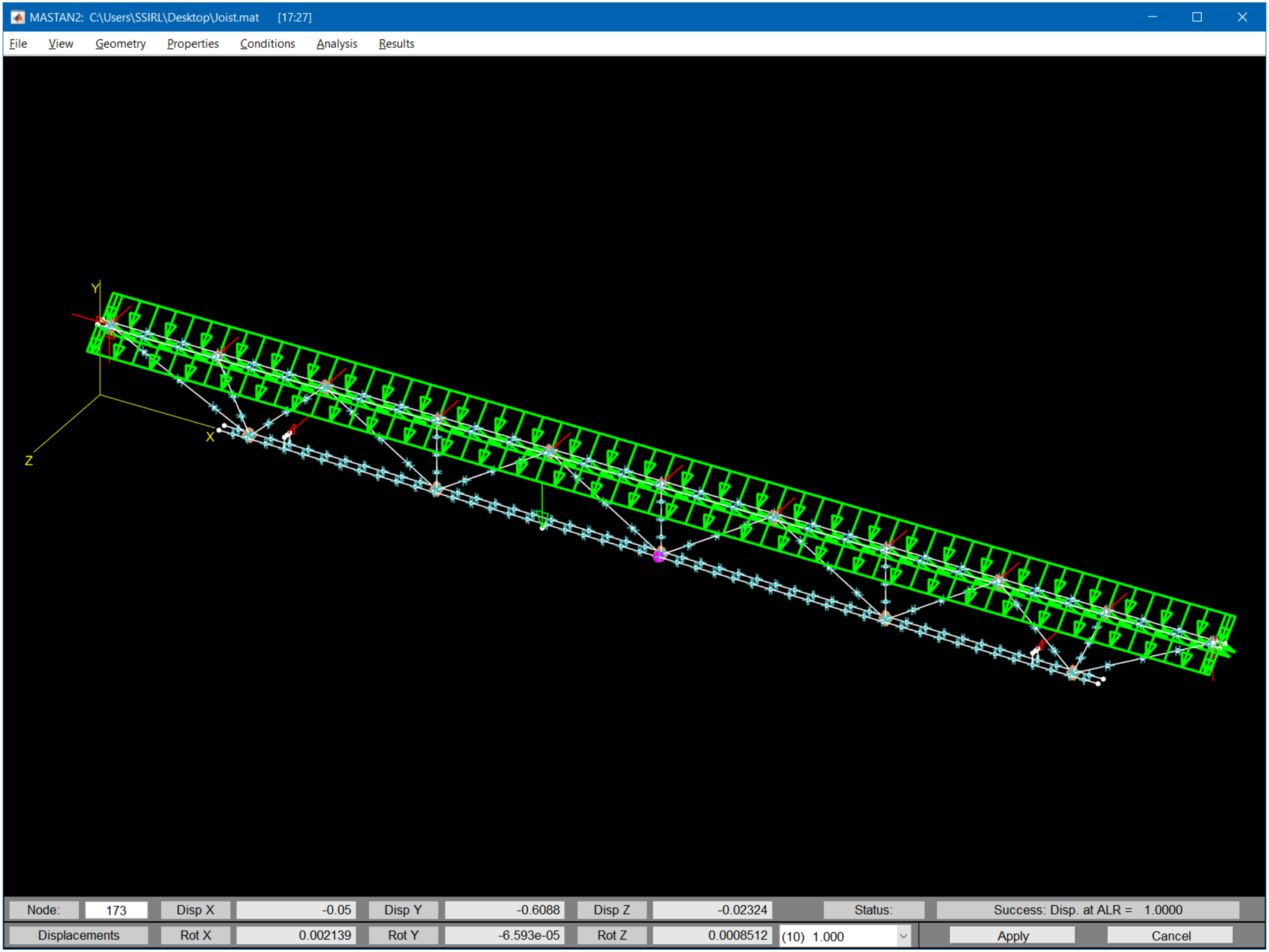
Eccentrically Loaded Joist Elastic Results

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

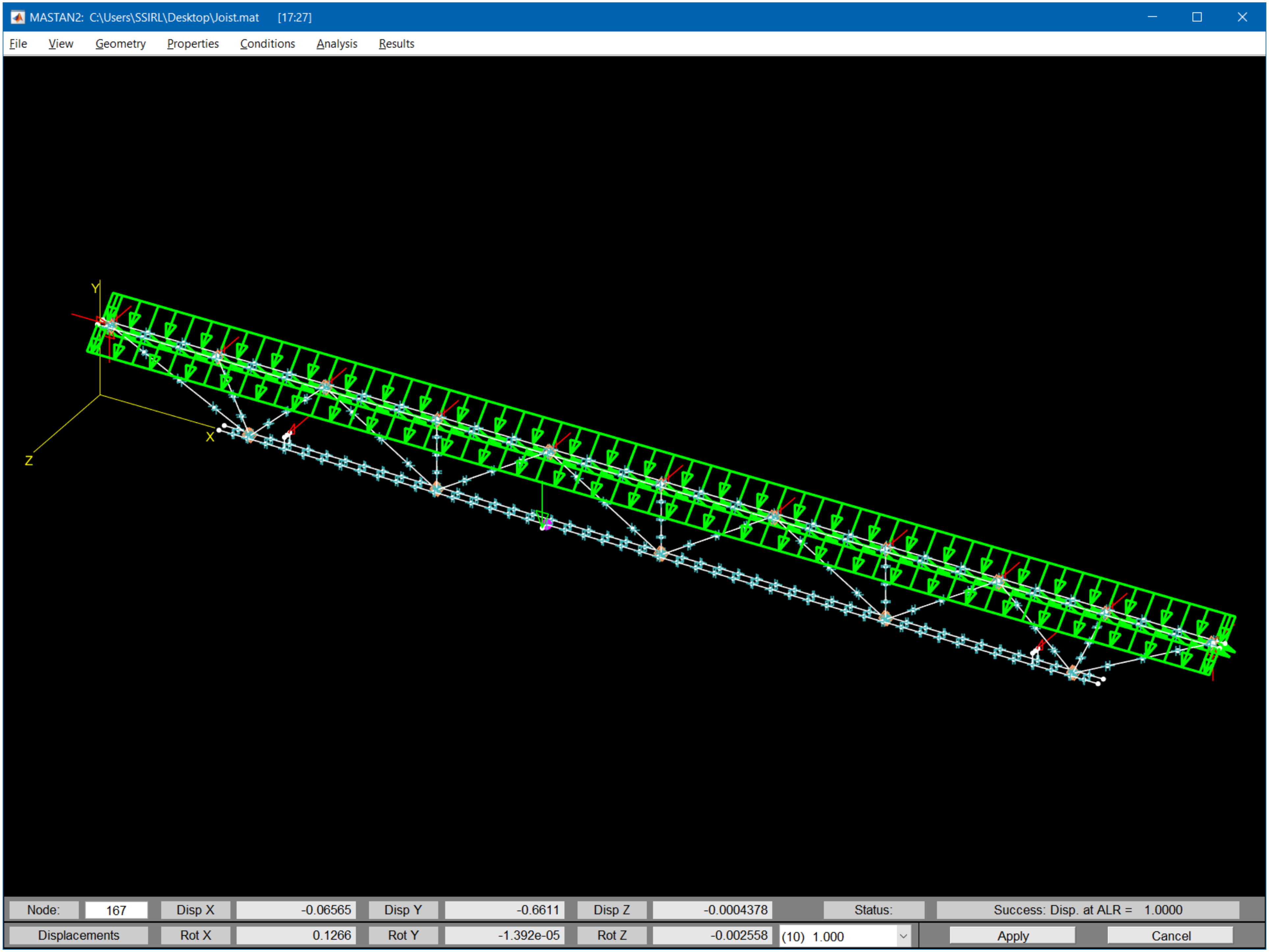
2) At the bottom menu bar, click on the **Apply** button.

3) On the undeflected shape, click on the midspan node of interest, node **173**, to see how the change to the midspan deflection from the eccentric load.

4) On the undeflected shape, click on the node of the bottom chord attached to the eccentric loading arm, node **167**, to see how the hanging load moved the chord.


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:27]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
Node:
173
Disp X
-0.05
Disp Y
-0.6088
Disp Z
-0.02324
Status:
Success: Disp. at ALR = 1.0000
Displacements
Rot X
0.002139
Rot Y
-6.593e-05
Rot Z
0.0008512
(10) 1.000
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:27]
File View Geometry Properties Conditions Analysis Results
Y
X
Z
Node: 167
Disp X -0.06565
Disp Y -0.6611
Disp Z -0.0004378
Status: Success: Disp. at ALR = 1.0000
Displacements
Rot X 0.1266
Rot Y -1.392e-05
Rot Z -0.002558
(10) 1.000
Apply
Cancel

Stress Calculations

Using the results available within the MASTAN2 model, it is possible to calculate the internal stresses that the members are experiencing. Provided in this tutorial are the steps to pull the necessary values from MASTAN2 and the resulting stresses.

Additional details on the necessary calculations are available in the “Pour Stop” tutorial.

Getting Internal Forces

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

2) On the undeflected shape, click on the midspan element of interest, element **198**, 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 over left-hand corner until the position indicator just to the right displays **1.00L**.

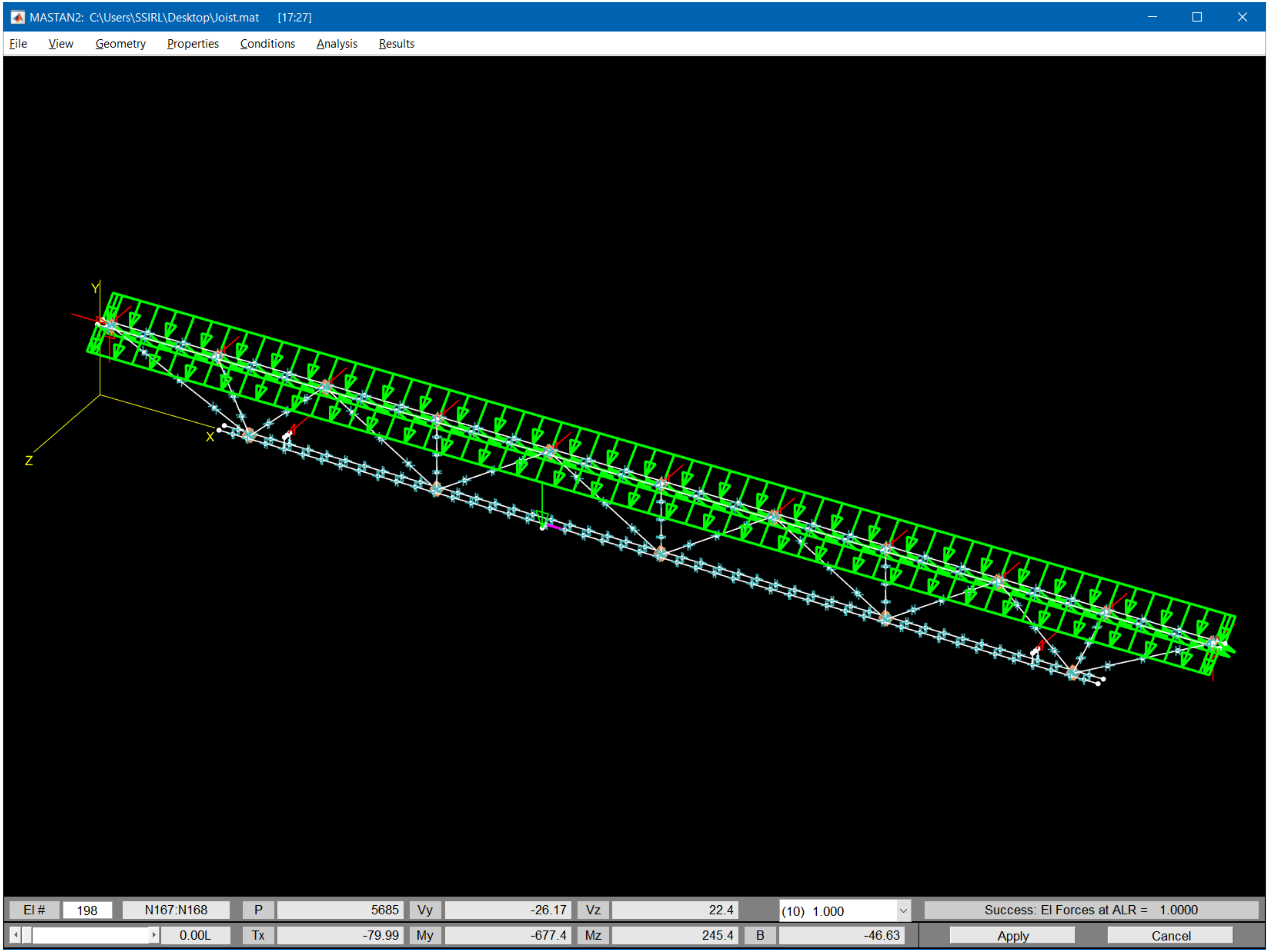
5) Click on the **Apply** button. These are the forces at the end of the member.

6) From this position, the bimoment is required to appropriately divide the longitudinal moment into the standard twisting and warping components for stress calculations.

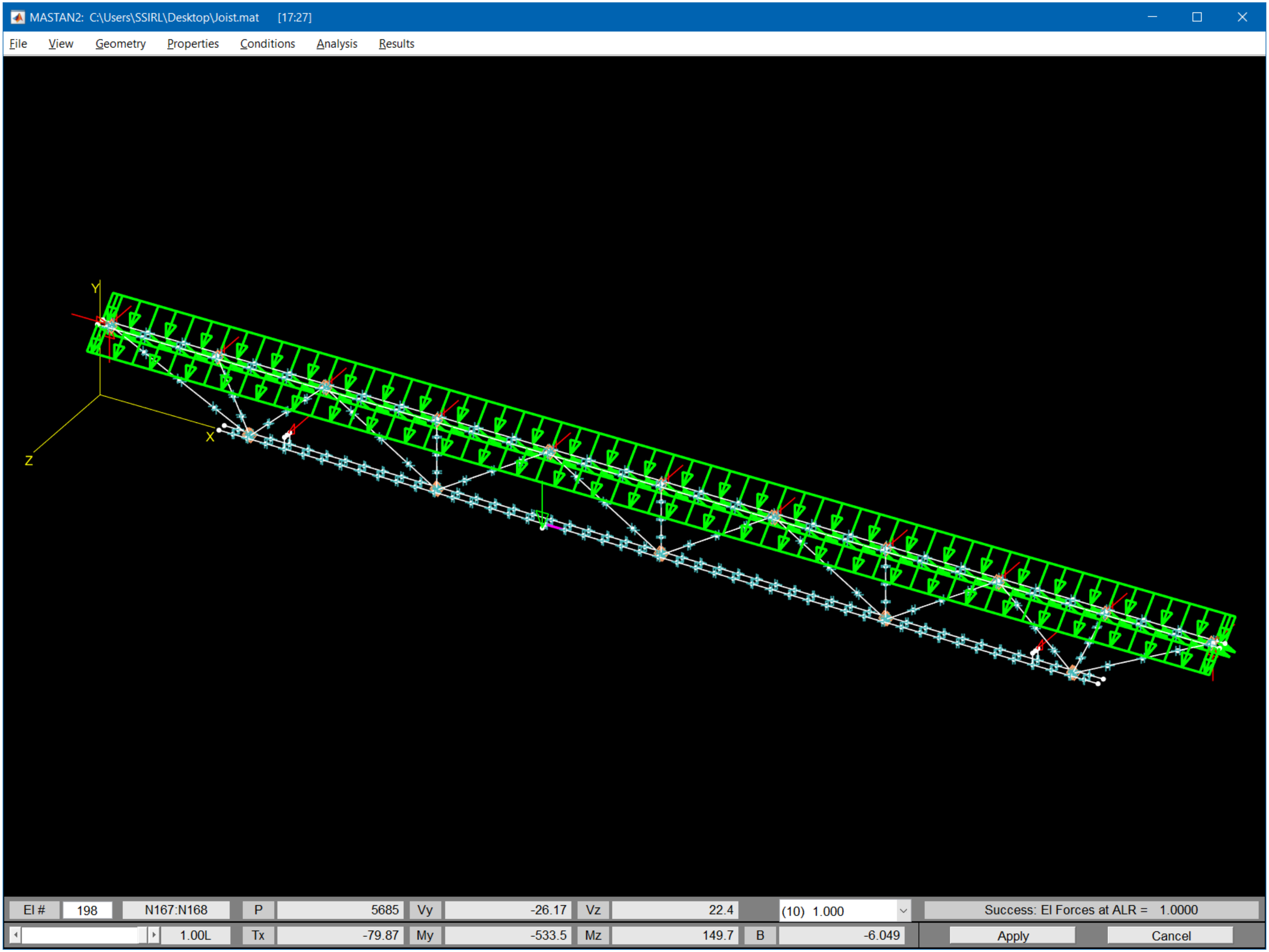
7) Repeat these steps to get the internal forces in element **192**. This is where the larger negative moment on the bottom chord now occurs.

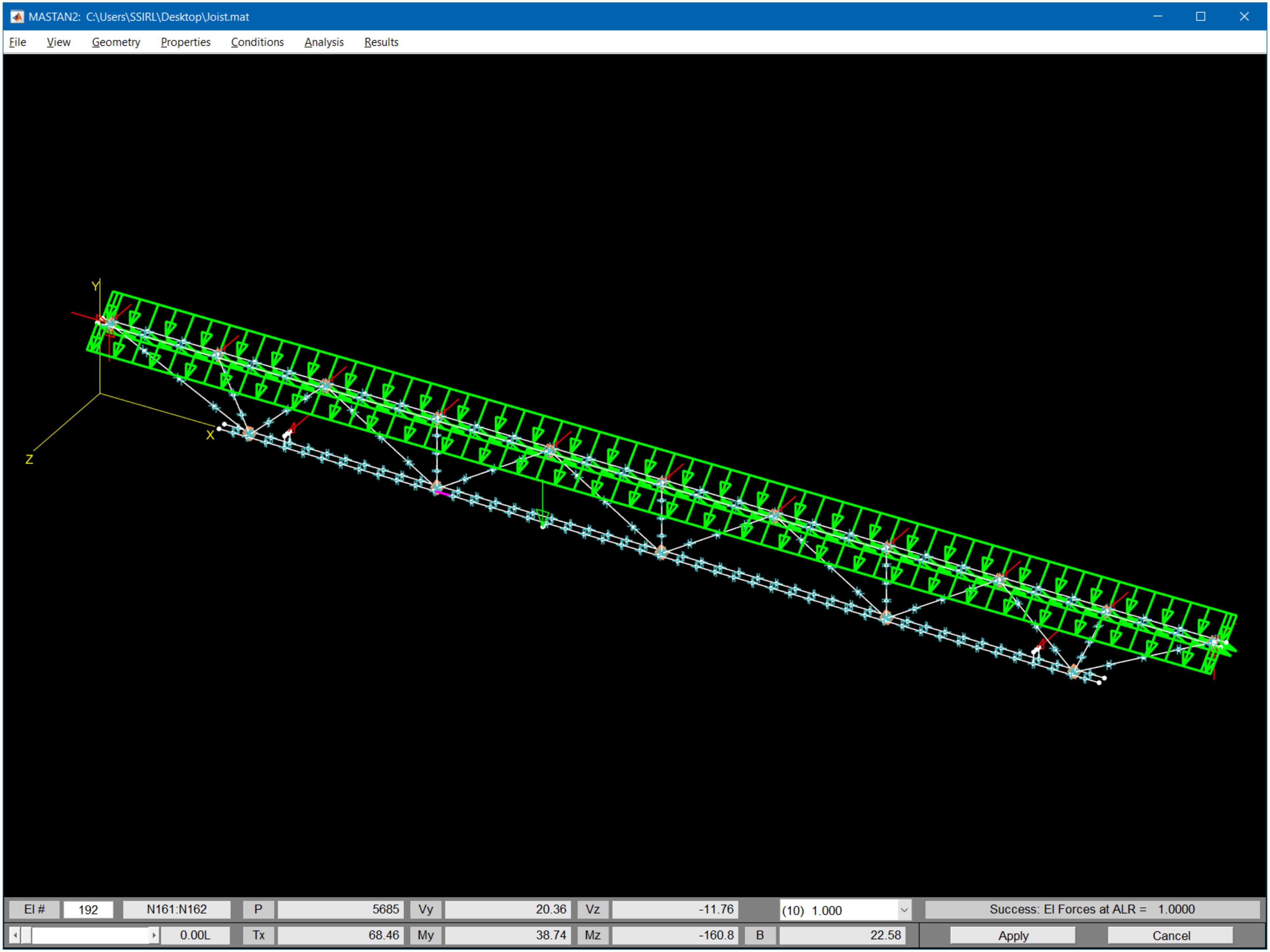
The forces at the start: The forces at the end:

MASTAN2

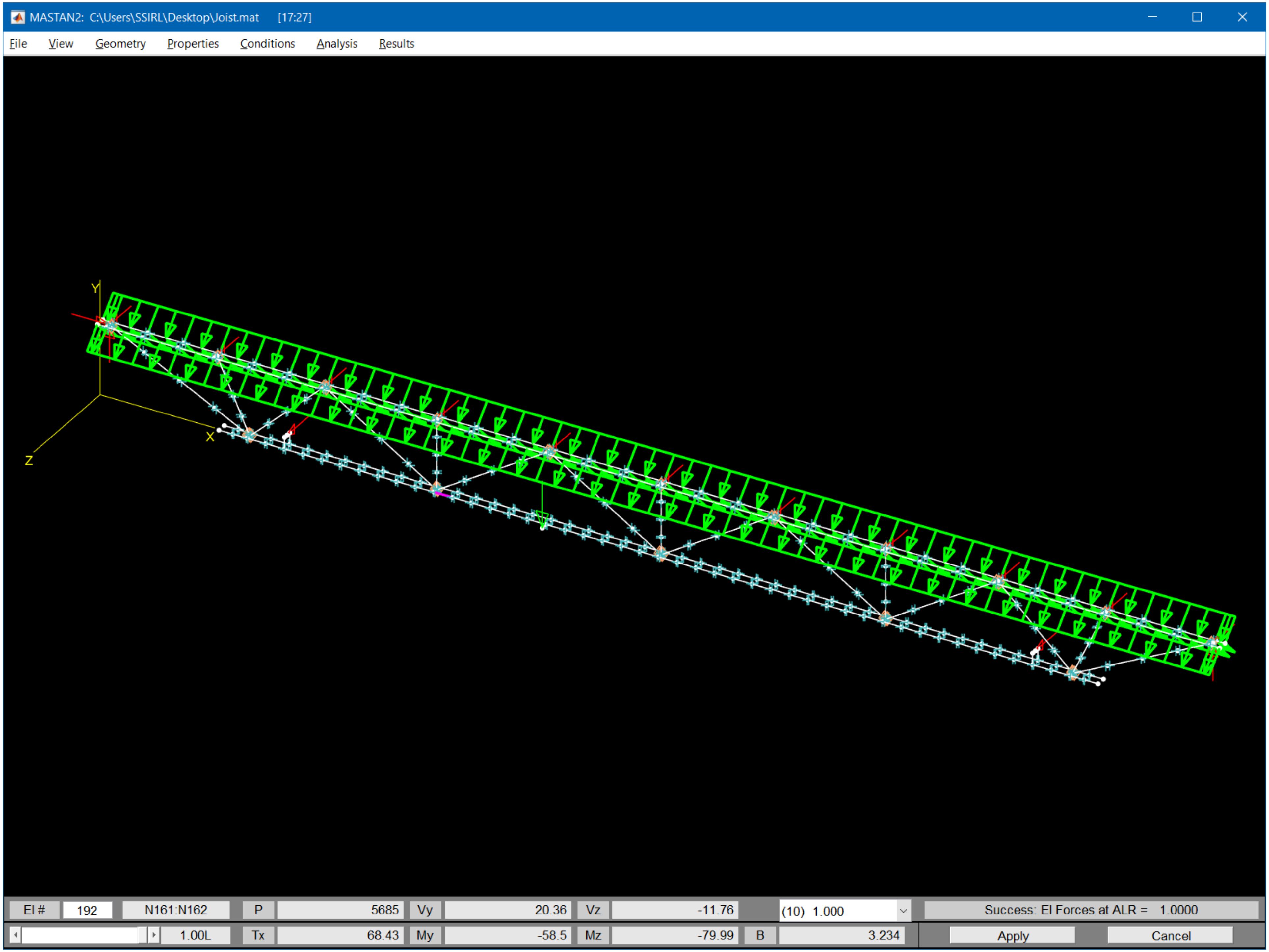

MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:27]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
El #
198
N167:N168
P
5685
Vy
-26.17
Vz
22.4
(10) 1.000
Success: El Forces at ALR = 1.0000
0.00L
Tx
-79.99
My
-677.4
Mz
245.4
B
-46.63
Apply
Cancel

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:27]
File View Geometry Properties Conditions Analysis Results
Y
X
Z
El #
198
N167:N168
P
5685
Vy
-26.17
Vz
22.4
(10) 1.000
Success: El Forces at ALR = 1.0000
1.00L
Tx
-79.87
My
-533.5
Mz
149.7
B
-6.049
Apply
Cancel


MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
El #
192
N161:N162
P
5685
Vy
20.36
Vz
-11.76
(10) 1.000
Success: El Forces at ALR = 1.0000
0.00L
Tx
68.46
My
38.74
Mz
-160.8
B
22.58
Apply
Cancel

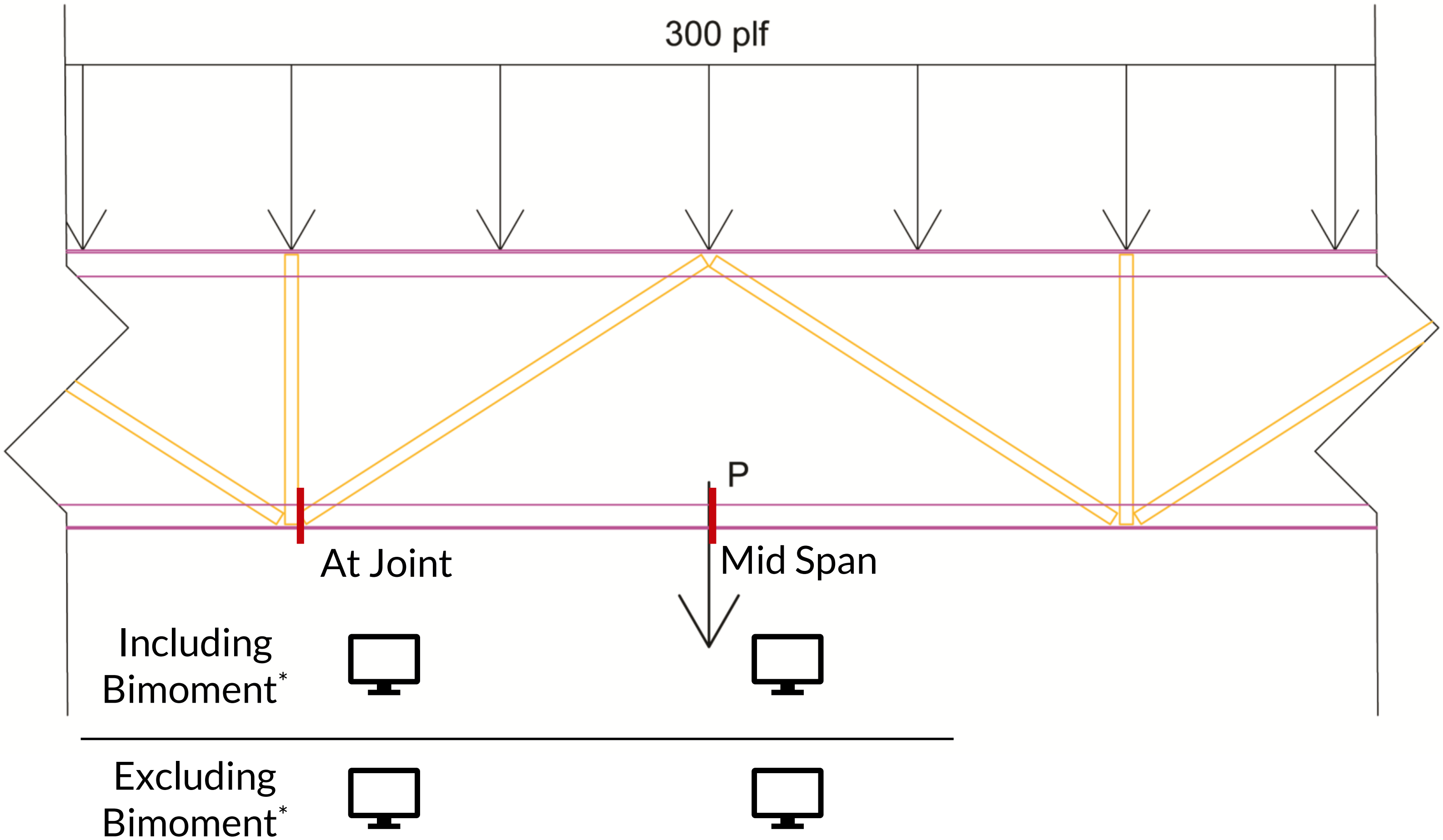
MASTAN2



MASTAN2: C:\Users\SSIRL\Desktop\Joist.mat [17:27]
File
View
Geometry
Properties
Conditions
Analysis
Results
Y
X
Z
El #
192
N161:N162
P
5685
Vy
20.36
Vz
-11.76
(10) 1.000
Success: El Forces at ALR = 1.0000
1.00L
Tx
68.43
My
-58.5
Mz
-79.99
B
3.234
Apply
Cancel

Stress Calculation Results

The resulting stresses are available for the locations identified on the sketch. The forces are calculated immediately after the web connection node and after the point load.



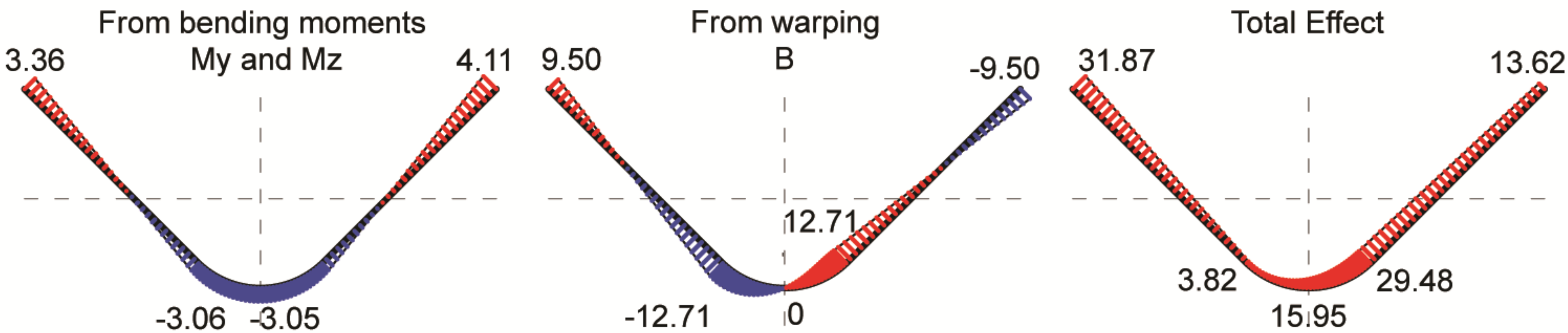
* Numerically bimoment exists in the model because the warping constant, Cω, is non-zero. Additional meshing of the model would refine the distribution of bimoment, but bimoment would still exist locally in the model at applied torque or supports and rapidly decrease to approximately zero along the majority of the length of the bottom chord. Since the evaluation of angles would often excludes the effects of warping, the internal stresses are provided having been calculated using the forces observed in this model including and excluding the bimoment values.

MASTAN2

Stresses at Joint including Bimoment

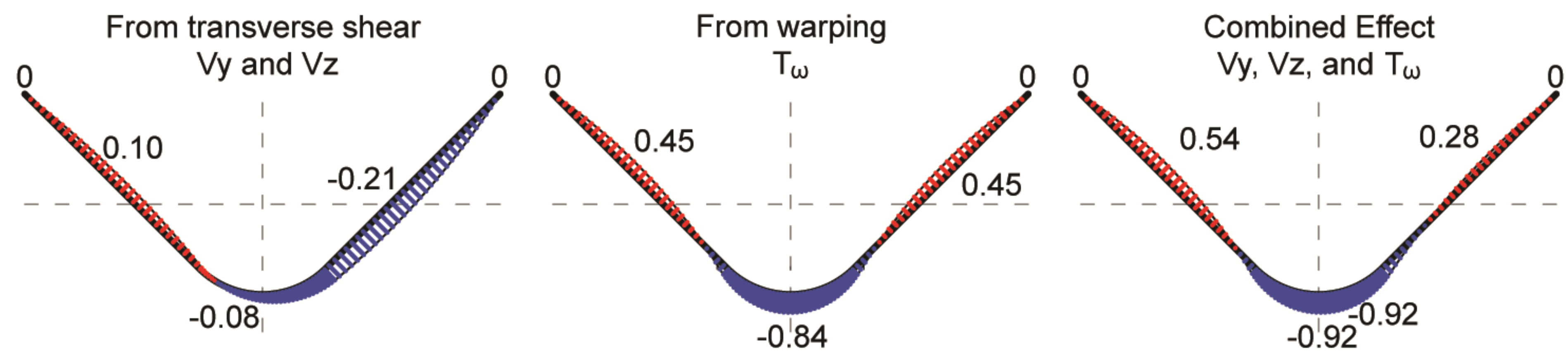
Normal Stresses (ksi)
Red - Tension, Blue - Compression

Diagram not shown: From axial force, P - Uniform 19.00 ksi tension



Shear Stresses (ksi)
Red - To the right, Blue - To the left

Diagram not shown: From torsional moment, T_T - Variable across thickness ±6.04 ksi

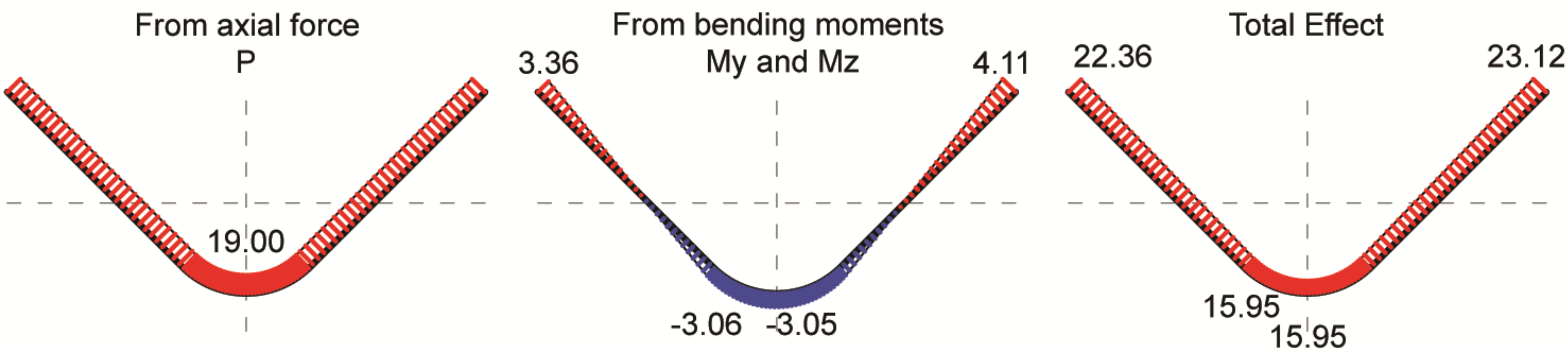


MASTAN2

Stresses at Joint excluding Bimoment

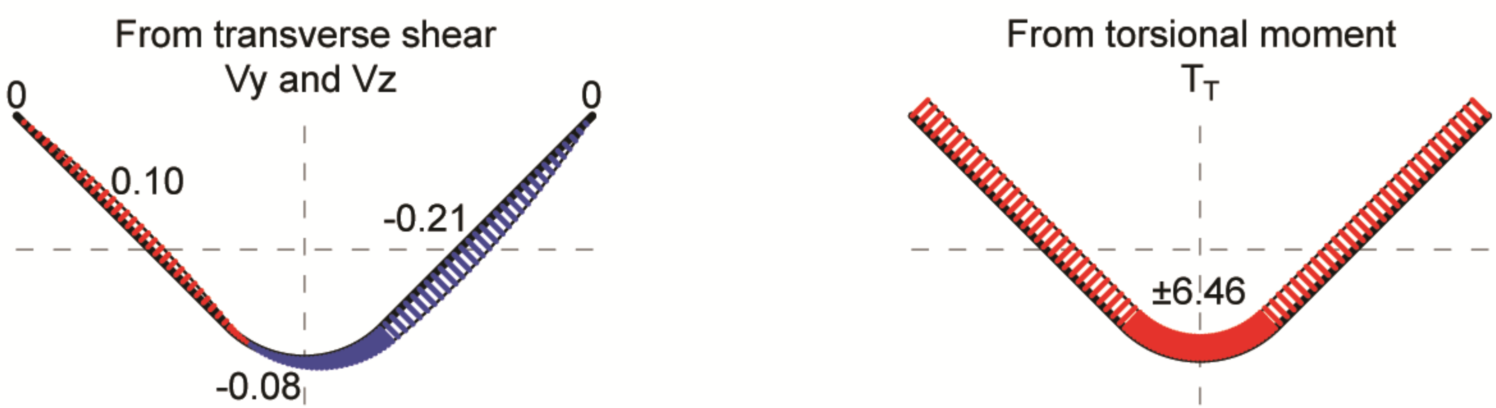
Normal Stresses (ksi)

Red - Tension, Blue - Compression



Shear Stresses (ksi)

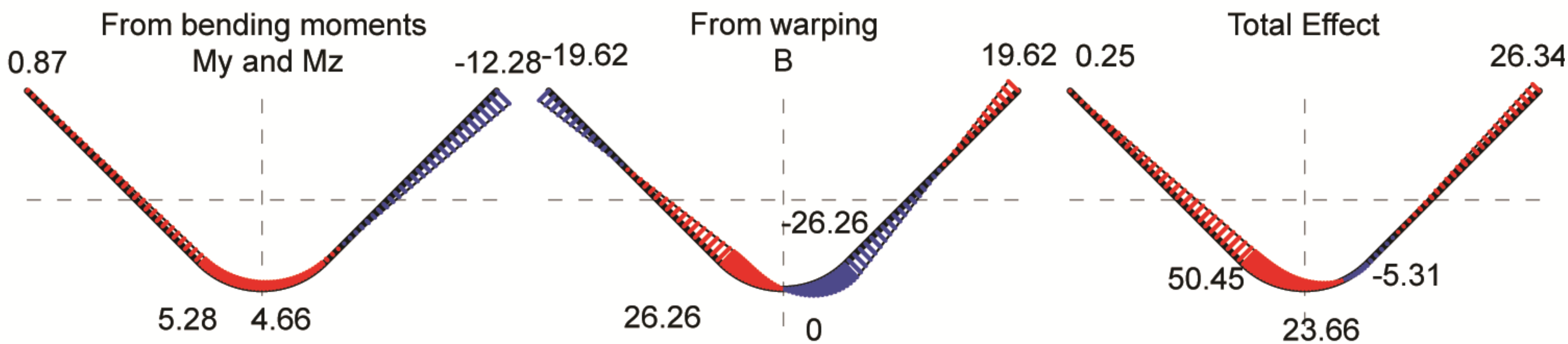
Red - To the right, Blue - To the left



Stresses at Midspan including Bimoment

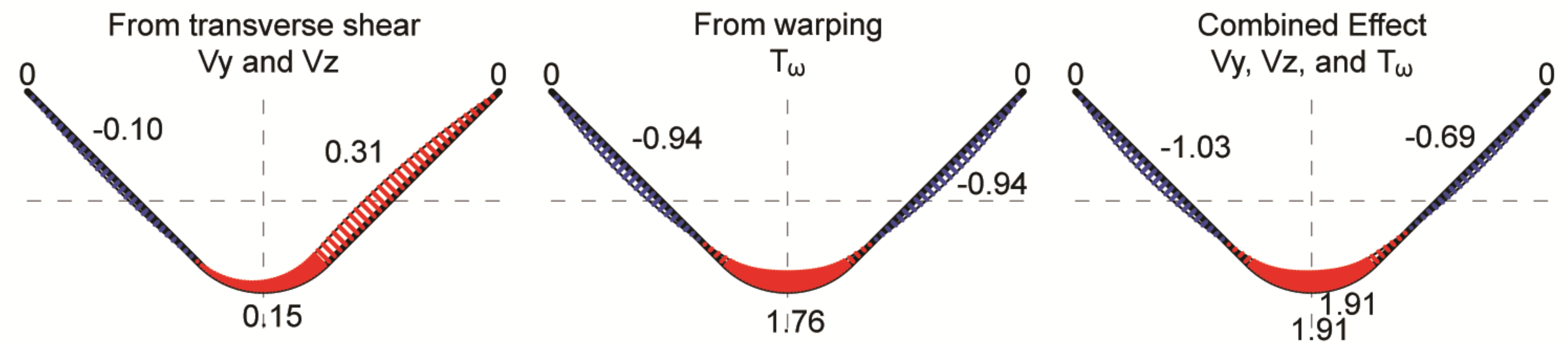
Normal Stresses (ksi)
Red - Tension, Blue - Compression

Diagram not shown: From axial force, P - Uniform 19.00 ksi tension

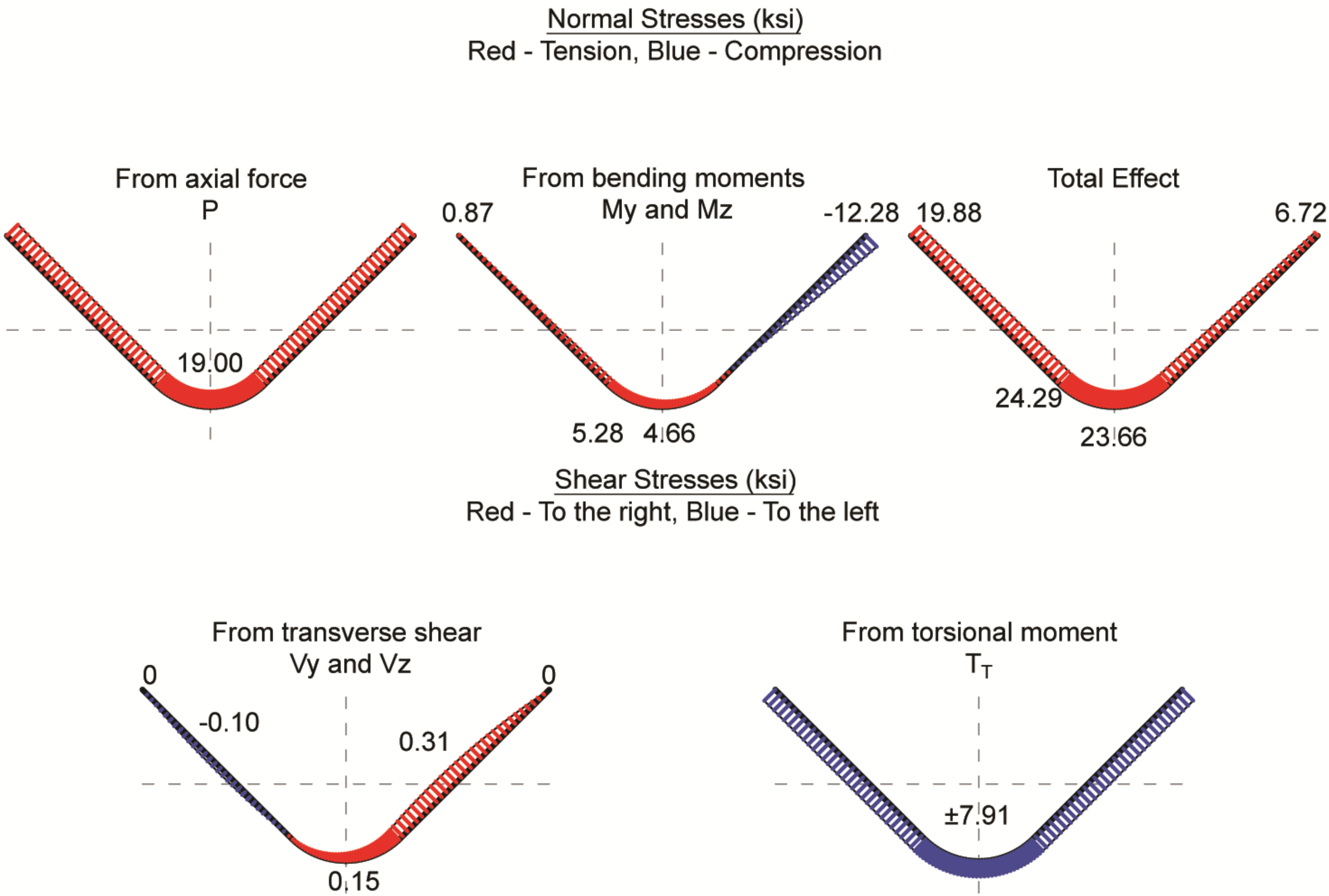


Shear Stresses (ksi)
Red - To the right, Blue - To the left

Diagram not shown: From torsional moment, T_T - Variable across thickness ±7.03 ksi



Stresses at Midspan excluding Bimoment



MASTAN2

This completes the tutorial.

American Iron and Steel Institute

25 Massachusetts Avenue, NW
Suite 800
Washington, DC 20001
www.steel.org

The New Steel®

Research Report RP-21-03