Tutorial for MASTAN2

version 3.0

Developed by:

Ronald D. Ziemian

Professor of Civil Engineering

Bucknell University

William McGuire

Professor of Civil Engineering, Emeritus

Cornell University

JOHN WILEY & SONS, INC.

New York Chichester Weinheim

Brisbane Singapore Toronto

Continue

Exit

Tutorial Topics

- Introduction
- Getting started
- Window layout
- Step-by-step example
- Samples of MASTAN2 models
- Overview of commands
- Programming user defined code
- Additional information

< click on a topic >

Introduction

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 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.

Getting Started

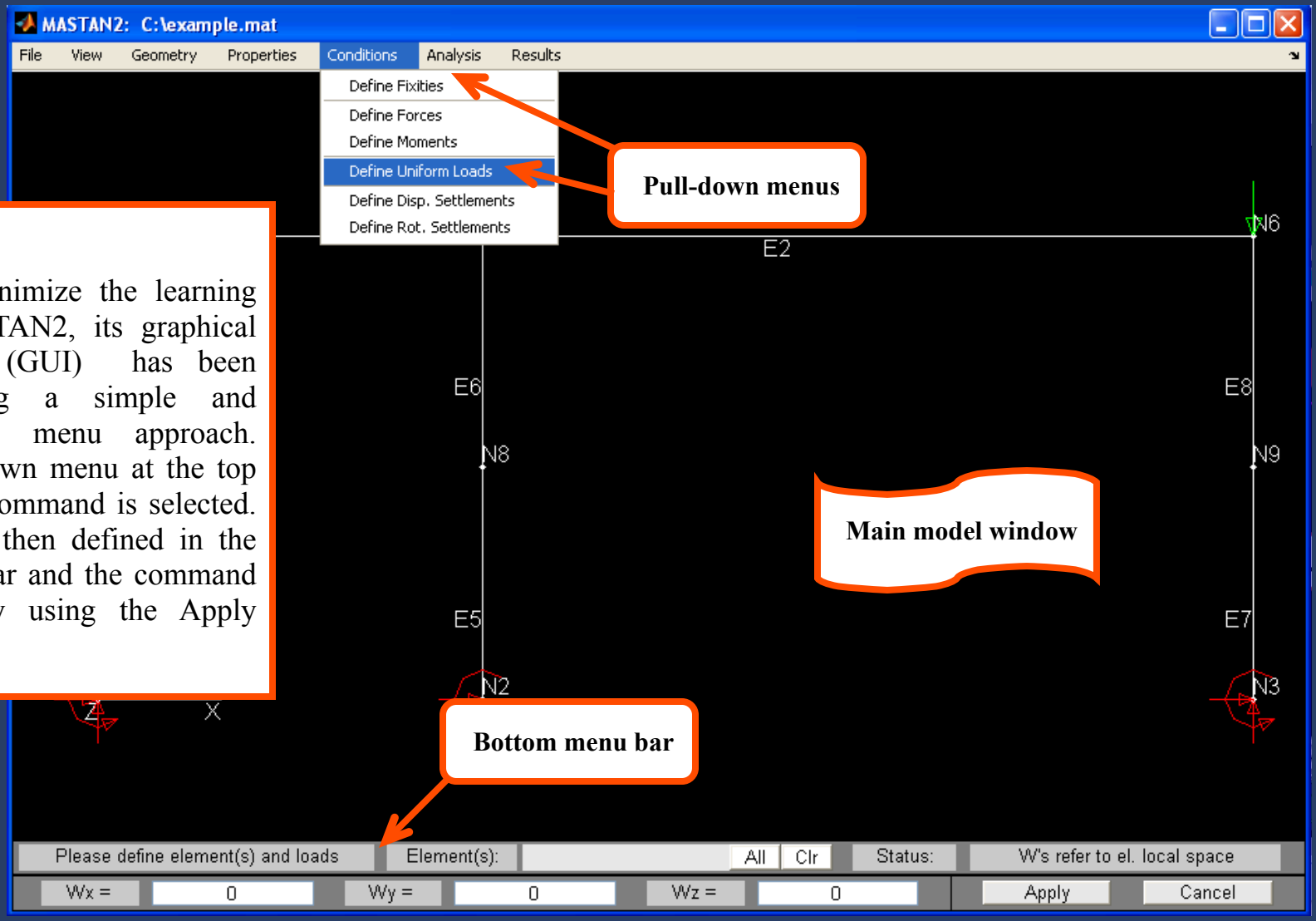
Two versions of MASTAN2 have been developed and may be installed. One requires you to have access to MATLAB (recommended) and the other does not. Please note that Installation Method 1 is required if you plan to develop and implement additional or alternative analysis routines that will directly interact with the MASTAN2.

Method 1 (Users who have access to MATLAB): Double click on the msav3p.zip file and extract all files into a MASTAN2 folder on your computer. Start the MATLAB program and set the **Current Directory** to the location of this MASTAN2 folder. To avoid having to do this each time you startup MATLAB, you can permanently add this folder to the MATLAB search path by selecting **File** and then **Set Path…** After using either of these procedures, type **mastan2** (only lower case letters with no spaces) at the MATLAB command line prompt (>>) and the MASTAN2 graphical user interface (GUI) should start. If the GUI does not start, and you get an error message that reads ***??? Undefined function or variable 'mastan2'***, you have not properly set the current directory or path to point to your MASTAN2 folder.

Method 2 (PC-Users who do not have access to MATLAB): A stand-alone version of MASTAN2 is also available. Double click on the msav3exe.zip file and then double click on the install.exe file. This will start an installer with simple step-by-step instructions. When the installation is complete, two icons will appear on your computer's desktop. Double click on the MASTAN2v3 icon to start MASTAN2. Note that it may take up to a minute for the program to initially start. The second icon provides access to an interactive tutorial. Note that this stand-alone version provides all the same functionality except that you cannot prepare user defined code that will interact with MASTAN2.

Window Layout

Overview:
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.



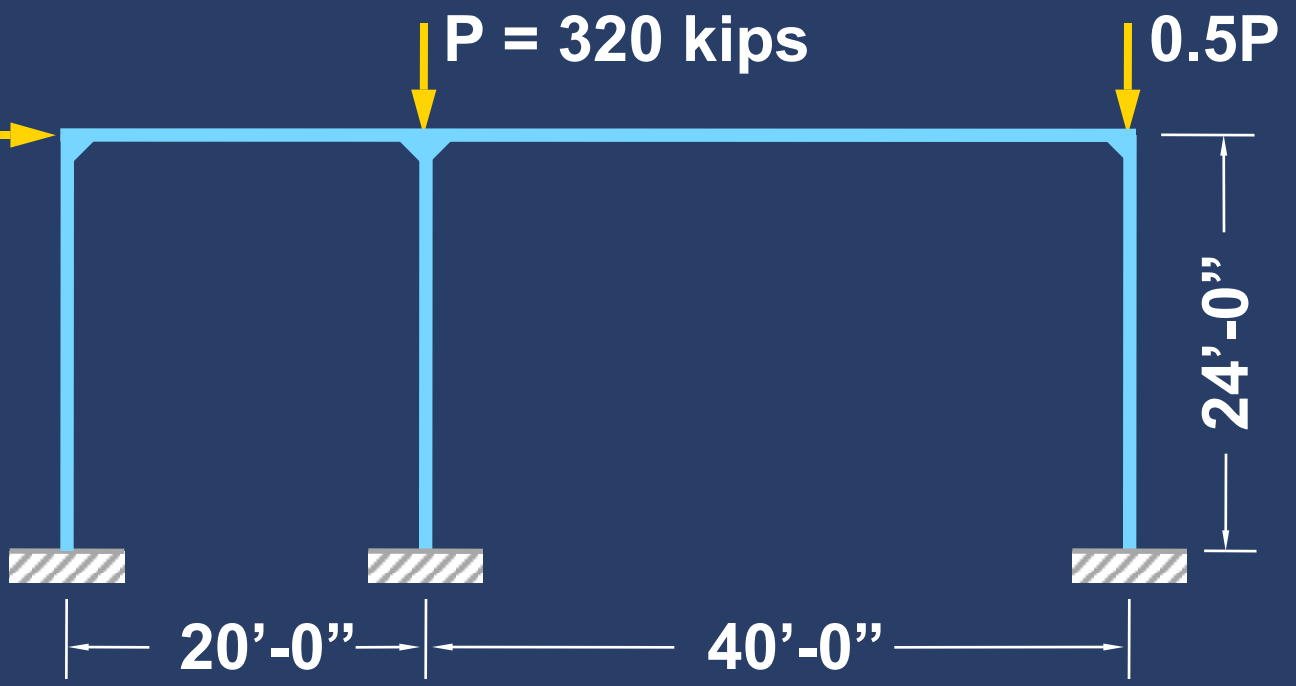
Step-by-Step Example

- Problem description
- Geometry definition
- Section and material properties
- Loads and support conditions
- First-order elastic analysis
- Results: diagrams, reports, and response curves
- Other methods of analysis

< click on a topic >

Problem Description

A two-bay single story frame will be used to illustrate several of the preprocessing, analysis, and postprocessing capabilities of MASTAN2.



Columns: W10x45
- $A = 13.3\ in^2$
- $I = 248\ in^4$
- $Z = 54.9\ in^3$

Girders: W27x84
- $A = 24.8\ in^2$
- $I = 2850\ in^4$
- $Z = 244\ in^3$

All members: A36 Steel
- $E = 29{,}000\ ksi$
- $\sigma_y = 36\ ksi$

Geometry Definition

Part I: Frame Definition

1. From the **Geometry** menu select **Define Frame**.
2. At the bottom menu bar, click in the edit box to the left of **bays @** and change the **0** to **2**. Click in the edit box just to the right of **bays @** and change the **0** to **240**.
3. Click in the edit box to the left of **stories @** and change the **0** to **1**. Click in the edit box just to the right of **stories @** and change the **0** to **288**.
4. Click on the **Apply** button.
5. A two-bay single story frame is now defined.

MASTAN2

Click these boxes to view the resulting windows

Notes:

a. Edit boxes will accept math expressions. For example, typing **24*12** is the same as typing **240**. In all cases, only one value may be executed in any edit box.

b. A three dimensional structure is defined by providing the number of frames (a value greater than 1) and the appropriate spacing.

c. Any consistent set of units may be used to define a model.


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
N4
N5
N6
E4
E5
E1
E2
E3
Y
N1
Z
X
N2
N3
Please enter repetitions and dimensions of structure
Status:
Success: Frame generated.
2
bays @
240
1
stories @
288
1
frames @
0
Apply
Cancel

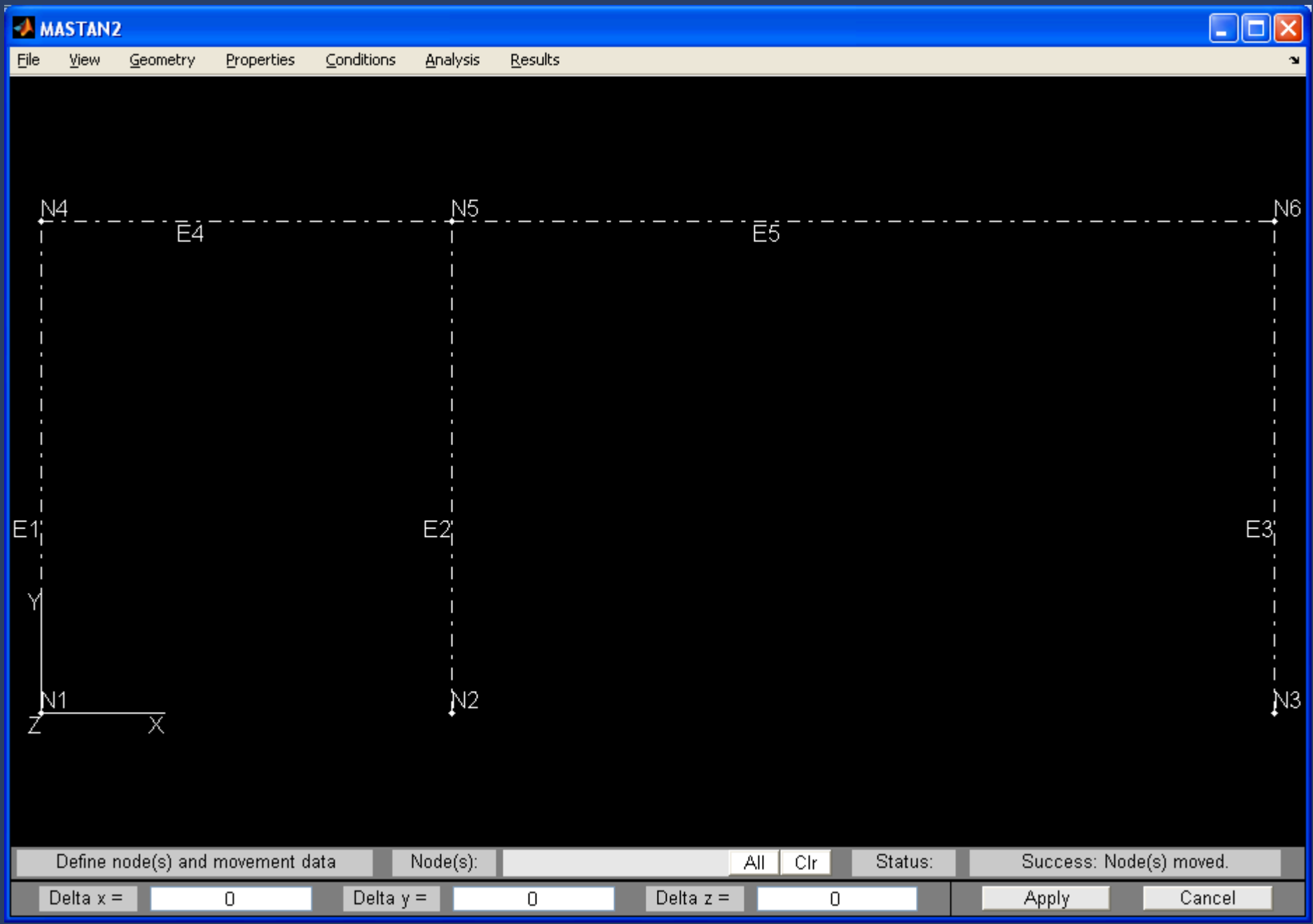
Geometry Definition (cont.)

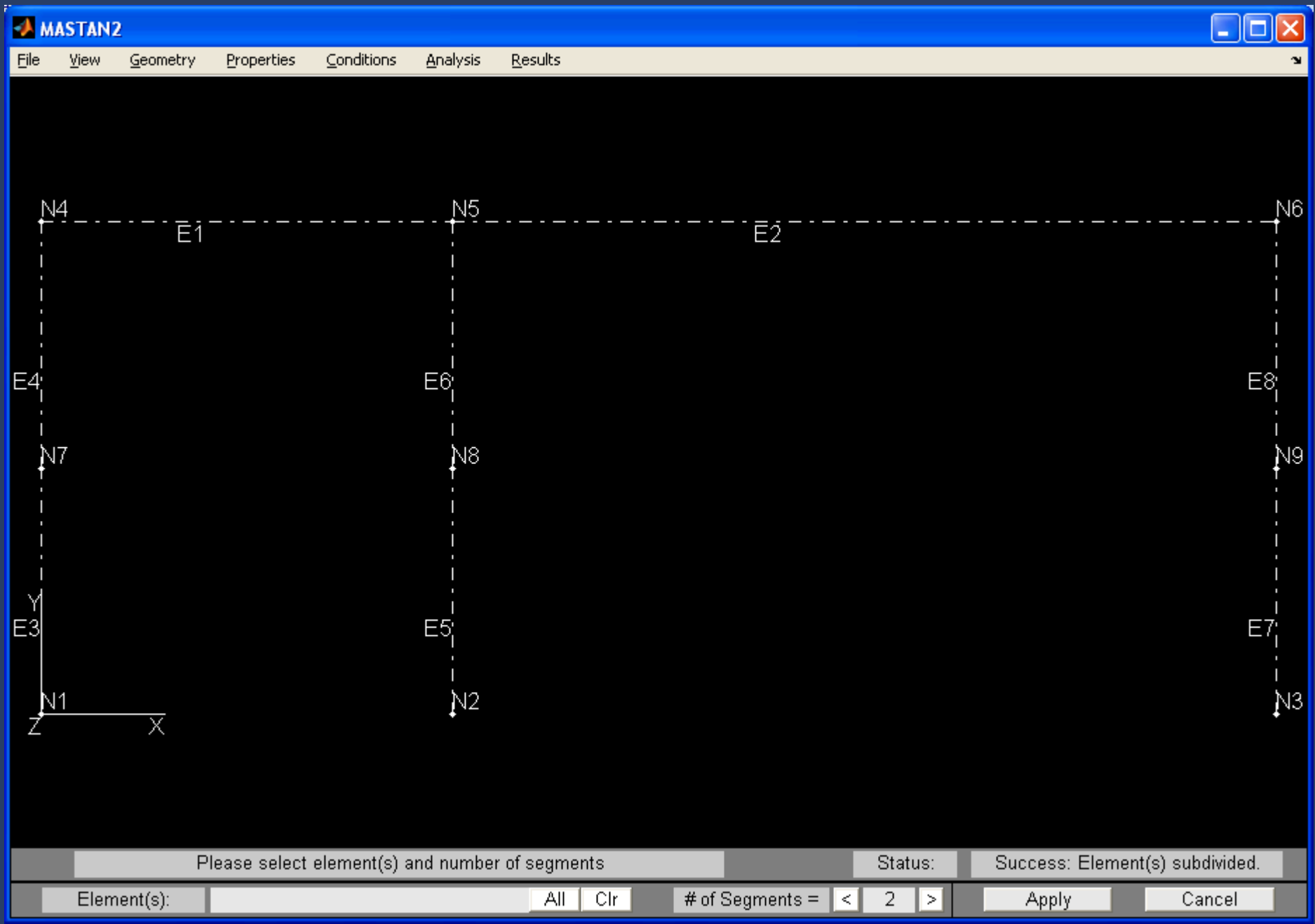
Part II: Refinement

1. From the **Geometry** menu select **Move Node(s)**.
2. At the bottom menu bar, click in the edit box to the right of **Delta x =** and change the **0** to **240**.
3. Create the list of nodes by clicking on the two rightmost nodes. Note that selected nodes (or elements) turn magenta and their numbers are added to the **Node(s):** list.
4. Click on the **Apply** button.
5. From the **View** menu select **Fit**. MASTAN2
6. From the **Geometry** menu select **Subdivide Element(s)**.
7. Create the list of elements by clicking on each vertical element.
8. Since the number of segments is already set at **2**, click on the **Apply** button. MASTAN2

Note:

To remove a node or element number from a list, click on it again. To remove all numbers from the node or element list, click on the **Clr** box to the right of **Node(s):** or **Element(s):**.


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
N4
N5
N6
E4
E5
E1
E2
E3
Y
N1
N2
N3
Z
X
Define node(s) and movement data
Node(s):
All
Clr
Status:
Success: Node(s) moved.
Delta x =
0
Delta y =
0
Delta z =
0
Apply
Cancel


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
N4
N5
N6
E1
E2
E4
E6
E8
N7
N8
N9
Y
E3
E5
E7
N1
N2
N3
Z
X
Please select element(s) and number of segments
Status:
Success: Element(s) subdivided.
Element(s):
All
Clr
# of Segments =
<
2
>
Apply
Cancel

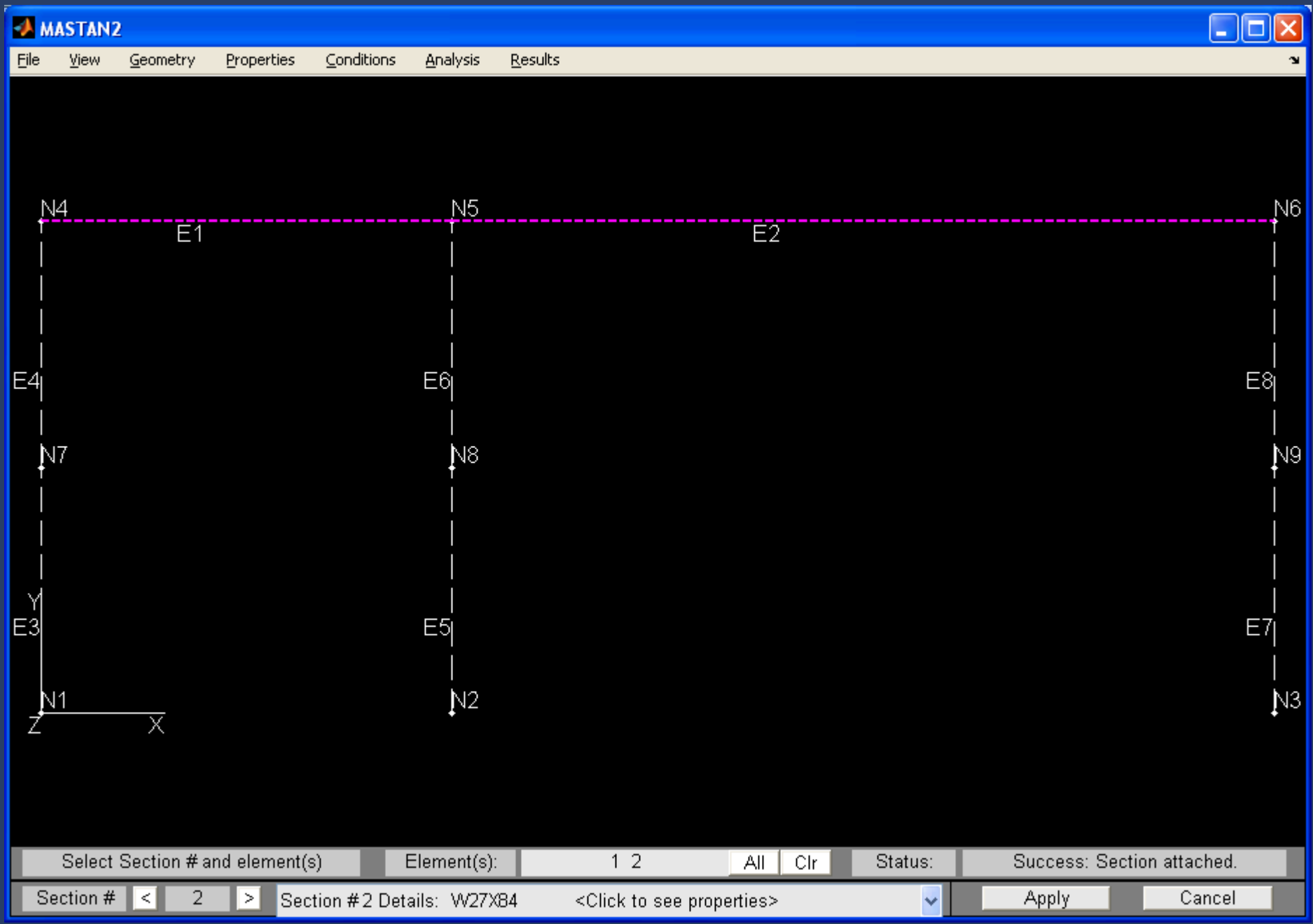
Section and Material Properties

Part I: Section Properties

1. From the **Properties** menu select **Define Section**.
2. At the bottom menu bar, click in the edit box just to the right of **Area =** and change the **0** to **13.3**. Similarly, define $\mathbf{I_{zz} = 248}$ and $\mathbf{Z_{zz} = 54.9}$. Click on the **Apply** button (Section 1 is now defined with the properties of a W10×45).
3. Repeat step 2 using **Area = 24.8**, $\mathbf{I_{zz} = 2850}$ and $\mathbf{Z_{zz} = 244}$. After clicking the **Apply** button, Section 2 will be defined with the properties of a W27×84.
4. From the **Properties** menu select **Attach Section**.
5. Create the list of elements to be assigned Section 1 by clicking on each vertical element. Click on the **Apply** button (note that elements with assigned section properties turn from dash-dot to dashed).
6. Advance the **Section #** by clicking on the **>** box. Select the **Clr** button located to the right of **Element(s):** to clear the list of element numbers.
7. Assign Section 2 properties to all horizontal elements by repeating step 5.

Notes: 1. Section properties refer to the element's local coordinate system with *x* being along its length axis, the *y*-axis oriented as shown by the element's web direction (see **View-Labels-Element Web**) and the *z*-axis defined by the right hand cross product of these *x*- and *y*-axes. 2. Although selecting a section from the **Database** will automatically type in all relevant properties, you must still click on the **Apply** button to define the section.

MASTAN2


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
N4
N5
N6
E1
E2
E4
E6
E8
N7
N8
N9
Y
E3
E5
E7
N1
N2
N3
Z
X
Select Section # and element(s)
Element(s):
1 2
All
Clr
Status:
Success: Section attached.
Section #
<
2
>
Section # 2 Details: W27X84
<Click to see properties>
Apply
Cancel

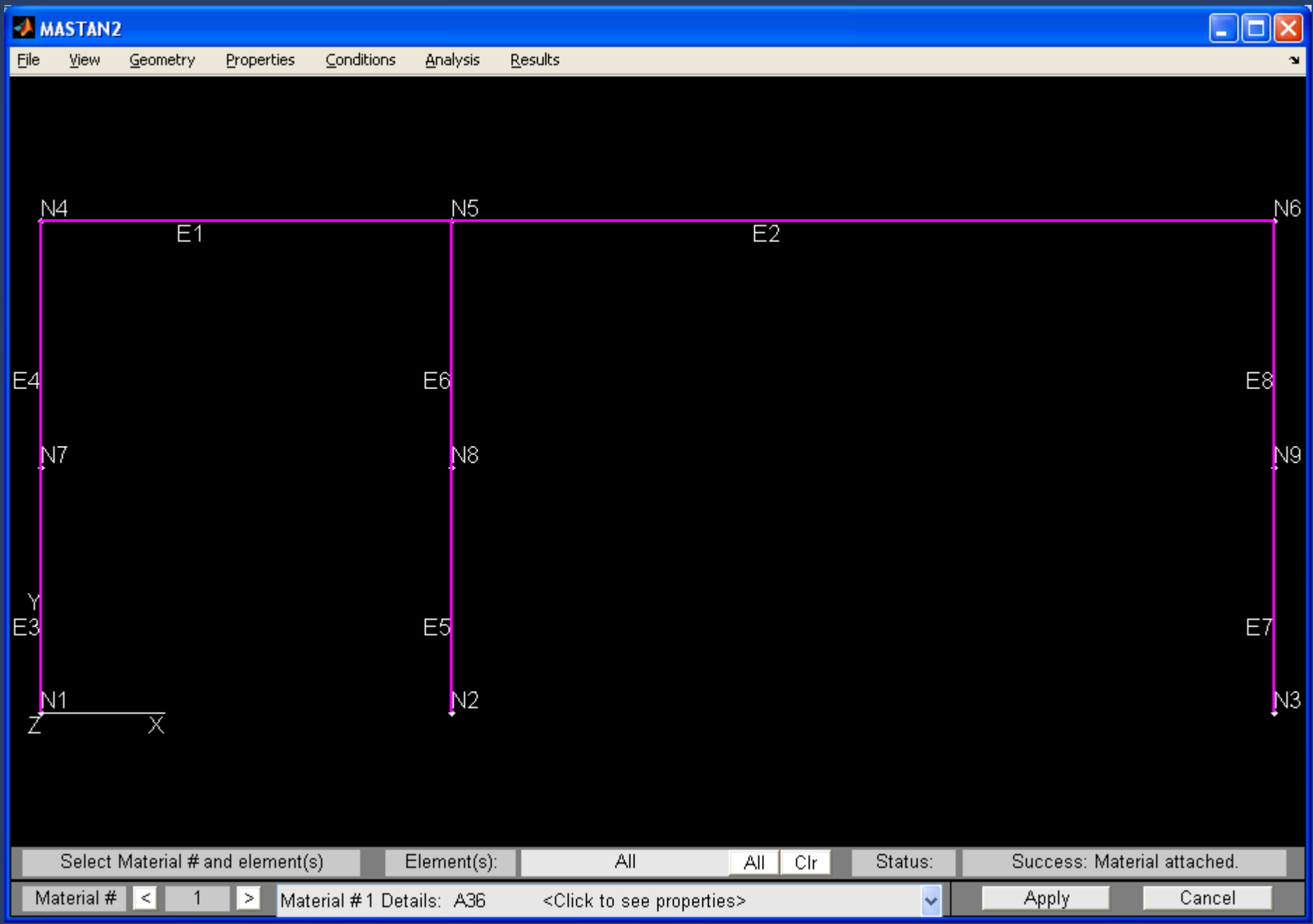
Section and Material Prop. (cont.)

Part II: 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, define **F_y = 36** and the **Name** of the material as **A36**. Click on the **Apply** button (Material #1 is now defined with the properties of A36 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 **Element(s):**. Click on the **Apply** button (note that elements with assigned section and material properties turn to solid). MASTAN2

Notes:

1. As indicated earlier, MASTAN2 will work for any consistent set of units. In this example all force units are in kips and all length units are in inches.
2. Similar to section properties, properties for more than one material can be defined and assigned to different elements.
3. Definition and attached elements of section and material properties may be confirmed with **Properties-Information-Section**. or **Properties-Information-Material**.


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
N4
N5
N6
E1
E2
E4
E6
E8
N7
N8
N9
Y
E3
E5
E7
N1
N2
N3
Z
X
Select Material # and element(s)
Element(s):
All
All
Clr
Status:
Success: Material attached.
Material #
<
1
>
Material # 1 Details: A36
<Click to see properties>
Apply
Cancel

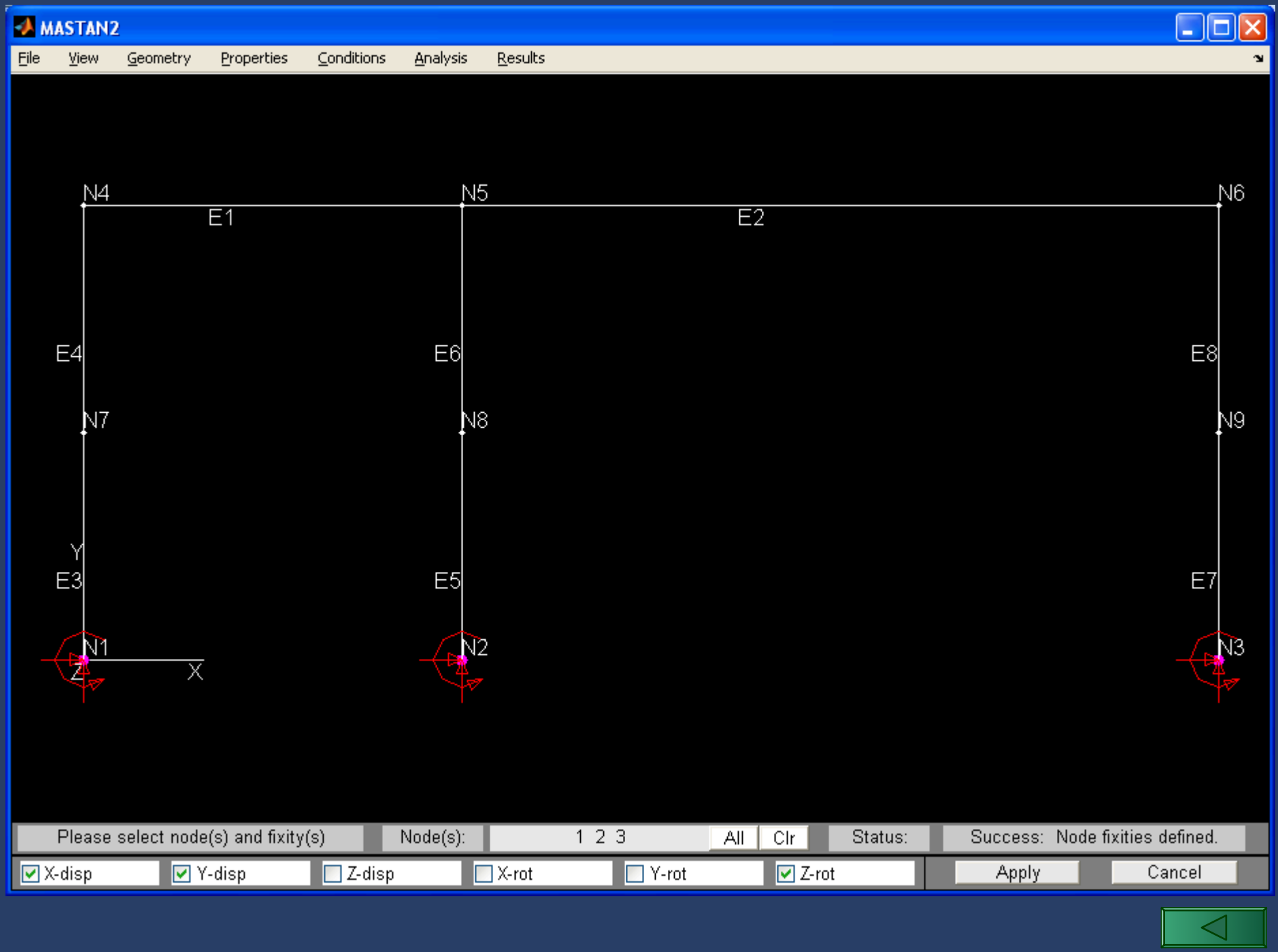
Loads and Support Conditions

Part I: 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 **X-disp**, **Y-disp**, and **Z-rot**.
3. Create the list of nodes to be assigned these fixities by clicking on the bottom three nodes of the model.
4. Click on the **Apply** button.
5. From the **View** menu select **Fit**. MASTAN2

Notes:

1. Red arrows indicate the degrees of freedom at a node that are restrained.
2. MASTAN2 provides the opportunity to analyze structures as two or three dimensional. For two dimensional analyses, only degrees of freedom in the x-y plane need to be restrained. On a related topic, additional section properties would be needed to analyze this system as three-dimensional.


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
N4
N5
N6
E1
E2
E4
E6
E8
N7
N8
N9
Y
E3
E5
E7
N1
N2
N3
Z
X
Please select node(s) and fixity(s)
Node(s):
1 2 3
All
Clr
Status:
Success: Node fixities defined.
X-disp
Y-disp
Z-disp
X-rot
Y-rot
Z-rot
Apply
Cancel

Loads and Support Cond. (cont.)

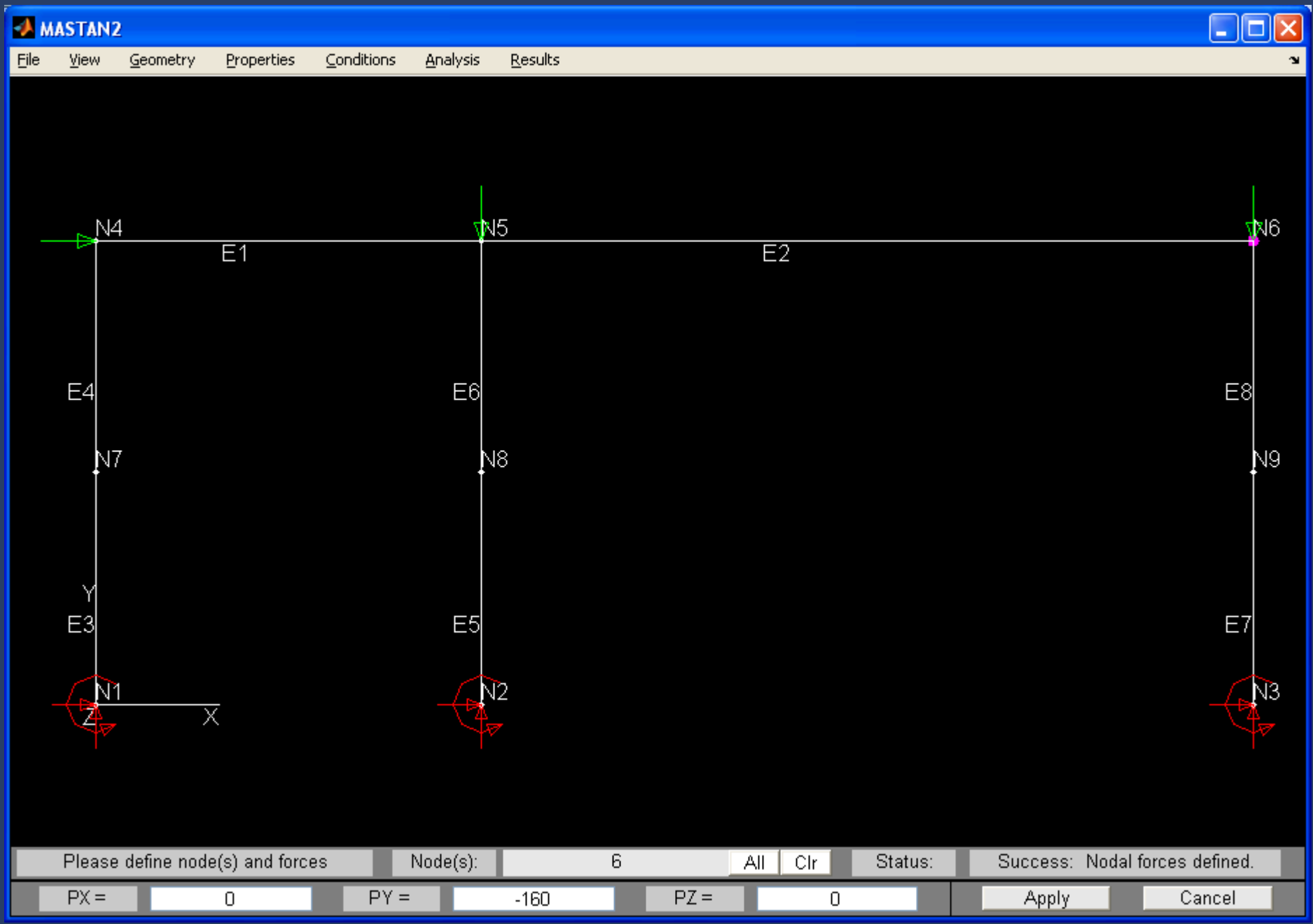
Part II: Loads

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 **32**.
3. Create the list of nodes to be assigned this force by clicking on the upper left node of the model. Click on the **Apply** button.
4. Click in the edit box just to the right of **PX =** and change the **32** to **0** and then click in the edit box just to the right of **PY =** and change the **0** to **-320**.
5. Create the list of nodes to be assigned this force by first clearing the node list by clicking on the **Clr** button and then clicking on the node at the top of the center column. Click on the **Apply** button.
6. Repeat steps 4 and 5 using **PY = -160** and applying this force to the upper right node of the model. From the **View** menu select **Fit**.

MASTAN2

Notes:

1. To remove a support or load condition from a node or group of nodes, first create the node list and then with all conditions blank (for support) or zero (for load), click on **Apply**.
2. Green arrows represent applied forces.
3. The conditions at a node may be checked with **Geometry-Information-Node**.


MASTAN2
File
View
Geometry
Properties
Conditions
Analysis
Results
N4
N5
N6
E1
E2
E4
E6
E8
N7
N8
N9
Y
E3
E5
E7
N1
N2
N3
Z
X
Please define node(s) and forces
Node(s):
6
All
Clr
Status:
Success: Nodal forces defined.
PX =
0
PY =
-160
PZ =
0
Apply
Cancel

First-Order Elastic Analysis

1. From the **Analysis** menu select **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.

Although the following steps are not required for us to proceed, this is a good time to perform them.

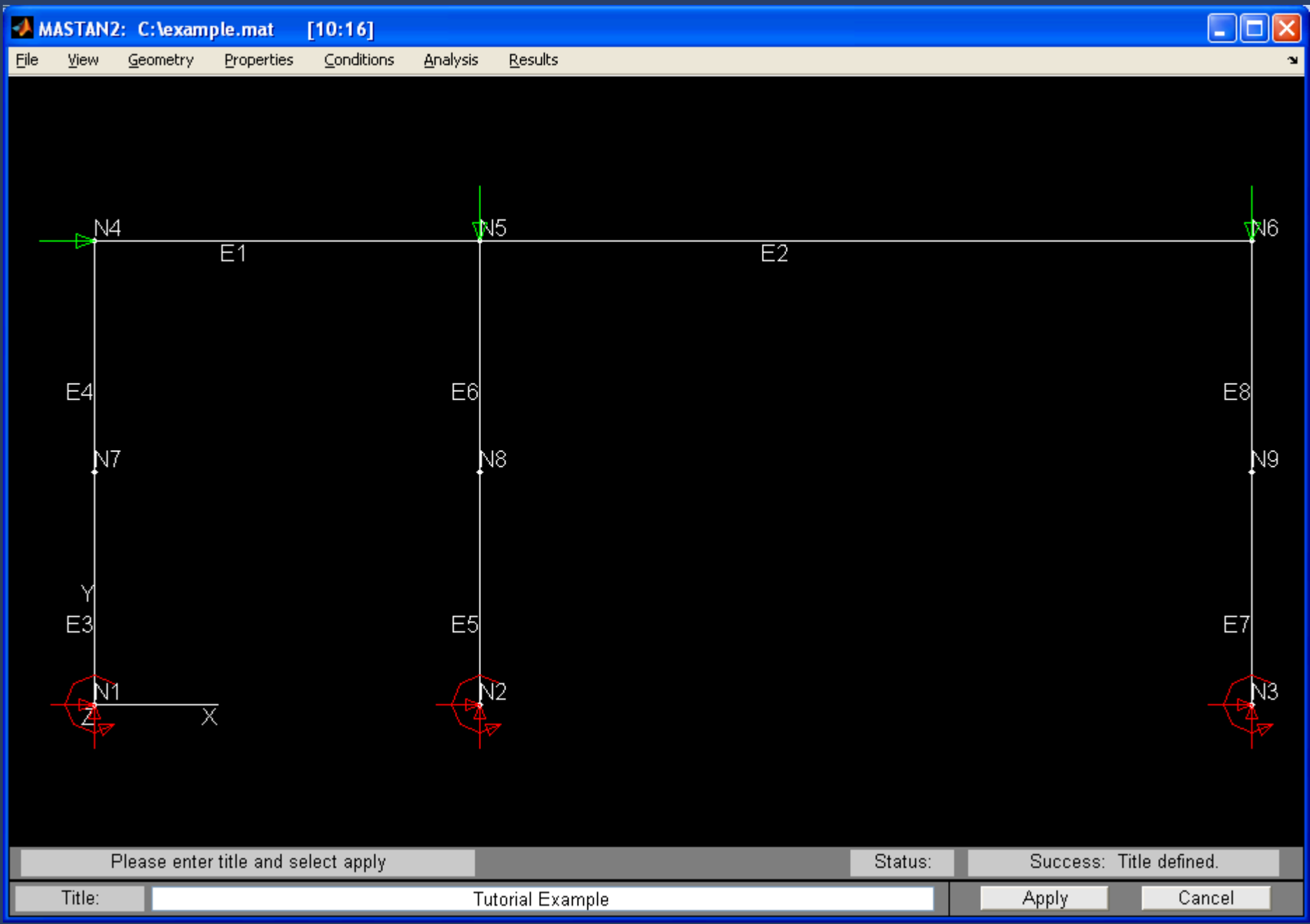
a. From the **File** menu select **Define Title**. At the bottom menu bar, click in the edit box to the right of **Title:** and type in a brief description of this effort. This text might include the model title, your name, and/or the assignment number. Click on the **Apply** button.

b. From the **File** menu select **Save As....** After selecting your destination folder, type in the filename **example** 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

Note:

Only alpha-numeric file names may be used.


MASTAN2: C:\example.mat [10:16]
File View Geometry Properties Conditions Analysis Results
N4
N5
N6
E1
E2
E4
E6
E8
N7
N8
N9
Y
E3
E5
E7
N1
N2
N3
Z
X
Please enter title and select apply
Status:
Success: Title defined.
Title:
Tutorial Example
Apply
Cancel

Results

MASTAN2 has several postprocessing capabilities. A sampling of them and their use are illustrated below.

- Deflected shape and node displacements/reactions
- Force diagrams and element force information
- Printing photos and creating a text report
- Plotting response curves with MSAPLOT

< click on a topic >

Deflections and Reactions

Part I: Deflected Shape

1. From the **Results** menu select **Diagrams** and submenu **Deflected Shape**.
2. At the bottom menu bar, click on the **Apply** button. MASTAN2

Part II: Displacement Values at a Node

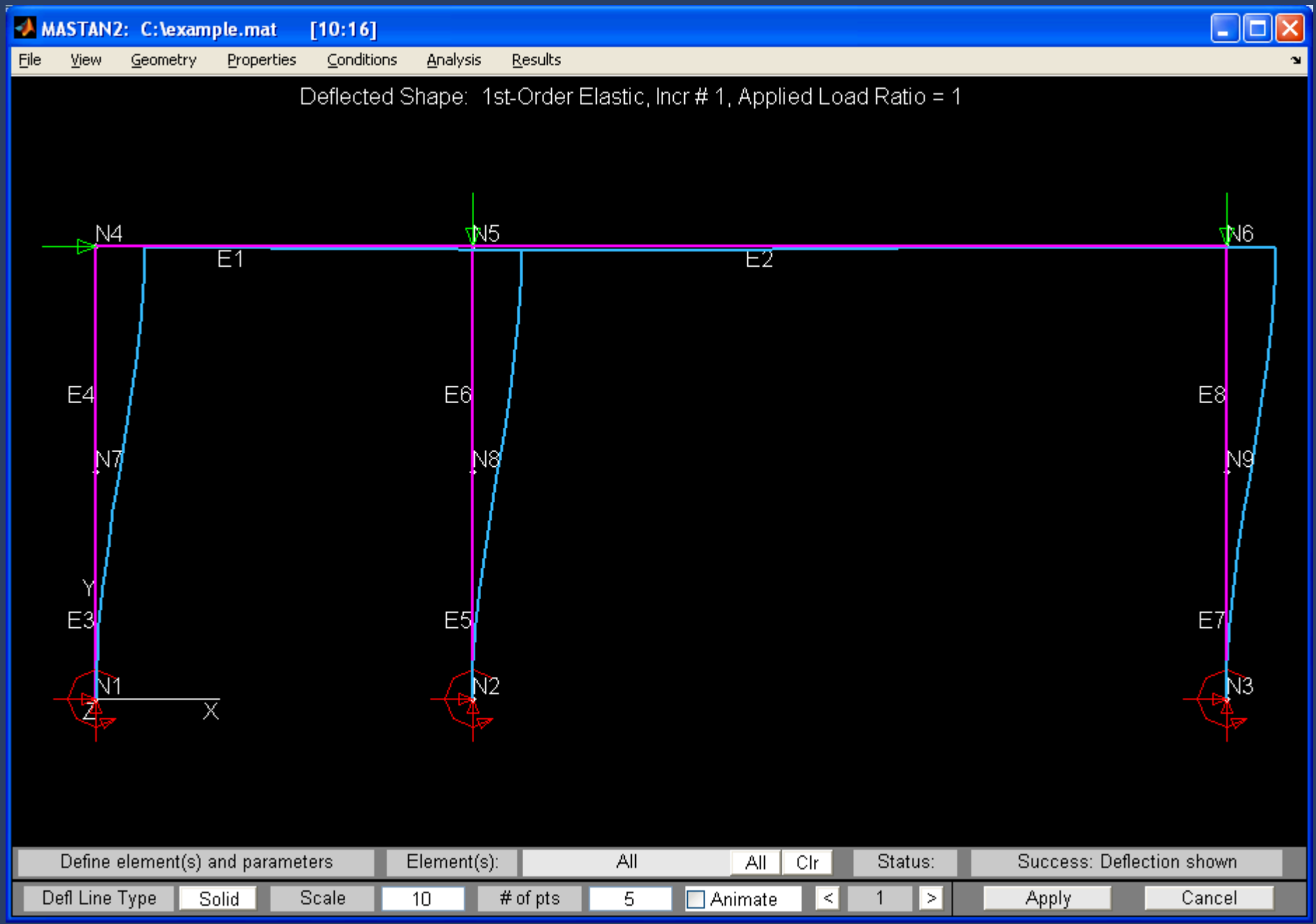
1. From the **Results** menu select **Node Displacements**.
2. On the undeflected shape, click on a node of interest and its displacement components are provided in the bottom menu bar. Repeat for other nodes.

Part III: Reactions at a Node

1. From the **Results** menu select **Node Reactions**.
2. Click on a node of interest and any applicable reaction components are provided in the bottom menu bar. Repeat for other nodes.

Notes:

1. The scale of the deflected shape may be changed by editing the number to the right of **Scale** and clicking on the **Apply** button.
2. A smoother diagram can be obtained by increasing the value to the right of **# of pts** and clicking on the **Apply** button.
3. As an alternative to step 2 in above Parts II and III, displacement and reaction components at a node can be obtained by typing the node number in the edit box to the right of **Node:** and then clicking on the **Apply** button.


MASTAN2: C:\example.mat [10:16]
File View Geometry Properties Conditions Analysis Results
Deflected Shape: 1st-Order Elastic, Incr # 1, Applied Load Ratio = 1
N4
N5
N6
E1
E2
E4
E6
E8
N7
N8
N9
Y
E3
E5
E7
N1
N2
N3
Z
X
Define element(s) and parameters
Element(s):
All
All
Clr
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
10
# of pts
5
Animate
<
1
>
Apply
Cancel

Element Force Diagrams and Values

Part I: Moment Diagram

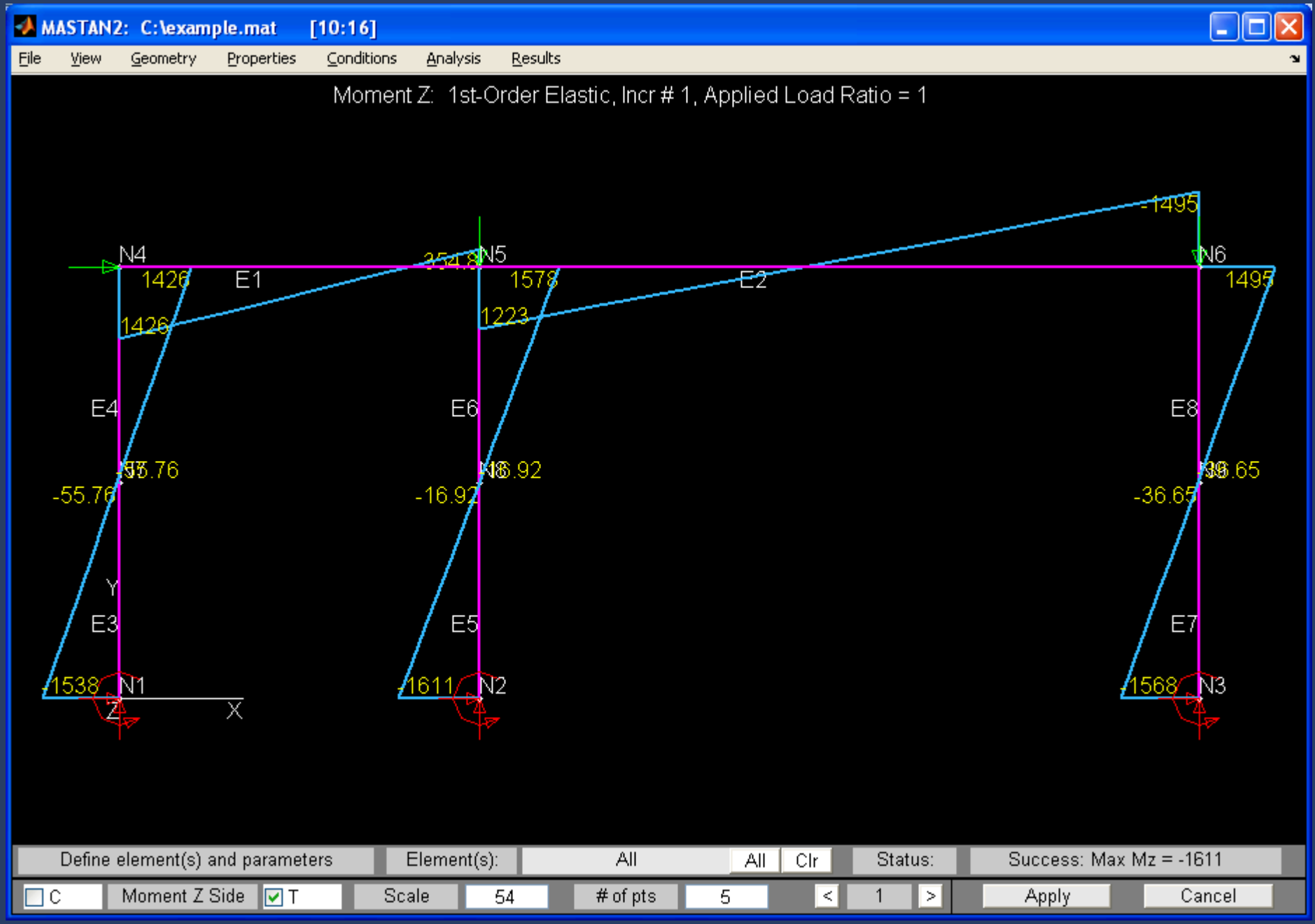
1. From the **Results** menu select **Diagrams** and submenu **Moment Z**.
2. At the bottom menu bar, click on the **C** or **T** box between **Moment Z Side** depending on whether you want the moment diagram drawn on the compression or tension side of the member.
3. Click on the **Apply** button. From the **View** menu select **Fit**. MASTAN2

Part II: Internal Element Forces and Moments

1. From the **Results** menu select **Element Results**.
2. Click on an element of interest and its internal forces at the start node of the element are provided in the bottom menu bar. Repeat for other elements.
3. To view element forces at a location along the length of the element including the end node, move the slider at the lower left of the bottom menu so it reads the desired fraction of the element length and click **Apply**.

Notes:

1. Moment diagram values may be turned on and off with **View-Labels-Diagram Values**.
2. As an alternative to step 2 in Part II, element forces can be obtained by typing the element number in the edit box to the right of **El #** and then clicking on the **Apply** button.


MASTAN2: C:\example.mat [10:16]
File View Geometry Properties Conditions Analysis Results
Moment Z: 1st-Order Elastic, Incr # 1, Applied Load Ratio = 1
N4
N5
N6
N1
N2
N3
E1
E2
E3
E4
E5
E6
E7
E8
1426
1426
1578
1223
1495
-1495
-55.76
-16.92
-36.65
-1538
-1611
-1568
Y
Z
X
Define element(s) and parameters
Element(s):
All
All
Clr
Status:
Success: Max Mz = -1611
C
Moment Z Side
T
Scale
54
# of pts
5
<
1
>
Apply
Cancel

Photos and Text Reports

I. Printing Photos

1. To print a photo of the main model window, select **Print Photo…** from the **File** menu. Note that the title is also printed at the base of the photo.

II. Creating Text Reports

1. From the **File** menu select **Create Report...**.
2. At the bottom menu bar, click on the check boxes just to the left of the desired information.
3. Click on the **Apply** button and this information is printed to the main text window. Use the scroll button to move up or down in the report.
4. To save the text report to a file that can be read and, in turn, printed by any word processor or text editor, click on the **Save Text** button and provide a destination folder and file name.
5. Click on the **Cancel** button to return to the main model window.

MASTAN2

Note:

Information printed to the main text window will remain, even after the **Cancel** button is clicked, until the **Clear** button is clicked. In this way, additional information such as the results from a different analysis can be added later.

MASTAN2: C:\example.mat [10:16]

File View Geometry Properties Conditions Analysis Results

```
***********   MASTAN2 v3.0.7   ***********

Time:  10:23:19      Date:  01/10/2007

Problem Title:    Tutorial Example
**************

============================
Input for Structural Analysis
============================

General Information Categories:

(i)   Number of Nodes = 9
(ii)  Number of Elements = 8
(iii) Number of Sections = 2
(iv)  Number of Materials = 1
(v)   Number of Supports = 3
(vi)  Applied Loads

(i)   Node Information

  Coordinates
     Node          X               Y               Z
       1     0.0000e+000     0.0000e+000     0.0000e+000
       2     2.4000e+002     0.0000e+000     0.0000e+000
       3     7.2000e+002     0.0000e+000     0.0000e+000
       4     0.0000e+000     2.8800e+002     0.0000e+000
       5     2.4000e+002     2.8800e+002     0.0000e+000
       6     7.2000e+002     2.8800e+002     0.0000e+000
       7     0.0000e+000     1.4400e+002     0.0000e+000
       8     2.4000e+002     1.4400e+002     0.0000e+000
       9     7.2000e+002     1.4400e+002     0.0000e+000

  Fixities
     Node      X       Y       Z      X Rot    Y Rot    Z Rot
       1     FIXED   FIXED   FREE     FREE     FREE     FIXED
       2     FIXED   FIXED   FREE     FREE     FREE     FIXED
       3     FIXED   FIXED   FREE     FREE     FREE     FIXED
       4     FREE    FREE    FREE     FREE     FREE     FREE
```

Select Requested Information and Apply | Status: | Success: Data printed to screen.

☑ General Info. ☑ Geometry ☑ Properties ☑ Conditions Save Text Clear Text

☑ Displacements ☑ Element Results ☑ Reactions Incr # < 1 > Apply Cancel

Plotting with MSAPLOT

1. To use the plotting module that is provided with MASTAN2, select **MSAPlot** from the **Results** menu.

Part I. Axes Definition

1. From the MSAPlot **Curves** menu select **Define X-Data**.
2. At the center of the bottom menu bar, click on the pop-up menu and select **Displacement**.
3. Click in the edit box to the right of **Node #** and type **4**.
4. Click on the **Apply** button (x-axis is now defined but nothing plotted).
5. Repeat steps 1 to 4, using **Define Y-Data** to monitor the **Applied Force or Moment** above the center column. Set **Node #** to **5**, **d.o.f.** to **y** (vertical force), and the scale to **-1** (to plot in upper right quadrant).

Notes:

1. In MSAPlot, all node and element numbers must be typed; clicking on a node or element in the MASTAN2 window will not automatically enter its number in a MSAPlot menu.
2. If an error is made while using **Define**, redefine the parameters and select **Apply**.
3. By also using **Define Z-Data**, MSAPlot can create three-dimensional plots.

Plotting with MSAPLOT (cont.)

Part II. Generate a Curve

1. From the MSAPlot **Curves** menu select **Generate Curve(s)**.
2. Click in the edit box to the right of **Label** and type **1st-Order Elastic** (or some other description to appear in the plot's legend).
3. Click on the **Apply** button and the response curve is drawn. MASTAN2

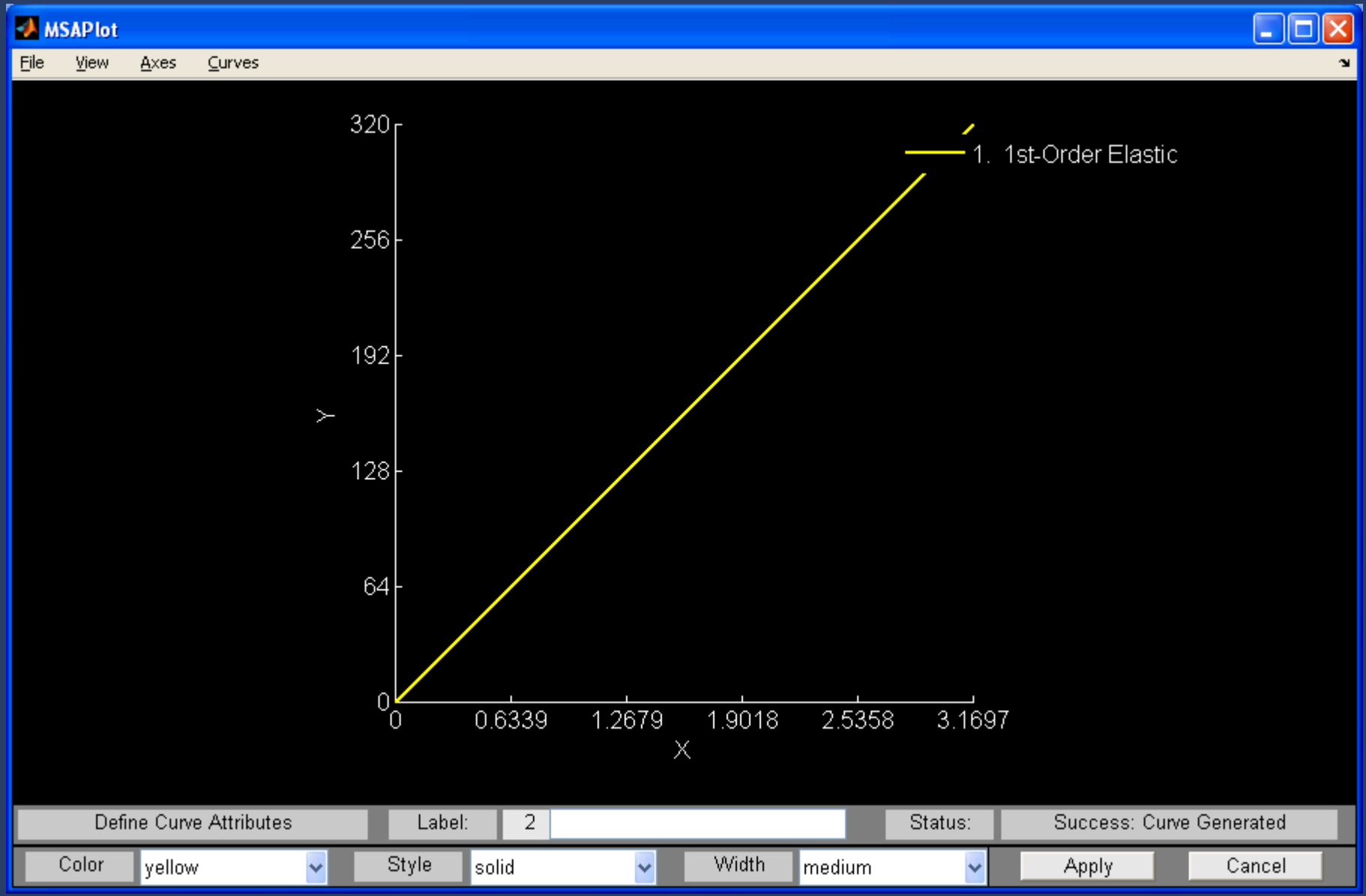
Part III. Plot Attributes

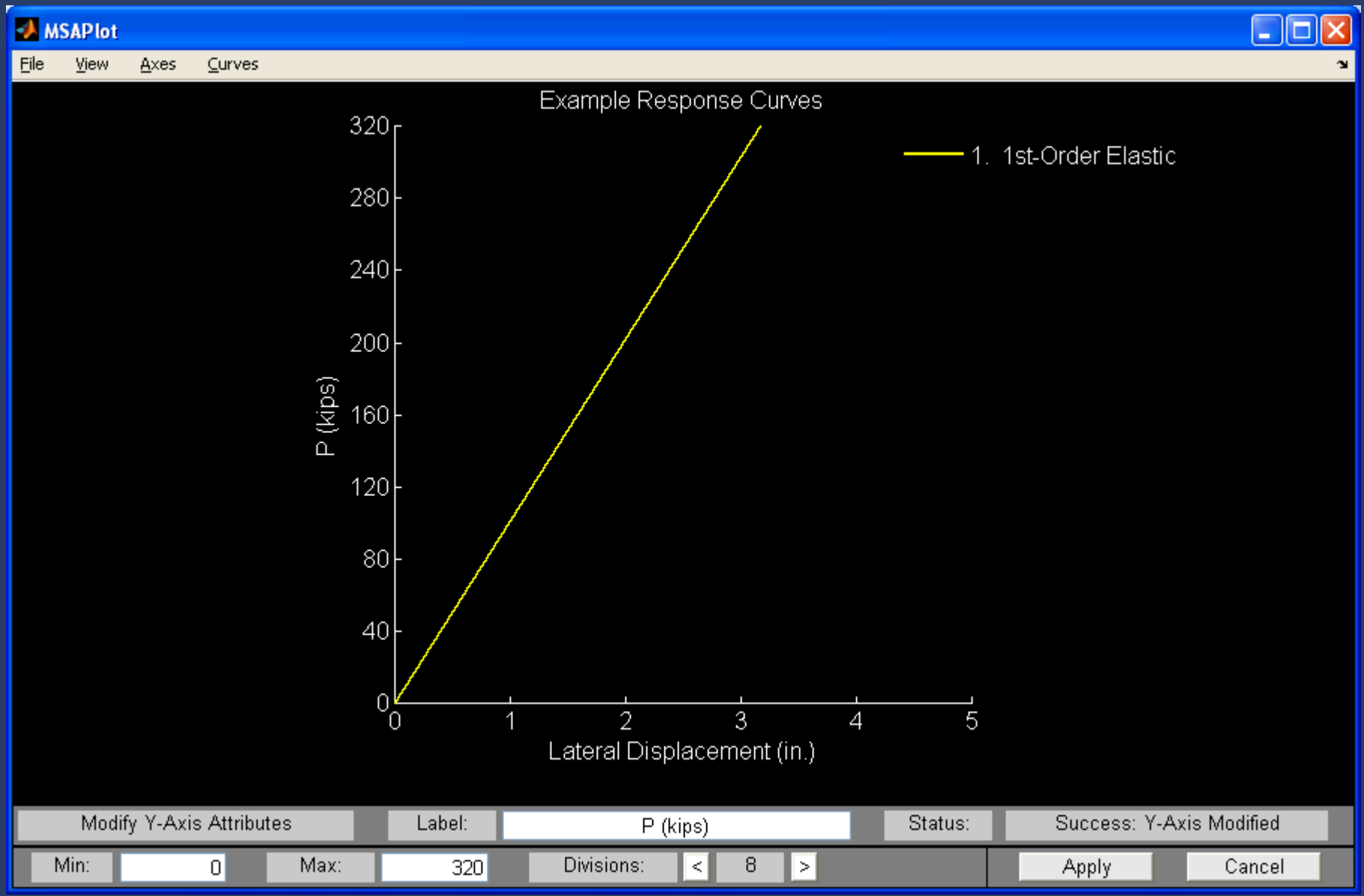
1. From the **Axes** menu select **Plot Title**.
2. At the bottom menu bar, click on edit box and enter a title.
3. Click on the **Apply** button.
4. From the **Axes** menu select **X-Attributes**.
5. Click on the edit box to the right of **Label** and change **X** to **Lateral Displacement (in.)**. Click on the edit box to the right of **Max:** and type **5**.
6. Click on the **Apply** button.
7. Repeat steps 4 to 6, using **Y-Attributes** to define the y-label as **P (kips)** and increasing the number of **Divisions** to **8**.

MASTAN2

Note:

The legend can be dragged to anywhere on the screen by clicking on it and holding the mouse button down to move it.


MSAPlot
File
View
Axes
Curves
320
256
192
128
64
0
Y
0
0.6339
1.2679
1.9018
2.5358
3.1697
X
1. 1st-Order Elastic
Define Curve Attributes
Label:
2
Status:
Success: Curve Generated
Color
yellow
Style
solid
Width
medium
Apply
Cancel


MSAPlot
File
View
Axes
Curves
Example Response Curves
1. 1st-Order Elastic
P (kips)
320
280
240
200
160
120
80
40
0
0
1
2
3
4
5
Lateral Displacement (in.)
Modify Y-Axis Attributes
Label:
P (kips)
Status:
Success: Y-Axis Modified
Min:
0
Max:
320
Divisions:
<
8
>
Apply
Cancel

Other Methods of Analysis

MASTAN2 provides seven different methods of analysis. These will be illustrated by using the current example problem and plotted results.

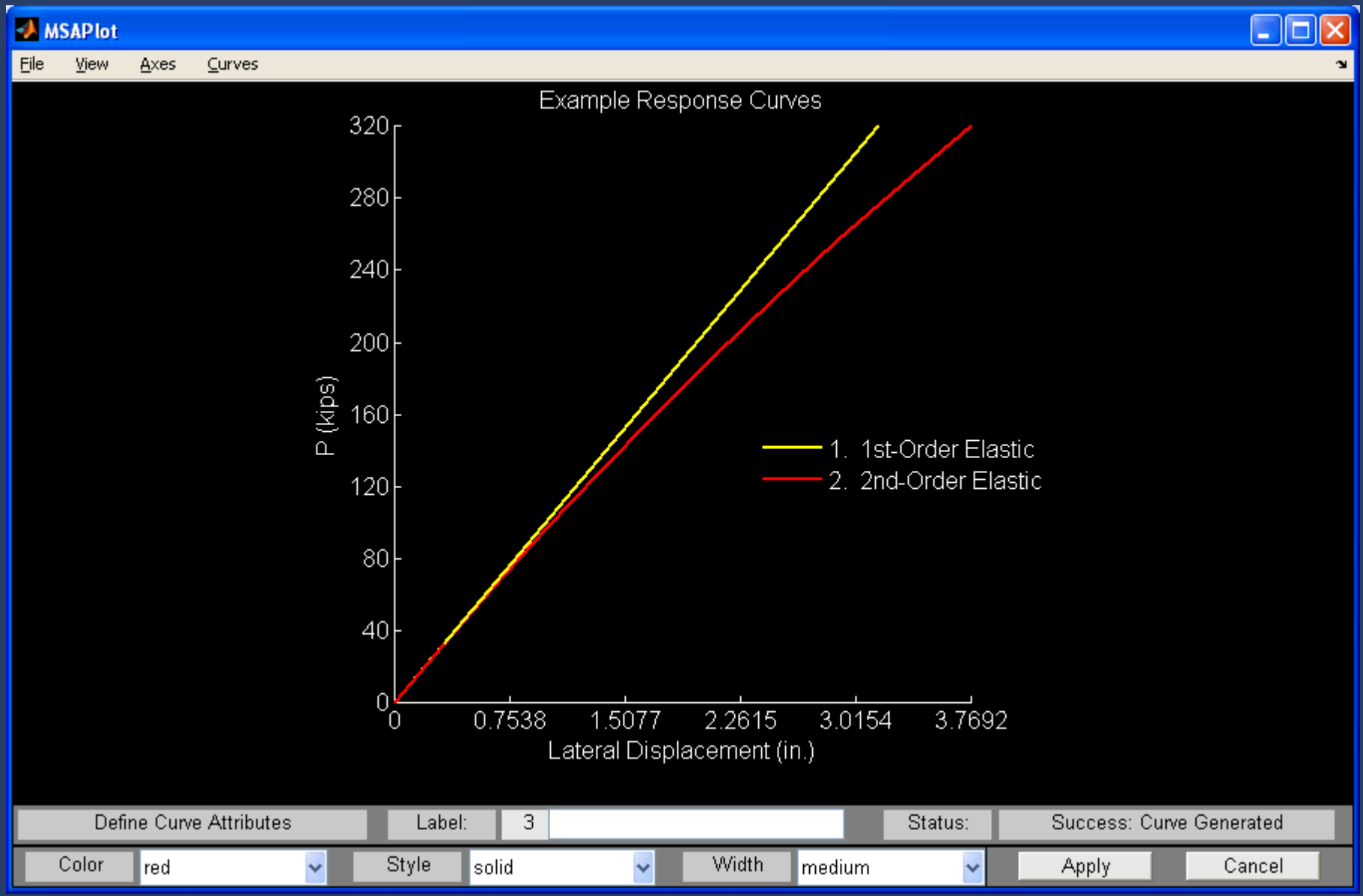
Part I. Second-order Elastic

1. From the MASTAN2 **Analysis** menu select **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 **MSAPlot**.
5. From the MSAPlot **Curves** menu select **Generate Curve(s)**.
6. At the bottom menu bar, click in the edit box to the right of **Label** and type **2nd-Order Elastic**.
7. Click on the pop-up menu just to the right of **Color** and select **red**.
8. Click on the **Apply** button and the response curve is added to the plot.

MASTAN2

Notes:

1. Steps 4 to 8 assume that the x- and y-data plot parameters were defined as previously described.
2. Diagrams, specific node and element results, and reports can be generated for all methods of analysis in the same manner as they were for the first-order elastic analysis.


MSAPlot
File
View
Axes
Curves
Example Response Curves
320
280
240
200
160
120
80
40
0
P (kips)
0
0.7538
1.5077
2.2615
3.0154
3.7692
Lateral Displacement (in.)
1. 1st-Order Elastic
2. 2nd-Order Elastic
Define Curve Attributes
Label:
3
Status:
Success: Curve Generated
Color
red
Style
solid
Width
medium
Apply
Cancel

Other Methods of Analysis (cont.)

Part II. First-order Inelastic

1. From the MASTAN2 **Analysis** menu select **1st-Order Inelastic**.
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 edit box to the right of **Max # of Incrs:** and change the **10** to **20**. The analysis will stop when either excessive deflections are detected or 20 load increments are applied or a maximum applied load ratio (**Max. Appl. Ratio**) of **1.0** is reached.
4. Click on the **Apply** button to perform the analysis. Note the analysis stops as a result of **Excessive Deflections** (most likely indicating the formation of a mechanism). Click on **No** to discontinue the analysis.
5. Note that the analysis stopped after 14 load increments. Click on the pop-up menu just to the left of **Apply** and select **Continue Prev**.
6. Click on the edit box to the right of **Max # of Incrs:** and change **20** to **15**. This will let the analysis run for one additional increment.

< move to next slide for additional instructions >

Other Methods of Analysis (cont.)

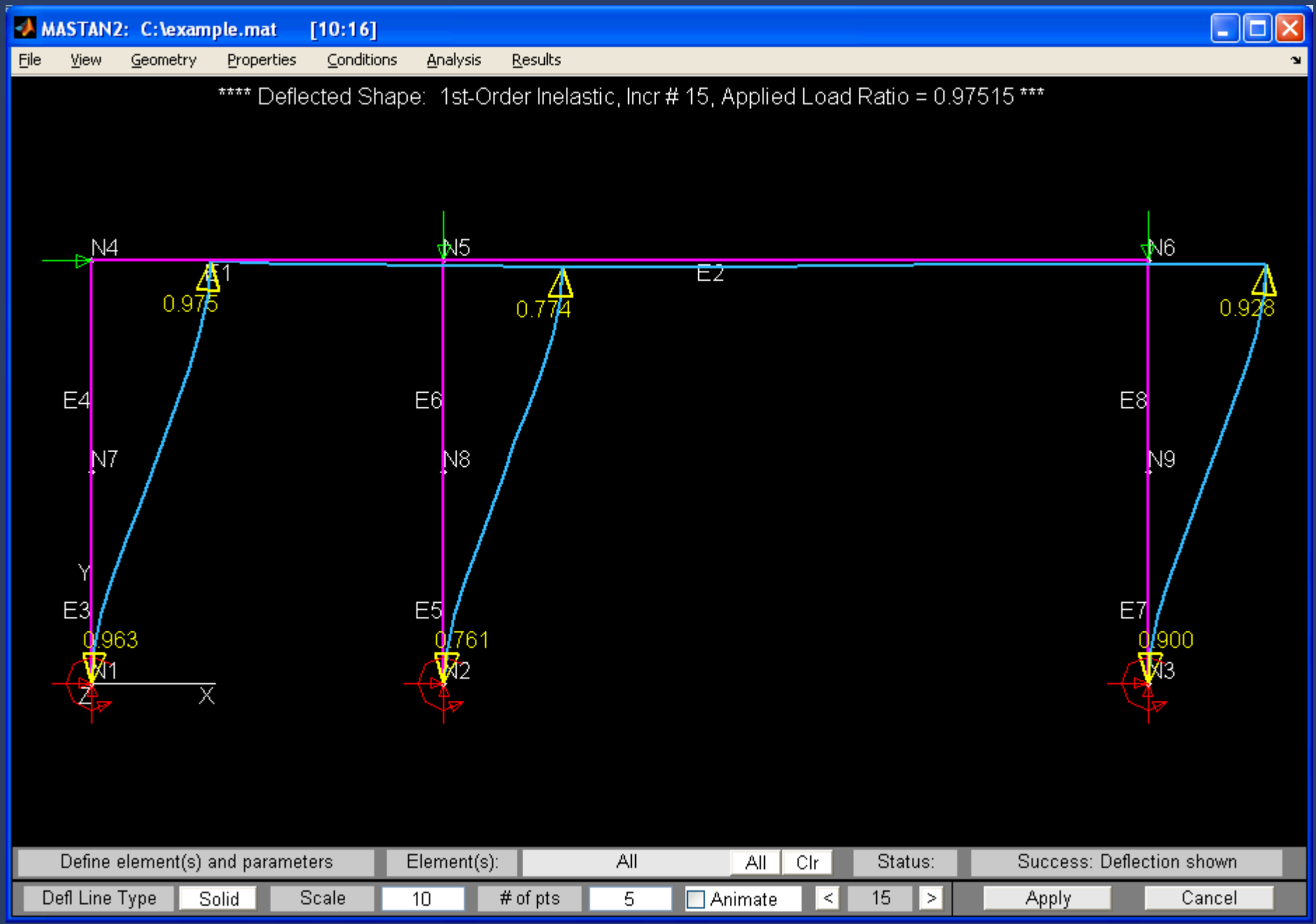
Part II. First-order Inelastic (cont.)

7. Click on the **Apply** button to continue the analysis. Note that the analysis stops again as result of **Excessive Deflections**. This time click on **Yes** to continue the analysis. As expected, the analysis immediately stops because the maximum number of load increments (**15**) has been reached.
8. From the **Results** menu select **Diagrams** and submenu **Deflected Shape**.
9. At the bottom menu bar, click on the **Apply** button. From the **View** menu select **Fit**. The deflected shape is shown along with the location of plastic hinges. Values indicate the load ratios when the hinges formed.
10. Click on the **<** at the lower right of the bottom menu and then click on **Apply** to view deflected shapes for previous load increments.
11. From the **Results** menu select **MSAPlot**.
12. At the bottom menu bar, click in the edit box to the right of **Label** and type **1st-Order Inelastic**.
13. Change the **Color** to **blue** and click on the **Apply** button. The response curve for this analysis is added to the plot.

MASTAN2

Note:

When diagrams are drawn, a descriptive label appears at the top of the MASTAN model window.


MASTAN2: C:\example.mat [10:16]
File View Geometry Properties Conditions Analysis Results
**** Deflected Shape: 1st-Order Inelastic, Incr # 15, Applied Load Ratio = 0.97515 ***
N4
N5
N6
E1
E2
0.975
0.774
0.928
E4
E6
E8
N7
N8
N9
Y
E3
E5
E7
0.963
0.761
0.900
N1
N2
N3
Z
X
Define element(s) and parameters
Element(s):
All
All
Clr
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
10
# of pts
5
Animate
<
15
>
Apply
Cancel

Other Methods of Analysis (cont.)

Part III. Second-order Inelastic

1. From the MASTAN2 **Analysis** menu select **2nd-Order Inelastic**.
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 edit box to the right of **Max # of Incrs:** and change **10** to **20**. The analysis will stop when either an instability is detected or 20 load increments are applied or a maximum applied load ratio (**Max. Appl. Ratio**) of **1.0** is reached.
4. Click on the pop-up menu just to the right of **Solution Type:** and select **Predictor-Corrector**.
5. Click on the pop-up menu just to the right of **Modulus:** and select **Et**.
6. Click on the **Apply** button to perform the analysis. Note the analysis stops as a result of an instability (Limit Reached).
7. Click on the pop-up menu just to the right of **Apply** and select **Continue Prev**.

< move to next slide for additional instructions >

Other Methods of Analysis (cont.)

Part III. Second-order Inelastic (cont.)

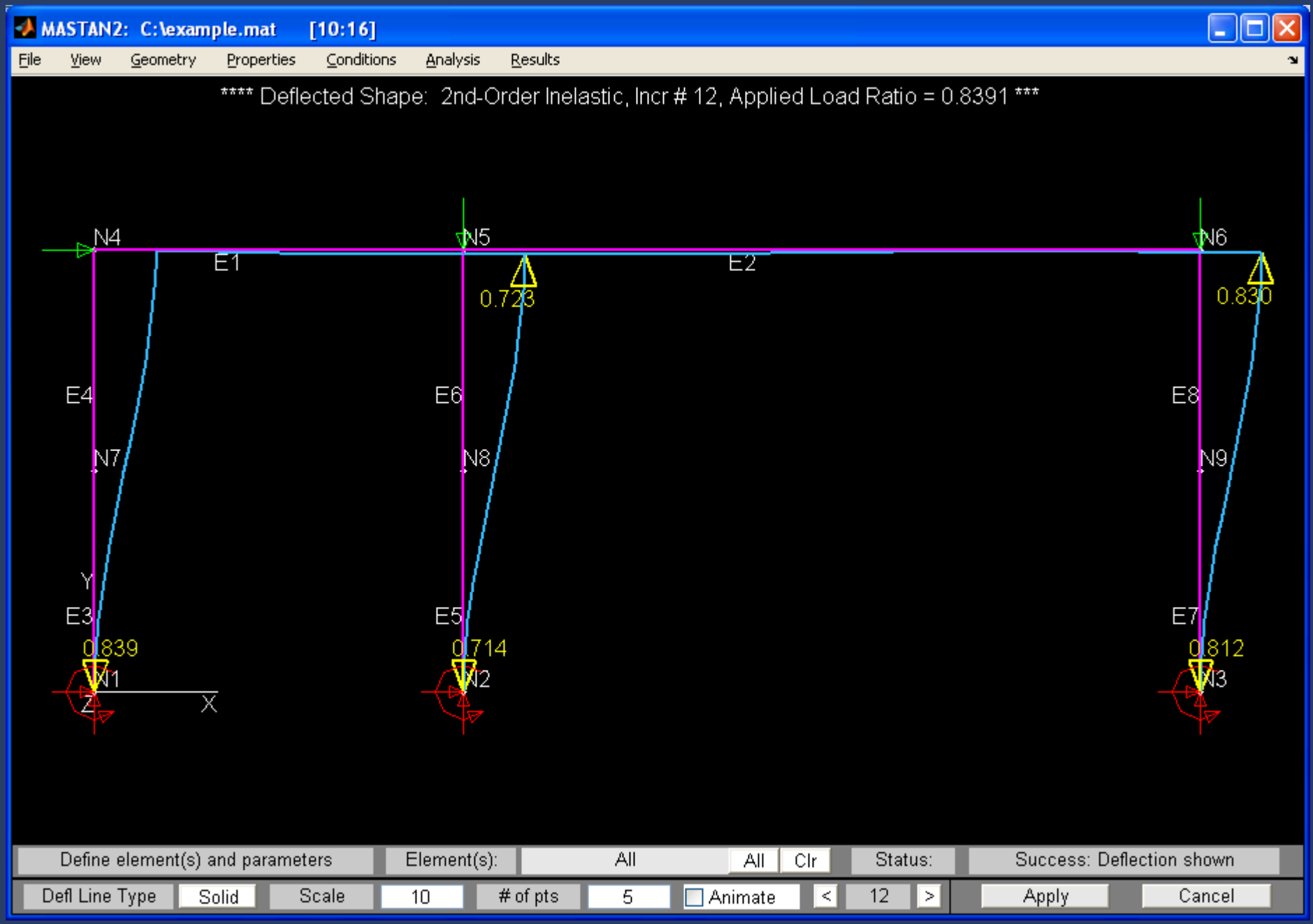
8. Click on the **Apply** button to perform a post-limit point analysis. Only let the analysis run for one or two unloading increments and then click on the **Stop** button. Alternatively, set the **Max # of Incrs:** to **14**.
9. From the **Results** menu select **Diagrams** and submenu **Deflected Shape**.
10. At the bottom menu bar, click on the **<** at the lower right of the bottom menu until the increment number reads **12** (the limit load increment).
11. Click on the **Apply** button. From the **View** menu select **Fit**. The deflected shape and location of plastic hinges are shown. Note that an instability has occurred without a kinematic mechanism.
12. From the **Results** menu select **MSAPlot**.
13. At the bottom menu bar, click in the edit box to the right of **Label** and type **2nd-Order Inelastic**.
14. Change the **Color** to **green** and click on the **Apply** button. The response curve for all four methods of analysis are shown in the plot.

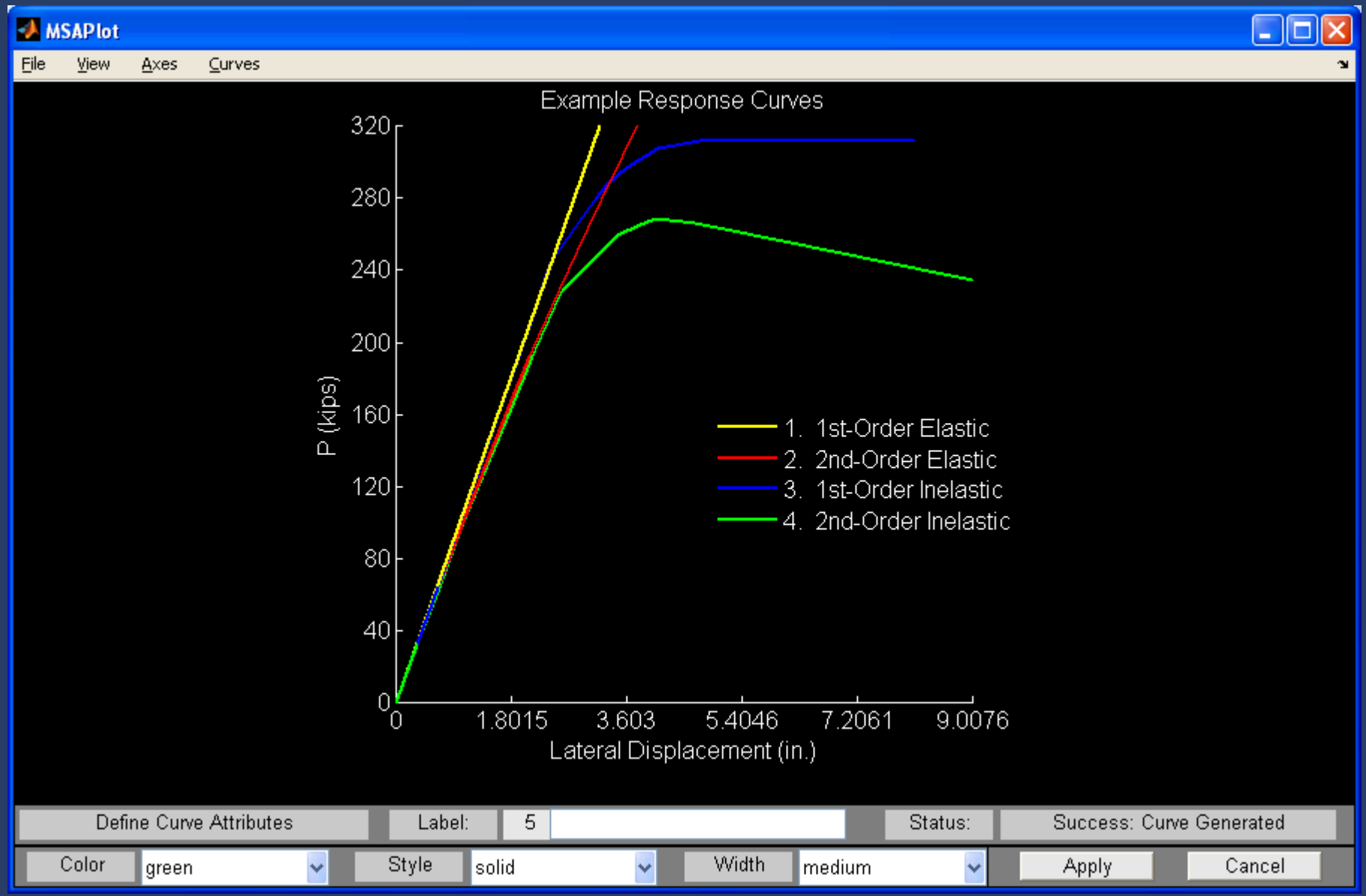
MASTAN2

MASTAN2

Note:

When diagrams are drawn for the limit load, the descriptive label at the top of the MASTAN2 model window is encased in *** 's.


MASTAN2: C:\example.mat [10:16]
File View Geometry Properties Conditions Analysis Results
**** Deflected Shape: 2nd-Order Inelastic, Incr # 12, Applied Load Ratio = 0.8391 ***
N4
N5
N6
E1
E2
0.723
0.830
E4
E6
E8
N7
N8
N9
Y
E3
E5
E7
0.839
0.714
0.812
N1
N2
N3
Z
X
Define element(s) and parameters
Element(s):
All
All
Clr
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
10
# of pts
5
Animate
<
12
>
Apply
Cancel


MSAPlot
File
View
Axes
Curves
Example Response Curves
320
280
240
200
160
120
80
40
0
P (kips)
0
1.8015
3.603
5.4046
7.2061
9.0076
Lateral Displacement (in.)
1. 1st-Order Elastic
2. 2nd-Order Elastic
3. 1st-Order Inelastic
4. 2nd-Order Inelastic
Define Curve Attributes
Label:
5
Status:
Success: Curve Generated
Color
green
Style
solid
Width
medium
Apply
Cancel

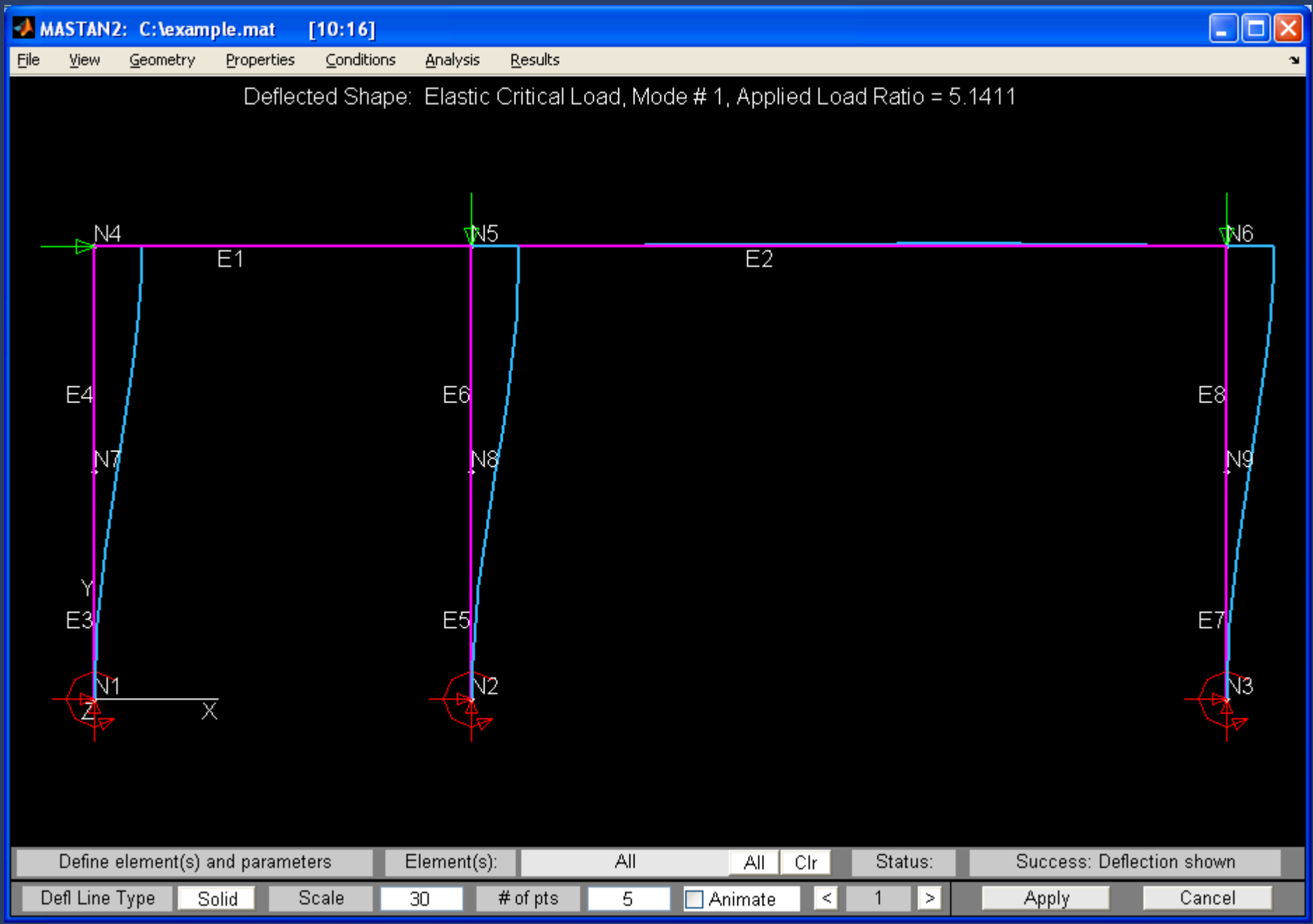
Other Methods of Analysis (cont.)

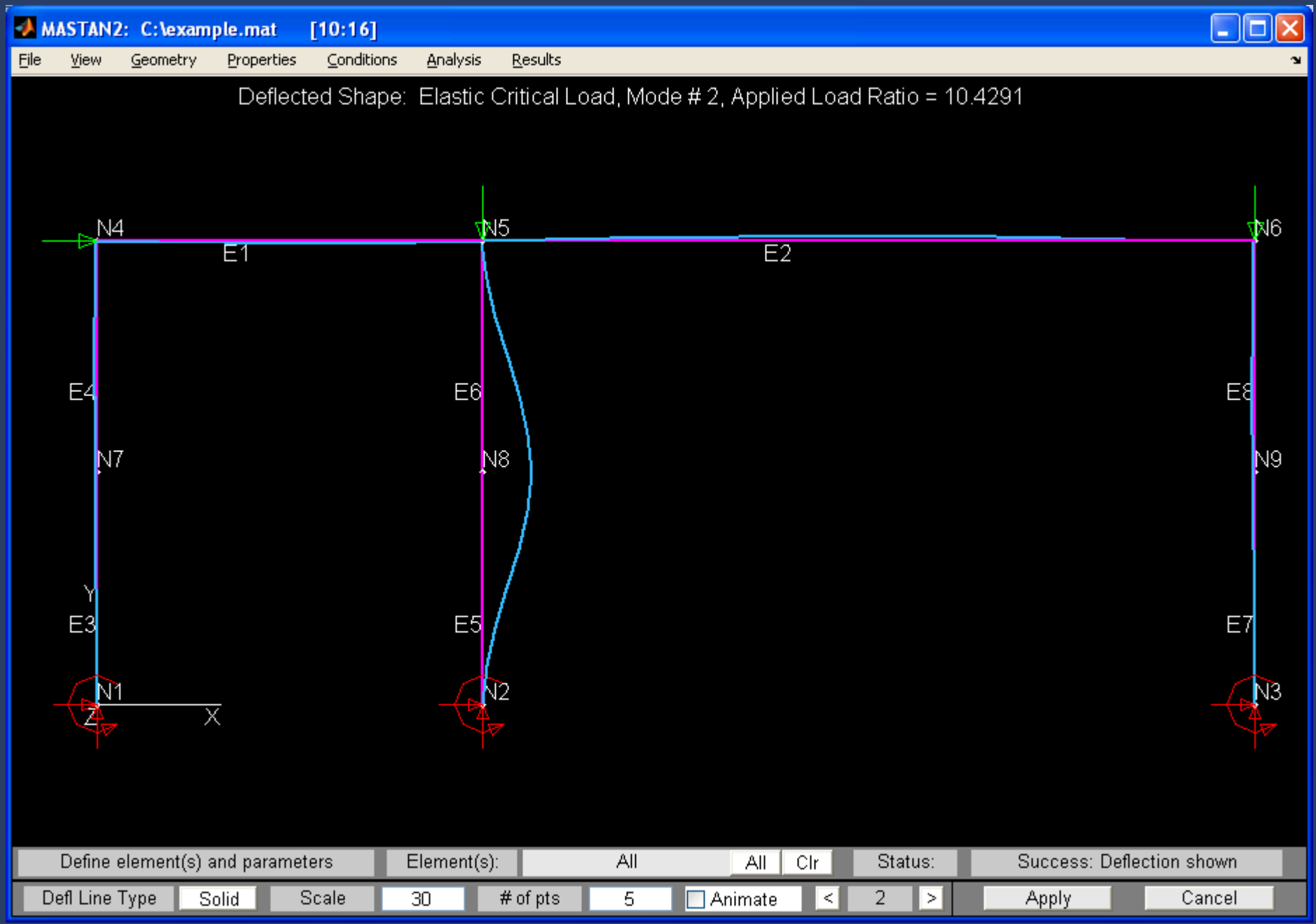
Part IV. Elastic and Inelastic Critical Loads

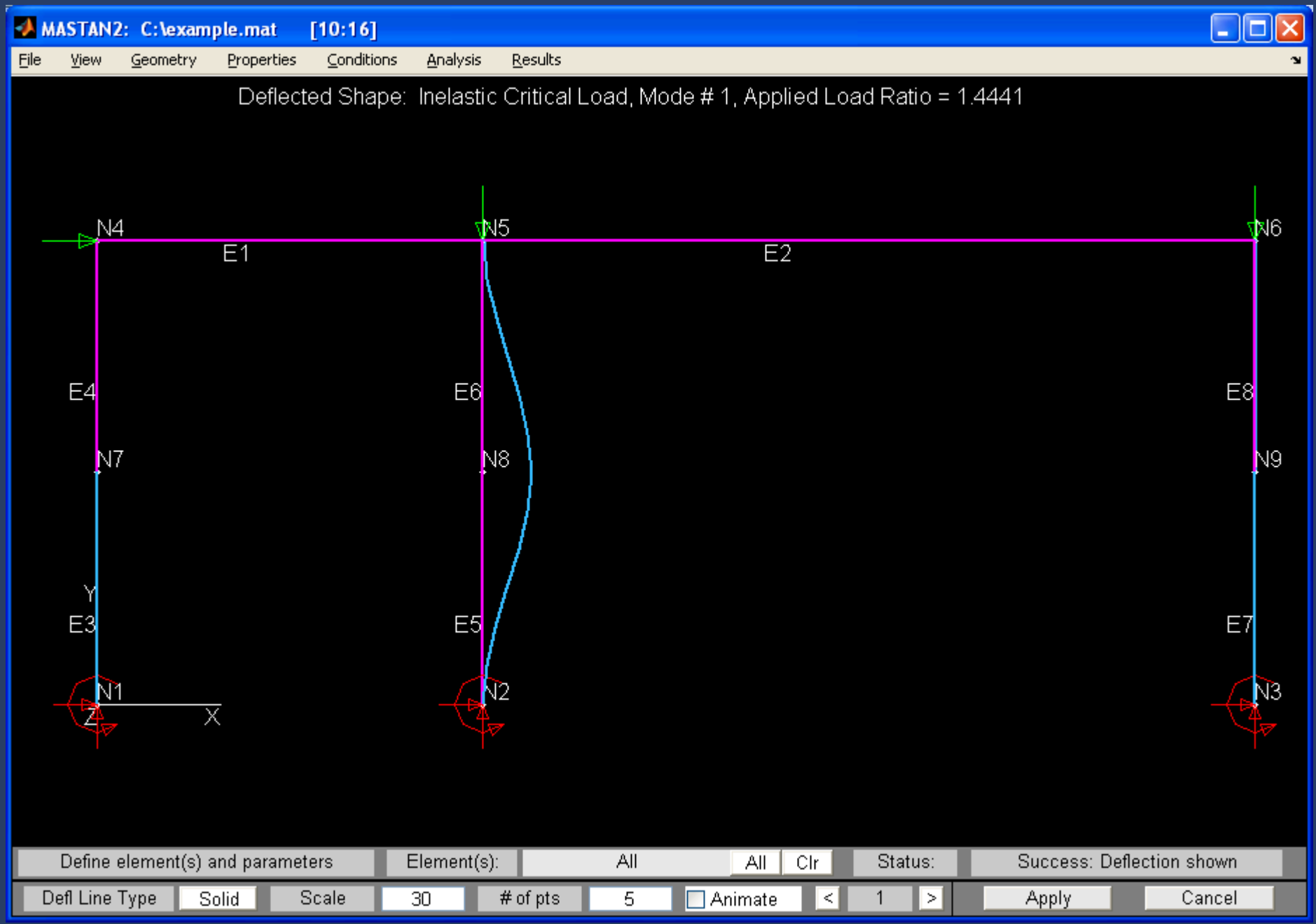
1. From the MASTAN2 **Analysis** menu select **Elastic Critical Load**.
2. At the bottom menu bar, click on the pop-up menu just to the right of **Analysis Type:** and select **Planar Frame (x-y)**.
3. Click on the > at the lower right of the bottom menu until the **Max. # of Modes:** number reads **3**.
4. Click on the **Apply** button to perform the analysis.
5. From the **Results** menu select **Diagrams** and submenu **Deflected Shape**.
6. At the bottom menu bar, click on the edit box to right of **Scale:** and replace **10** with **30** or **-30**, depending on the displaced direction.
7. Click on the **Apply** button and the first mode is shown.
8. To view higher modes, advance the mode number by using > at the lower right of the bottom menu and then click on **Apply**.
9. From the **Analysis** menu select **Inelastic Critical Load** and repeat steps 2, 4, 5, and 7. Note that only one inelastic mode can be calculated.

Note:

The analysis type, mode number, and critical load ratio are shown in the descriptive label located at the top of the main model window.


MASTAN2: C:\example.mat [10:16]
File View Geometry Properties Conditions Analysis Results
Deflected Shape: Elastic Critical Load, Mode # 1, Applied Load Ratio = 5.1411
N4
N5
N6
E1
E2
E4
E6
E8
N7
N8
N9
Y
E3
E5
E7
N1
N2
N3
Z
X
Define element(s) and parameters
Element(s):
All
All
Clr
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
30
# of pts
5
Animate
<
1
>
Apply
Cancel


MASTAN2: C:\example.mat [10:16]
File View Geometry Properties Conditions Analysis Results
Deflected Shape: Elastic Critical Load, Mode # 2, Applied Load Ratio = 10.4291
N4
N5
N6
E1
E2
E4
E6
E8
N7
N8
N9
Y
E3
E5
E7
N1
N2
N3
Z
X
Define element(s) and parameters
Element(s):
All
All
Clr
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
30
# of pts
5
Animate
<
2
>
Apply
Cancel


MASTAN2: C:\example.mat [10:16]
File View Geometry Properties Conditions Analysis Results
Deflected Shape: Inelastic Critical Load, Mode # 1, Applied Load Ratio = 1.4441
N4
N5
N6
E1
E2
E4
E6
E8
N7
N8
N9
Y
E3
E5
E7
N1
N2
N3
Z
X
Define element(s) and parameters
Element(s):
All
All
Clr
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
30
# of pts
5
Animate
<
1
>
Apply
Cancel

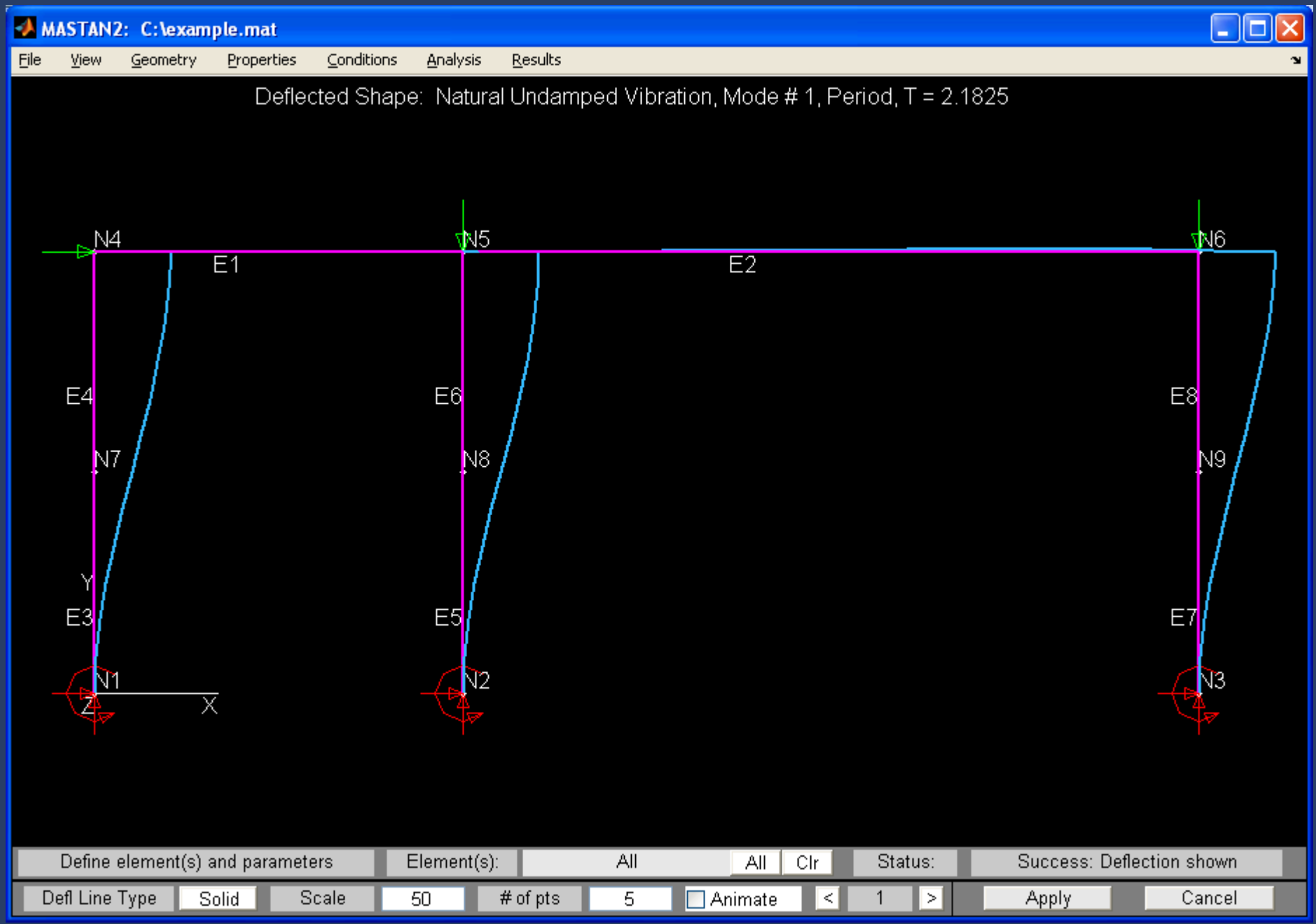
Other Methods of Analysis (cont.)

Part V. Elastic and Inelastic Natural Periods

1. From the MASTAN2 **Analysis** menu select **Natural Period**.
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 edit box to the right of **Mass Matrix Gravitational Acceleration** (GrAcc) and change the **0** to **386.4**.
4. To request three modes, click on the **>** at the lower right of the bottom menu until the **Max. # of Modes:** number reads **3**.
5. Click on the **Apply** button to perform the analysis.
6. From the **Results** menu select **Diagrams** and submenu **Deflected Shape**.
7. At the bottom menu bar, click on the edit box to right of **Scale:** and replace **30** with **50**.
8. Click on the **Apply** button and the first mode is shown. MASTAN2
9. To view animations and/or higher modes, check the Animate box, and as desired, advance the mode number by using **>** at the lower right of the bottom menu, and then click on **Apply**.

Note:

The analysis type, mode number, and natural period are shown in the descriptive label located at the top of the main model window.


MASTAN2: C:\example.mat
File View Geometry Properties Conditions Analysis Results
Deflected Shape: Natural Undamped Vibration, Mode # 1, Period, T = 2.1825
N4
N5
N6
E1
E2
E4
E6
E8
N7
N8
N9
Y
E3
E5
E7
N1
N2
N3
Z
X
Define element(s) and parameters
Element(s):
All
All
Clr
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
50
# of pts
5
Animate
<
1
>
Apply
Cancel

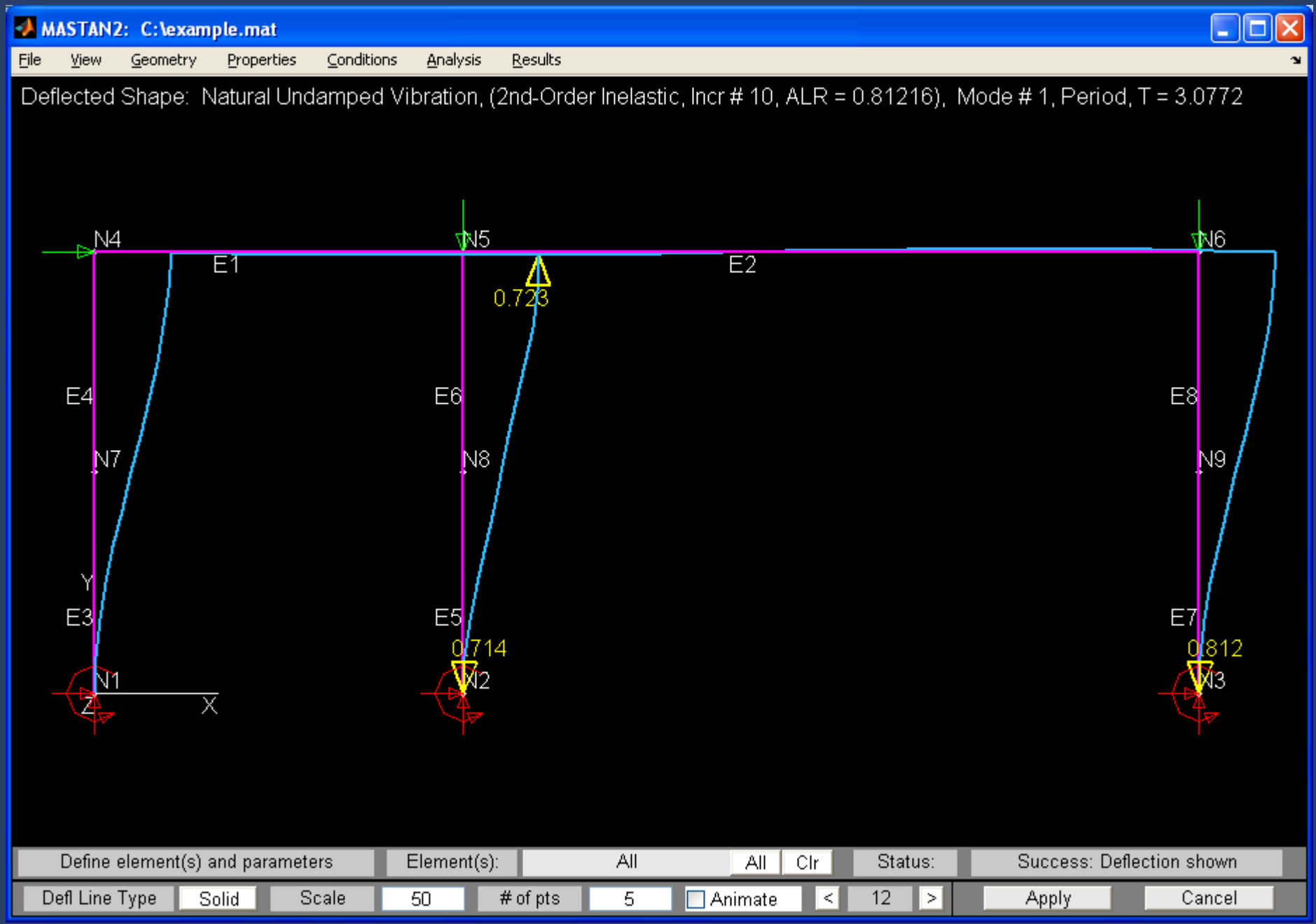
Other Methods of Analysis (cont.)

Part V. Elastic and Inelastic Natural Periods (cont.)

10. To obtain Inelastic Natural Periods, first go to the MASTAN2 **Analysis** menu, select **2nd-Order Inelastic**, and then click on **Apply**.
11. From the MASTAN2 **Analysis** menu select **Natural Period**.
12. At the bottom menu bar, click on the edit box to the right of **Stiffness Matrix** and select **Prev. Incr. Analysis Results**.
13. Click on the > at the right of **[K] from Incr #(ALR)** to request natural periods and mode shapes for all steps of the nonlinear analysis.
14. Click on the **Apply** button to perform the analysis.
15. From the **Results** menu select **Diagrams** and submenu **Deflected Shape**.
16. Click on the **Apply** button and the first mode displayed is for load step 12. From the **View** menu select **Fit**. MASTAN2
17. To view animations and/or different load steps, check the Animate box, and as desired advance the step number by using < or > at the lower right of the bottom menu, and then click on **Apply**.

Note:

Results of the inelastic natural period analysis my be plotted using MSAPlot. For example, a plot of the natural period versus applied load ratio may be generated.

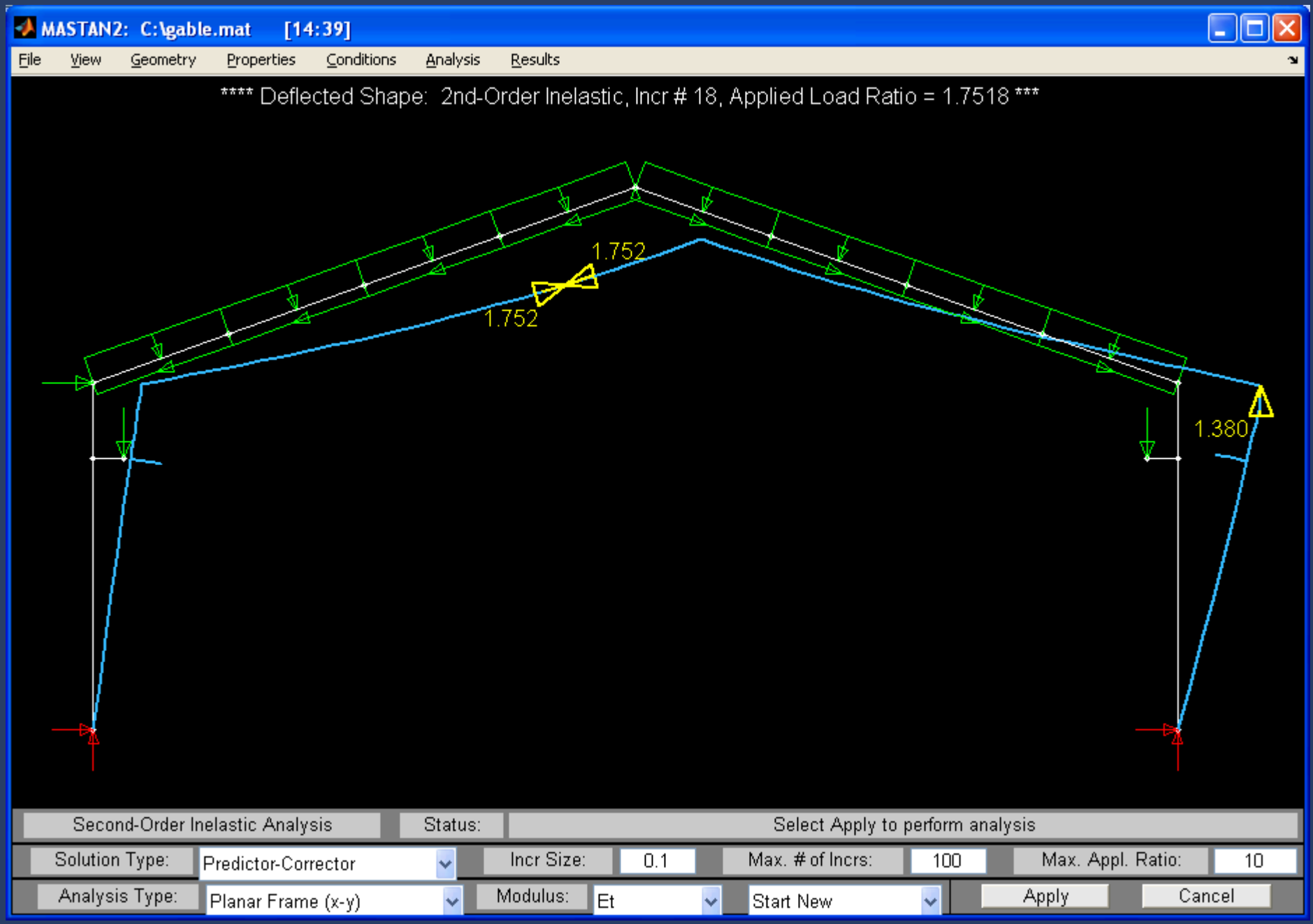

MASTAN2: C:\example.mat
File View Geometry Properties Conditions Analysis Results
Deflected Shape: Natural Undamped Vibration, (2nd-Order Inelastic, Incr # 10, ALR = 0.81216), Mode # 1, Period, T = 3.0772
N4
N5
N6
E1
E2
0.723
E4
E6
E8
N7
N8
N9
Y
E3
E5
E7
0.714
0.812
N1
N2
N3
Z
X
Define element(s) and parameters
Element(s):
All
All
Clr
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
50
# of pts
5
Animate
<
12
>
Apply
Cancel

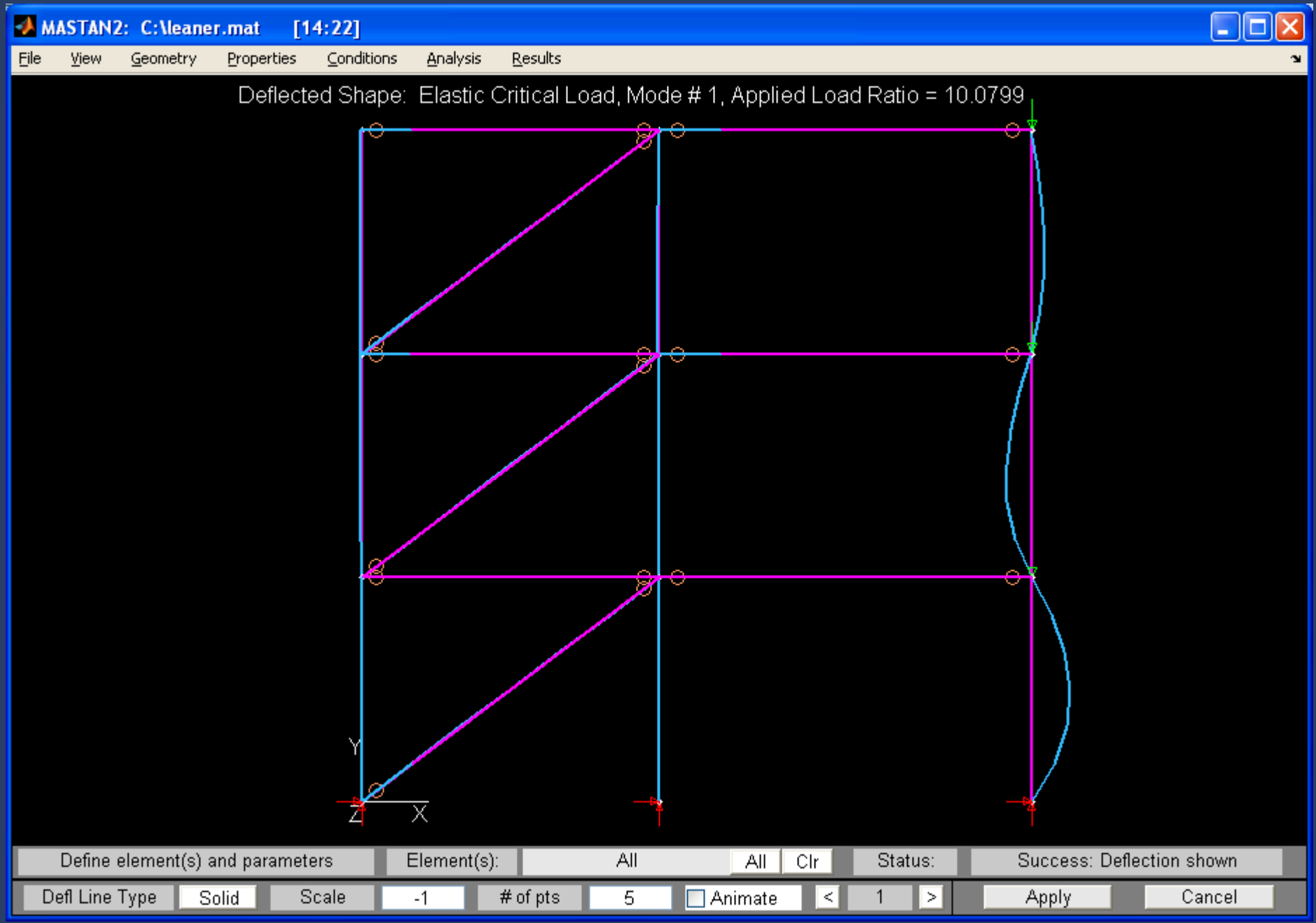
Samples of MASTAN2 Models

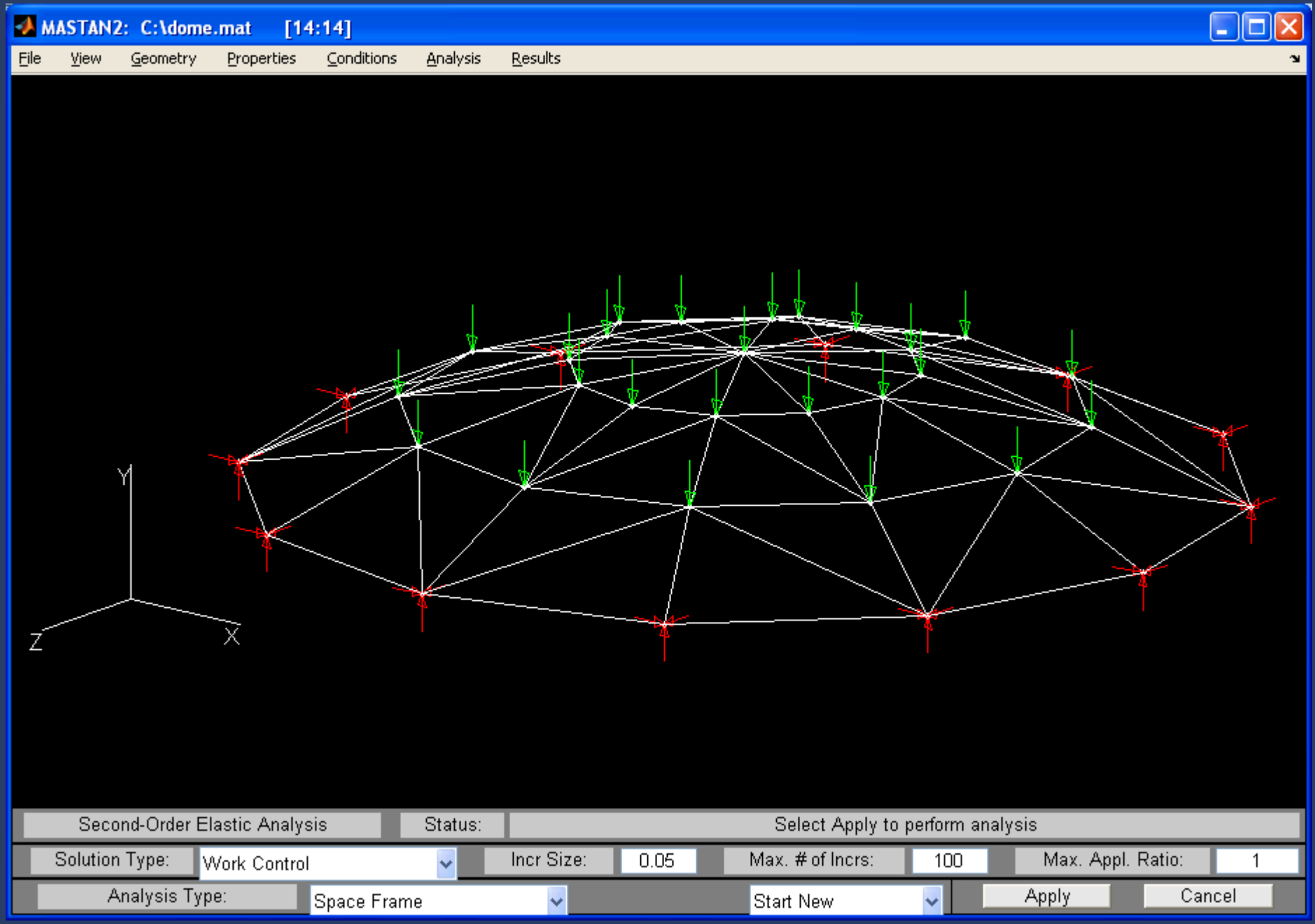
MASTAN2 can be used to model various two- and three-dimensional frames and trusses. Samples of these are provided below.

- Two-dimensional gable frame
- Two-dimensional braced frame with leaning columns
- Three-dimensional dome structure

< click on a description >


MASTAN2: C:\gable.mat [14:39]
File View Geometry Properties Conditions Analysis Results
**** Deflected Shape: 2nd-Order Inelastic, Incr # 18, Applied Load Ratio = 1.7518 ***
1.752
1.752
1.380
Second-Order Inelastic Analysis
Status:
Select Apply to perform analysis
Solution Type:
Predictor-Corrector
Incr Size:
0.1
Max. # of Incrs:
100
Max. Appl. Ratio:
10
Analysis Type:
Planar Frame (x-y)
Modulus:
Et
Start New
Apply
Cancel


MASTAN2: C:\leaner.mat [14:22]
File View Geometry Properties Conditions Analysis Results
Deflected Shape: Elastic Critical Load, Mode # 1, Applied Load Ratio = 10.0799
Y
X
Define element(s) and parameters
Element(s):
All
All
Clr
Status:
Success: Deflection shown
Defl Line Type
Solid
Scale
-1
# of pts
5
Animate
<
1
>
Apply
Cancel


MASTAN2: C:\dome.mat [14:14]
File View Geometry Properties Conditions Analysis Results
Y
Z
X
Second-Order Elastic Analysis
Status:
Select Apply to perform analysis
Solution Type:
Work Control
Incr Size:
0.05
Max. # of Incrs:
100
Max. Appl. Ratio:
1
Analysis Type:
Space Frame
Start New
Apply
Cancel

Overview of Commands

MASTAN2 Menus:

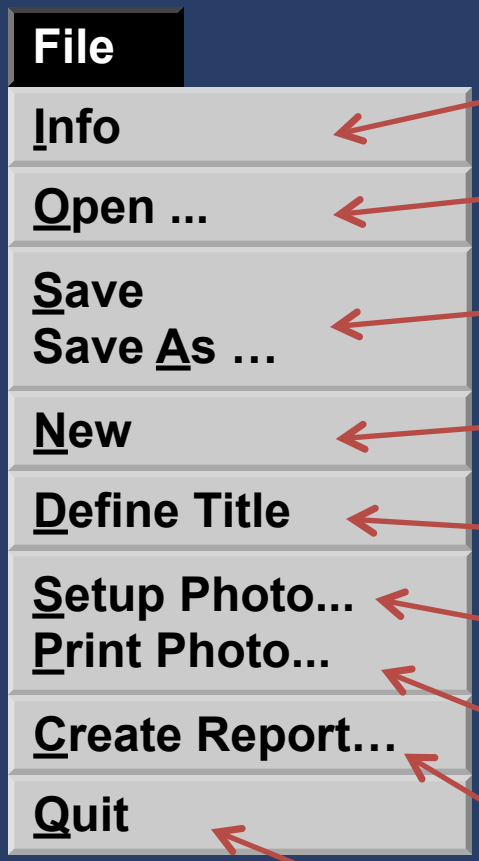
File	View	Geometry	Properties	Conditions	Analysis	Results

MSAPlot Menus:

File	View	Axes	Curves

< click on a menu button >

MASTAN2: File



Provide information about the program MASTAN2

Read an existing MASTAN2 file

Write a MASTAN2 file to disk

Clear existing model and all attributes

Provide a brief model description

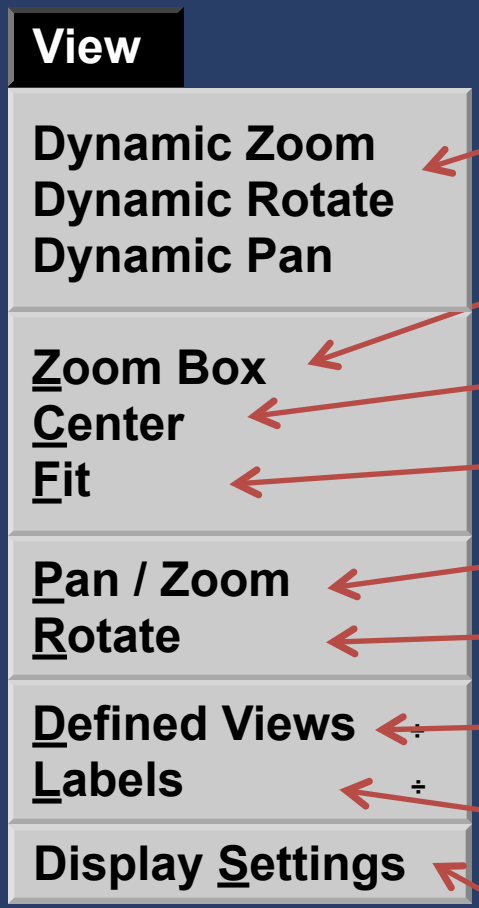
Define photo attributes of the current window

Print a photo of the current window

Write a text report

Exit MASTAN2

MASTAN2: View



After making selection, hold left mouse button down and moving pointer will continue to adjust view of model until mouse button is released

With mouse button down, define a rectangle to zoom in on part of the model

Click and define center of view

Scale view to fit all graphics in window

Manually adjust view of model

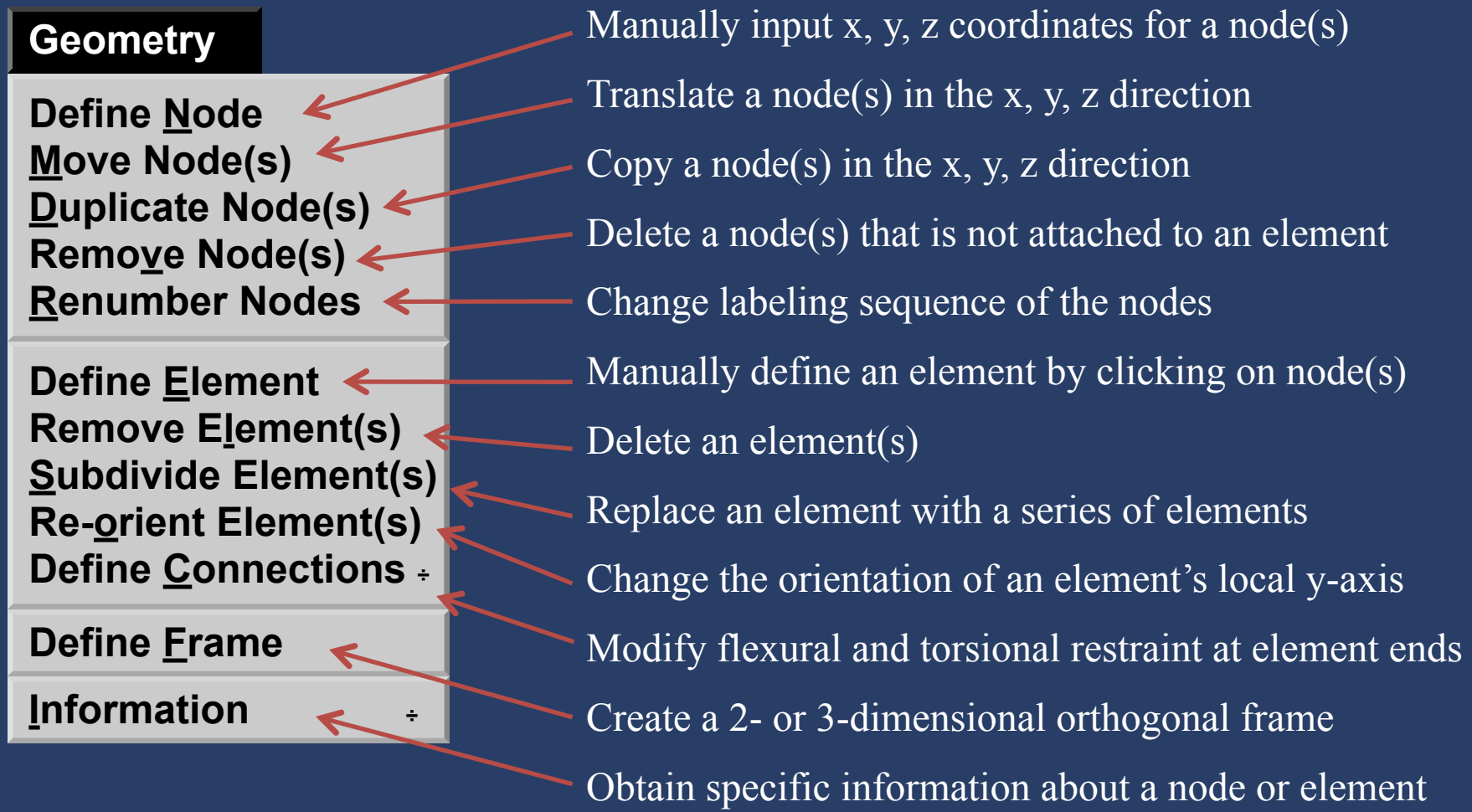
Incrementally rotate view about an axis

Select a pre-defined view

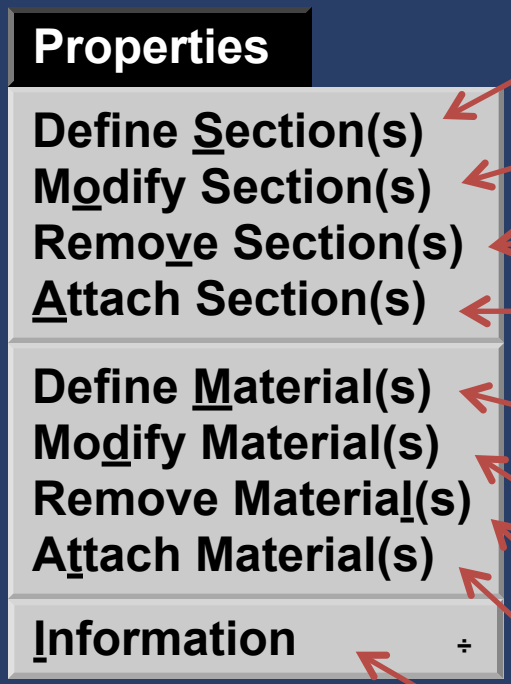
Turn on and off visual entities such as node and element numbers, web orientation vector, etc.

Control display parameters

MASTAN2: Geometry



MASTAN2: Properties



Define a section(s) by inputting key geometric properties, such as areas, moments of inertia, warping constant, and plastic section moduli

Change existing section properties

Delete a section(s)

Attach section(s) to elements

Define a material(s) by inputting key properties, such as modulus of elasticity, Poisson's ratio, yield strength, and weight density

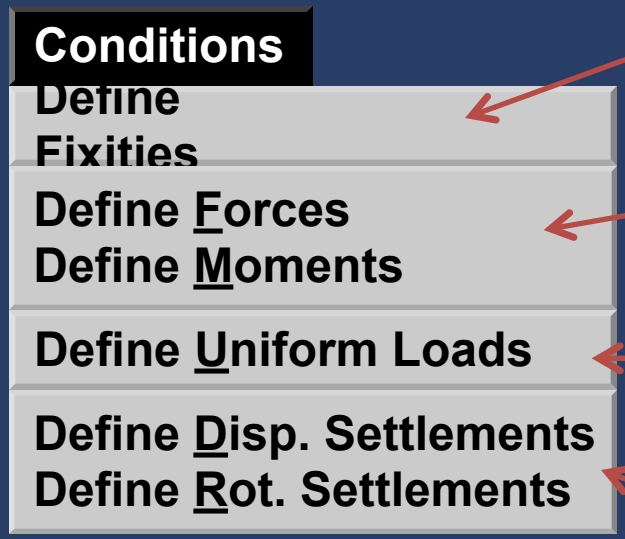
Change existing material properties

Delete a material(s)

Attach material(s) to elements

Obtain specific information about a section or material, including attached elements

MASTAN2: Conditions



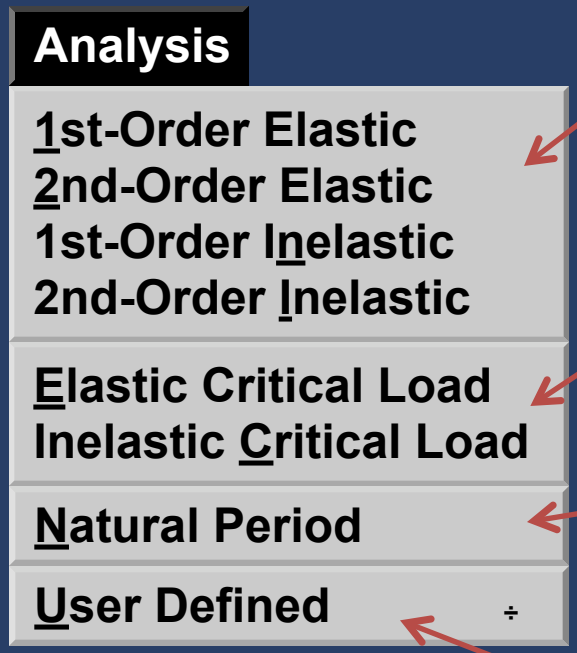
Restrain translational and rotational degrees of freedom at a node(s)

Apply concentrated forces and moments to a node(s)

Apply uniformly distributed loads along the three local axes of an element(s)

Prescribe nonzero translational and rotational values at nodal degrees of freedom

MASTAN2: Analysis



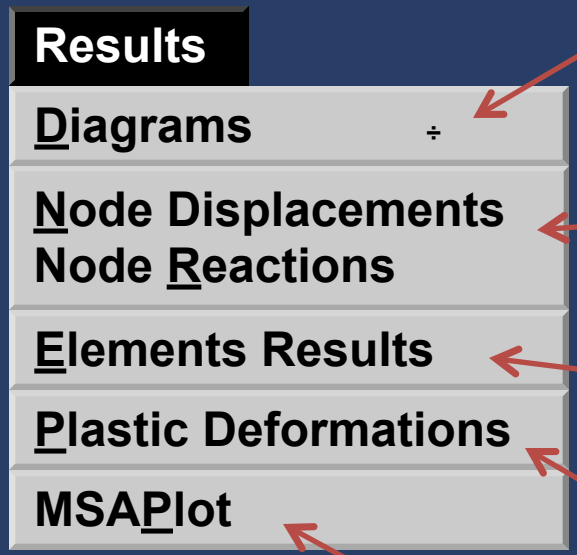
Define analysis parameters and perform selected method of analysis. Nonlinear analysis methods employ a user selected incremental solution scheme. Second-order effects are incorporated by using a geometric stiffness matrix and coordinate updating. Material nonlinear effects are modeled with a concentrated plastic hinge model.

Define analysis parameters and perform selected method of analysis. Critical load ratios and buckled mode shapes are determined using an eigenvalue analysis.

Define analysis parameters and calculate linear or nonlinear natural period(s) and mode shape(s) using an eigenvalue analysis. A lumped mass distribution is determined by dividing all force components in the y-direction by a user defined gravitational constant.

Define analysis parameters and perform a selected method of analysis that will employ user defined analysis modules. These files interact directly with MASTAN2 by using the common ud_*.m files that are provided with this software.

MASTAN2: Results



Define parameters and draw selected diagram. These include deformed shape and element force diagrams such as axial or shear forces, torque or bending moments, and bi-moments. Also provides an option to turn off an existing diagram.

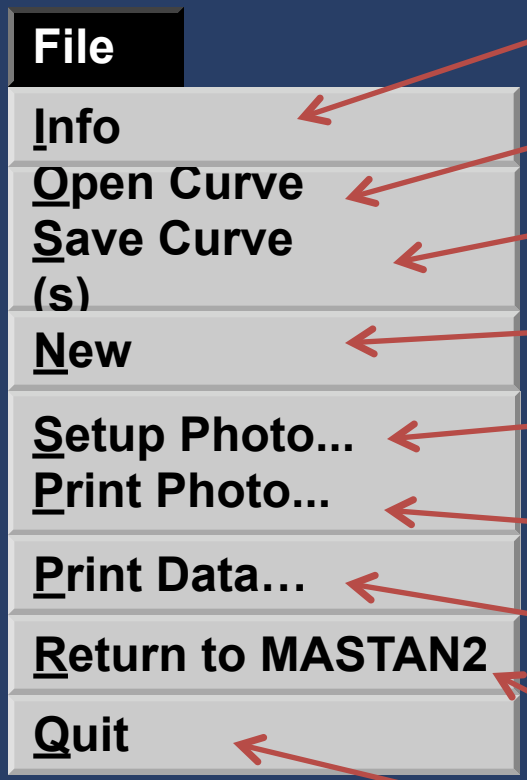
Provide displacement or reaction components at user selected node.

Provide internal forces and moments at any point along the length of a user selected element.

Provide inelastic axial displacement and major and/or minor axis rotations at a plastic hinge location of a user selected element. Reported values reference the element's local coordinate system.

Start an application that provides the opportunity to plot response curves from analysis results.

MSAPlot: File



Provide information about the program MSAPlot

Read an existing curve data file (text/ascii format)

Write a curve data file to disk

Clear all current curves and plot attributes

Define photo attributes of the current window

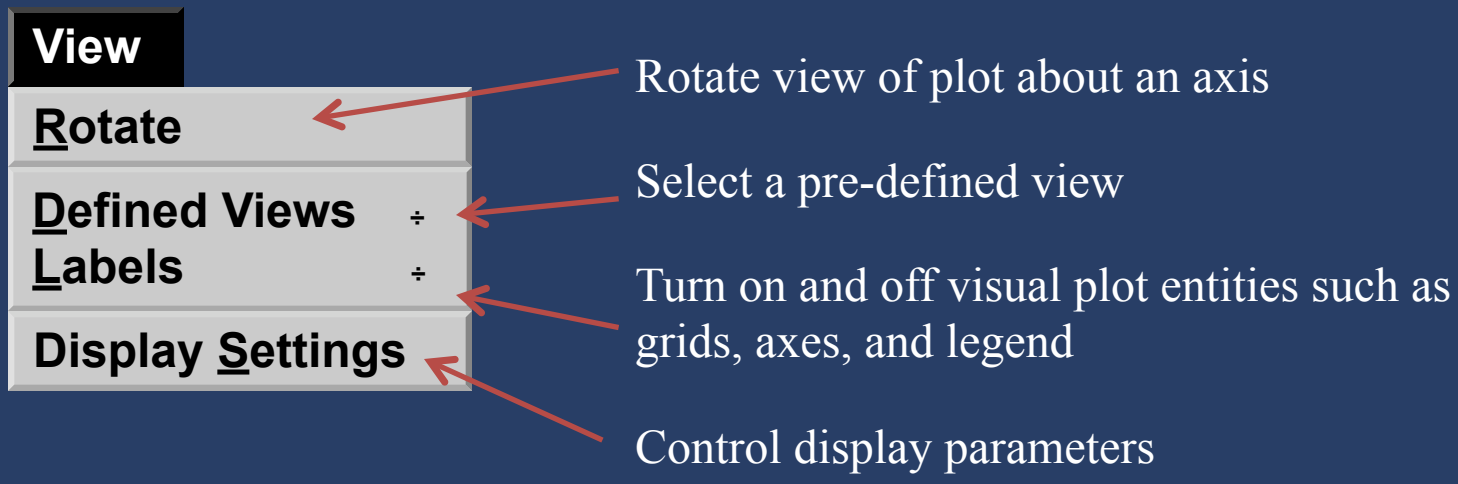
Print a photo of the current window

Write a text report

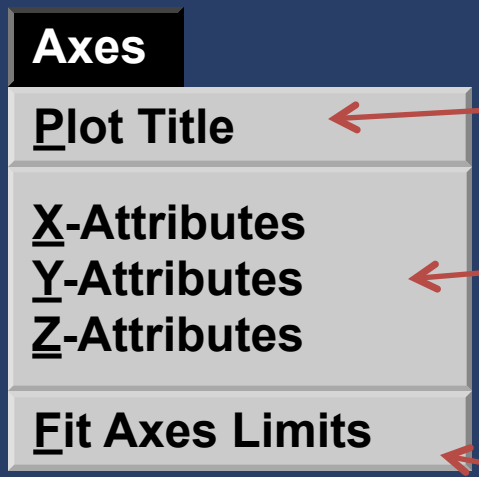
Bring MASTAN2 window to front

Exit MSAPlot

MSAPlot: View



MSAPlot: Axes

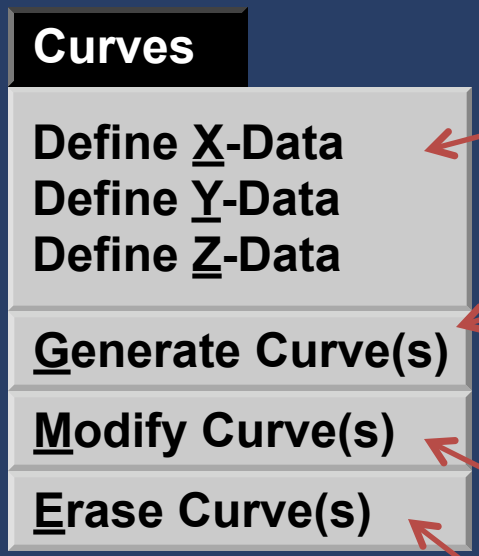


Provide a title that is located at the top of the plot

Define X-, Y-, or Z- axes attributes such as label, number of tick marks, and minimum/maximum limits

Scale all three axes to fit extremes of current curve data

MSAPlot: Curves



Define the response data that should be plotted on the X-, Y-, or Z- axis

Using the data-to-axis relationships defined in the above and the curve graphical attributes prescribed in this option, generate a two- or three-dimensional response curve

Change an existing curve's graphical attributes such as label, color, style, and line weight

Remove an existing curve from the plot

Programming

Users that have access to MATLAB can also employ MASTAN2 to execute their own MATLAB code. Twelve M-files (in text format) reside in the MASTAN2 folder that you copied onto your computer (see Method 1, *Getting Started*). These files contain functions that permit your code to interface with MASTAN2.

For example, the function contained in the file **ud_3d1el.m** is called when a user selects **Analysis--User Defined -- 1st-Order Elastic** and then applies a three-dimensional analysis. Since no code is originally provided in this function, the analysis cannot be performed and MASTAN2 responds with an appropriate message. However, you can make this analysis option functional by expanding the code contained in this file. Furthermore, the code you provide may also call other M-files that you prepare and hence, provide you the opportunity to write code in a modular style. The only limitation is that the first line of the twelve M-files (the function line containing the name of the routine and the input and output arrays) cannot be changed. These M-files are well commented and their use should be self-explanatory. It is important to note that the attributes or permission settings for these files may be originally set at Read Only. Before getting started, be sure to check this file property and remove it as required.

The twelve user-defined M-files and their corresponding analysis intent include:

ud_3d1el.m Three-dimensional 1-st Order Elastic
ud_2d1el.m Two-dimensional 1-st Order Elastic
ud_3d2el.m Three-dimensional 2nd-Order Elastic
ud_2d2el.m Two-dimensional 2nd-Order Elastic
ud_3d1in.m Three-dimensional 1-st Order Inelastic
ud_2d1in.m Two-dimensional 1-st Order Inelastic
ud_3d2in.m Three-dimensional 2nd-Order Inelastic
ud_2d2in.m Two-dimensional 2nd-Order Inelastic
ud_3decl.m Three-dimensional Elastic Critical Load
ud_2decl.m Two-dimensional Elastic Critical Load
ud_3dicl.m Three-dimensional Inelastic Critical Load
ud_2dicl.m Two-dimensional Inelastic Critical Load

Good Luck !

Additional Information

Additional information and updates for MASTAN2 may be provided at the following URL:

http://www.mastan2.com