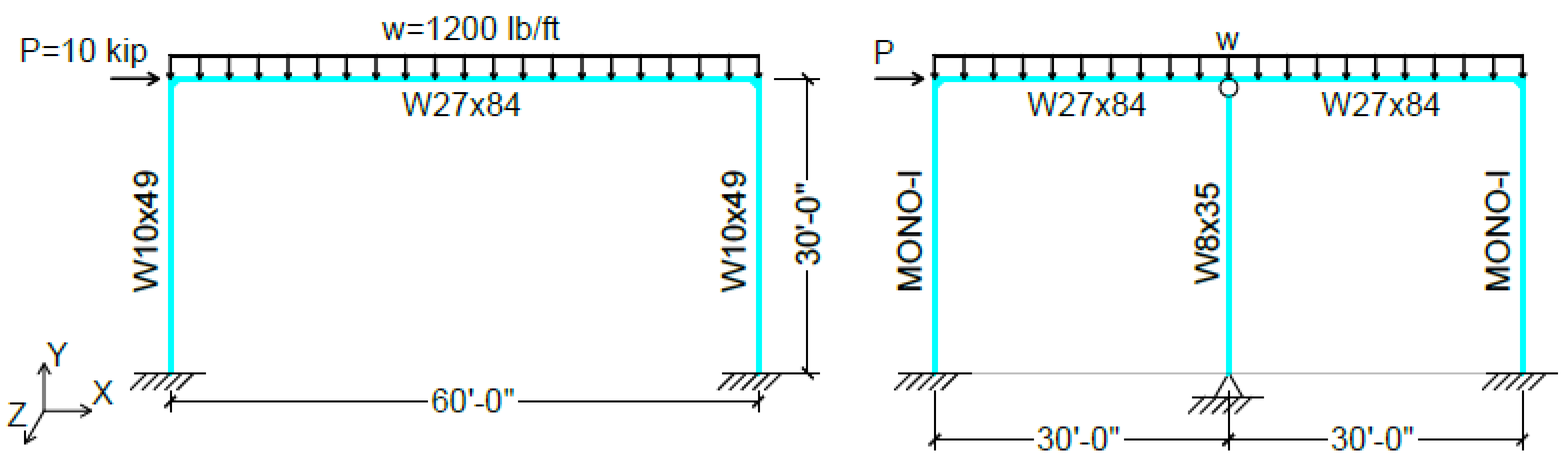
Tutorial for MASTAN2 v5.1 - Introductory Frame



MASTAN2

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

Section 1: Overview

Section 2: Getting Started

Section 3: 2-D Frame Analysis

Section 4: 3-D Frame Analysis

Section 5: Using MSASect

Section 6: Frame Analysis with Non-Doubly Symmetric 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.

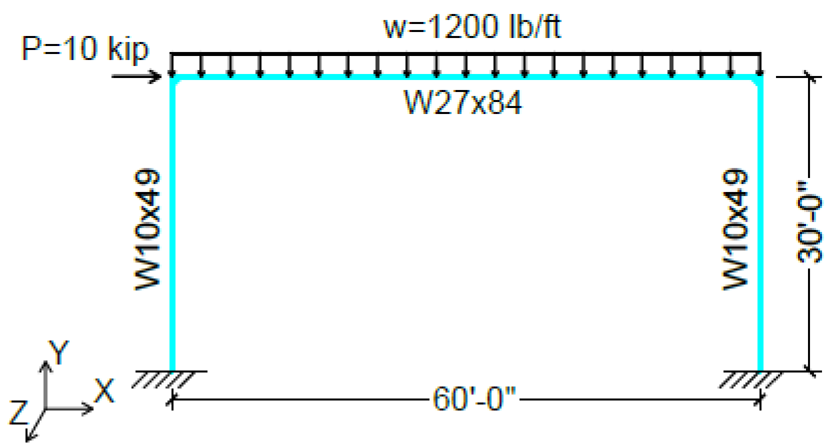
Section 1: Overview

Overview

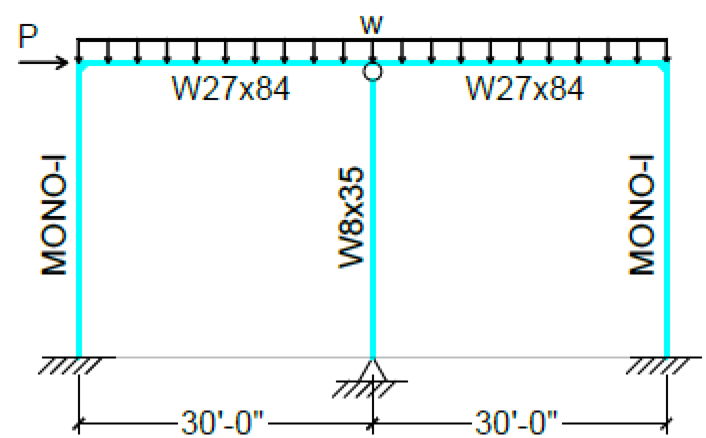
This tutorial provides step-by-step guidance for the sample frame structure. Enough details are provided that the example model with non-doubly symmetric sections can be completed following the instructions here. Not every feature available in MASTAN2 will be mentioned nor utilized in this tutorial.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

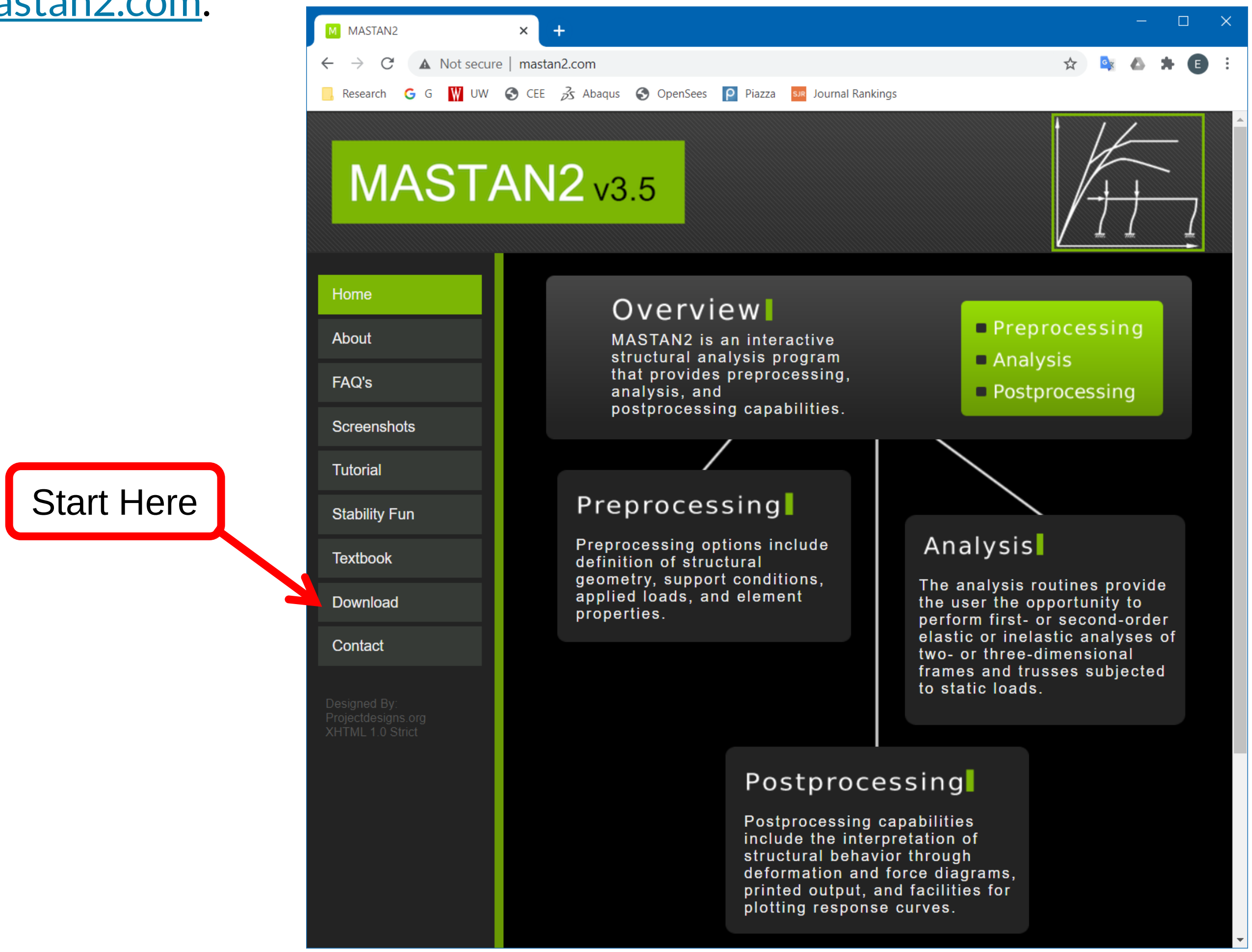
MASTAN2 General Information

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.

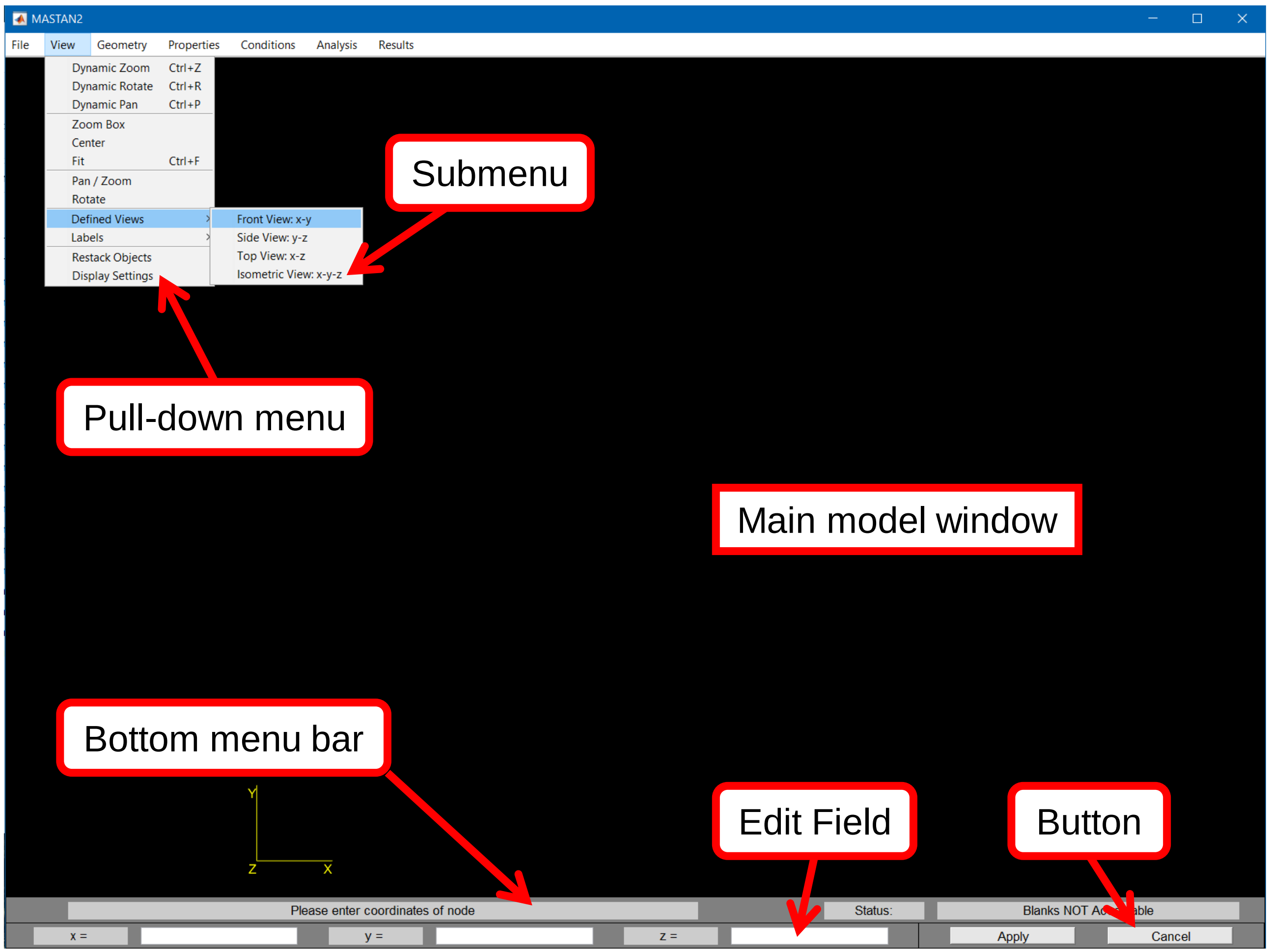
Launching MASTAN2

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

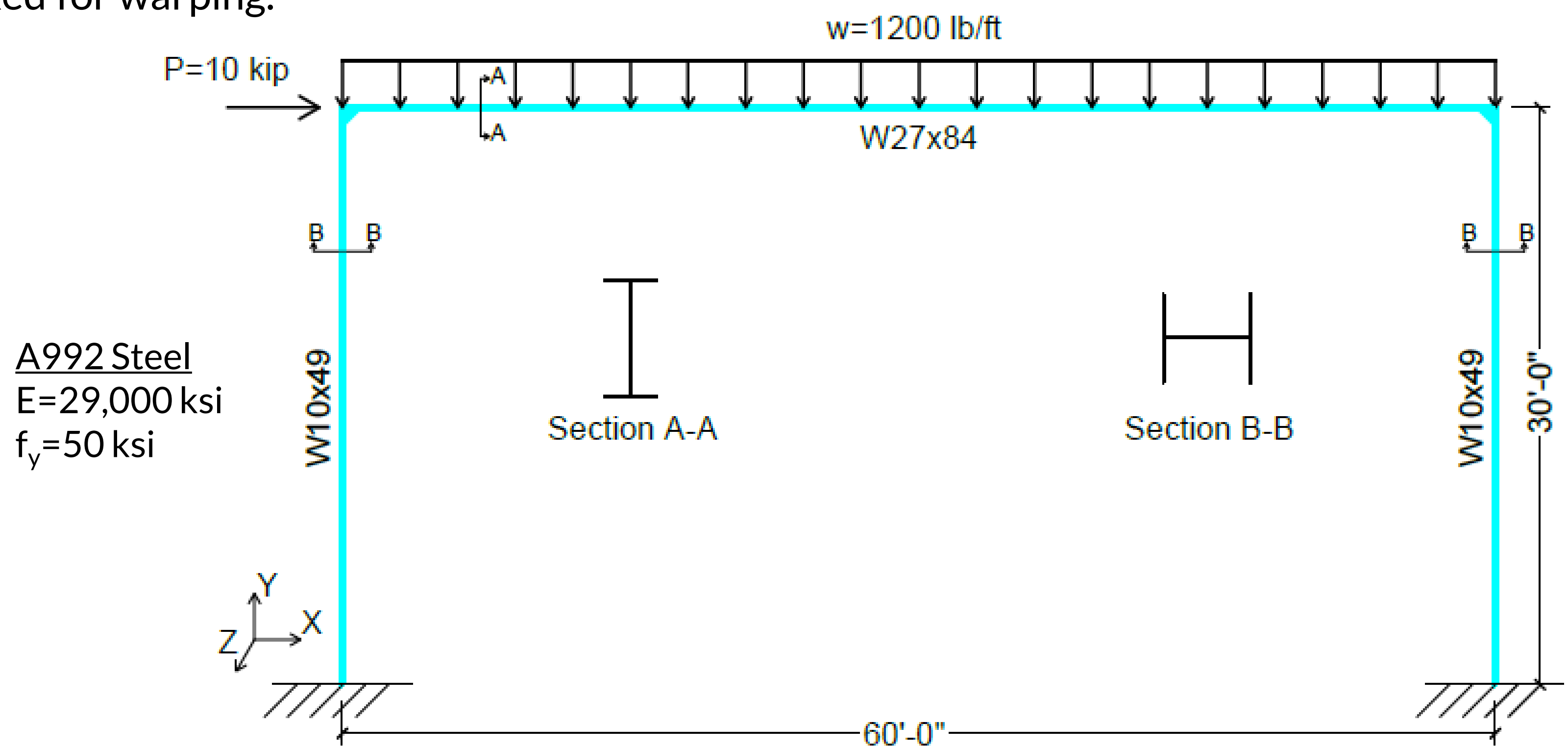
In order to minimize the learning time for MASTAN2, its graphical user interface (GUI) has been designed using a simple and consistent two menu approach. Using a pull-down menu at the top of the GUI, a command is selected. Parameters are then defined in the bottom menu bar and the command is executed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

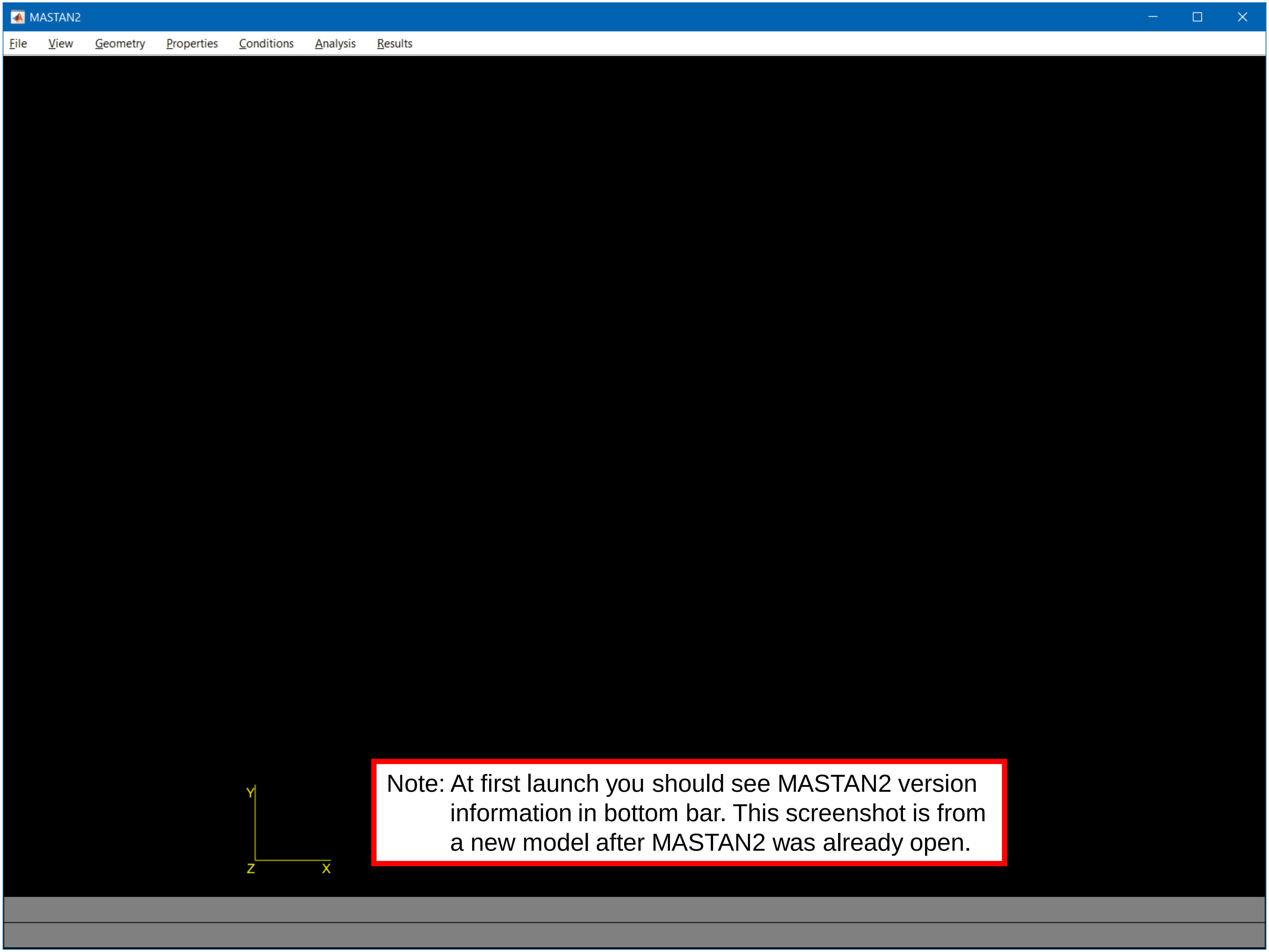
2) From the **Geometry** menu select **Define Frame**.

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**.

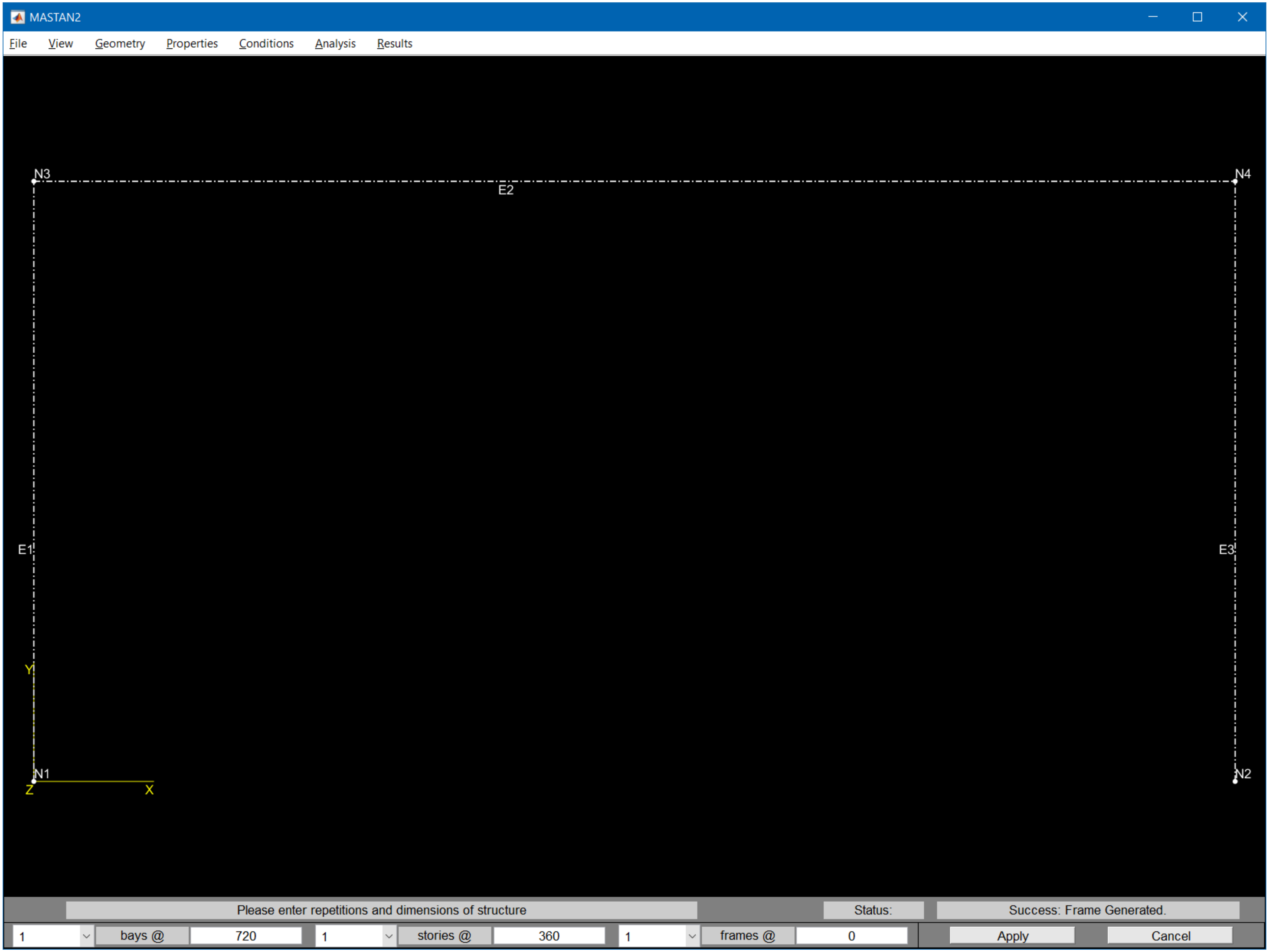
5) Click on the **Apply** Button. A one-bay single story frame is now defined.

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.



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)**.

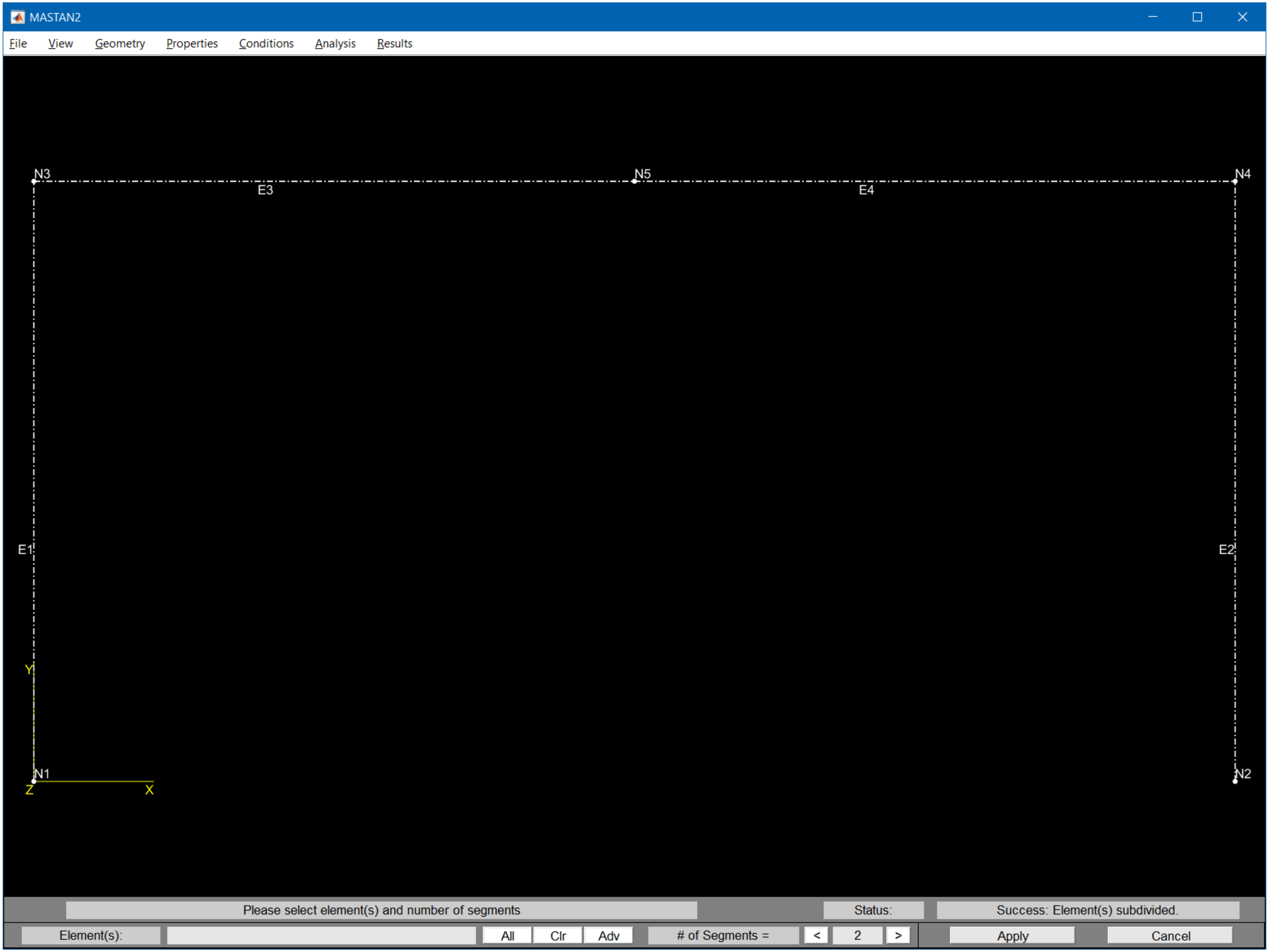
2) Create the list of elements by clicking on the horizontal element.

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

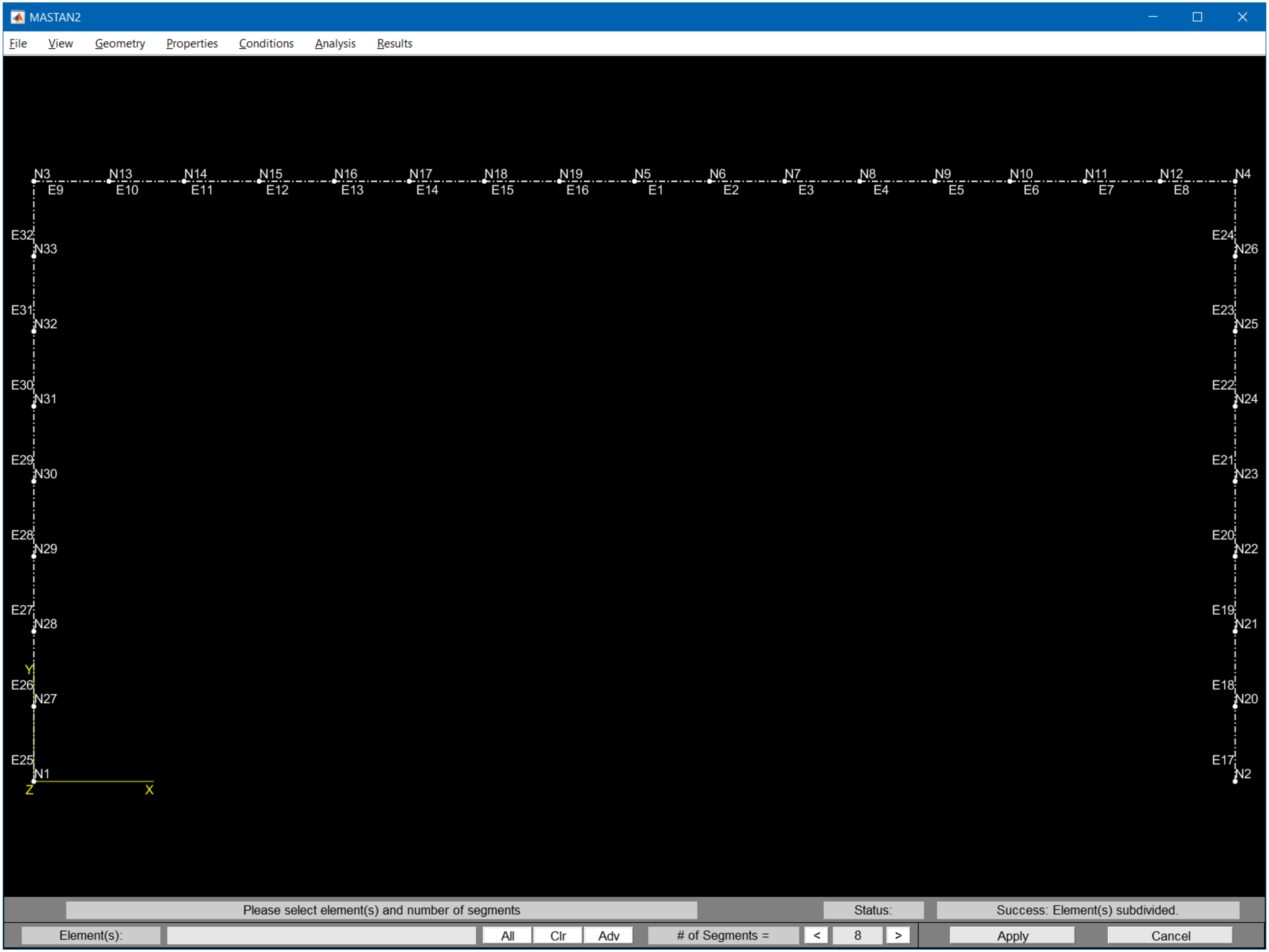
4) Create a new list of all elements by clicking the **All** button.

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

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


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
N3
N5
N4
E3
E4
E1
E2
Y
N1
N2
Z
X
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.

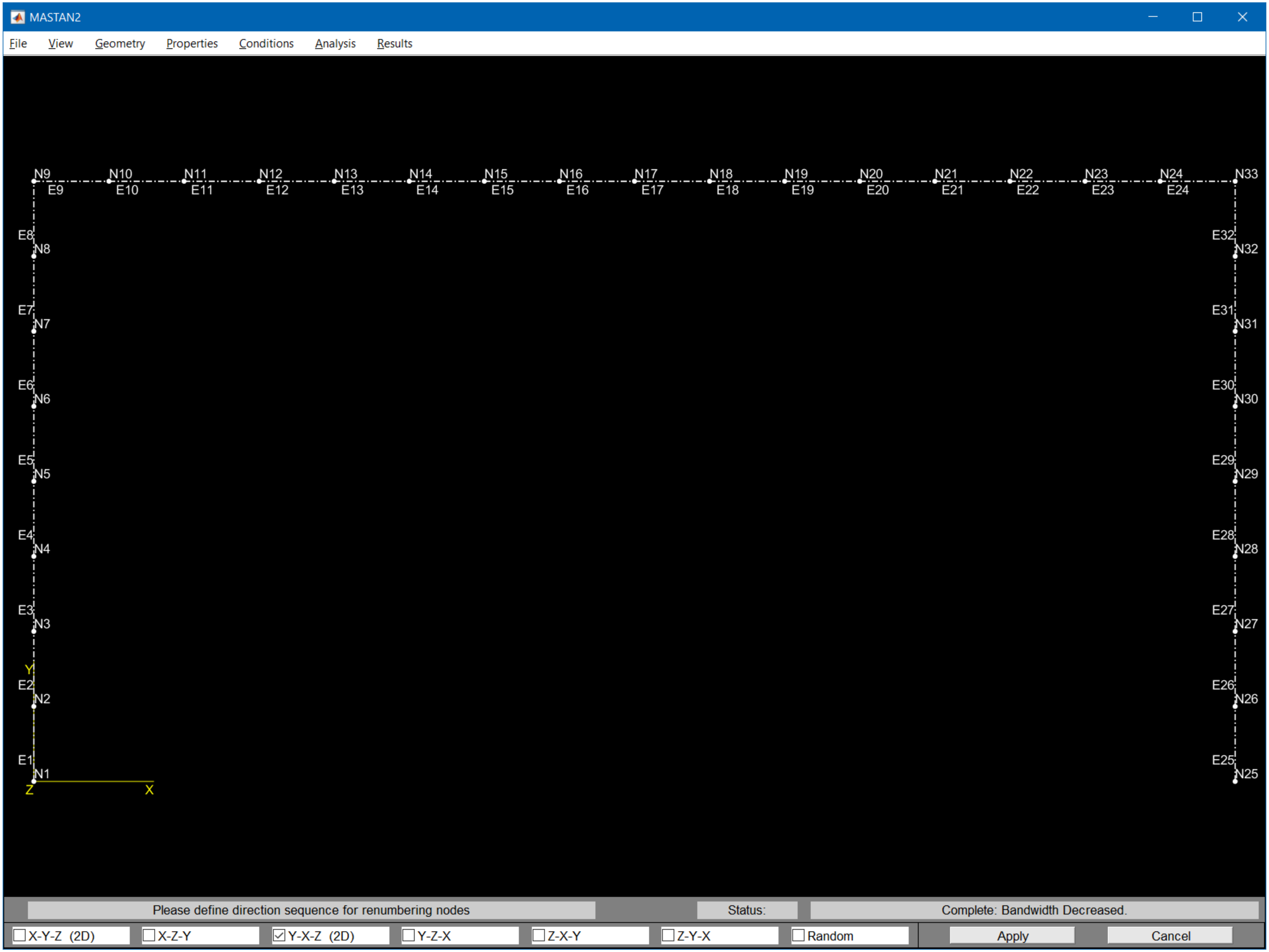
1) From the **Geometry** menu select **Renumber Elements**.

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.

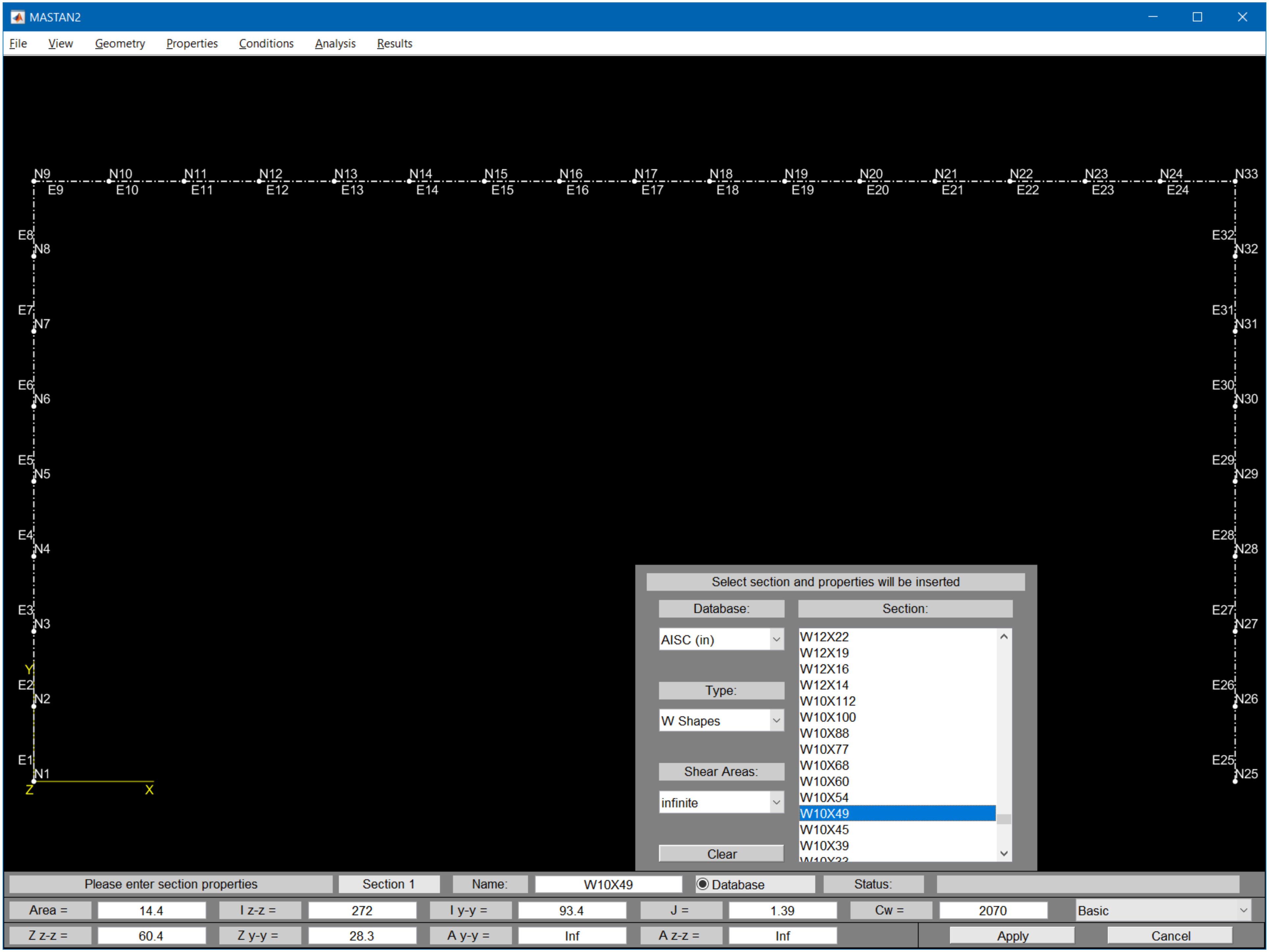
3) In the pop-up menu, scroll to find Section: **W10x49** and click on it.

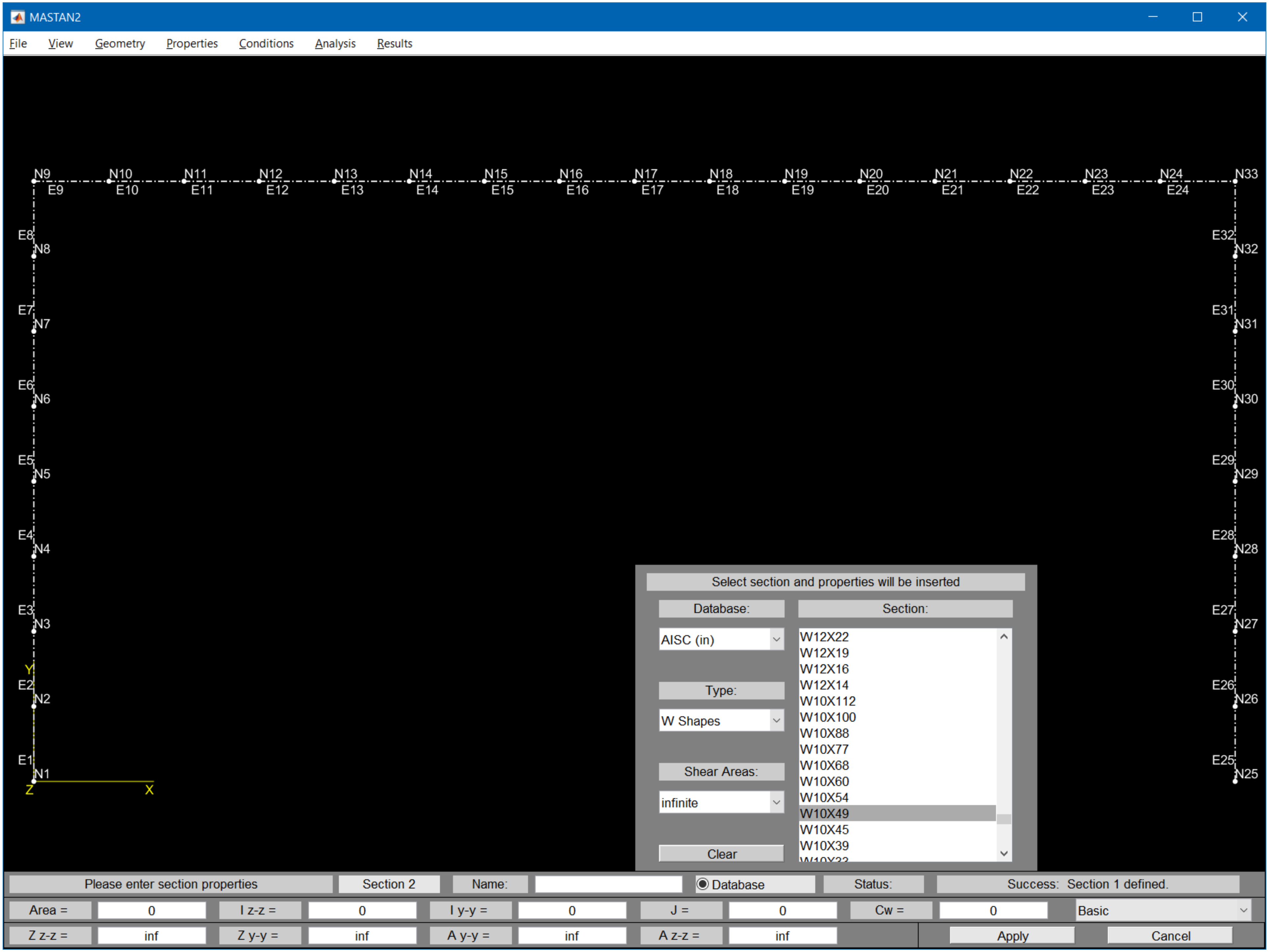
4) Then click on the **Apply** button. Section 1 is now defined with the properties of W10x49.

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.

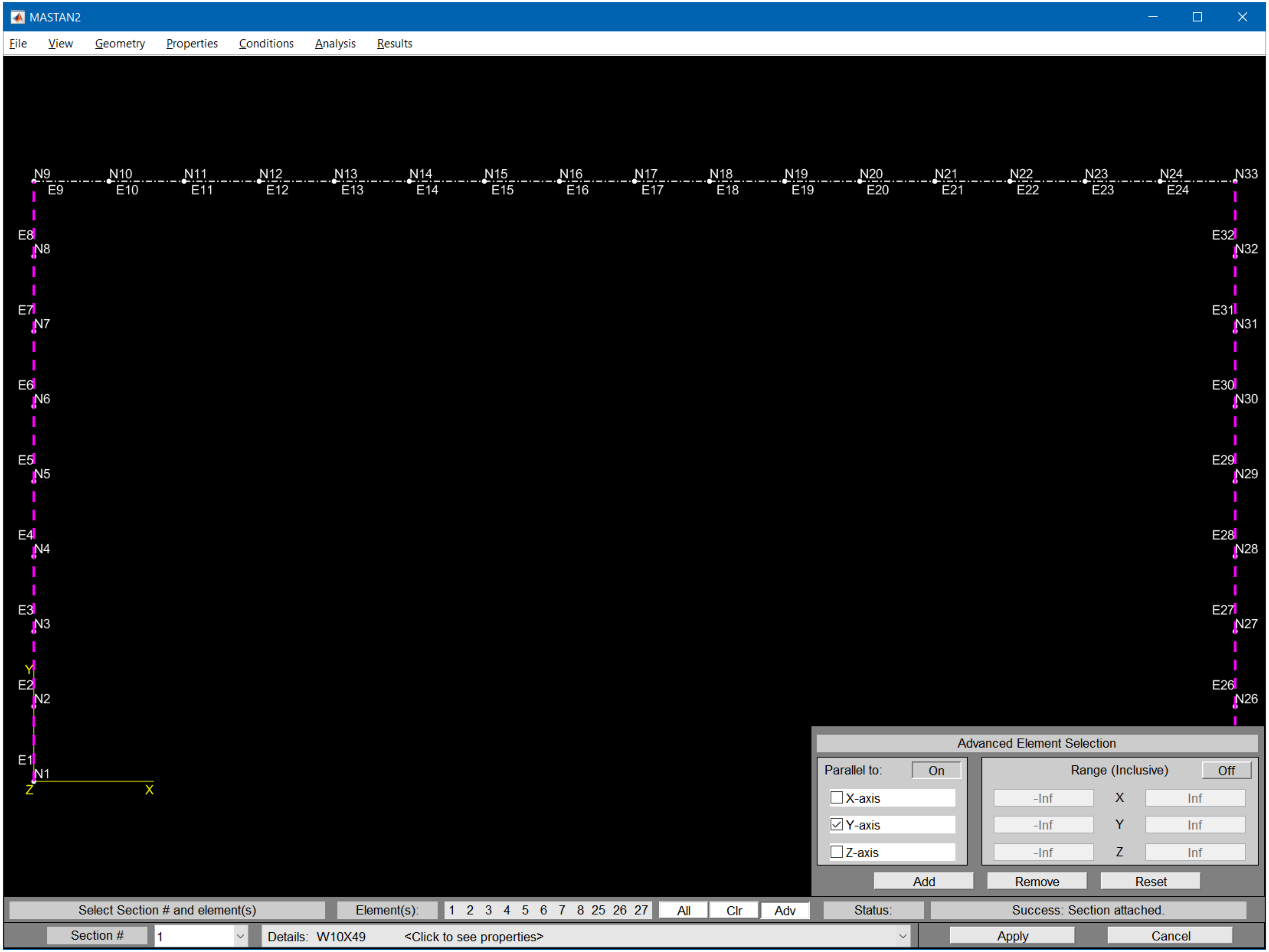
4) Click on the **Apply** button to assign Section 1. (Note that the element line style has changed from dash-dot to dashed.)

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

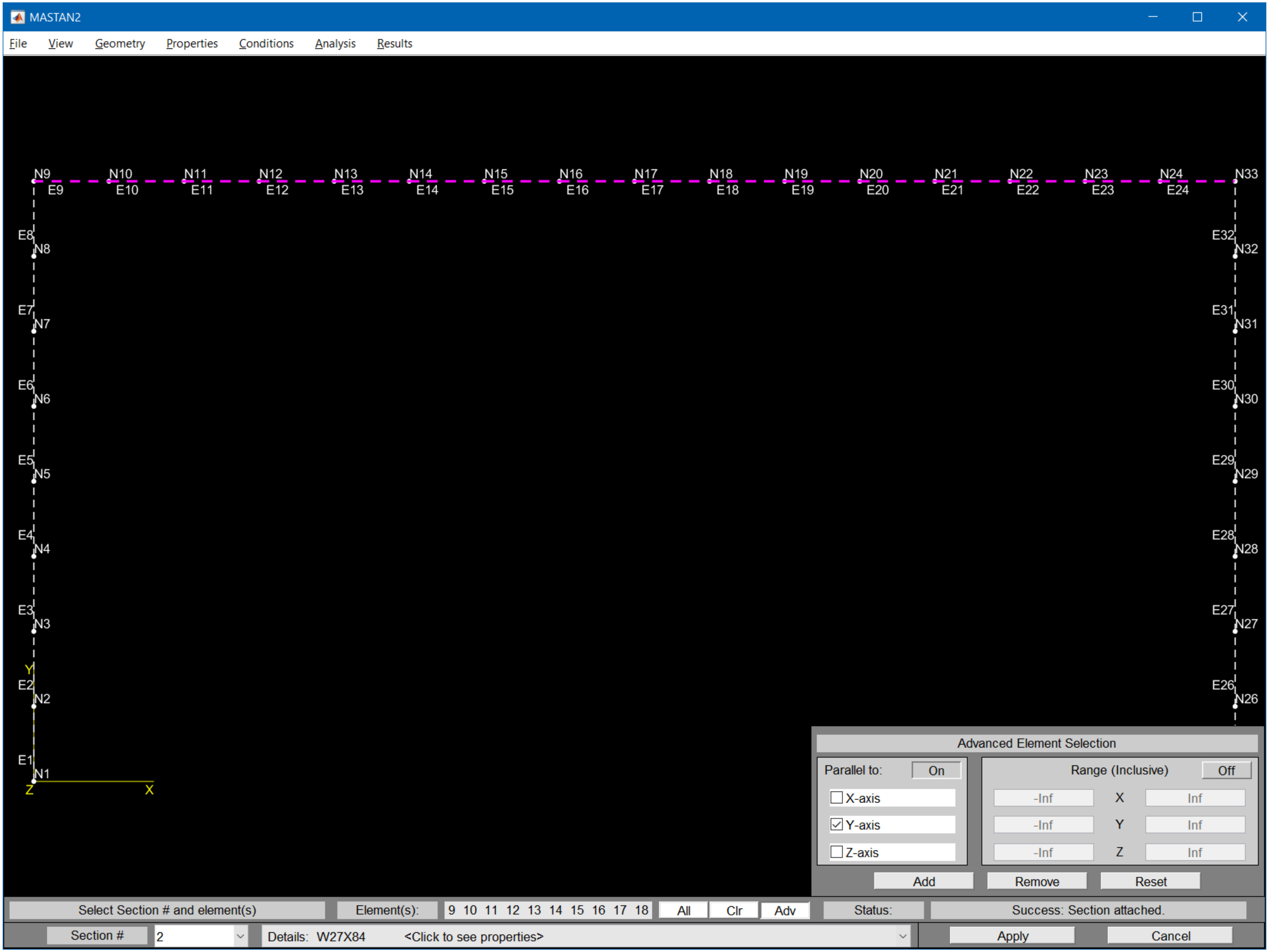
6) Create a list of the remaining elements by clicking the **All** button and then the **Remove** button in the pop-up menu. This should leave only the horizontal members selected.

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


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

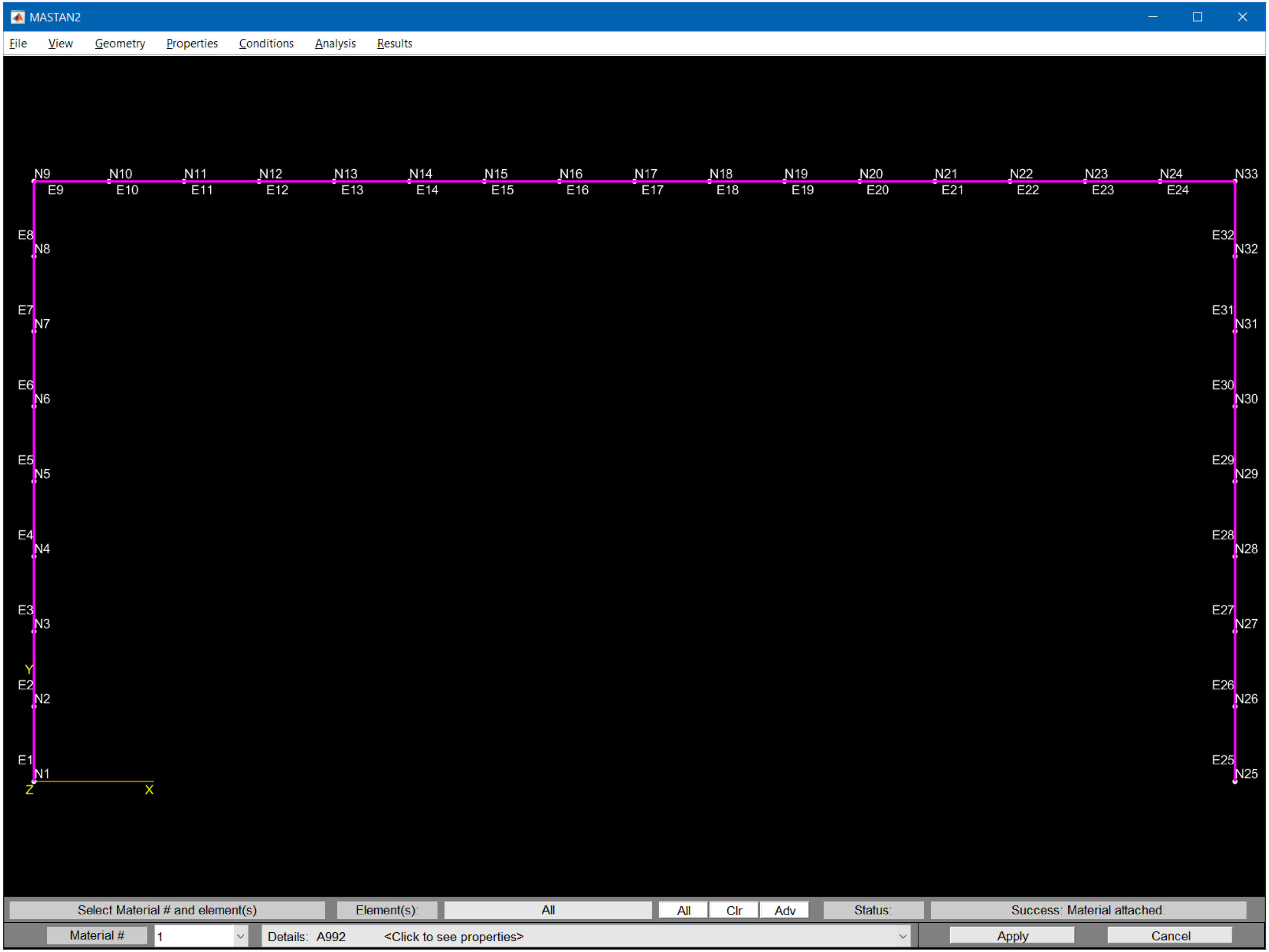
1) From the **Properties** menu select **Define Material**.

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
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 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**.

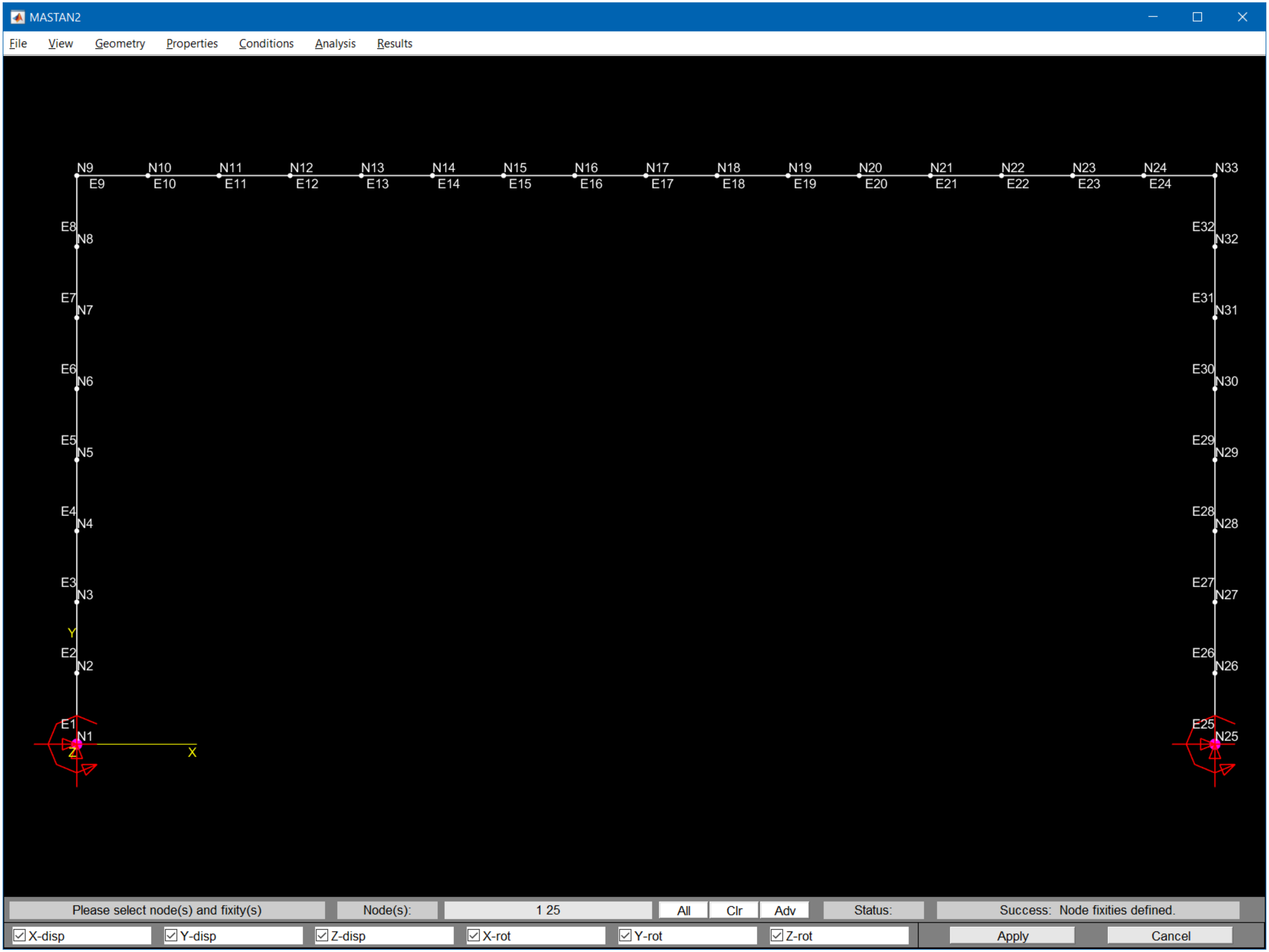
3) Create the list of nodes to be assigned these fixities by clicking on the bottom two nodes of the model, **1** and **25**.

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
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 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**.

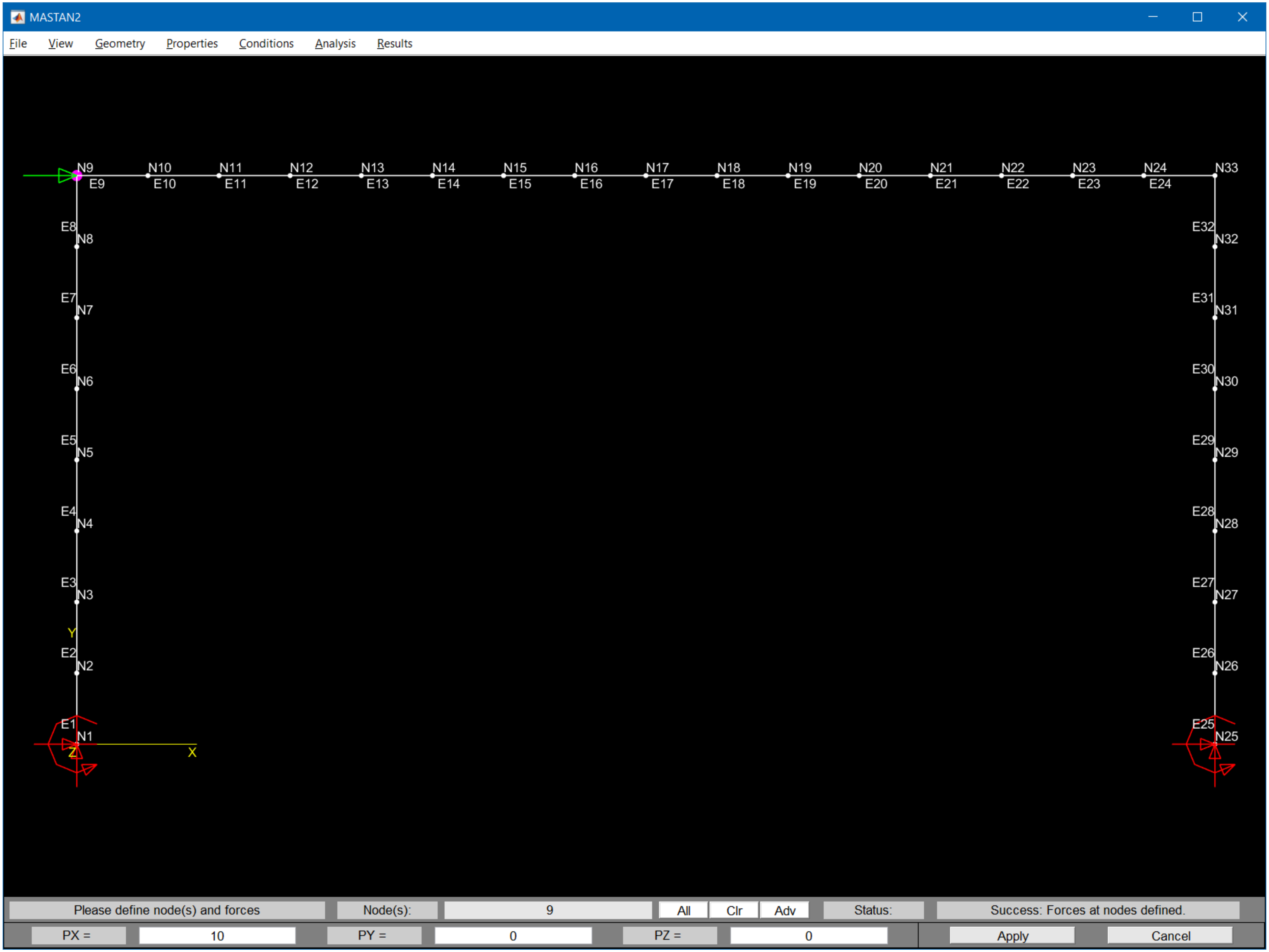
6) Since the loading input is already **Element(s) local x'-y'-z'**, click in the edit box just to the right of **wy' =** and change **0** to **-0.1**.

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.

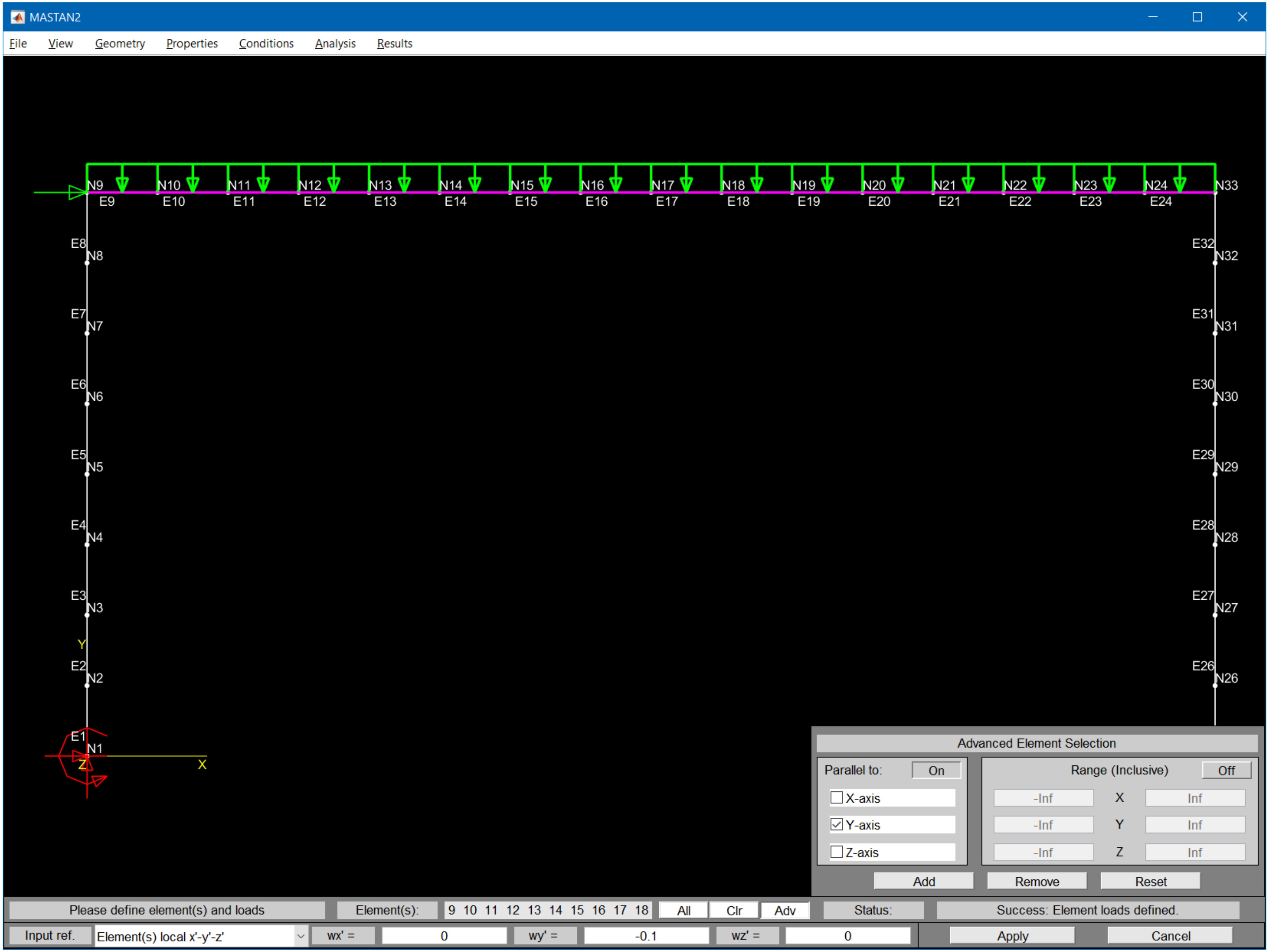
8) Click on the **Apply** button.

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

MASTAN2


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
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

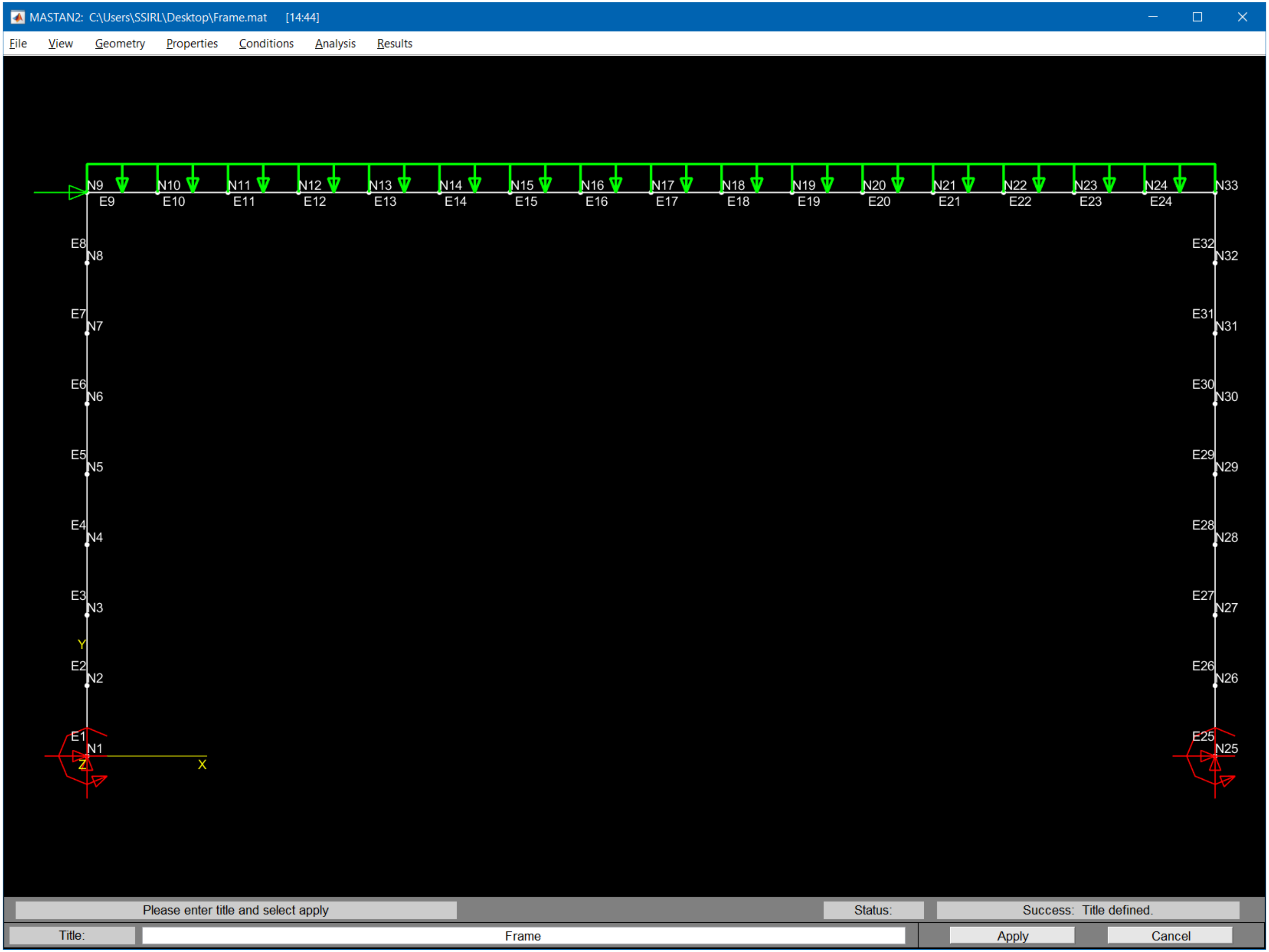
Naming and Saving

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
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**.

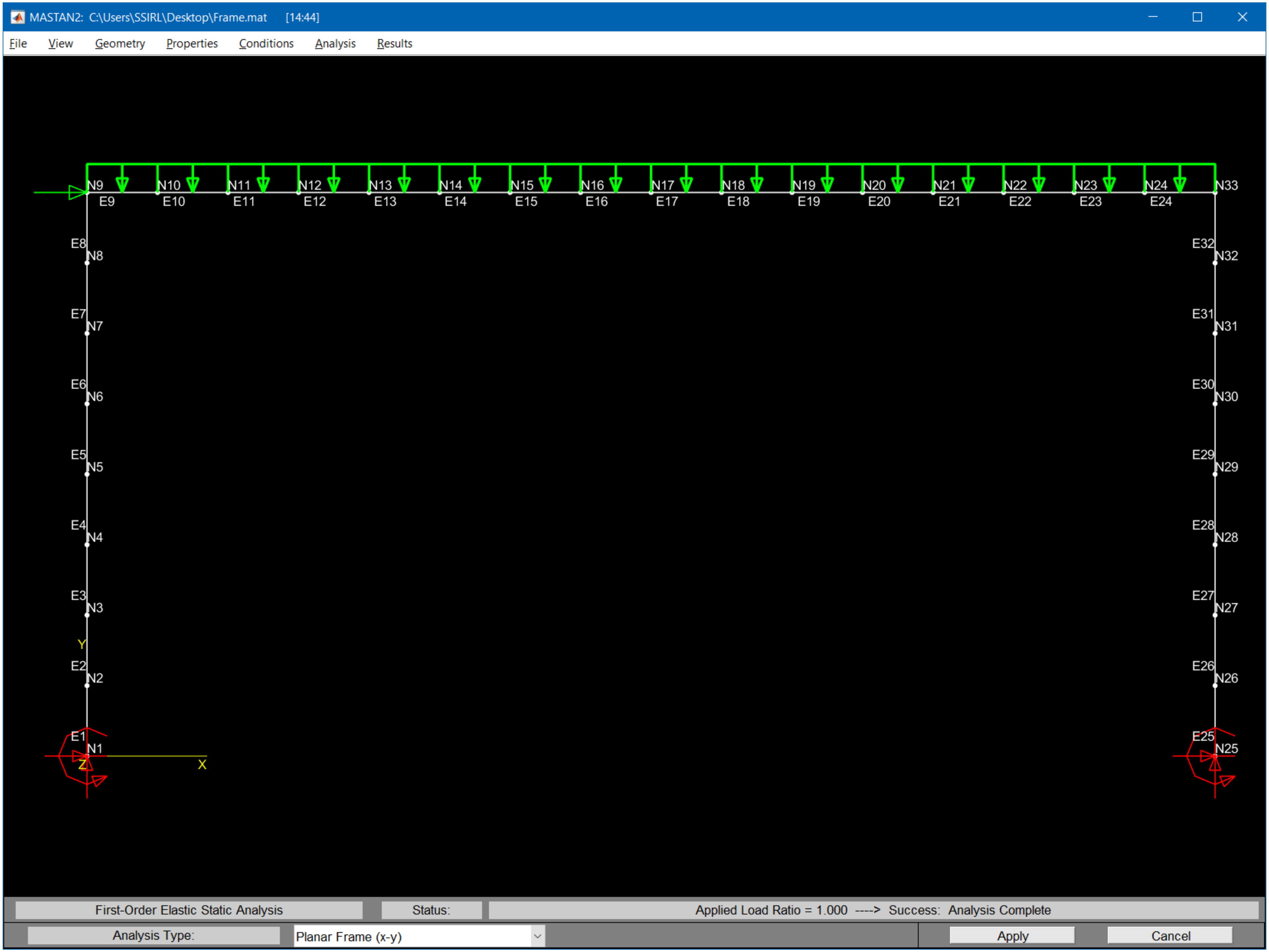
7) On the undeflected shape, click on the node of interest in the upper right corner, **33**, and its components are provided in the bottom menu bar.

Results:

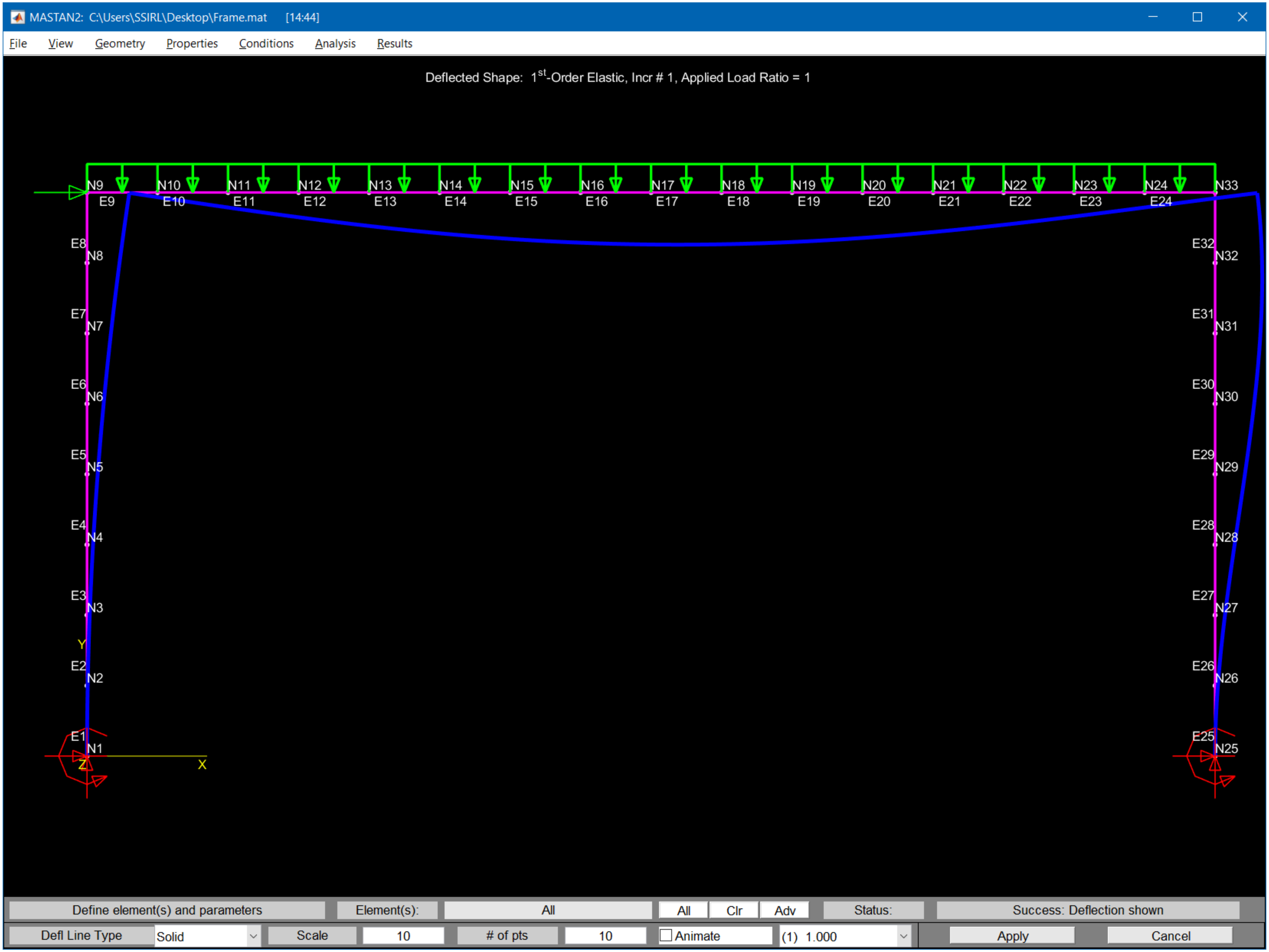
Disp X	Disp Y	Disp Z	Rot X	Rot Y	Rot Z
2.688	-0.03312	N/A	N/A	N/A	0.01235

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**.

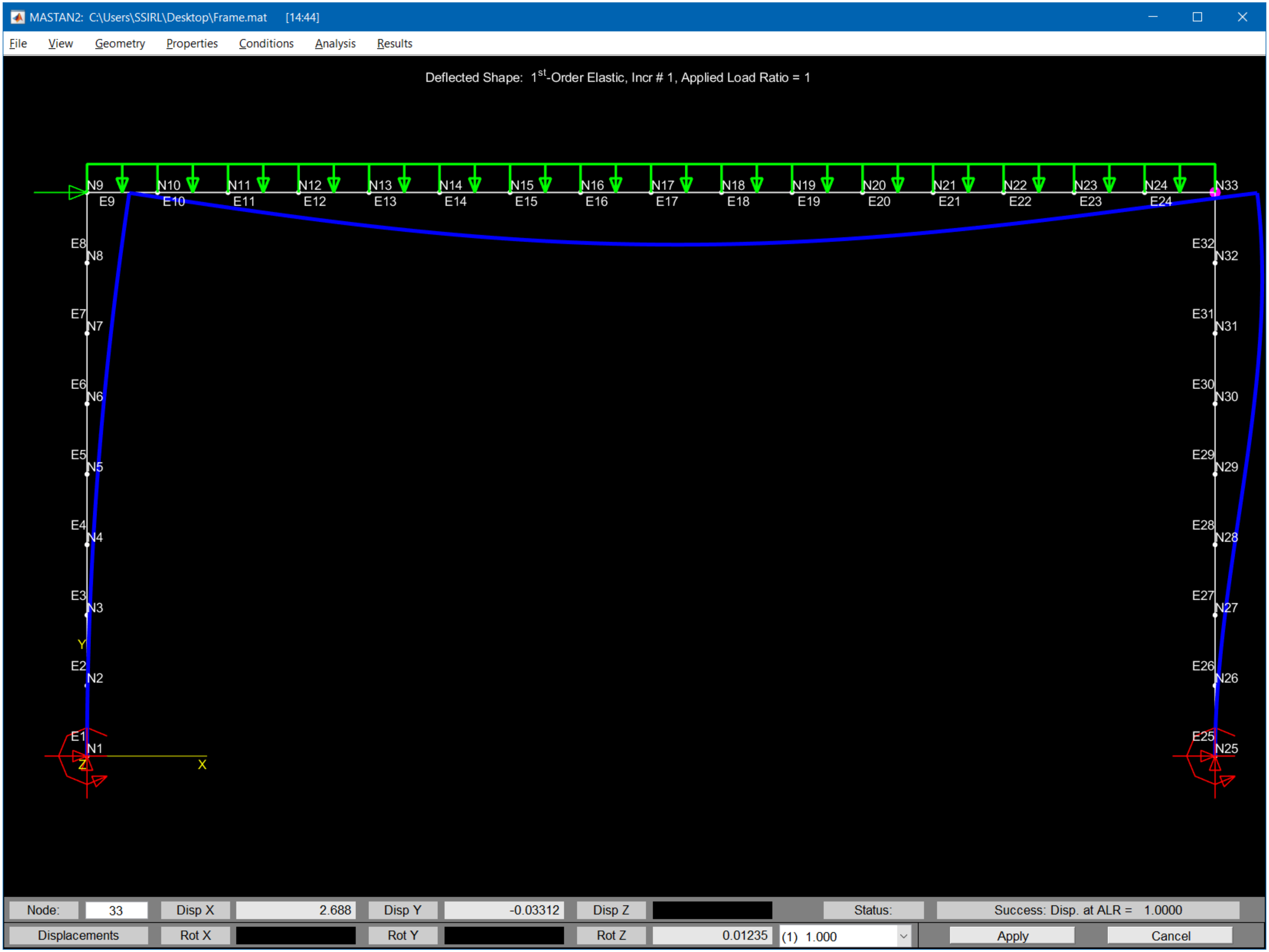
MASTAN2



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: 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
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
E1 E2 E3 E4 E5 E6 E7 E8
N1 N2 N3 N4 N5 N6 N7 N8
E25 E26 E27 E28 E29 E30 E31 E32
N25 N26 N27 N28 N29 N30 N31 N32
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)**.

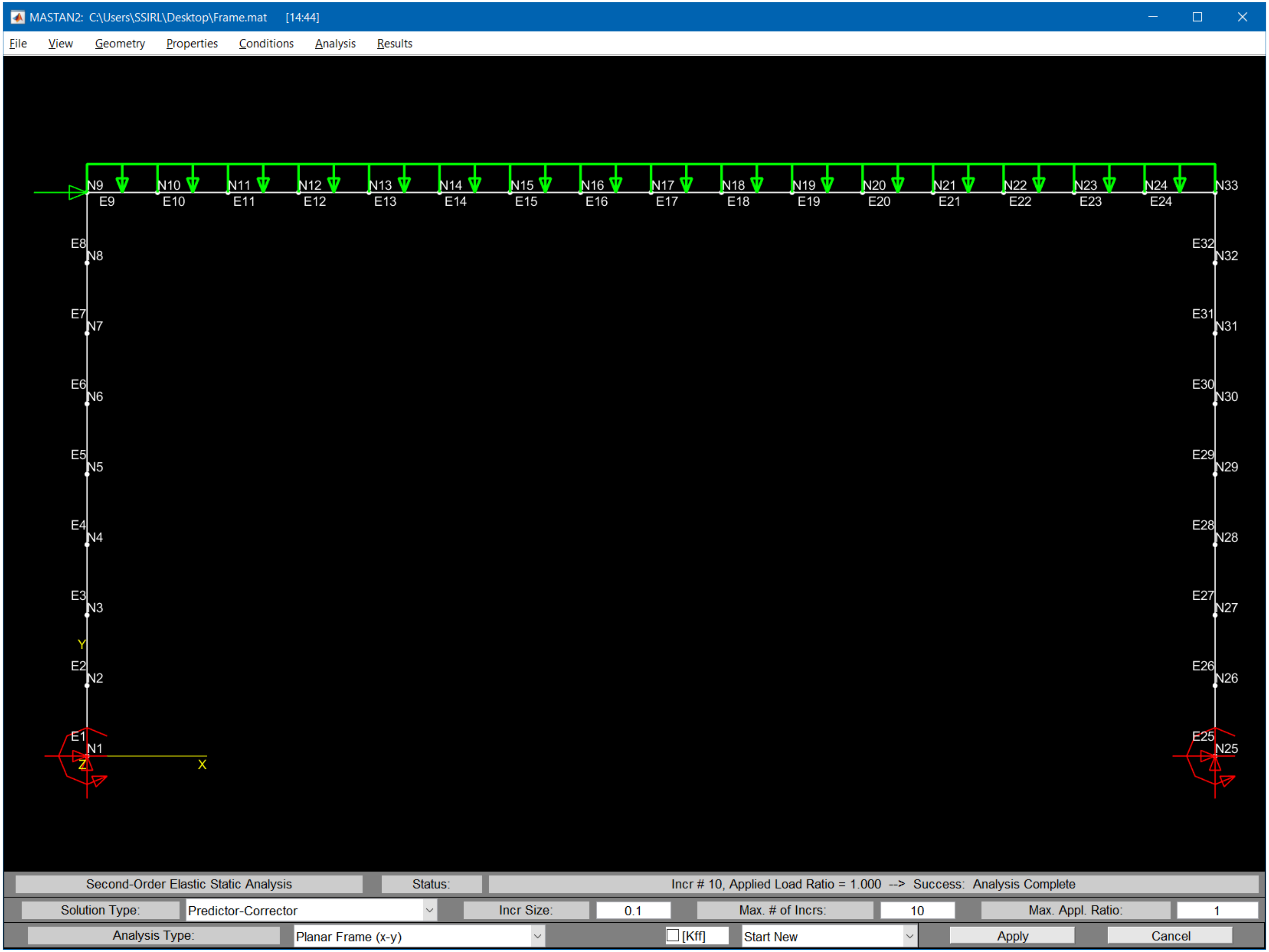
3) Click on the **Apply** button to perform the analysis.

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

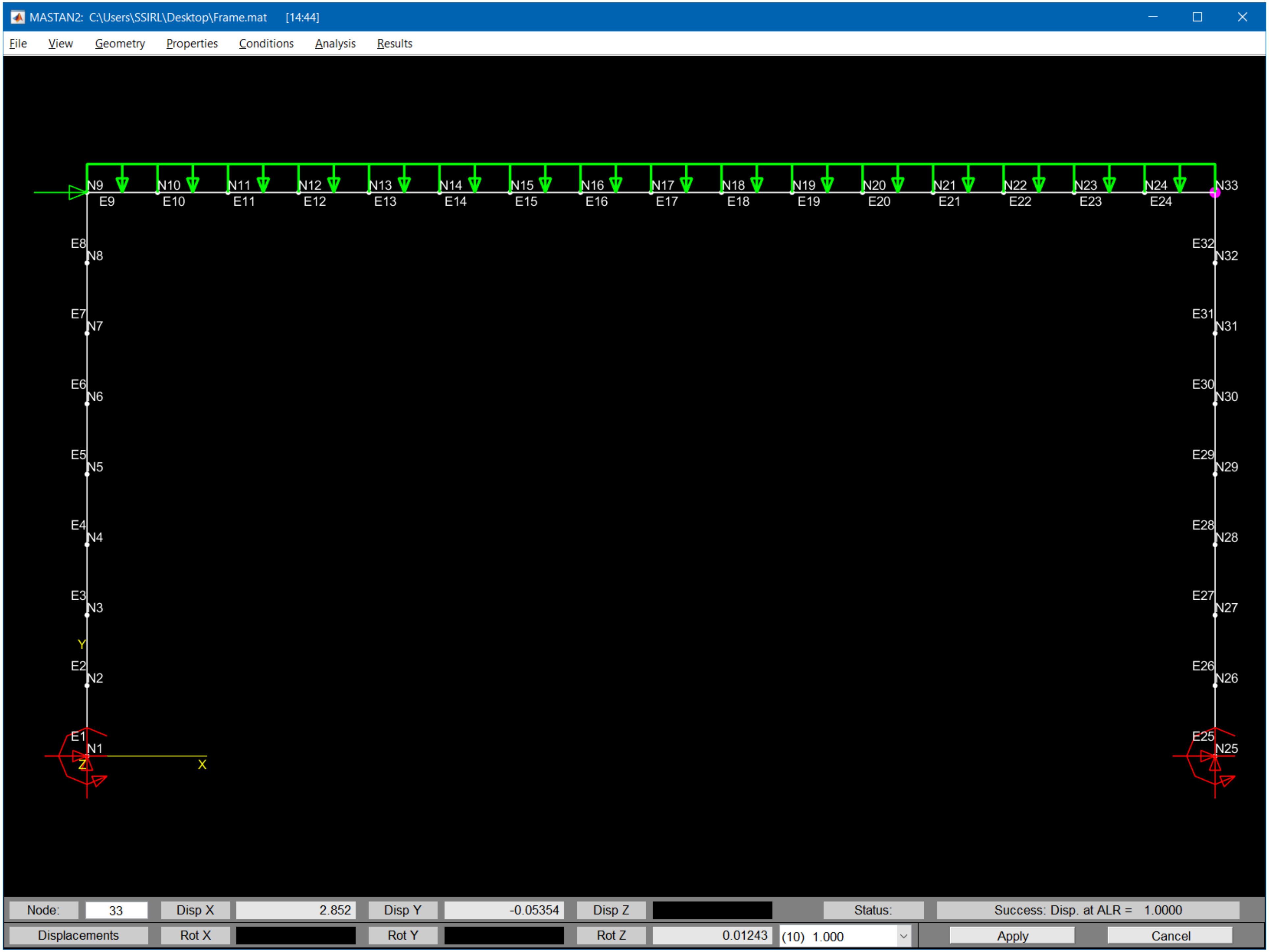
5) On the undeflected shape, click on the node of interest in the upper right corner, **33**, and its components are provided in the bottom menu bar.

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: 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 Y E2 N2 E1 N1 Z X
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26 E25 N25
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**.

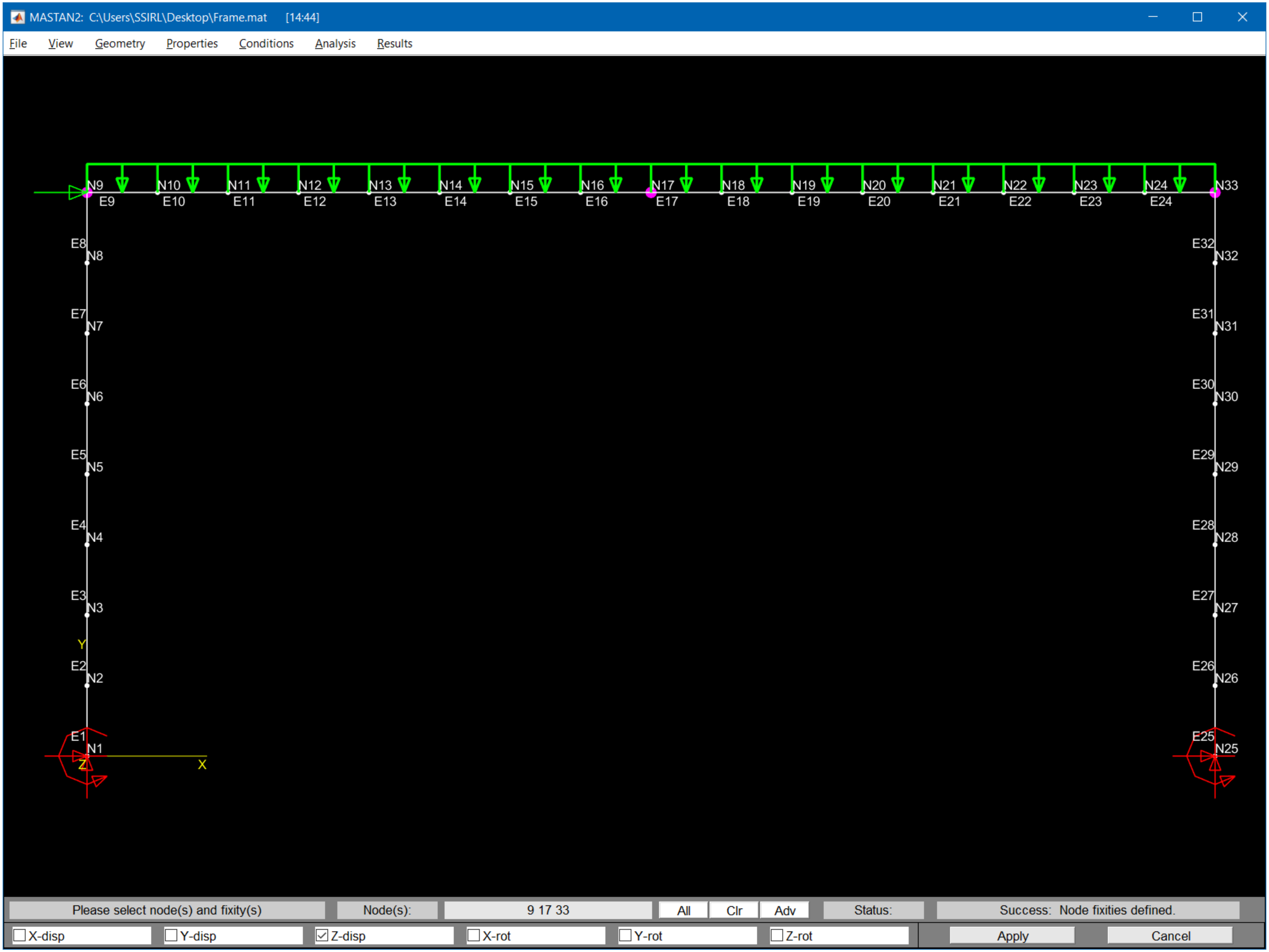
2) At the bottom menu bar, define the lateral support by clicking in the **check box** to the left of **Z-disp**.

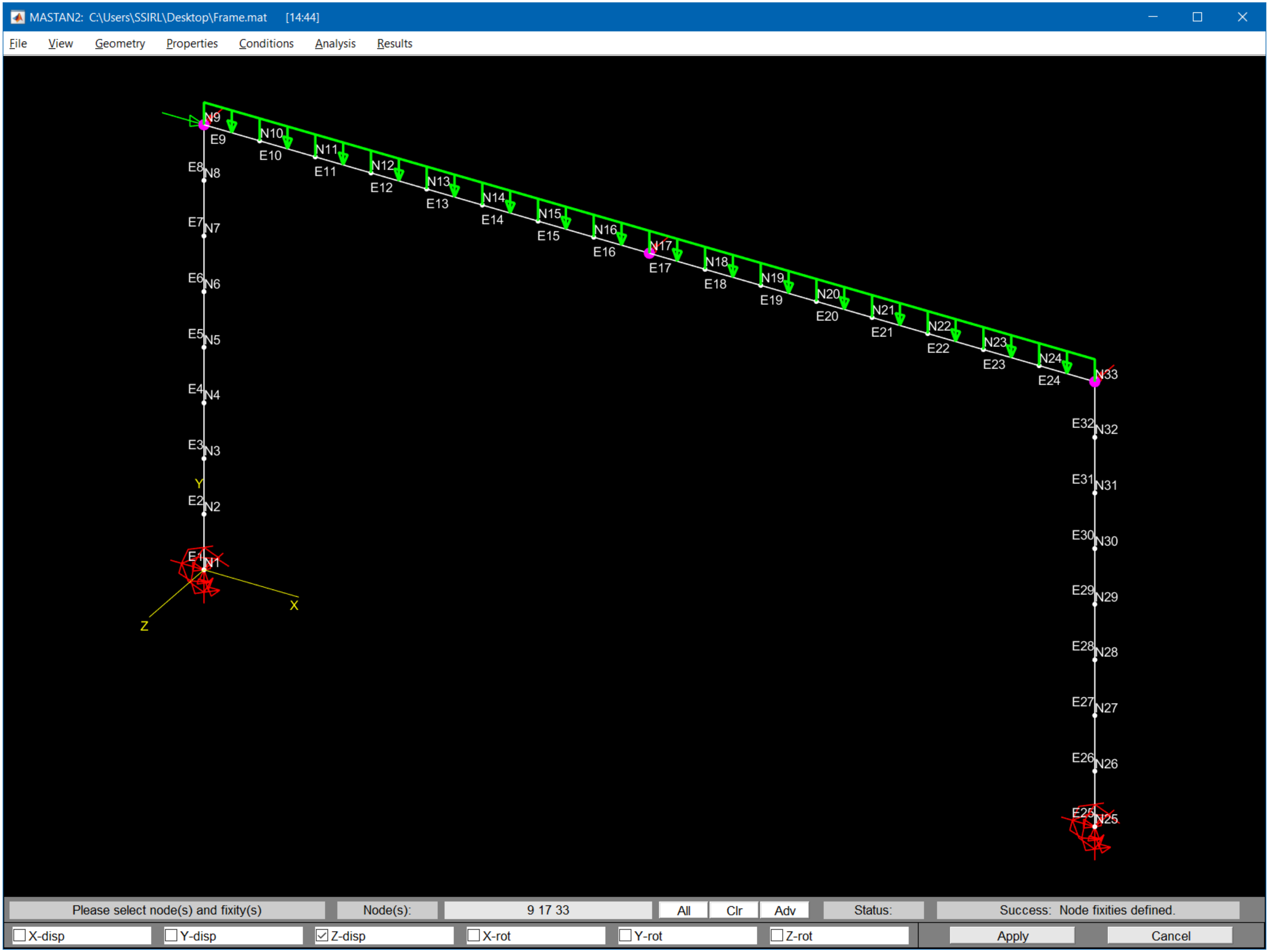
3) Create the list of nodes to be assigned these fixities by clicking on the top corner and middle nodes of the model: **9**, **17**, and **33**.

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

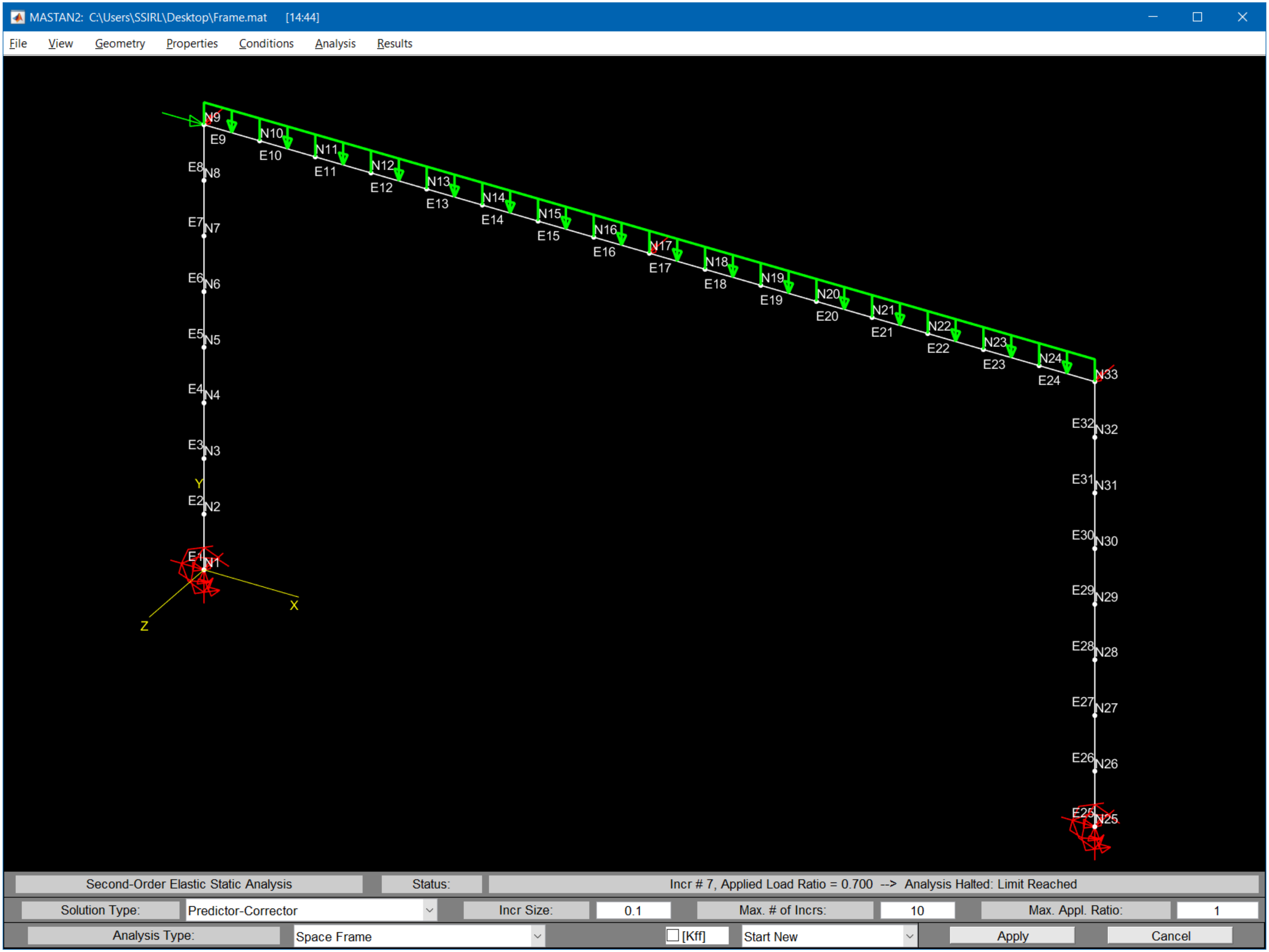
1) From the **Analysis** menu select **Static** and submenu option **2nd-Order Elastic**.

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
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.

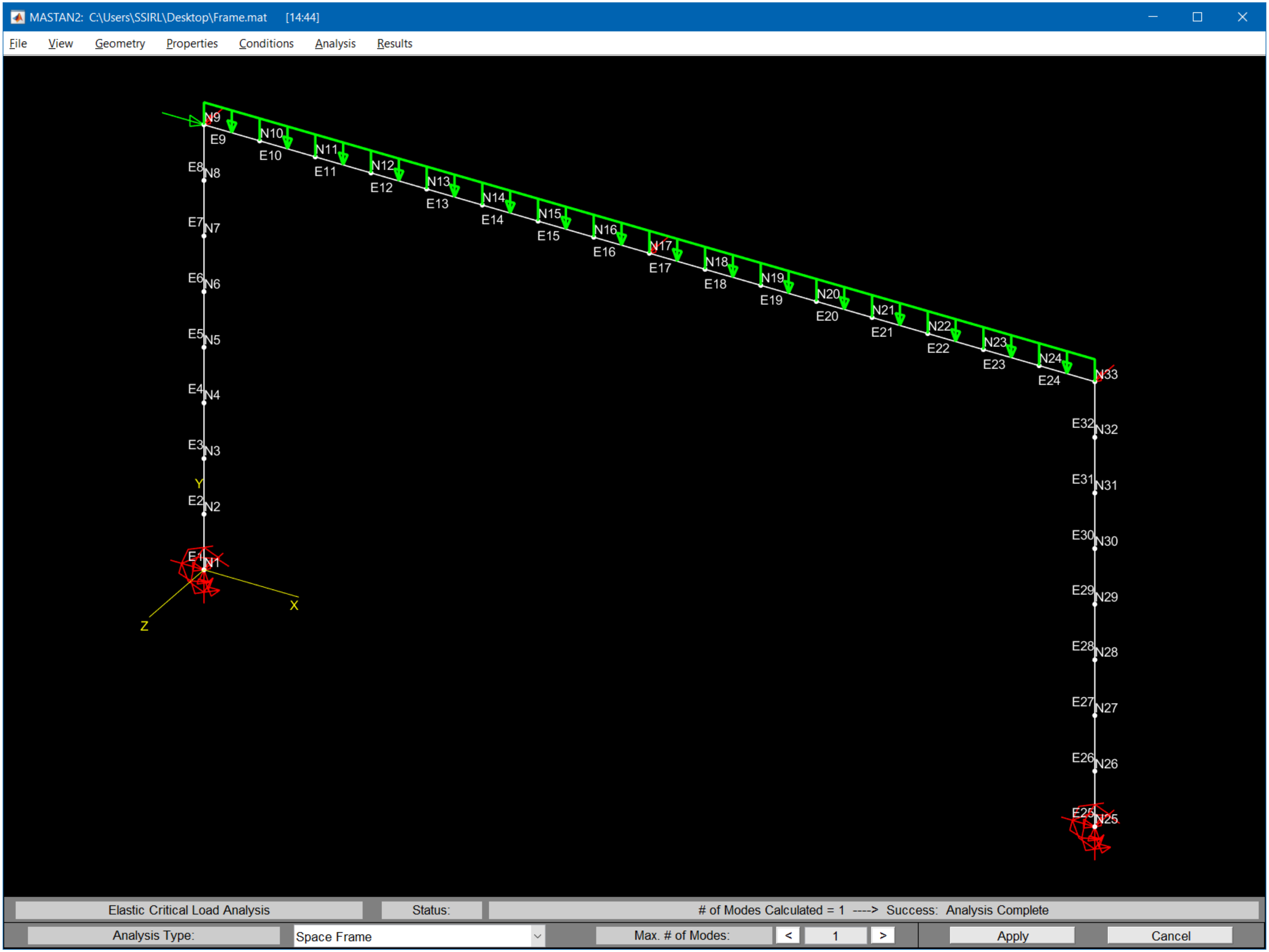
3) Click on the **Apply** button to perform the analysis.

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

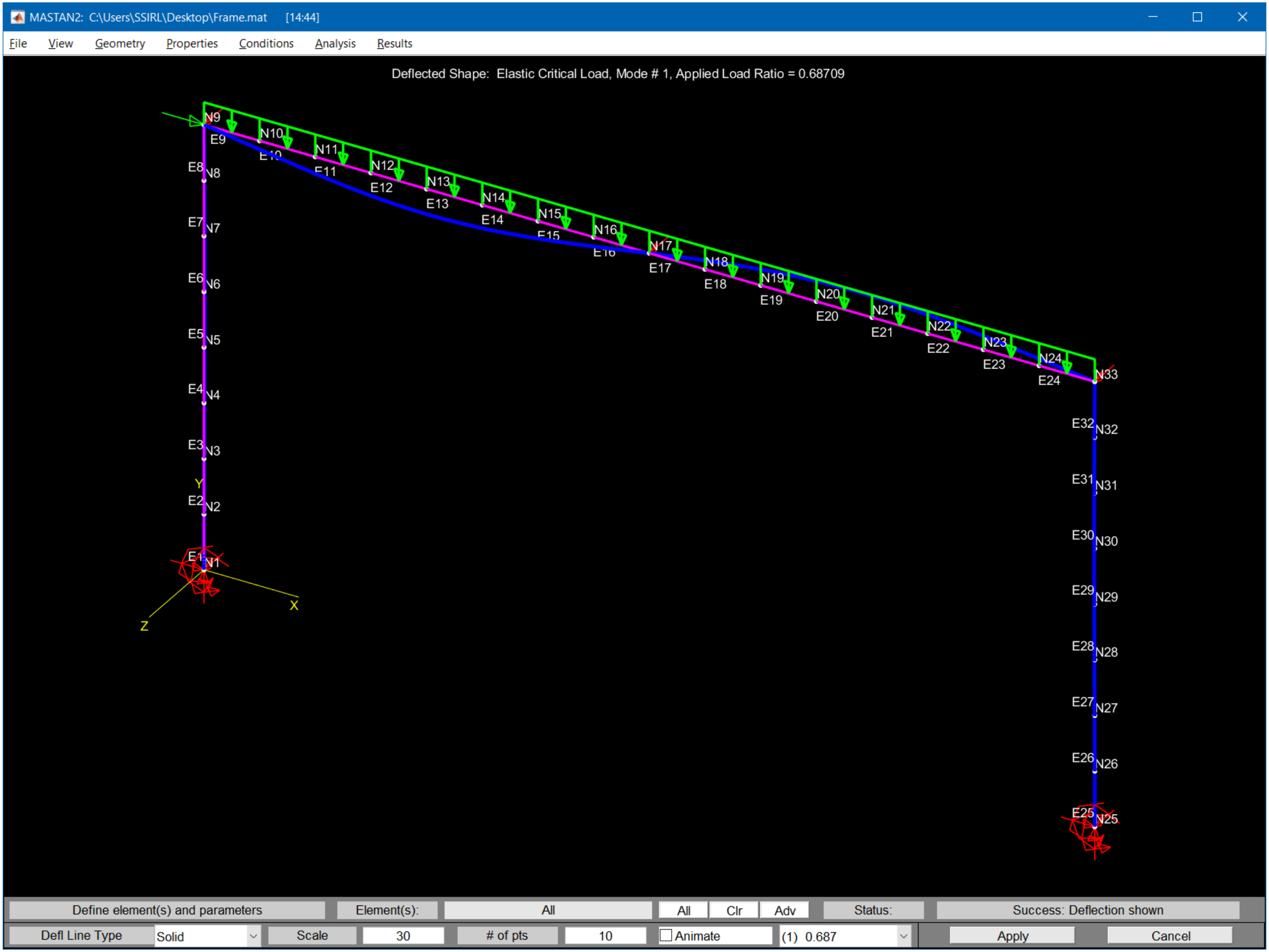
5) At the bottom menu bar, click the edit box to the right of **Scale**. Change **10** to **30** to amplify the deformed geometry in the visualization.

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: 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


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.

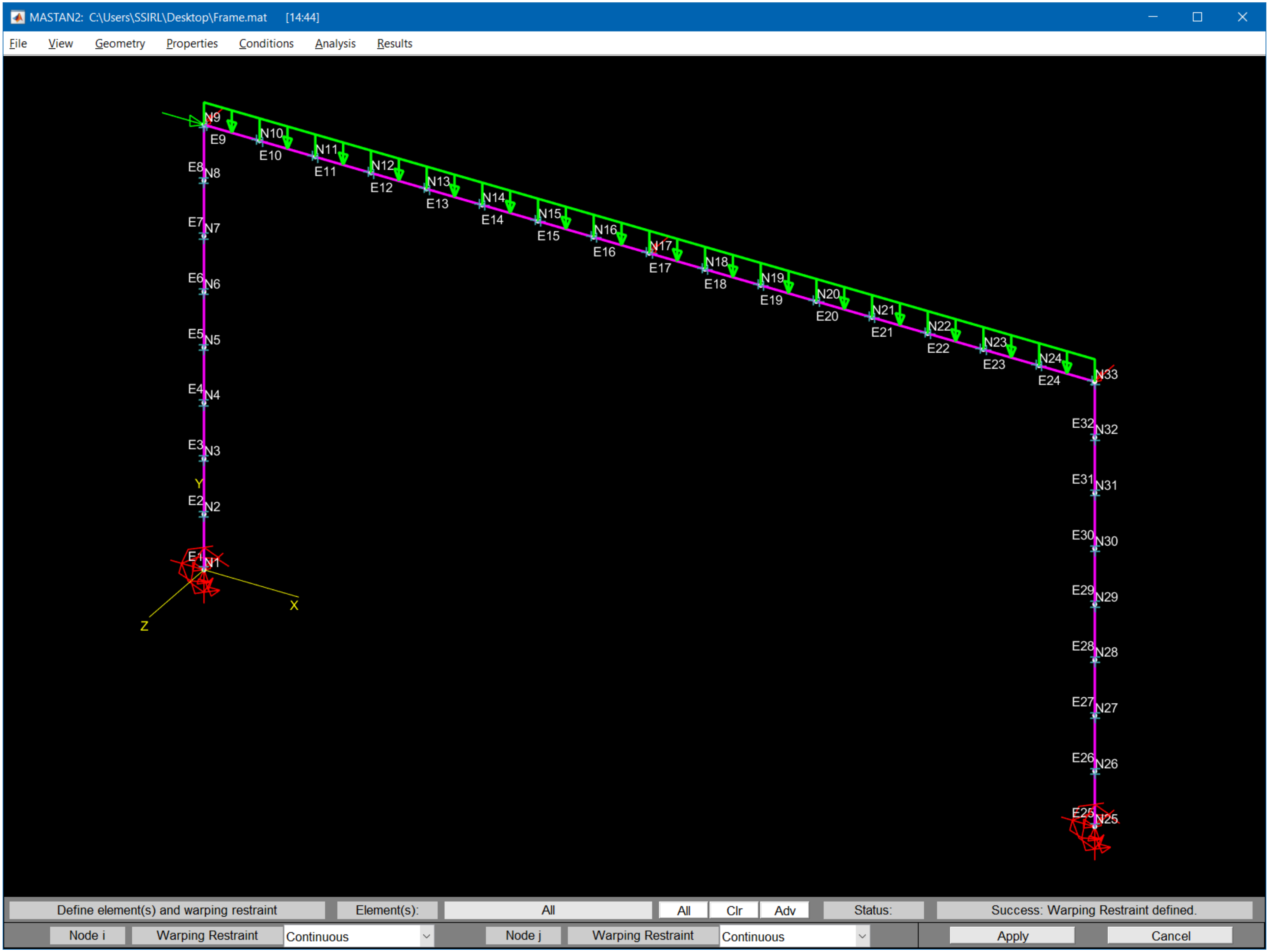
4) Click **Clr** to empty the list of elements. Click on the bottom element of each column and left end element of the beam to define the members that start with warping fixed and are continuous.

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.

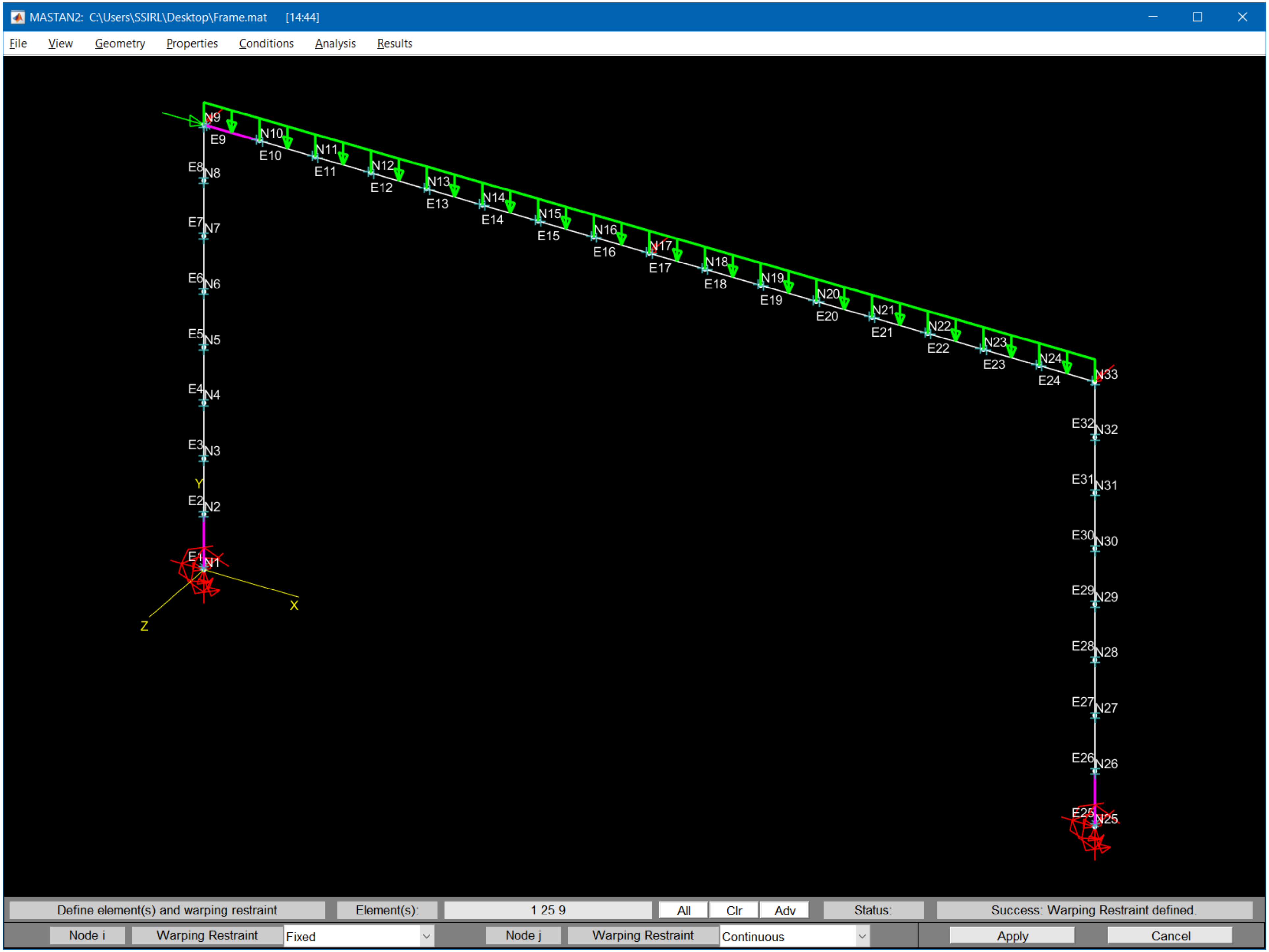
6) Click **Clr** to empty the list of elements. Click on the top element of each column and right end element of the beam.

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**.

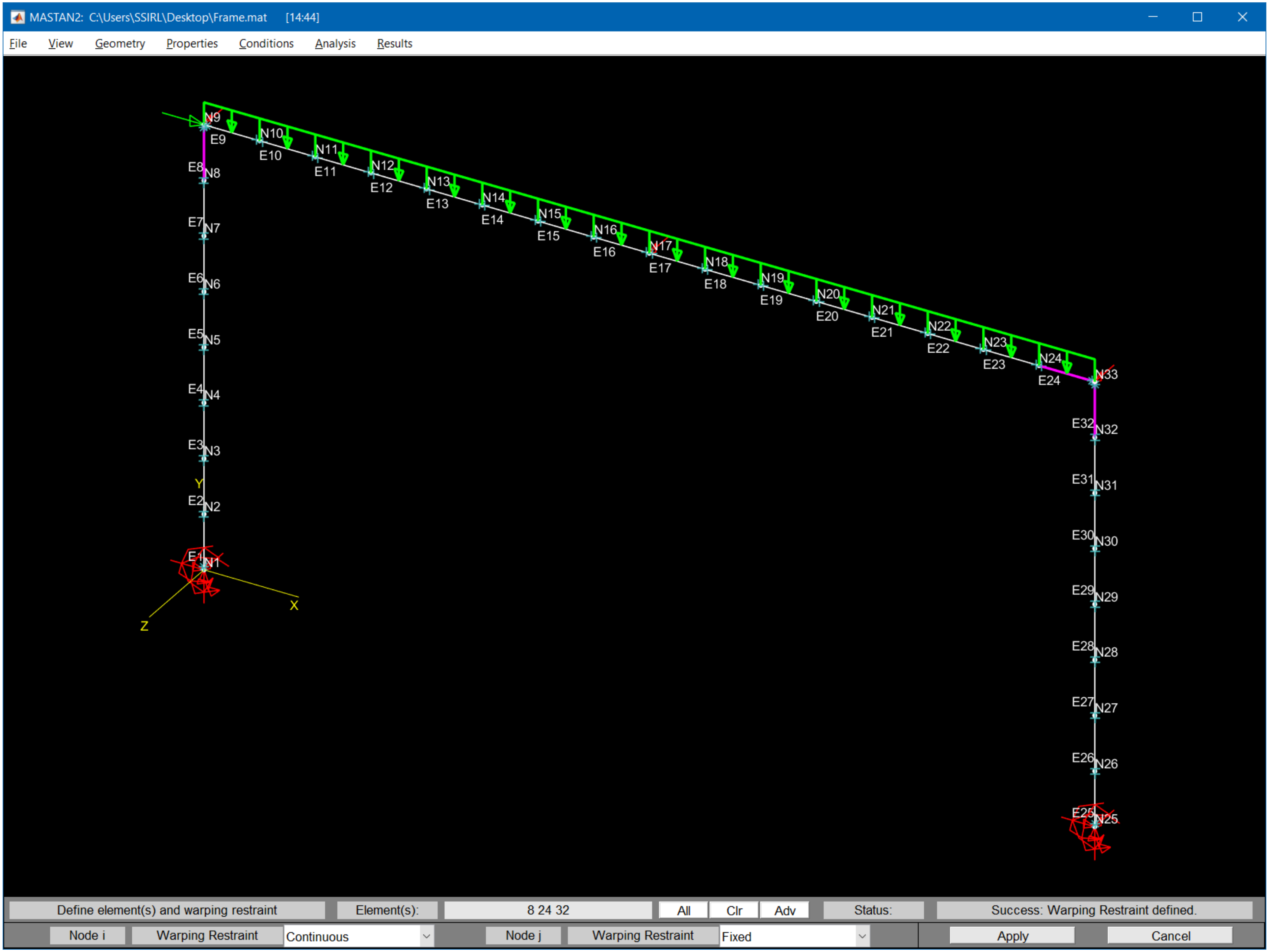
8) Click on the **Apply** button.


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
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
X
Z
E32 N32
E31 N31
E30 N30
E29 N29
E28 N28
E27 N27
E26 N26
E25 N25
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



MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File View Geometry Properties Conditions Analysis Results
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

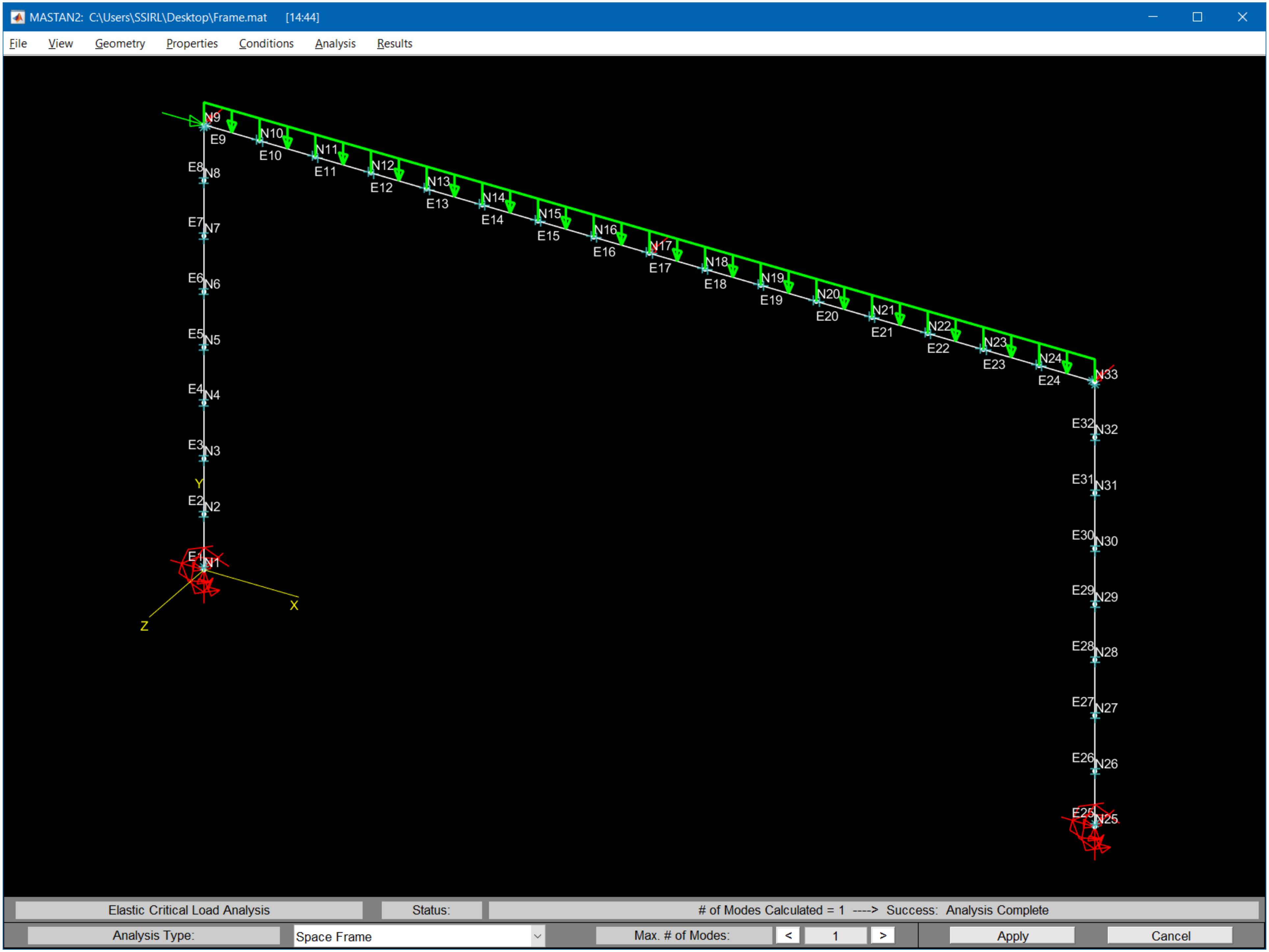

MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
N9
N10
N11
N12
N13
N14
N15
N16
N17
N18
N19
N20
N21
N22
N23
N24
N33
E9
E10
E11
E12
E13
E14
E15
E16
E17
E18
E19
E20
E21
E22
E23
E24
E8
N8
E7
N7
E6
N6
E5
N5
E4
N4
E3
N3
Y
E2
N2
E1
N1
X
Z
E32
N32
E31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

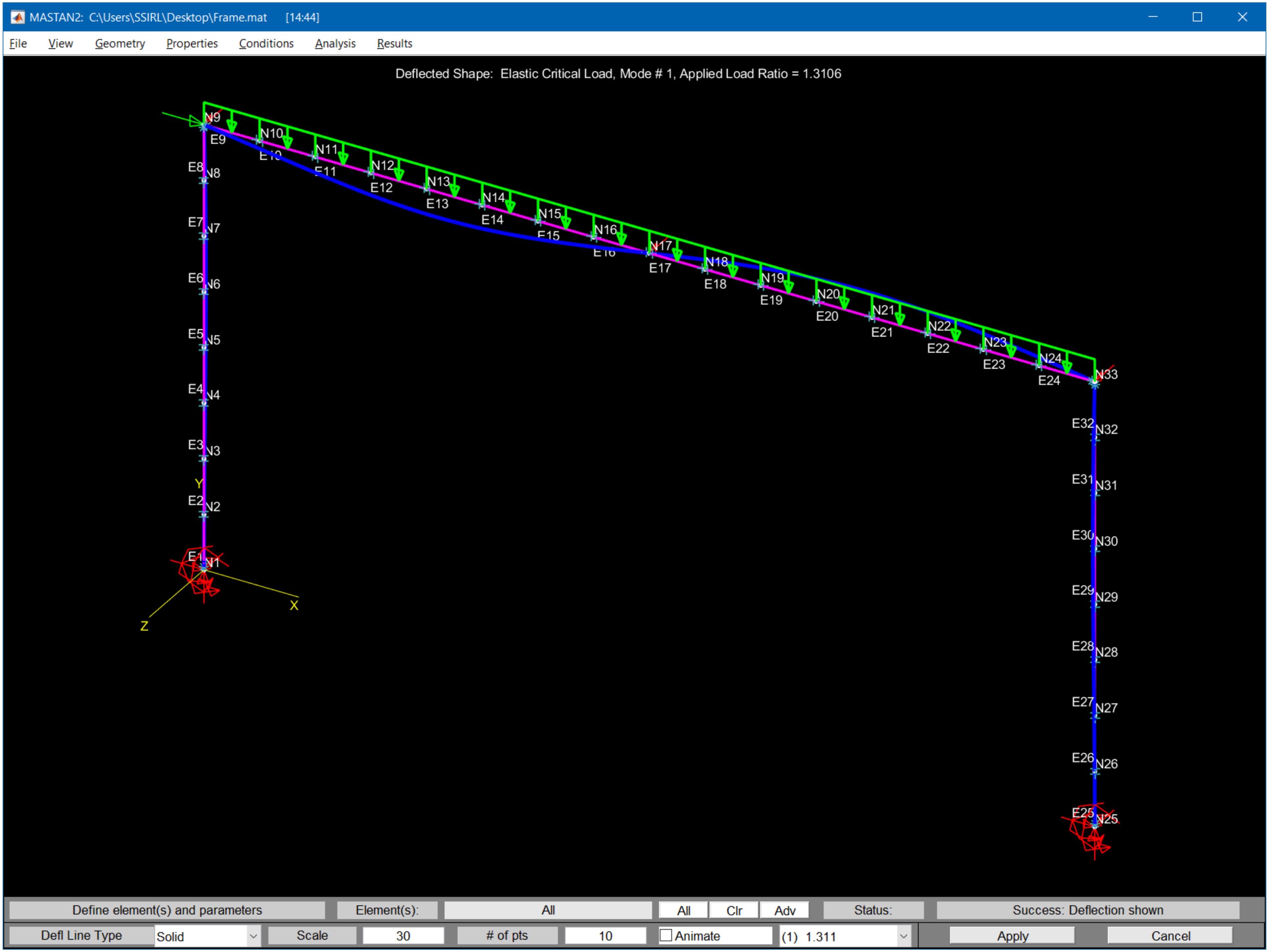
1) From the **Analysis** menu select **Eigen-Buckling** and submenu option **Elastic Critical Load**.
2) At the bottom menu bar, the **Analysis Type:** should already be set to **Space Frame** with the **Max. # of Modes:** set to **1** as desired.
3) Click on the **Apply** button to perform the analysis.
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
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
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**.

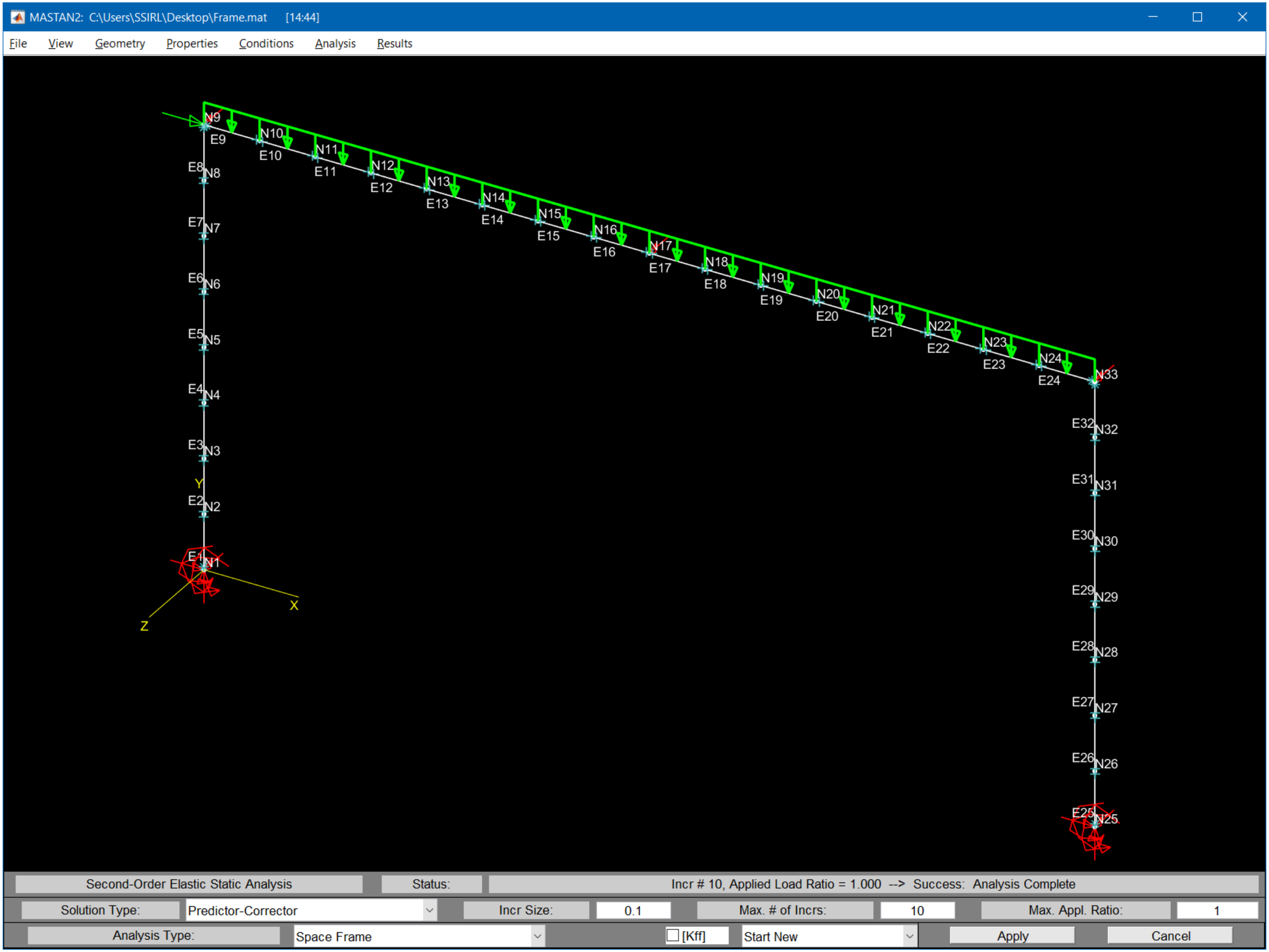
5) On the undeflected shape, click on the node of interest in the upper right corner, **33**, and its components are provided in the bottom menu bar.

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
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
Y
E2 N2
E1 N1
X
Z
E32 N32
E31 N31
E30 N30
E29 N29
E28 N28
E27 N27
E26 N26
E25 N25
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

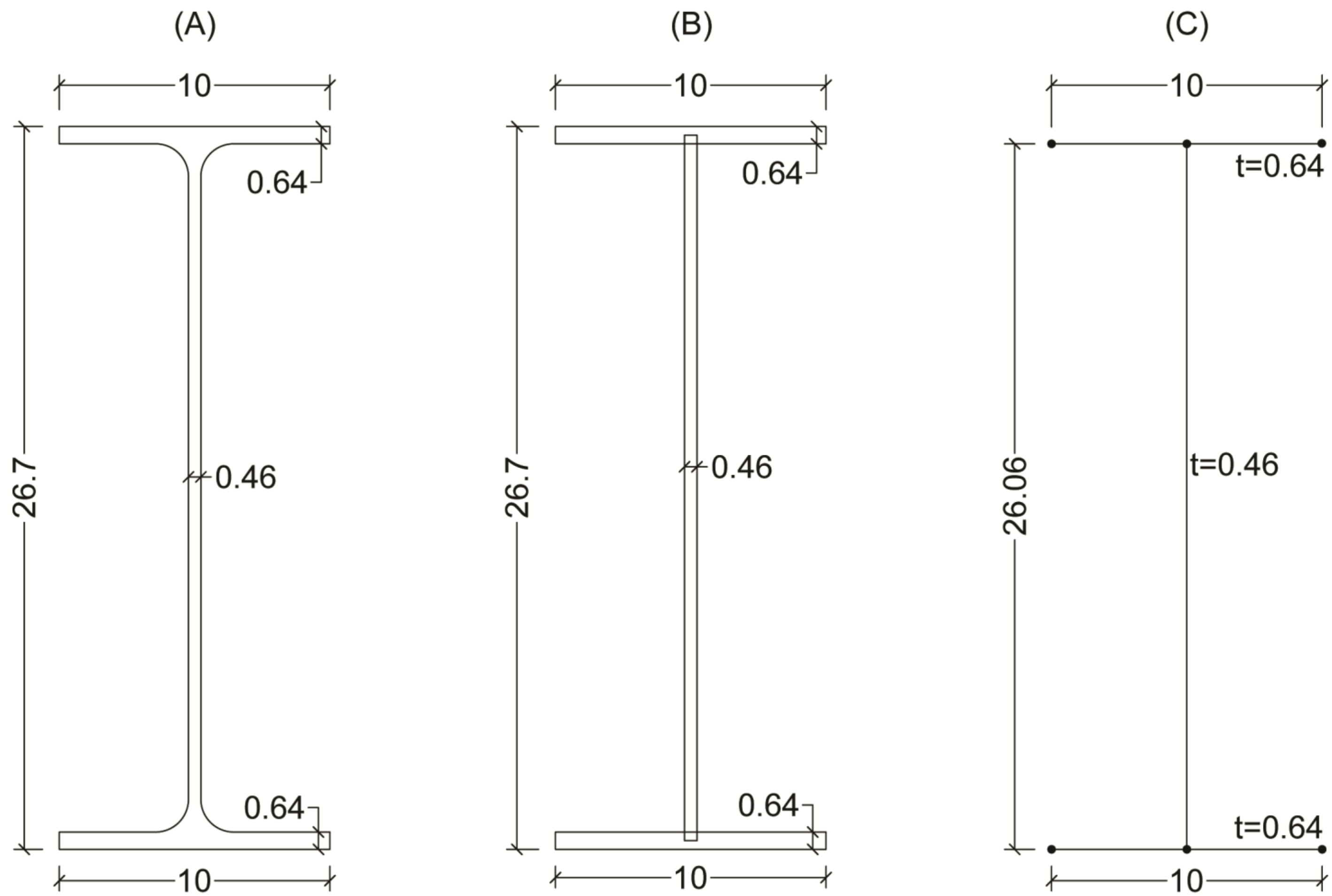
Section 5: Using MSASect

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.

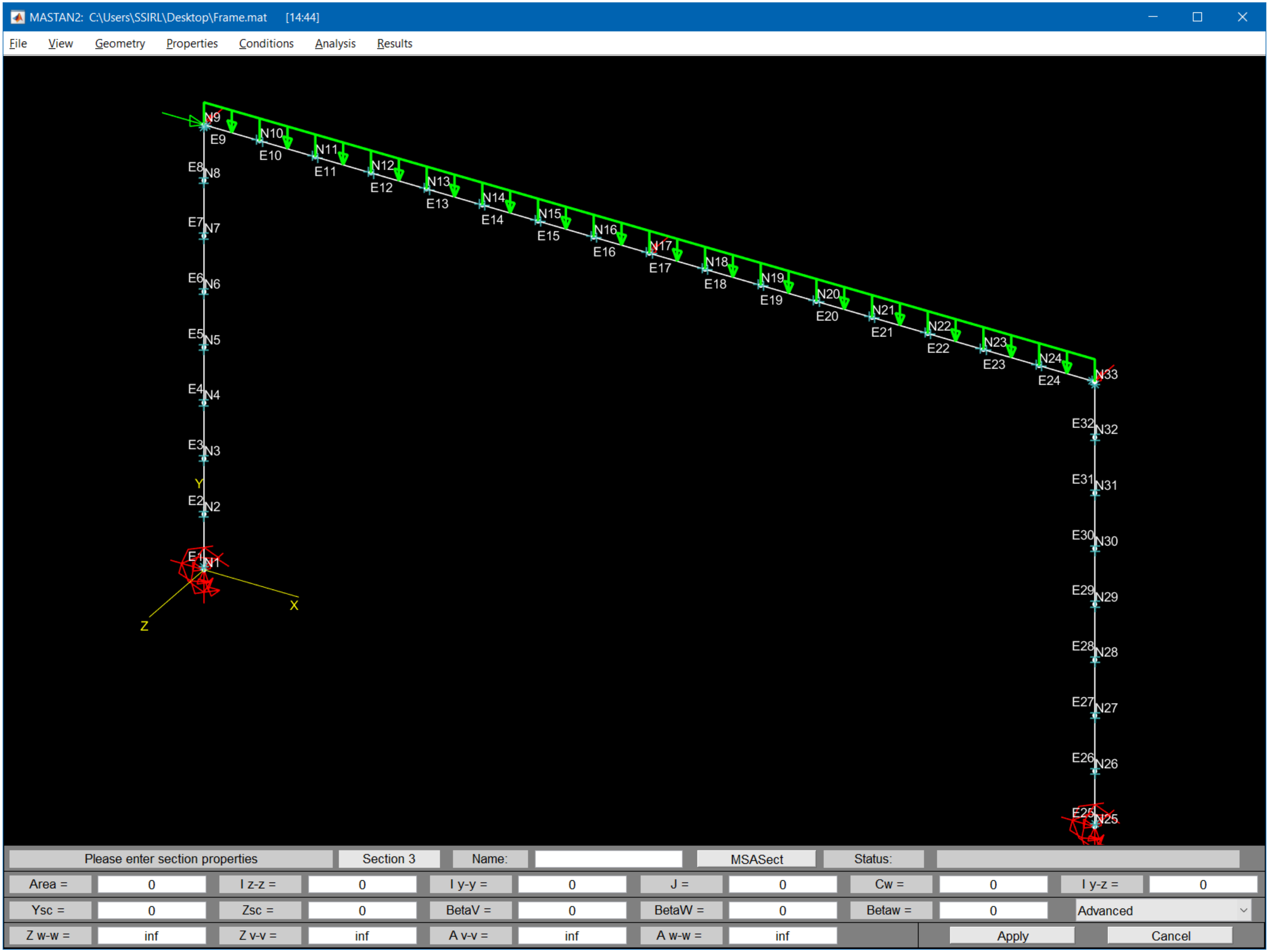
5) Click **Calculate** to determine the properties.

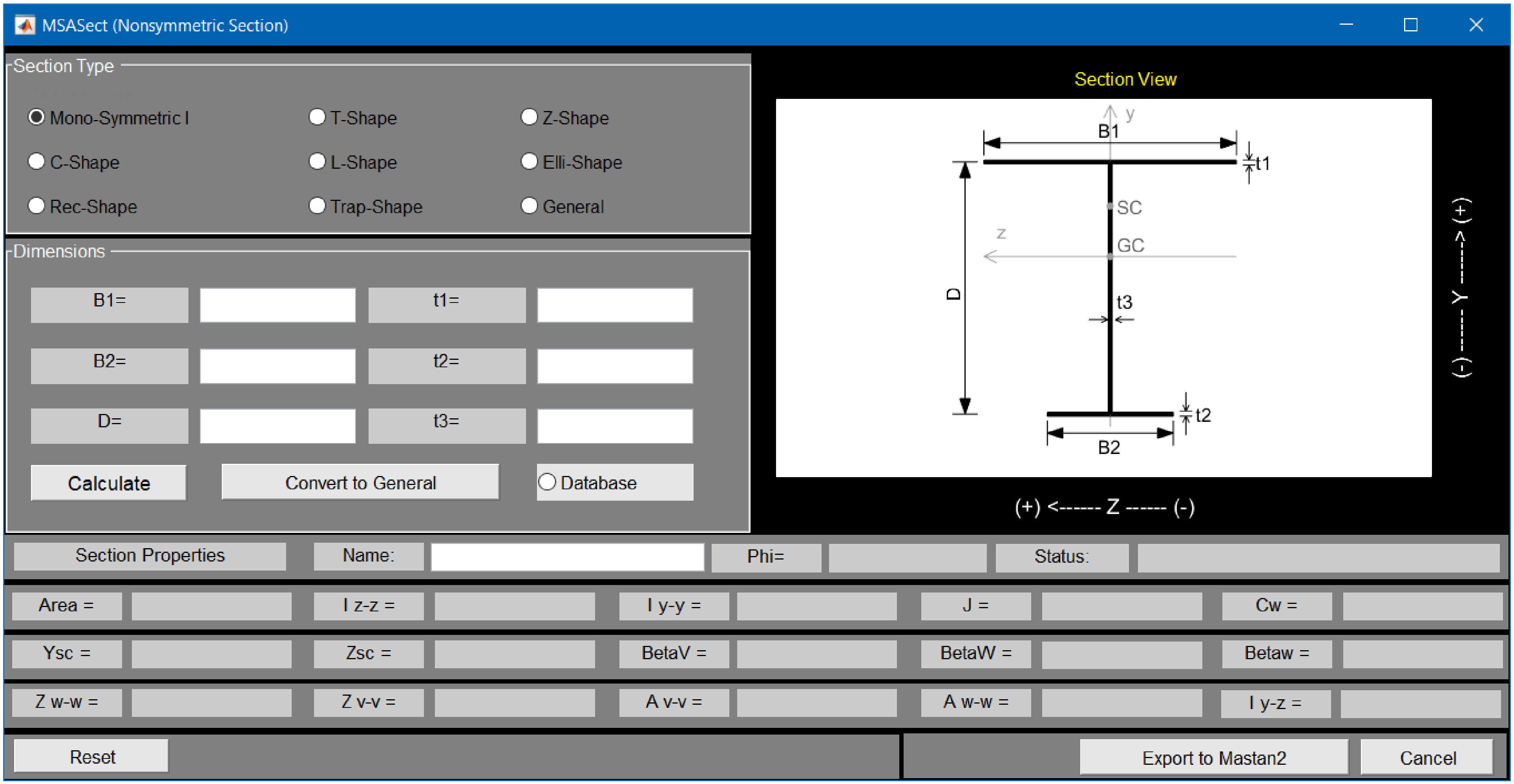
6) Click edit box to right of **Name:** and enter **W27x84Hand**.

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

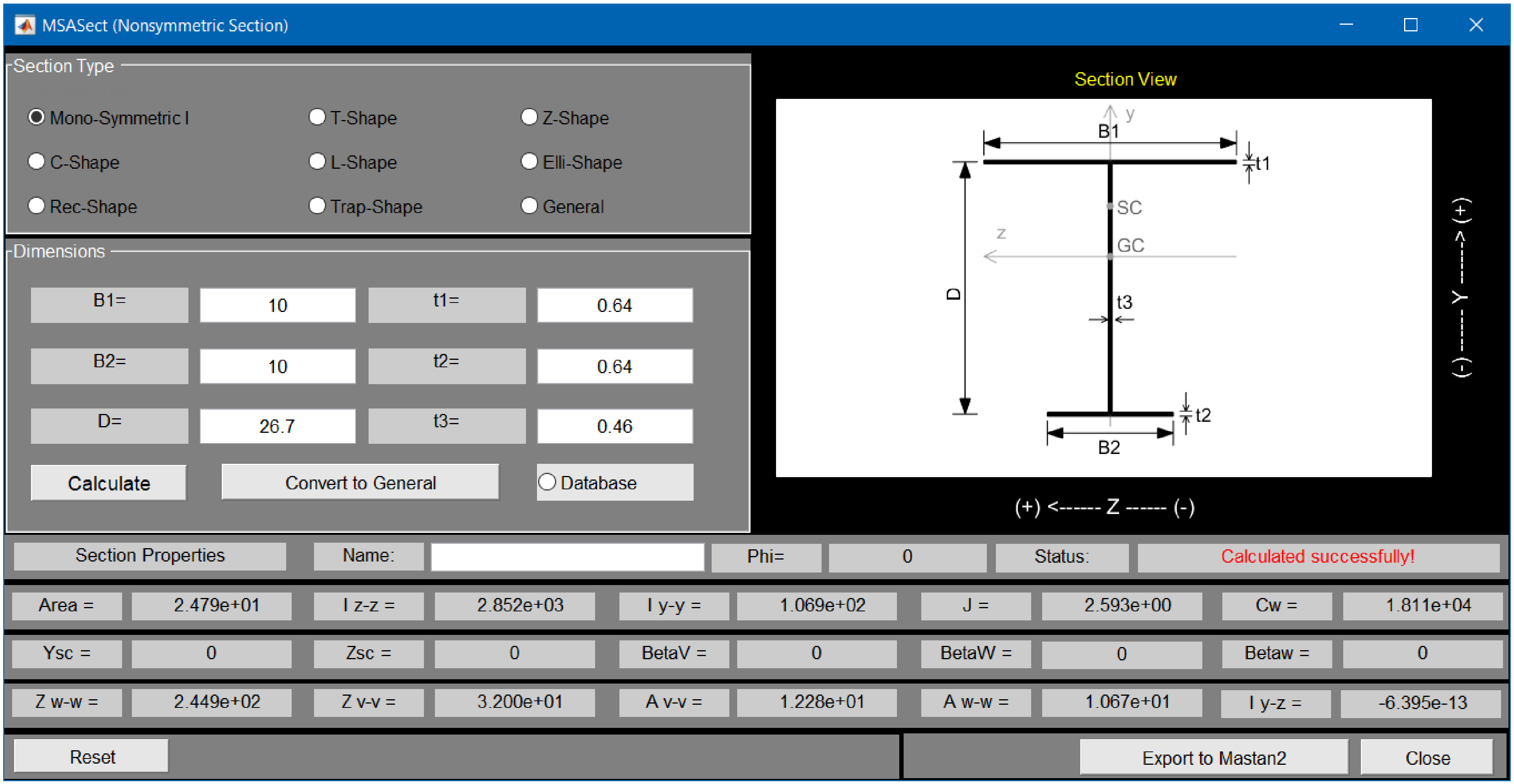
8) Click **Close** to return to the main window. There will often be a confirmation when closing it.

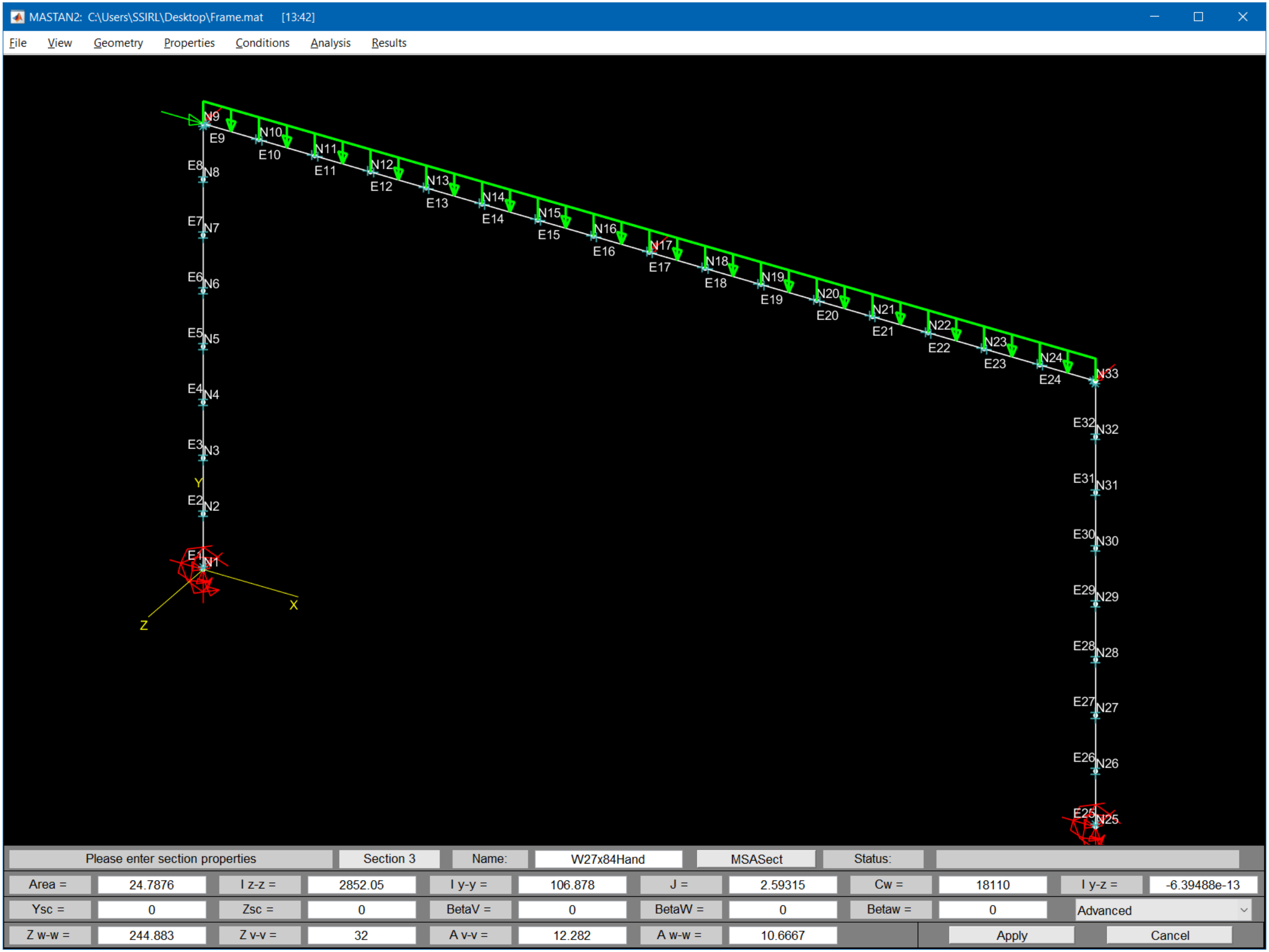
9) Click **Apply** to define Section 3.


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
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
(+) <------ 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
Cancel

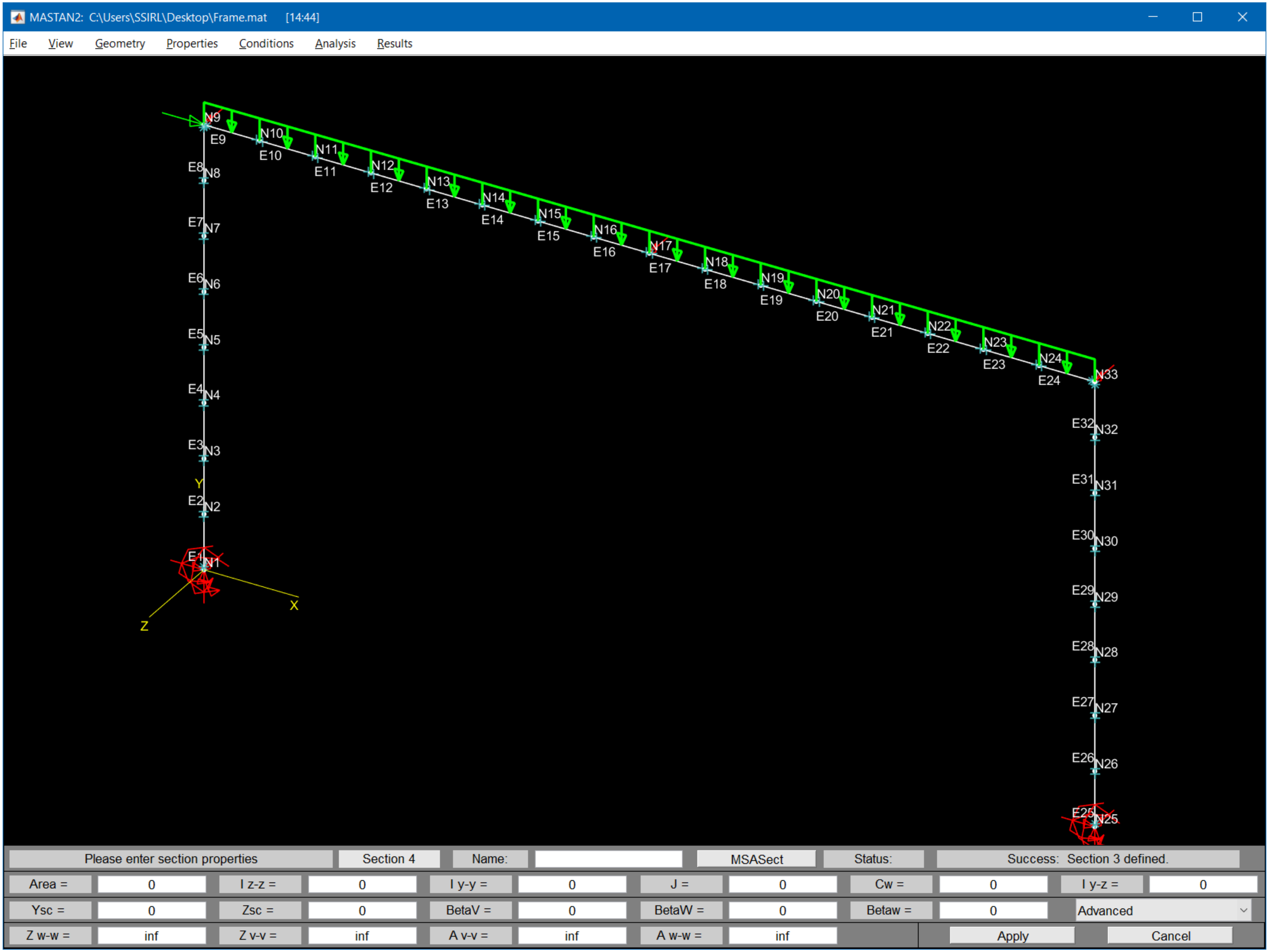
MASTAN2


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
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
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 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

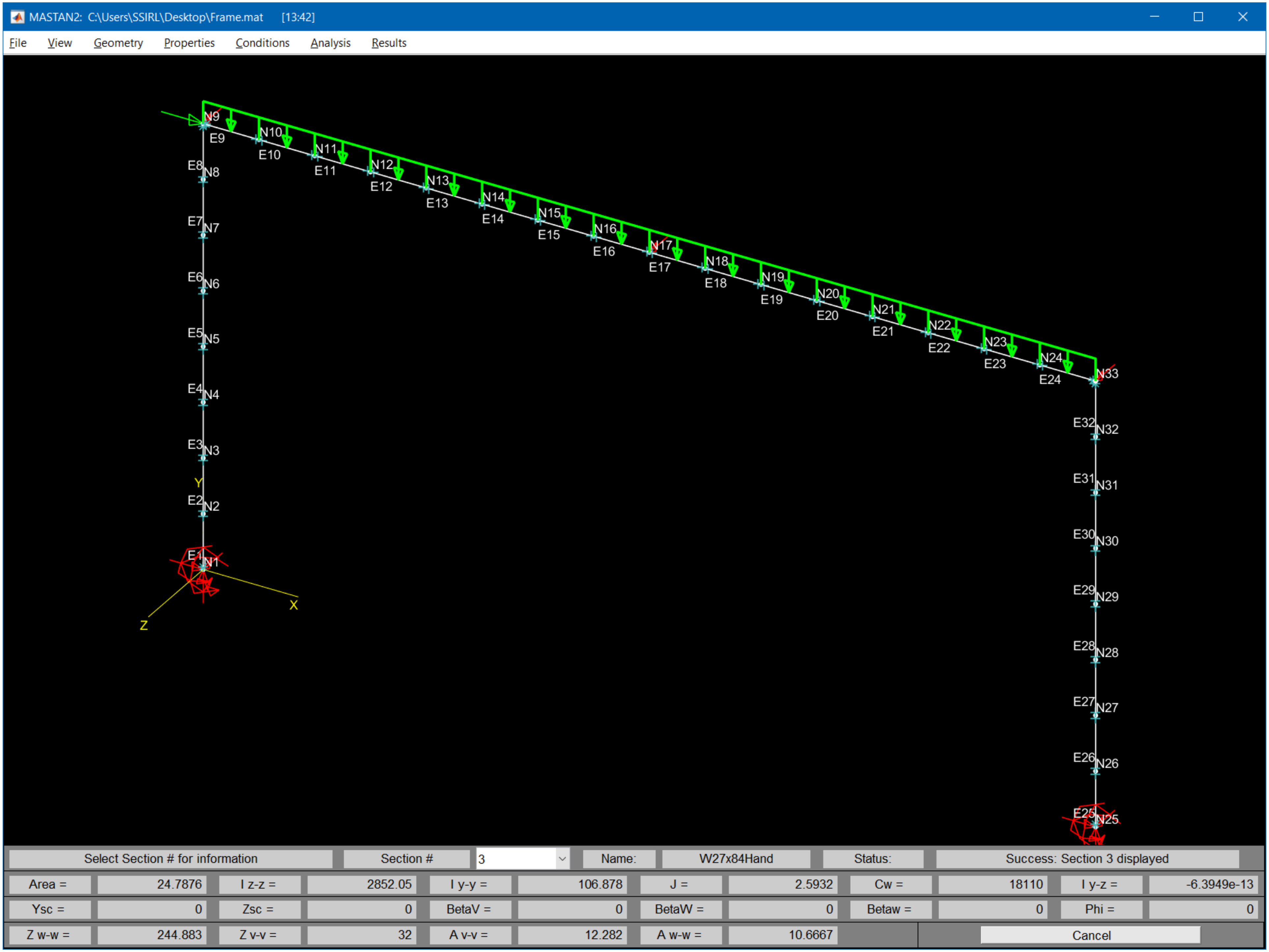
MSASect Results

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.

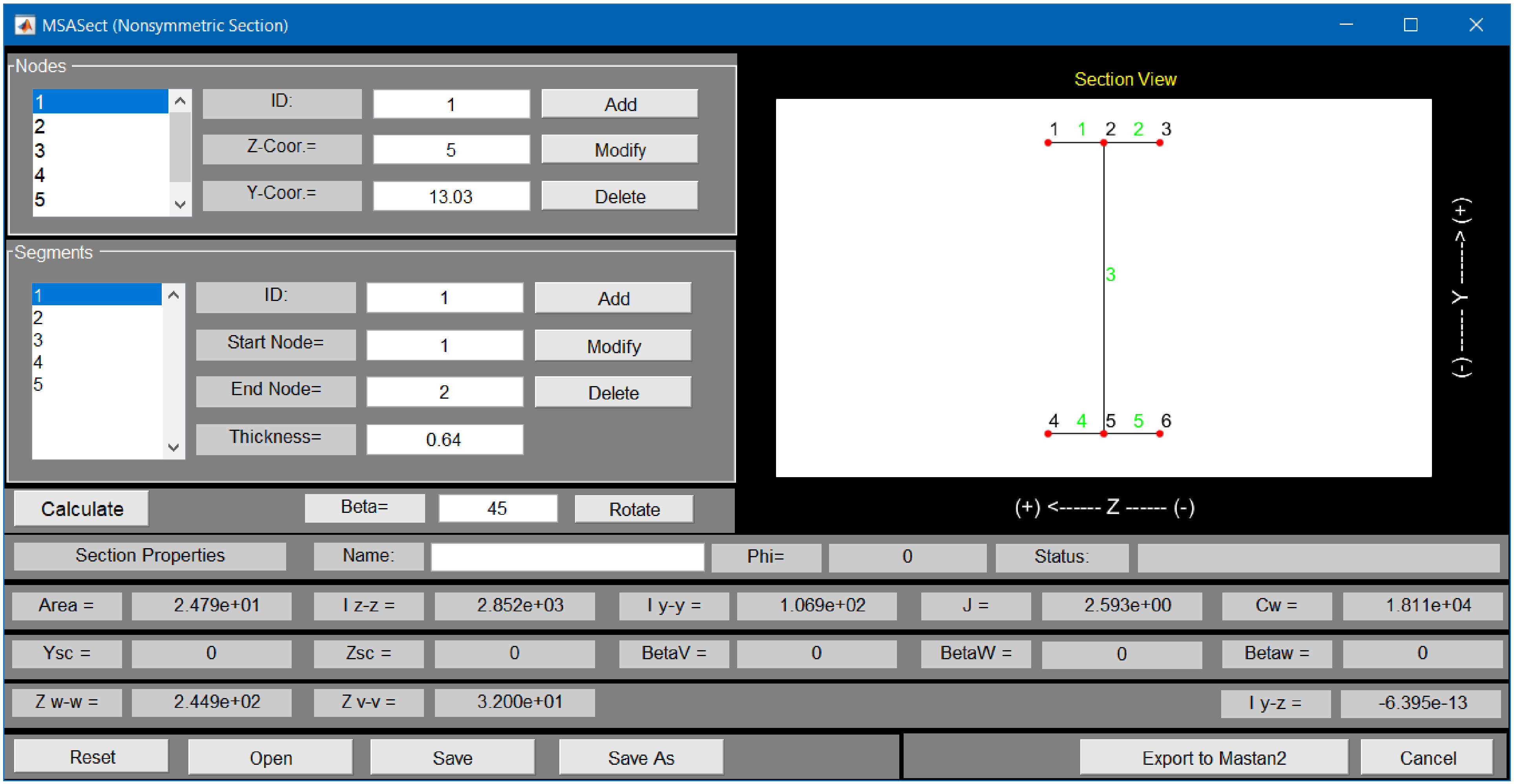
Property	Units	AISC	MSASect	Difference
A	in^2	24.7	24.79	0.4 %
Izz	in^4	2850	2852	0.1 %
Iyy	in^4	106	106.9	0.8 %
J	in^4	2.81	2.59	-7.7 %
Cw	in^6	18000	18110	0.6 %
Zzz	in^3	244	244.9	0.4 %
Zyy	in^3	33.2	32	-3.6 %

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

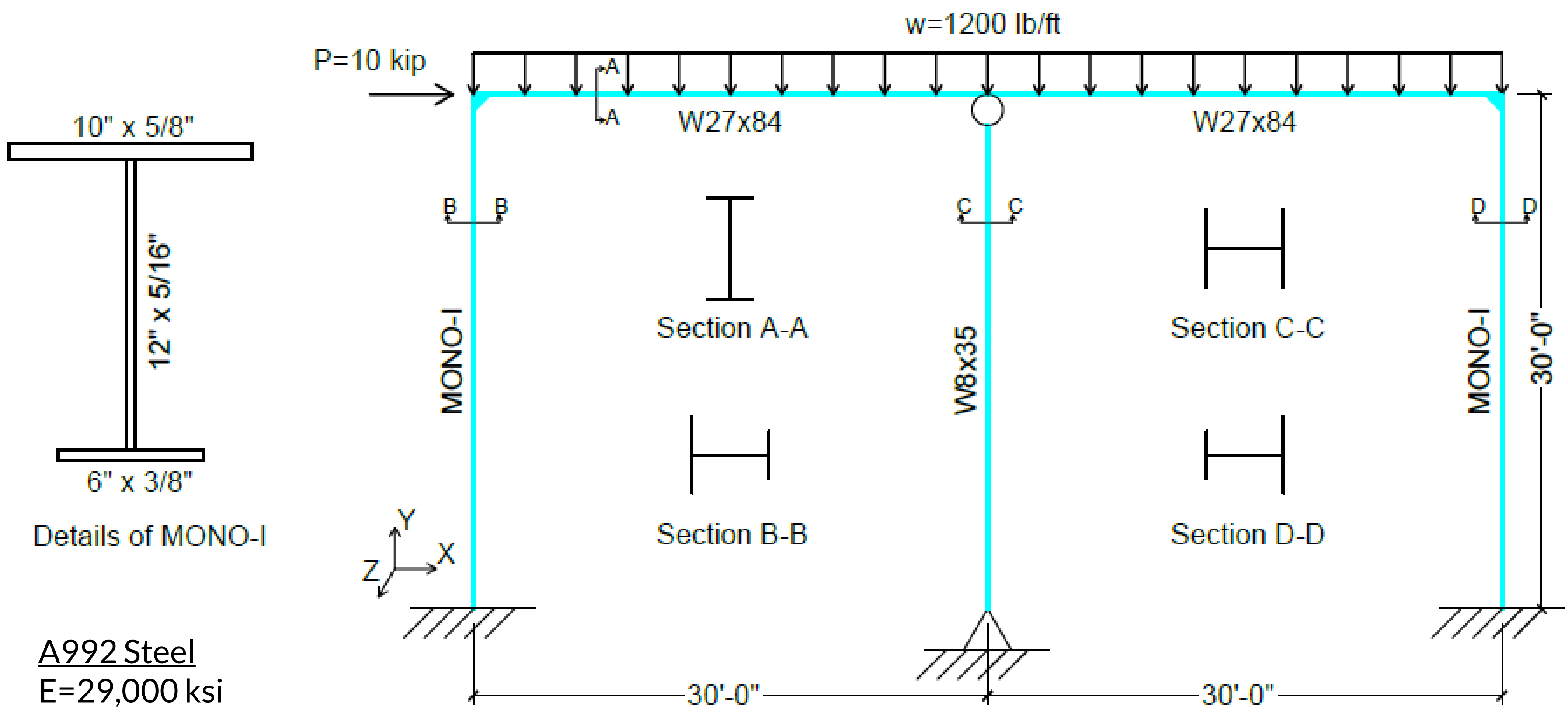
If one of the default cross sections does not cover your situation the General option allows for the input of nodes and line segments by the user. Clicking the radio button next to **General** and then the **Next** button will open an interface that allows for the input of nodes and line segments directly. 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**.

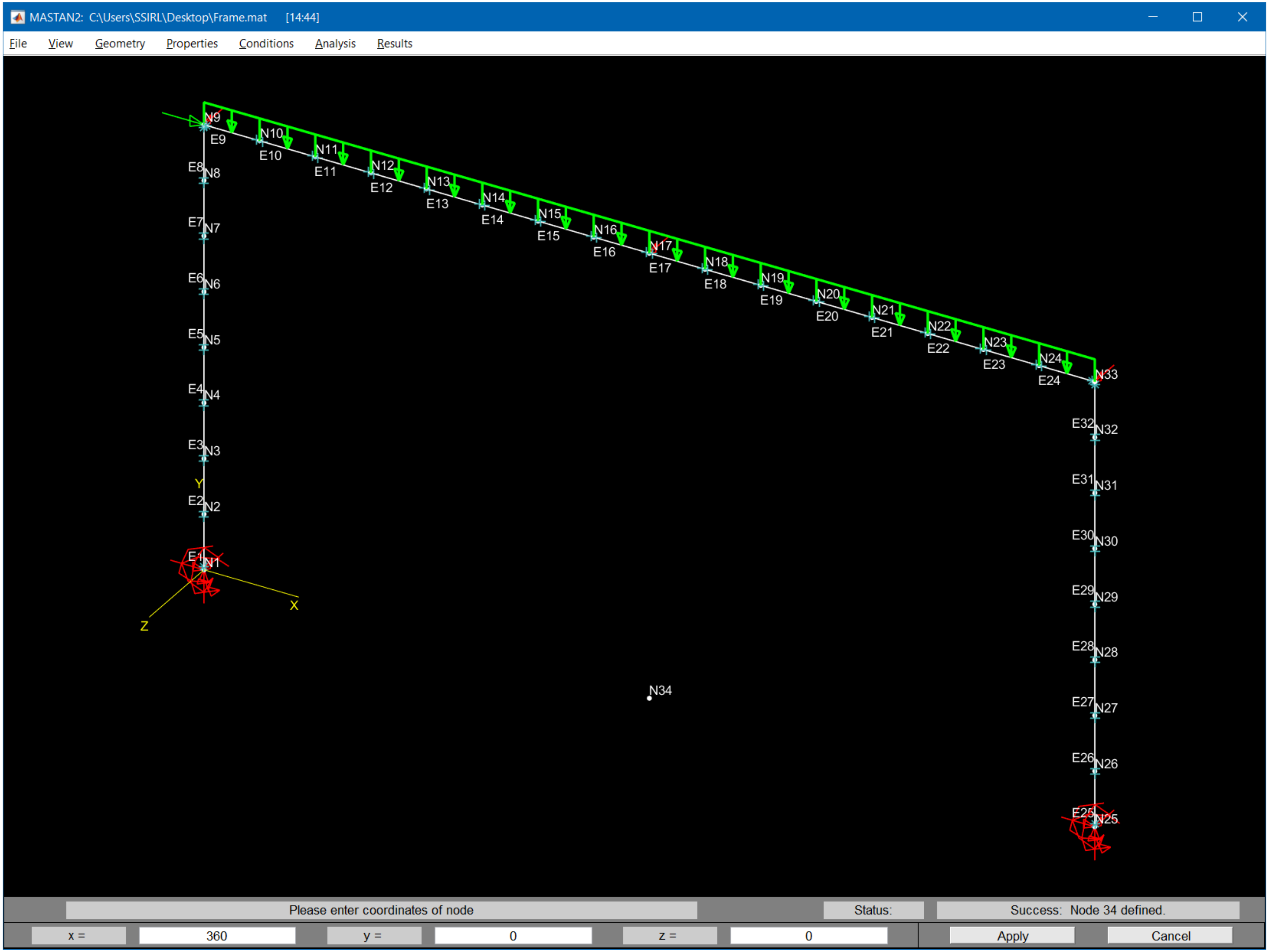
3) Click on the **Apply** Button.

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

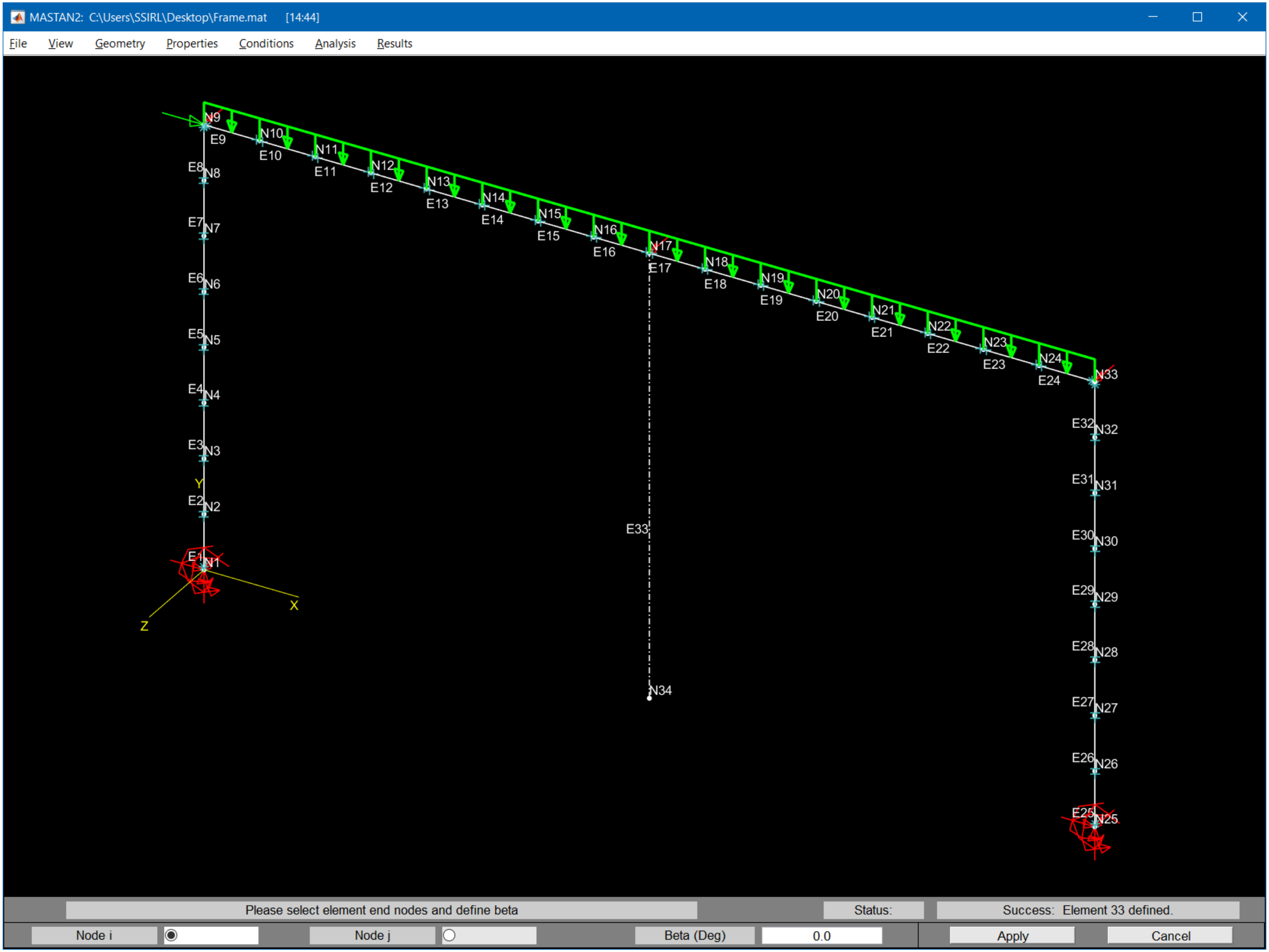
5) On the model, click the newly created node to define Node i. Then click the middle node of the top beam to define Node j. These nodes should be **34** and **17**, respectively.

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

MASTAN2


MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
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
N34
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.

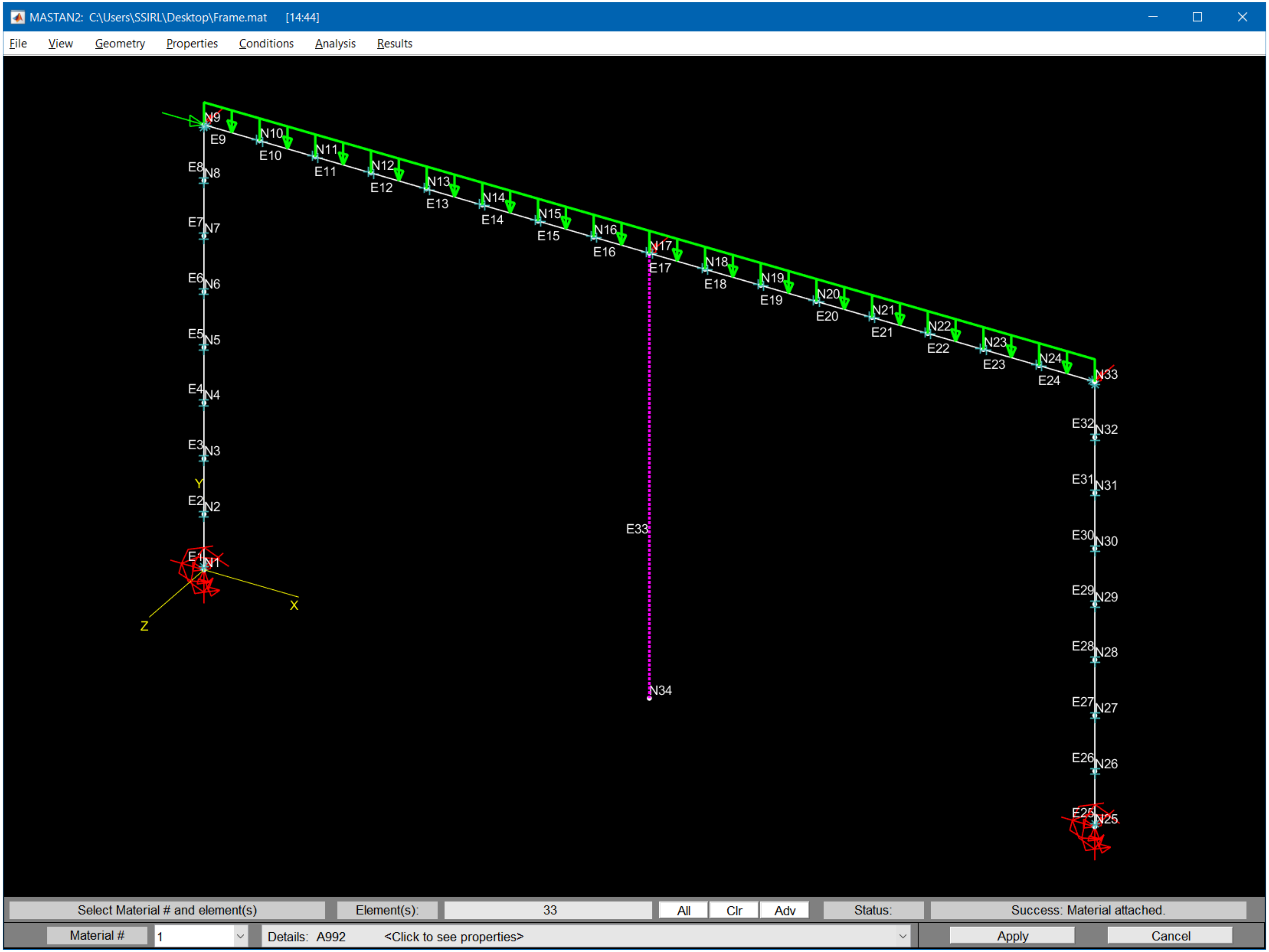
5) In the pop-up menu, scroll to find section **W8x35** and click on it. Then click on the **Apply** button. (Section 4 is now defined with the properties of W8x35.)

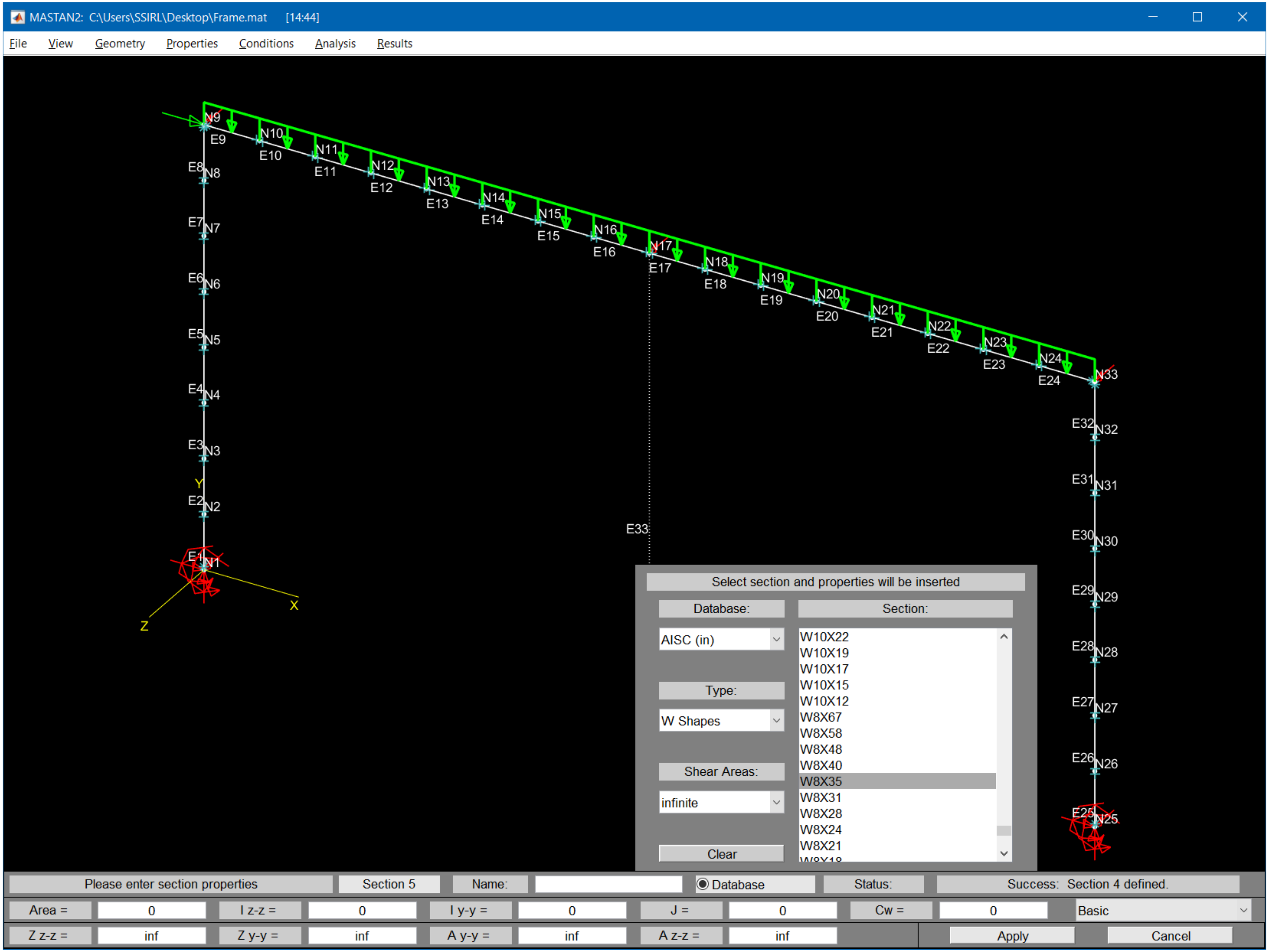
6) From the **Properties** menu select **Attach Section**.

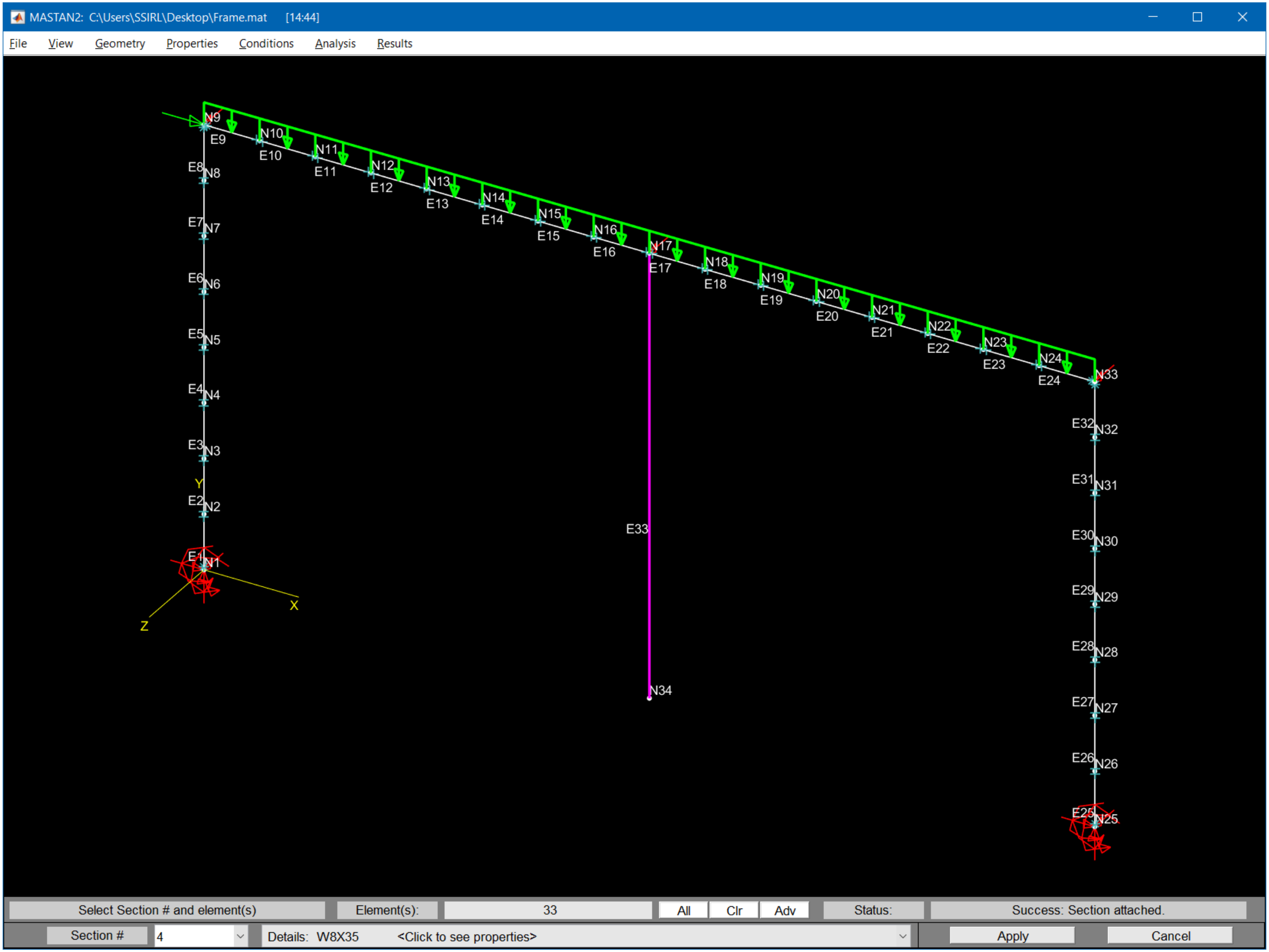
7) Create the list of elements to be assigned the properties of Section 4 by clicking on the new column, element **33**.

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
E9
N10
E10
N17
E17
E33
N34
N33
N25
N1
X
Y
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
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
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
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
E32 N32 E31 N31 E30 N30 E29 N29 E28 N28 E27 N27 E26 N26 E25 N25
E33
N34
Y
X
Z
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.)

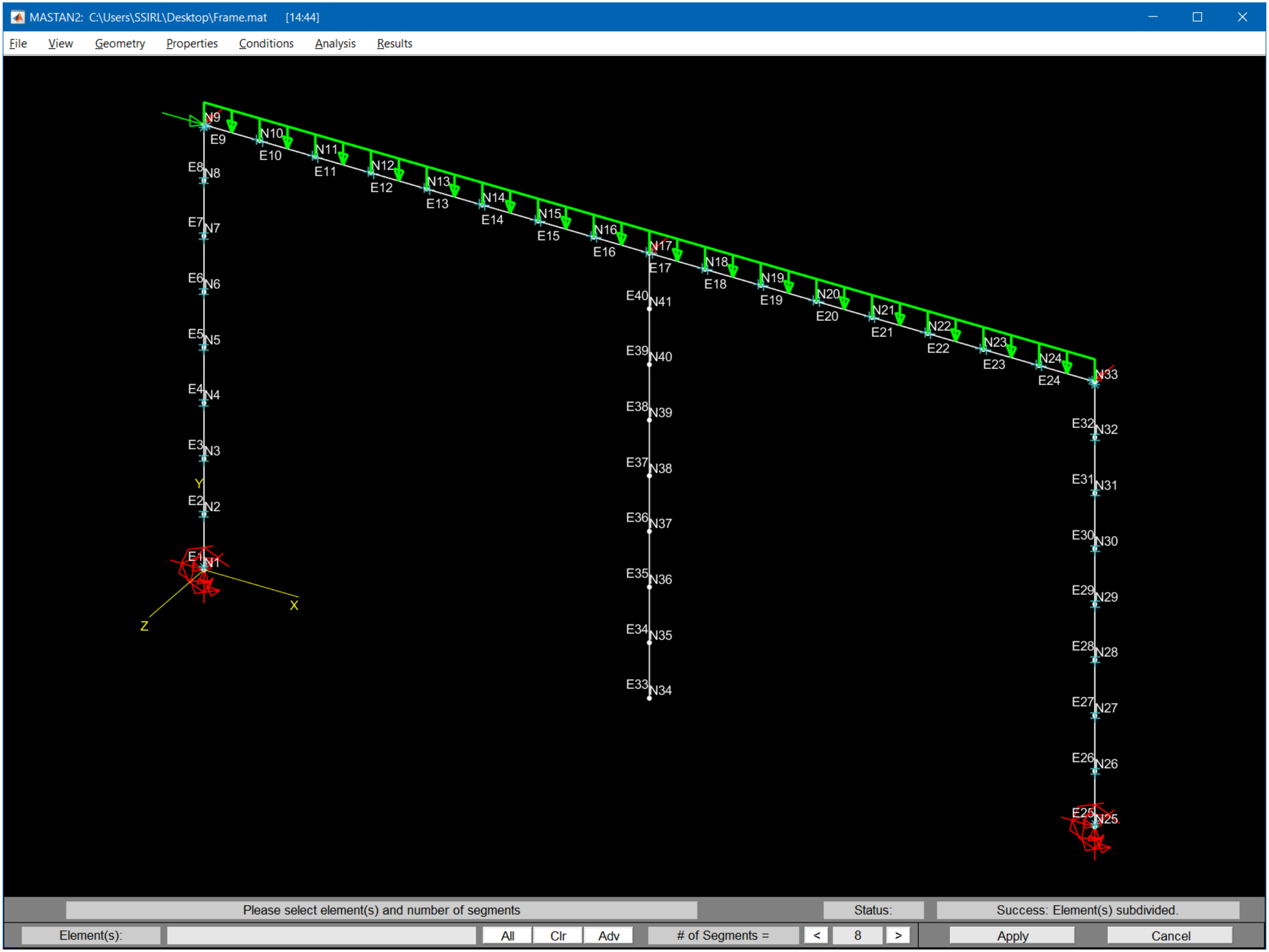
5) From the **Conditions** menu select Define **Fixities**.

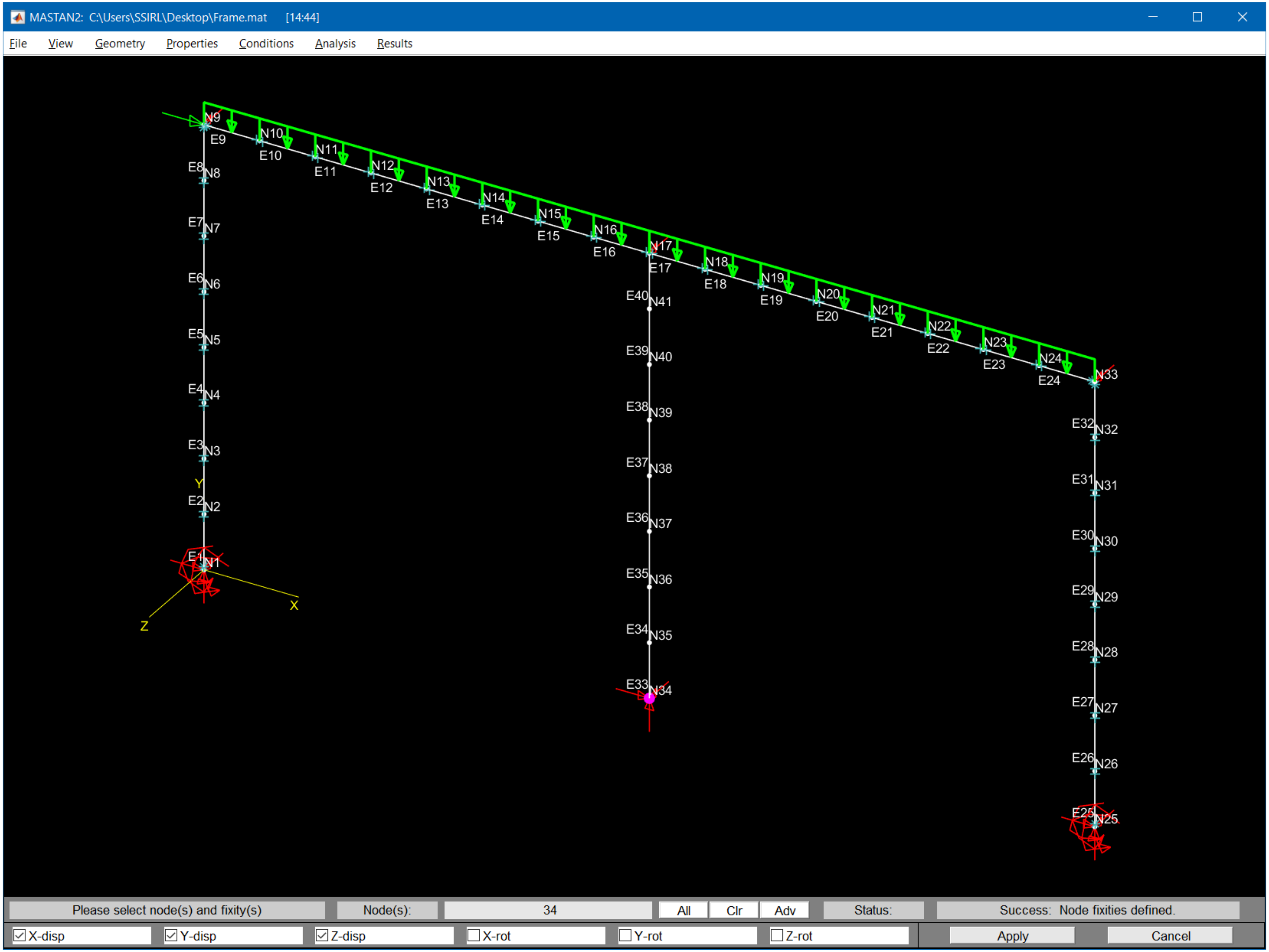
6) At the bottom menu bar, define a pin support by clicking in the **check boxes** just to the left of **X-disp**, **Y-disp**, and **Z-disp**.

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


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
E40 N41
E39 N40
E38 N39
E37 N38
E36 N37
E35 N36
E34 N35
E33 N34
E32 N32
E31 N31
E30 N30
E29 N29
E28 N28
E27 N27
E26 N26
E25 N25
X
Y
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

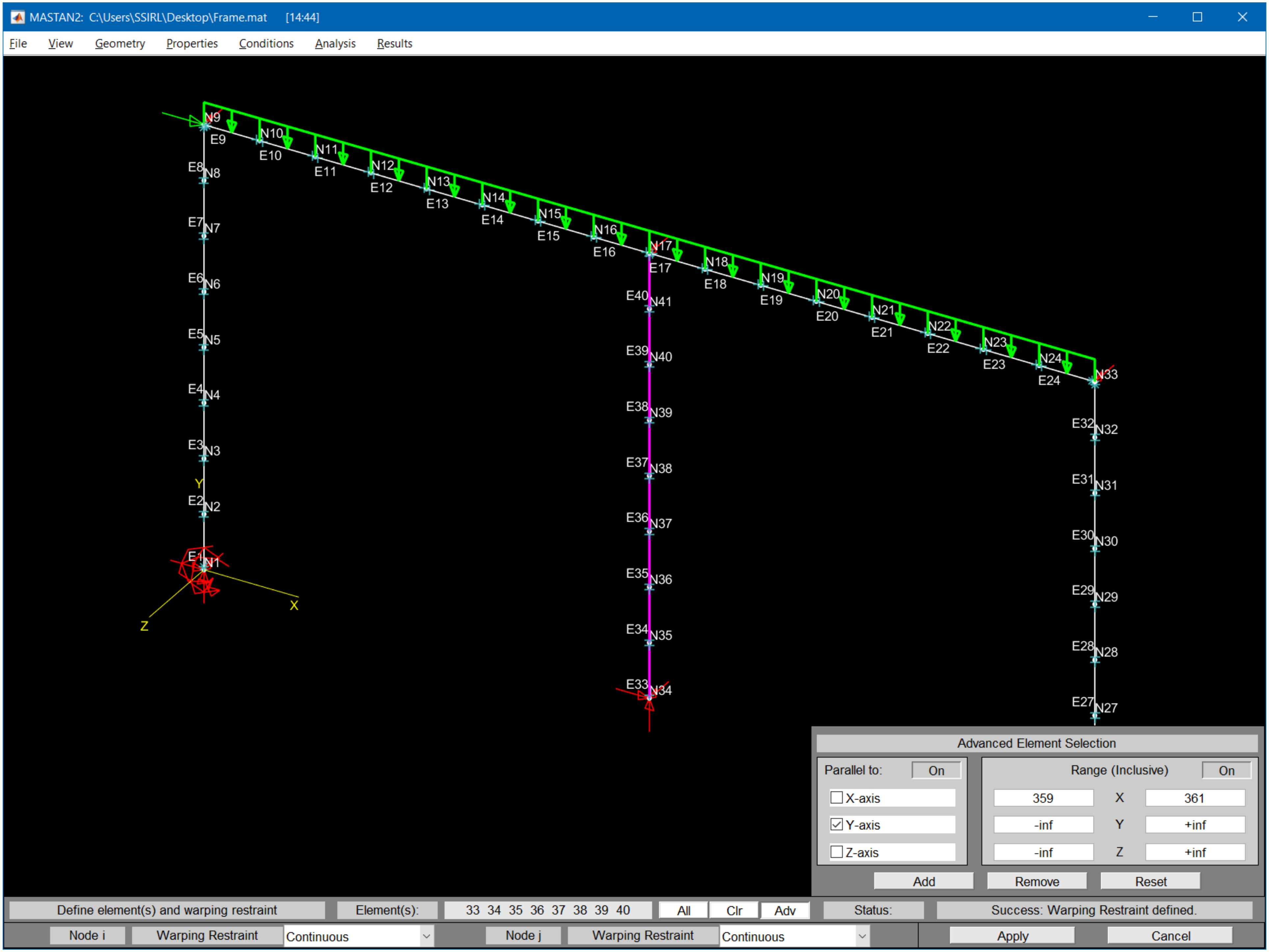
1) From the **Geometry** menu select **Define Connections** and submenu option **Torsion**.

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.

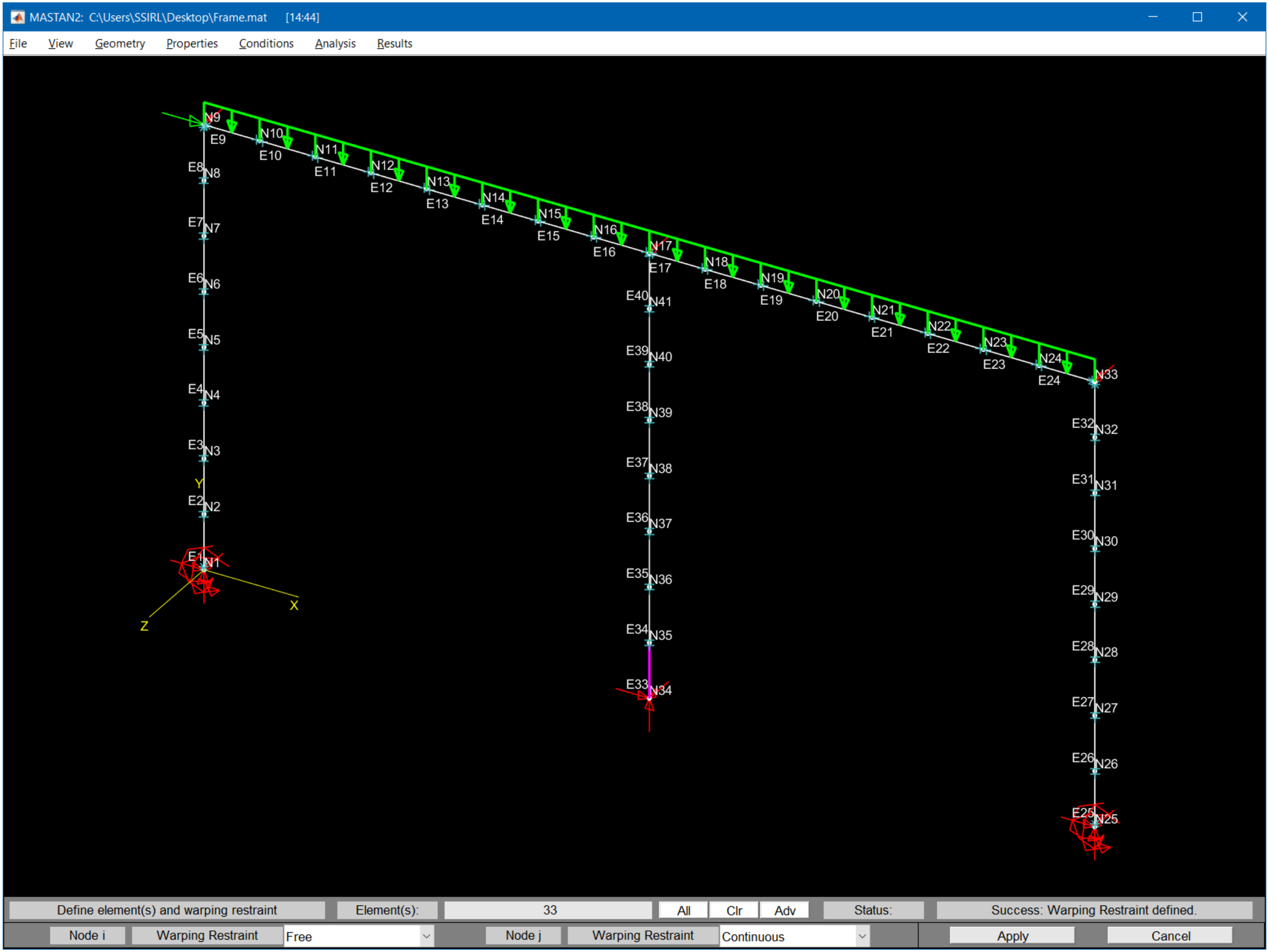
3) Click on the menu to the right of **Warping Restraint for Node i** and set the value to **Free**. Node j is set from the previous step.

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

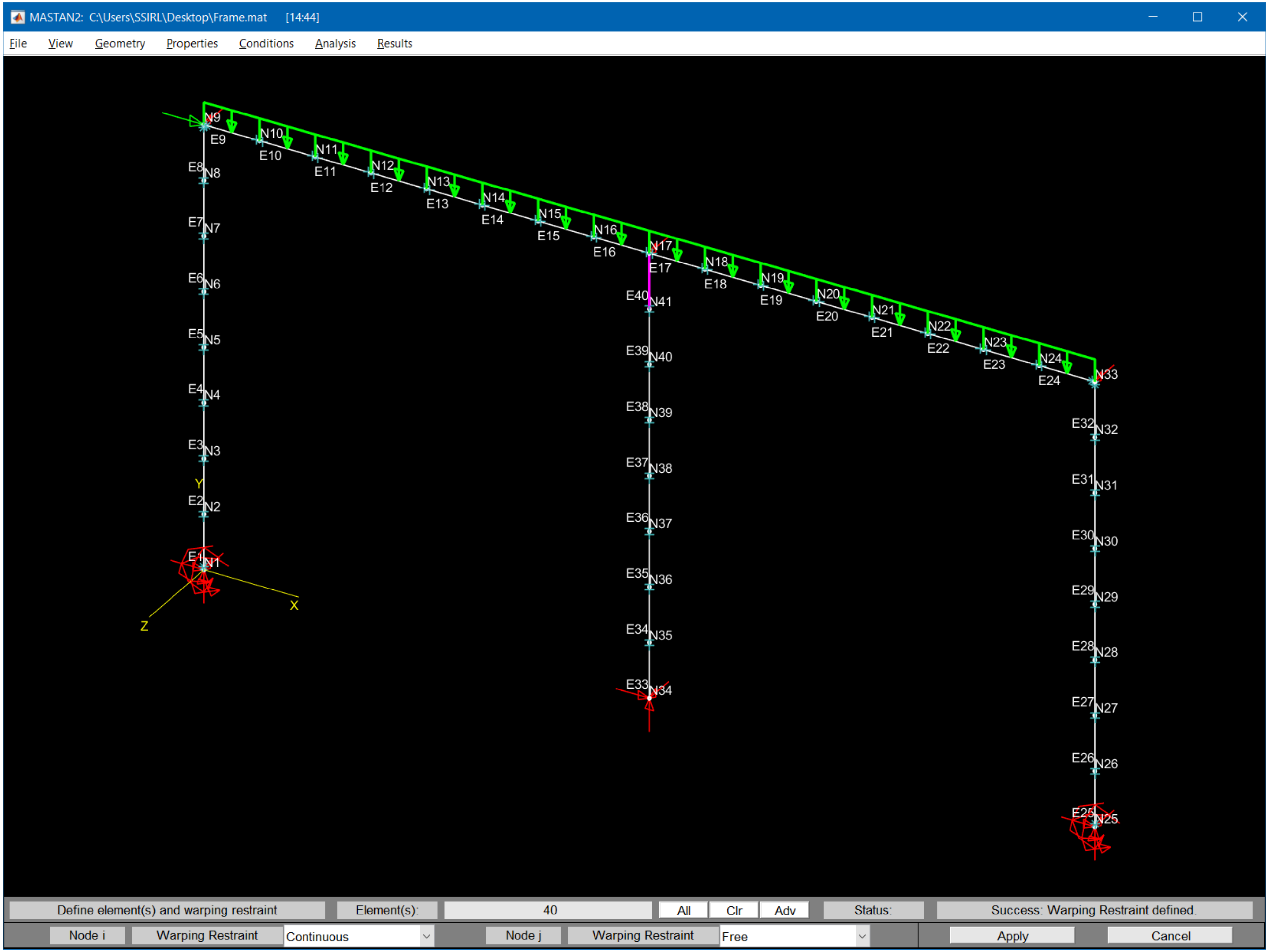
5) Click **Clr** to empty the list of elements. Click on the top element of the middle column.

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.

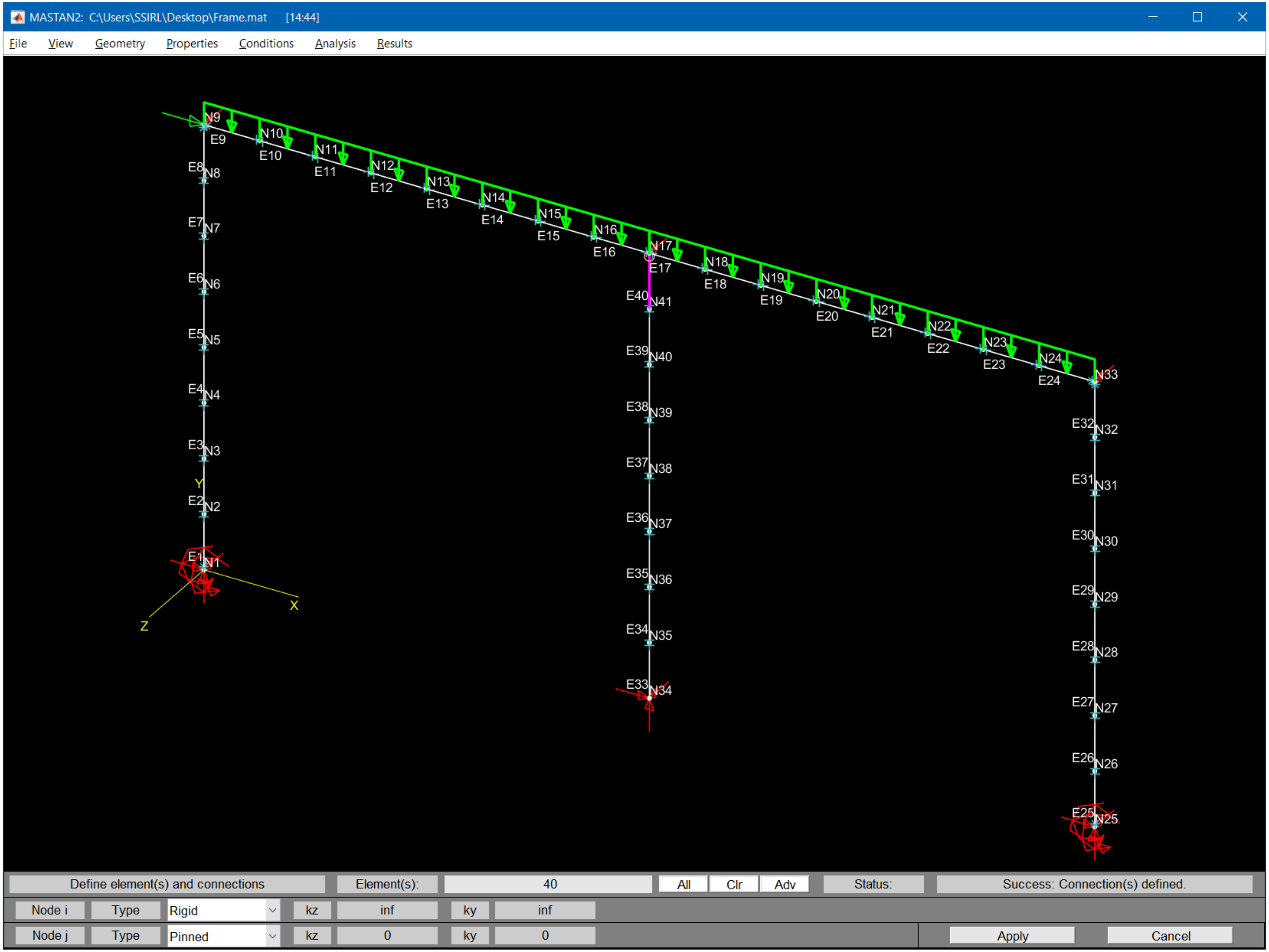

MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
Define element(s) and warping restraint
Element(s):
33
All
Clr
Adv
Status:
Success: Warping Restraint defined.
Node i
Warping Restraint
Free
Node j
Warping Restraint
Continuous
Apply
Cancel

MASTAN2


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: C:\Users\SSIRL\Desktop\Frame.mat [14:44]
File
View
Geometry
Properties
Conditions
Analysis
Results
X
Y
Z
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**.

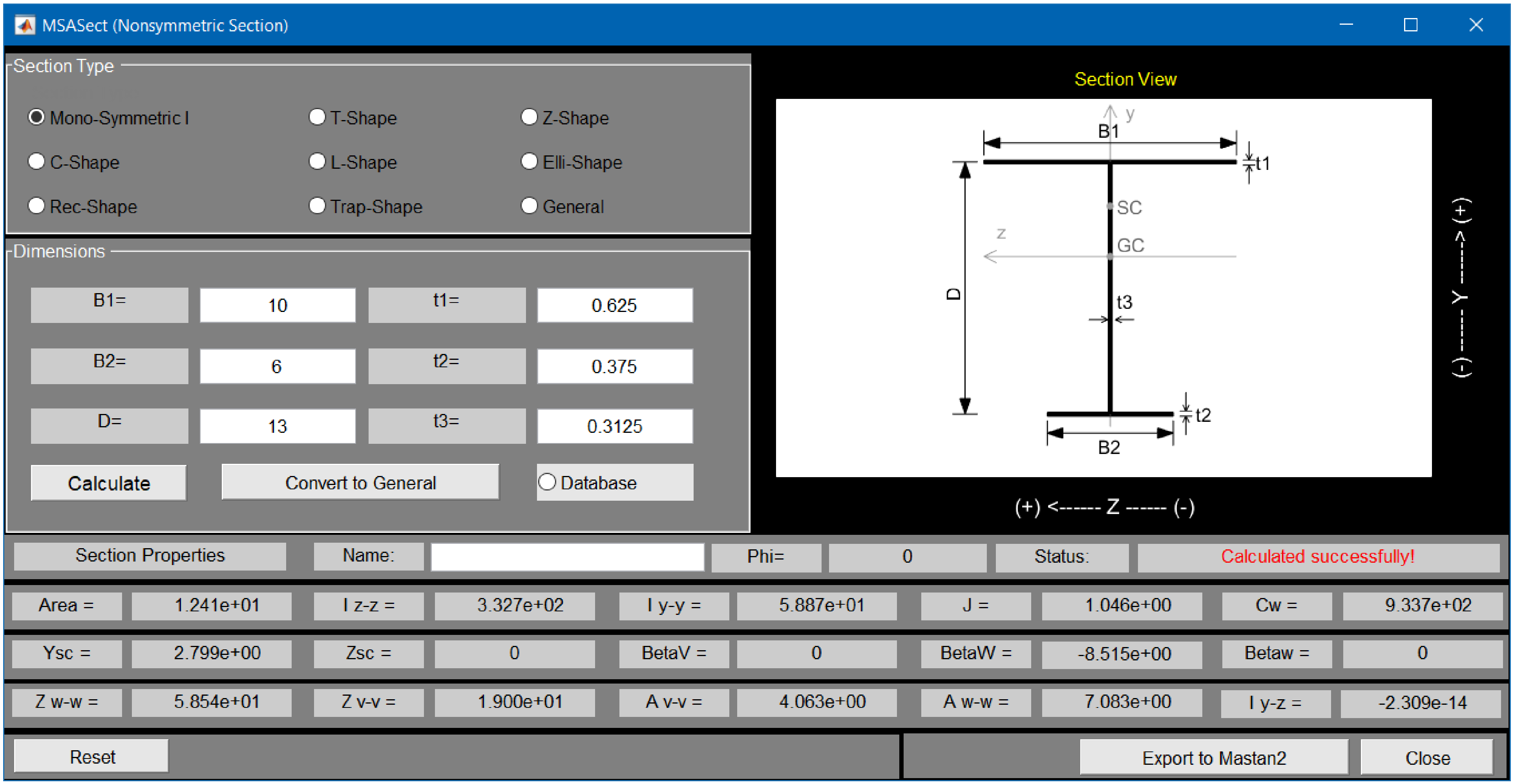
4) As the I-beam cross-section is selected by default, click the edit box to the right of **B1=** and enter **10**. Repeat to define **B2=6**, **D=13**, **t1=0.625**, **t2=0.375**, and **t3=0.3125**.

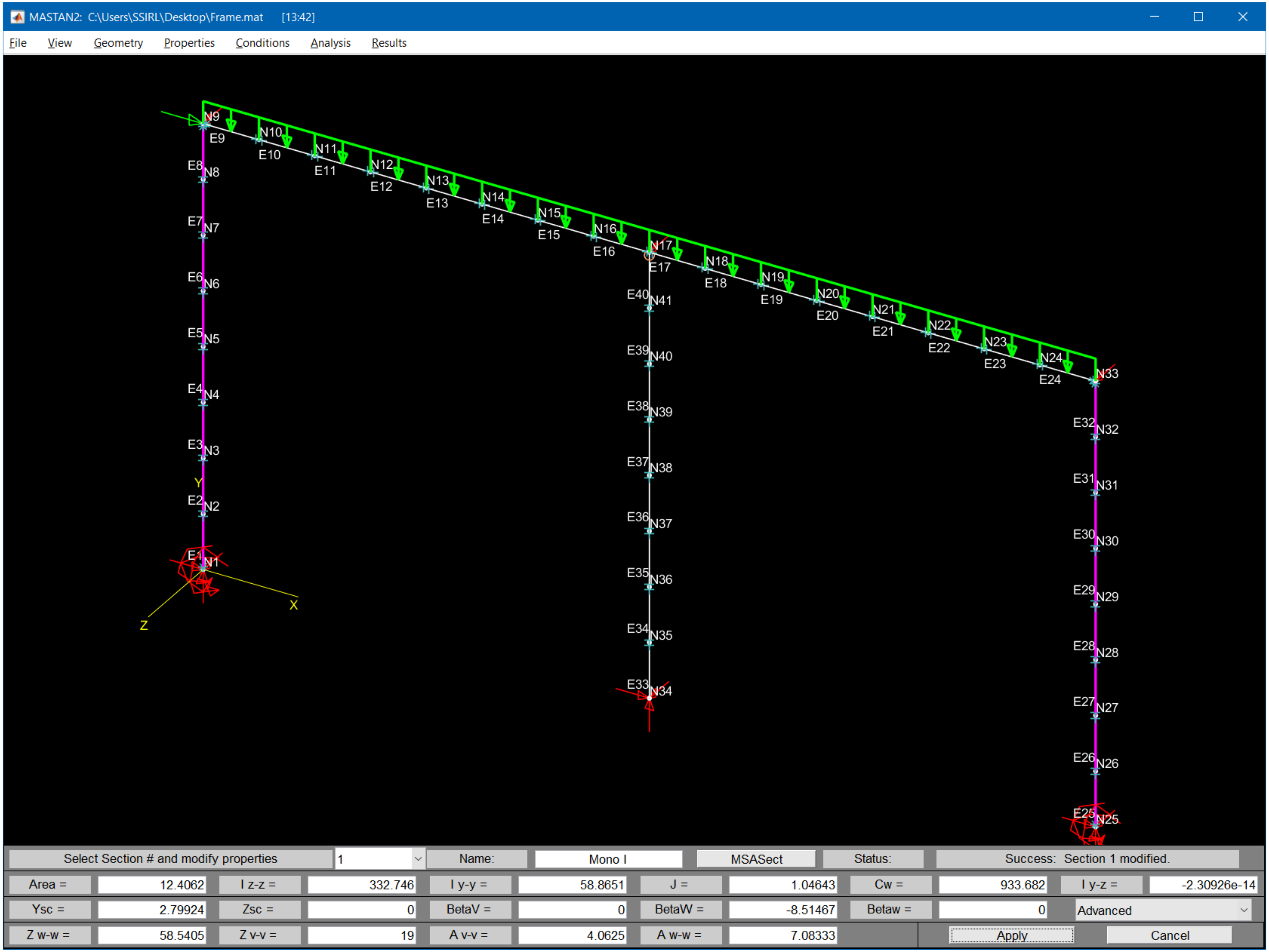
5) Click **Calculate** to determine the properties.

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)**.

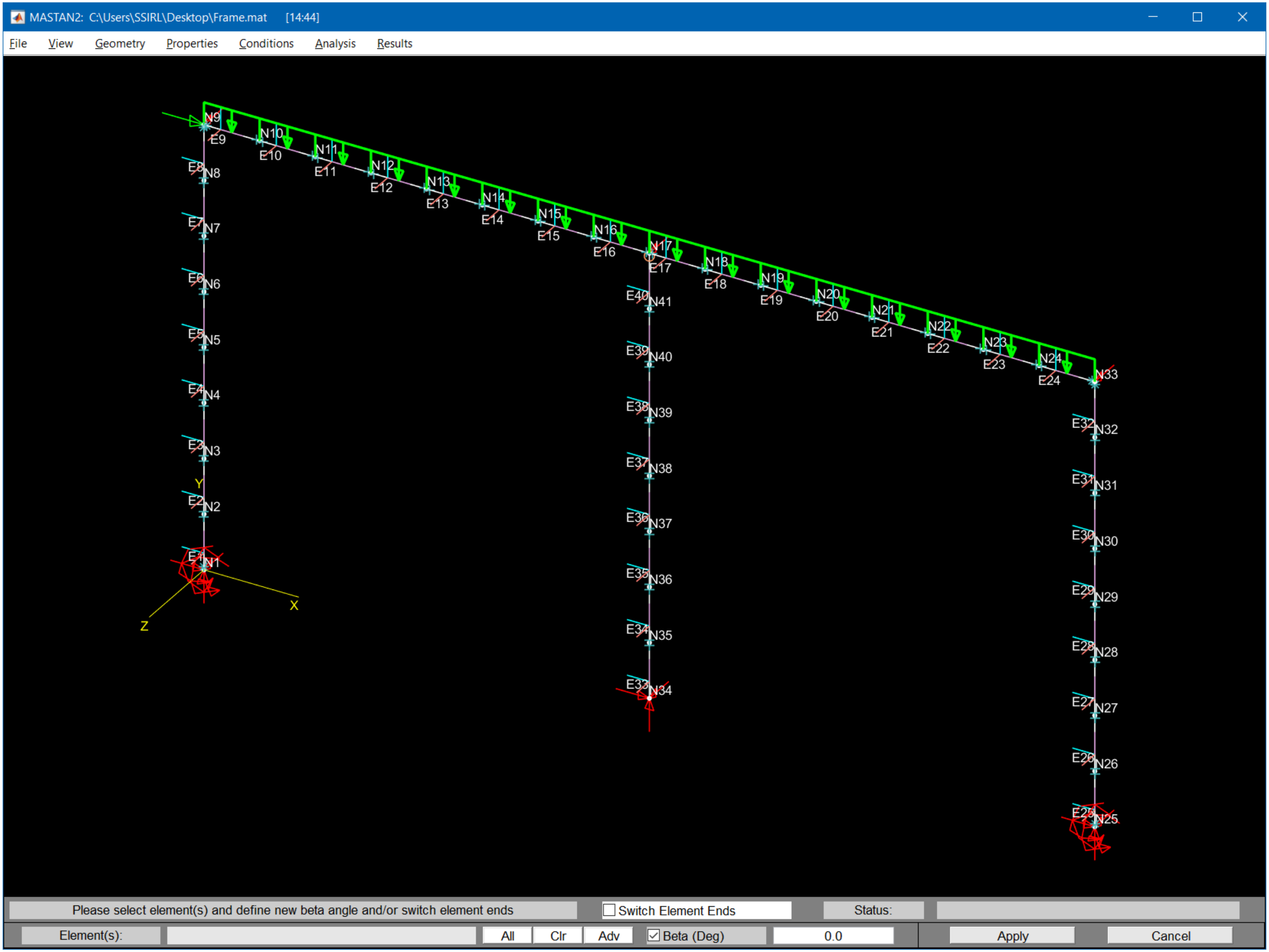
3) From the **View** menu select **Labels** and submenu option **Element local x’-y’-z’ axes**. Each axis is shown with a different color line drawn in the positive direction. The x axis is purple, the y axis is blue, and the z axis is red.

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

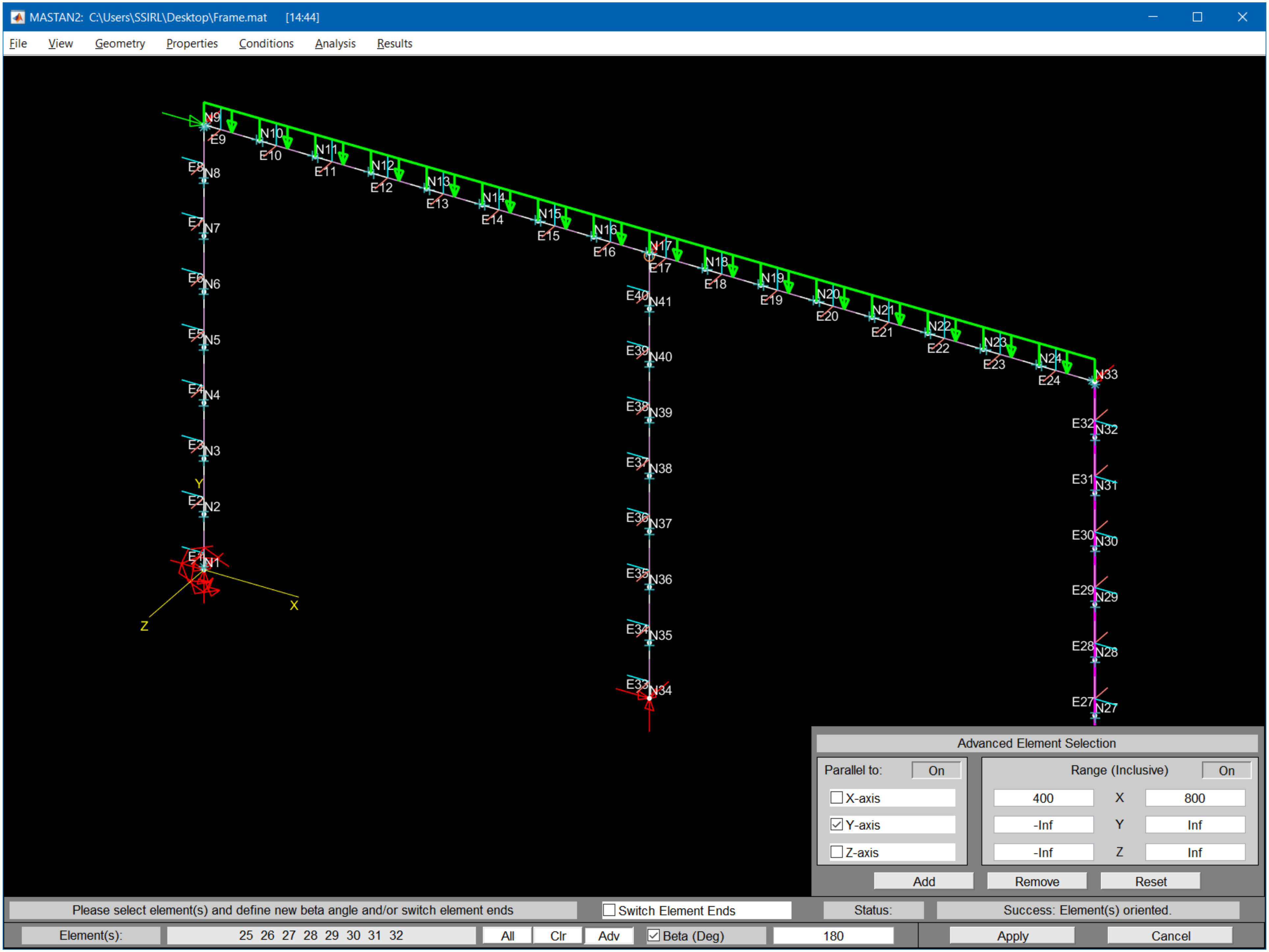
5) Use the buttons to the right of **Element(s):** to make the list of elements. Click the **Adv** button to open the pop-up menu. To select all the right column elements, click the edit box to the left of **X** and change **359** to **400**. Click the edit box to the right of **X** and change **361**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

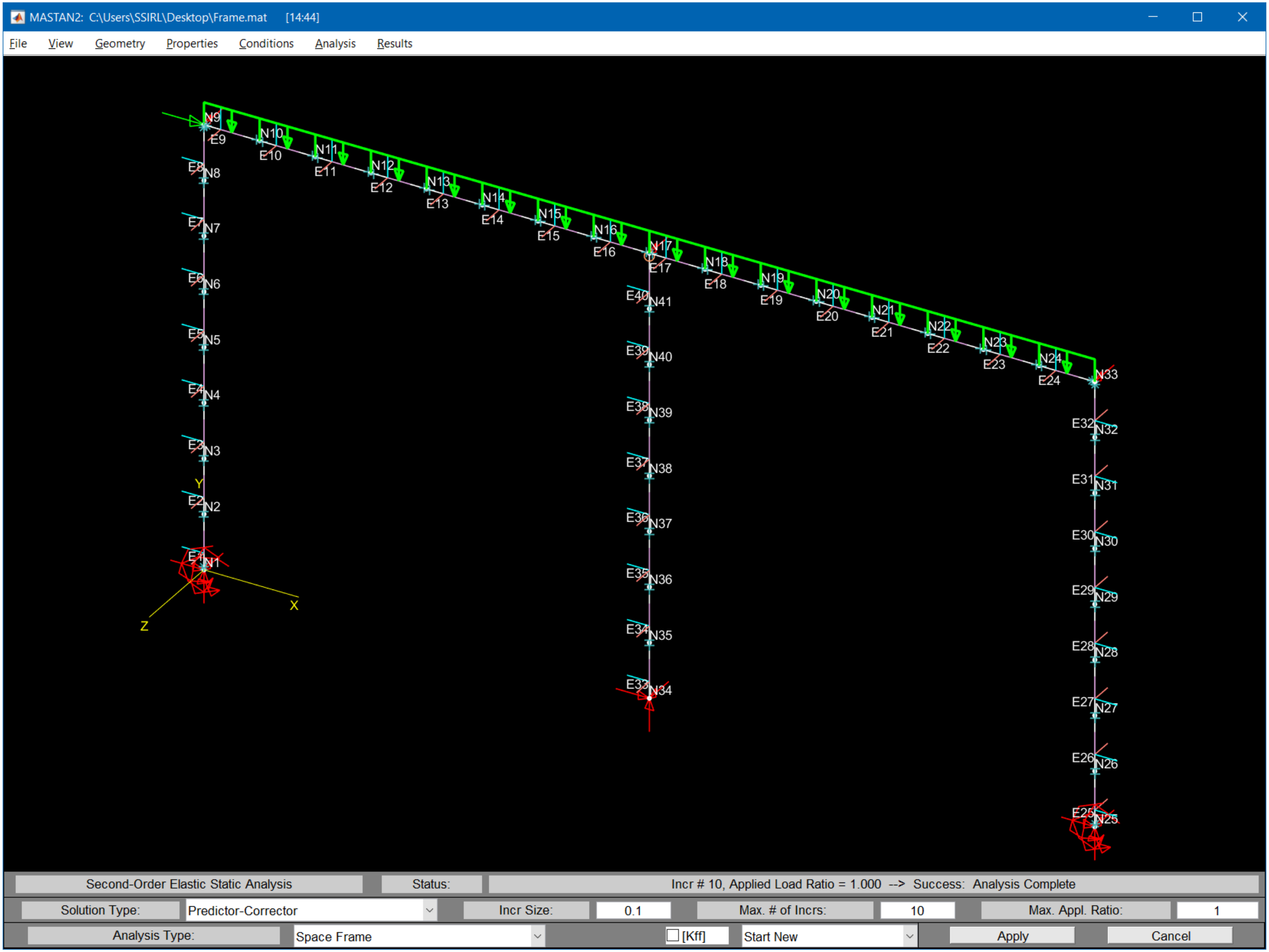
4) From the **Results** menu select **Node Displacements**.

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
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

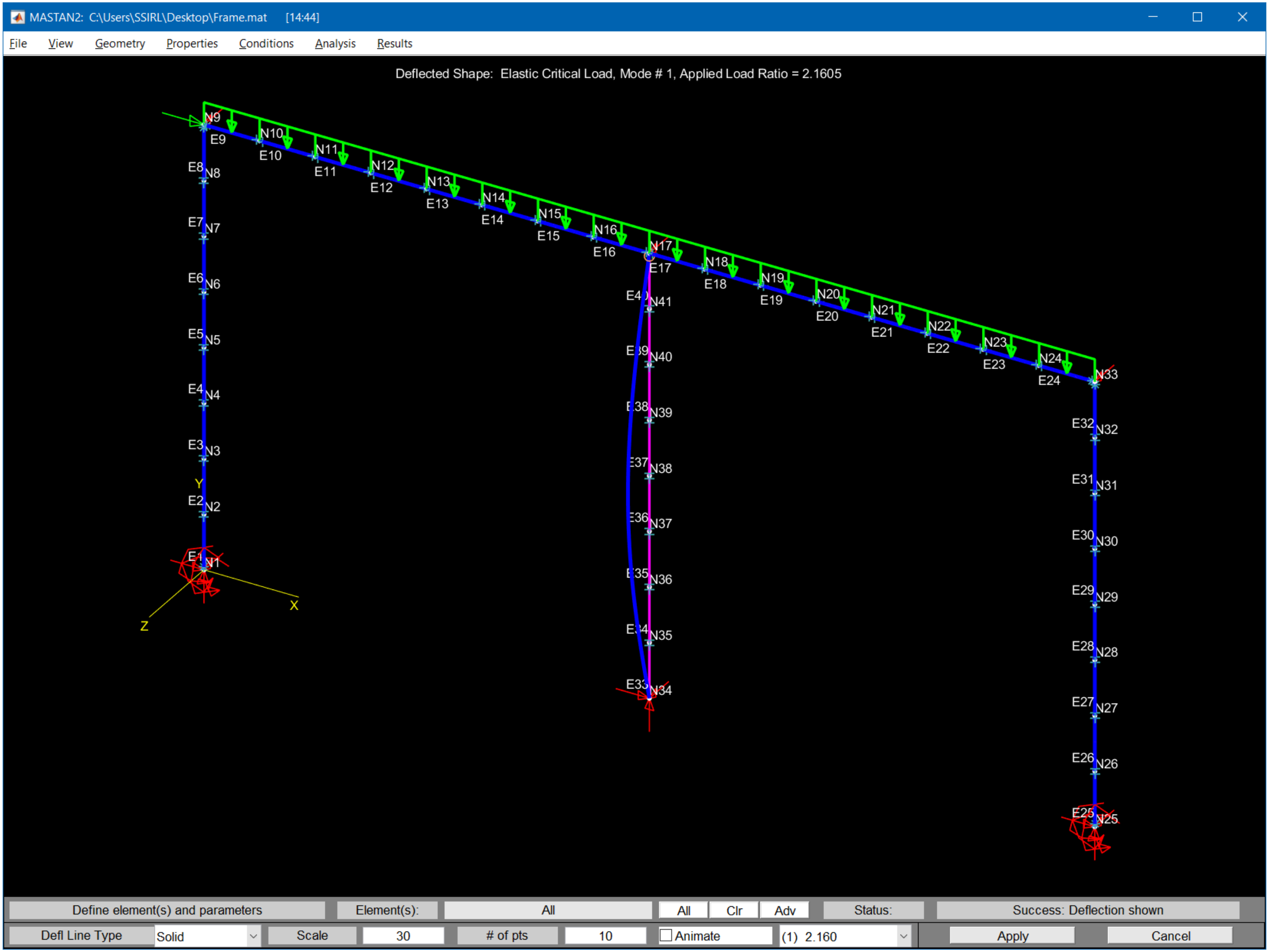
Additional Analysis

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.

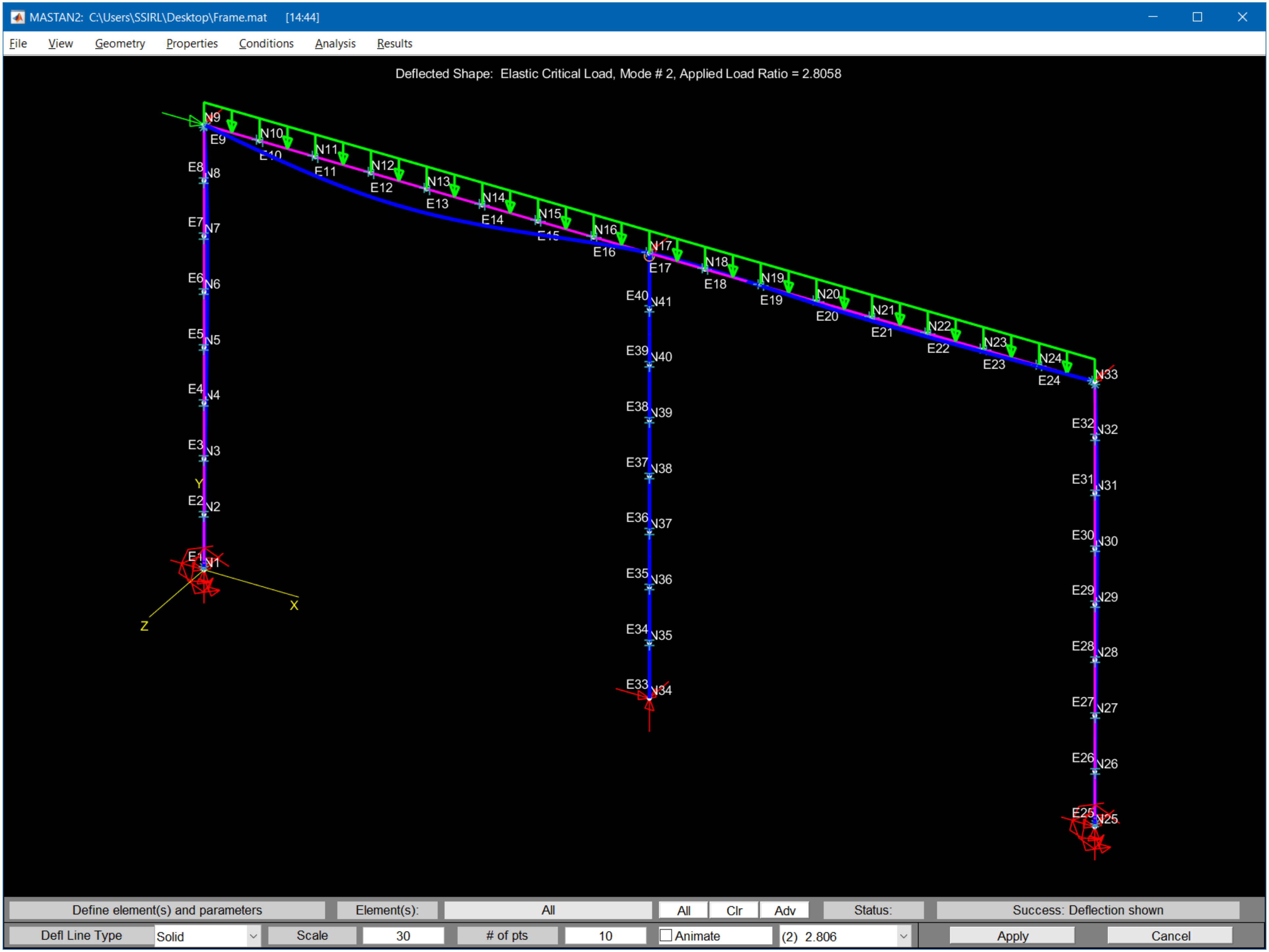
	Disp X	Disp Y	Rot Z
Basic	2.395	-0.02608	-1.304e-4
Advanced	2.395	-0.02608	-1.304e-4

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.

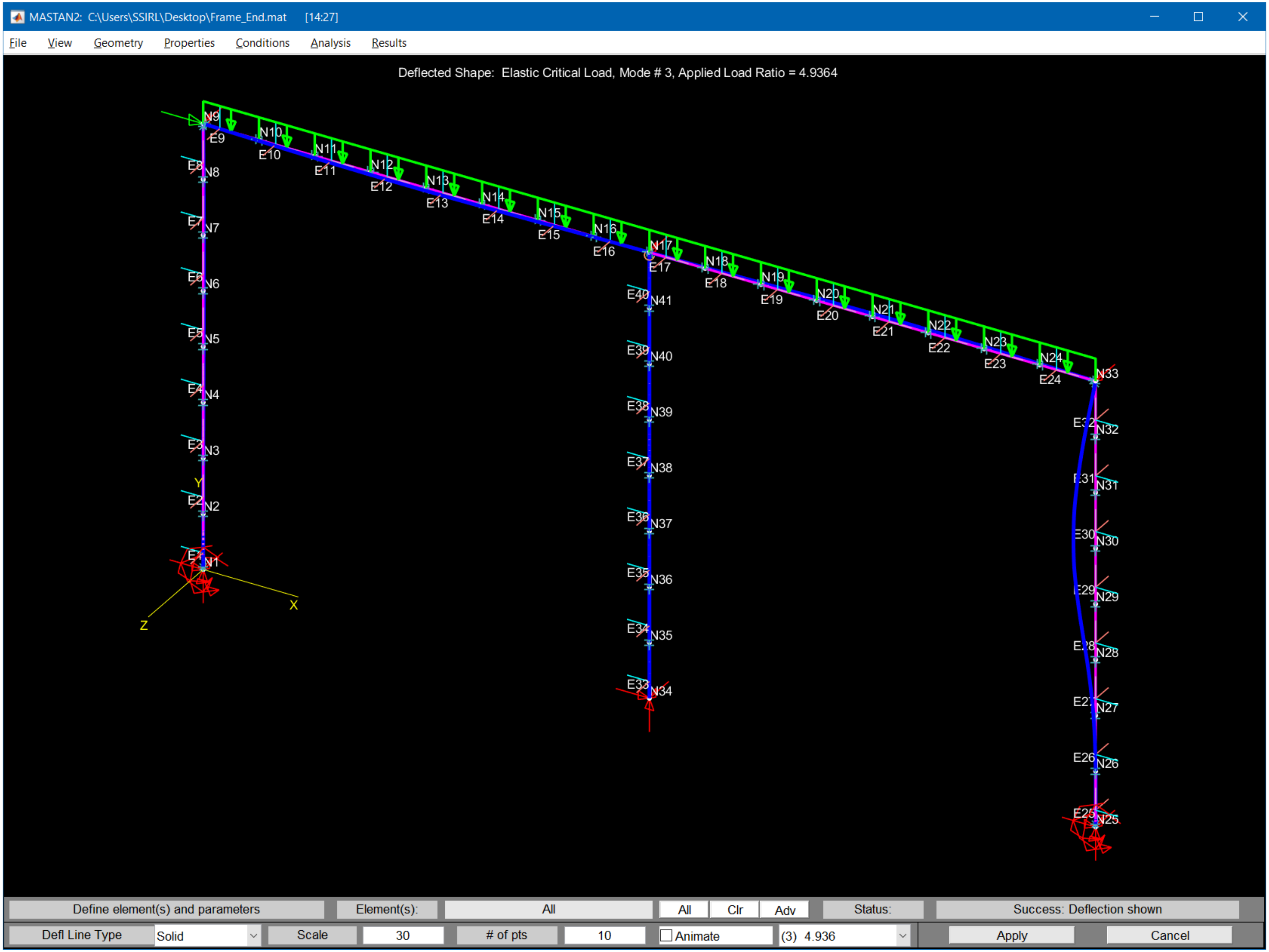
	Mode #1	Mode #2	Mode #3
Basic	2.160	2.806	4.936
Advanced	2.160	2.805	4.040


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

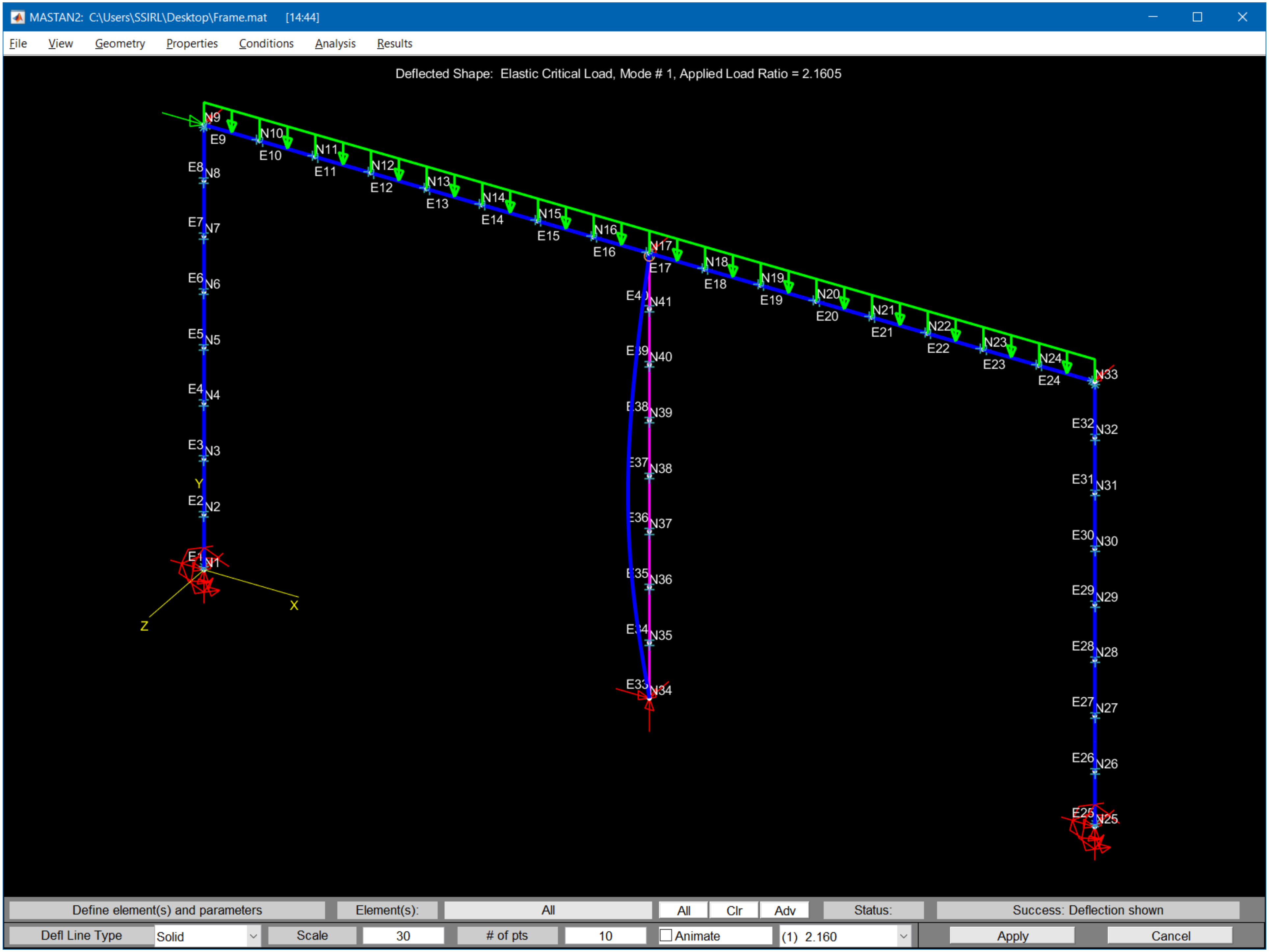
MASTAN2


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

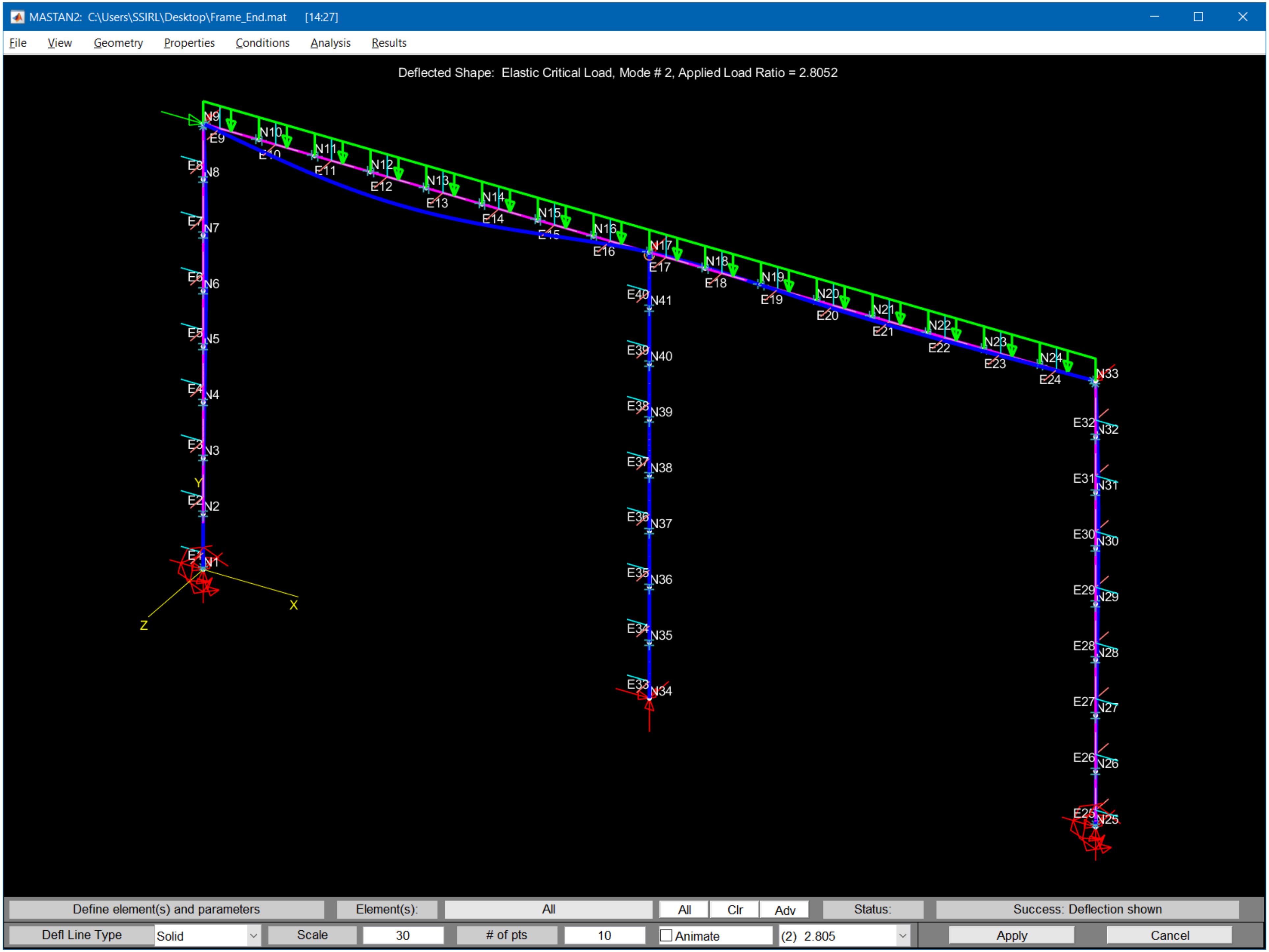
MASTAN2


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
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

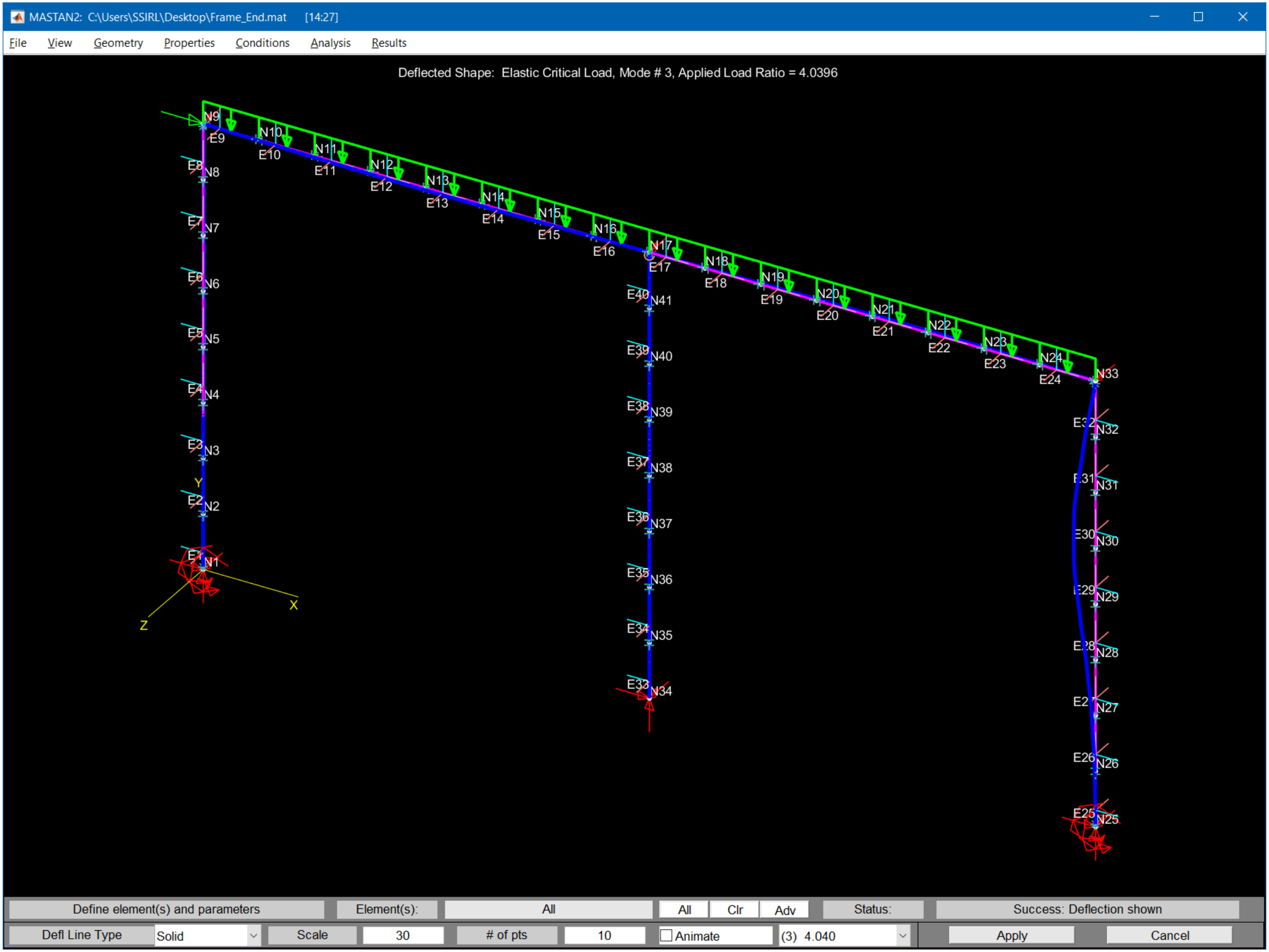
MASTAN2


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
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
(1) 2.160
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 # 2, Applied Load Ratio = 2.8052
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
(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.