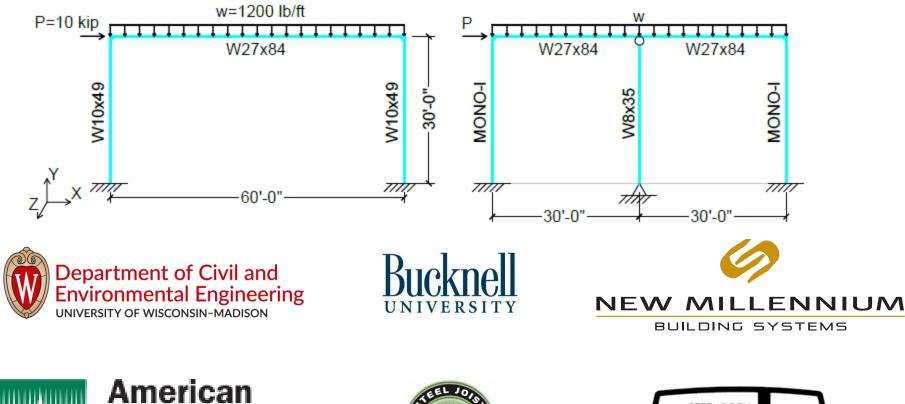
Tutorial for MASTAN2 v5.1 -**Introductory Frame**



MASTAN2

Iron and Steel Institute





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

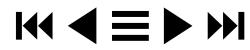
- Skip to Previous Section Title Page
 - Previous Slide
 - Return to Tutorial Sections Page
 - Next Slide
 - Skip to Next Section Title Page

- Open screenshot of MASTAN2 or
 - additional helpful information.



Section 1: Overview



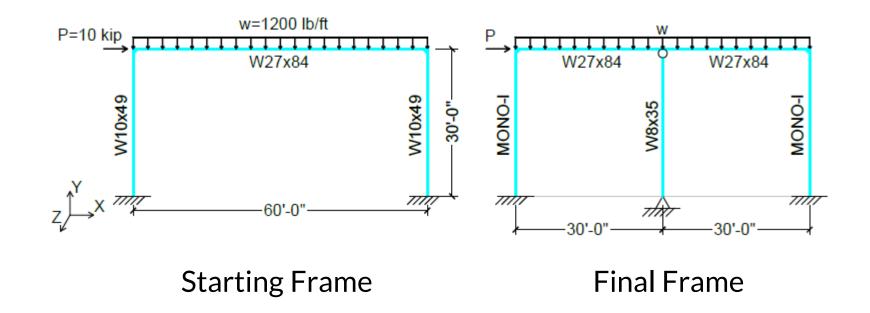


Overview

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 <u>http://www.mastan2.com/tutorial.html</u>.

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.





Section 2: Getting Started



MASTAN2 General Information

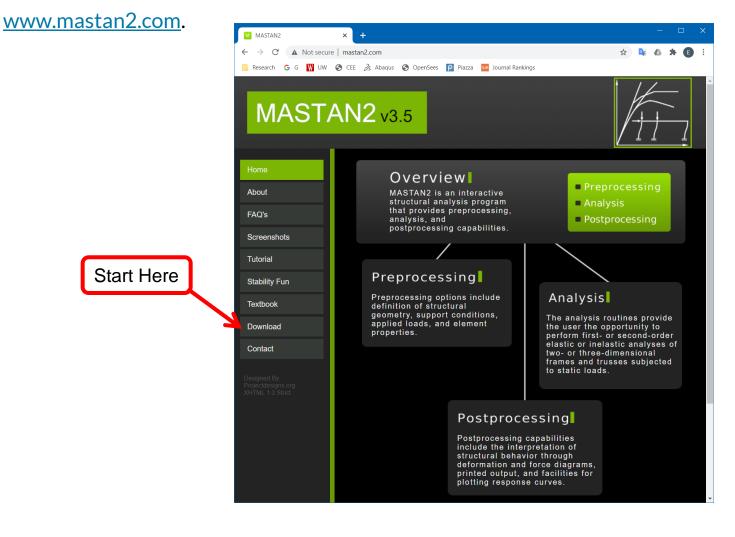
MASTAN2 is an interactive graphics program that provides preprocessing, analysis, and postprocessing capabilities. Preprocessing options include definition of structural geometry, support conditions, applied loads, and element properties. The analysis routines provide the user the opportunity to perform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

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

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

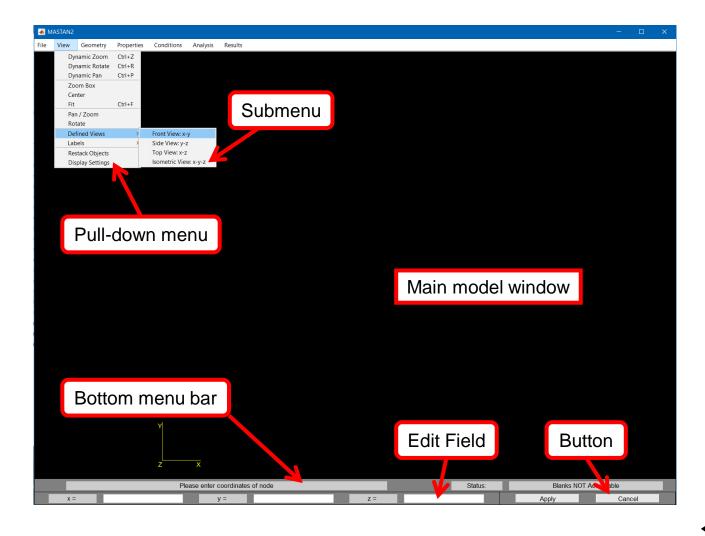
Launching MASTAN2

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



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.

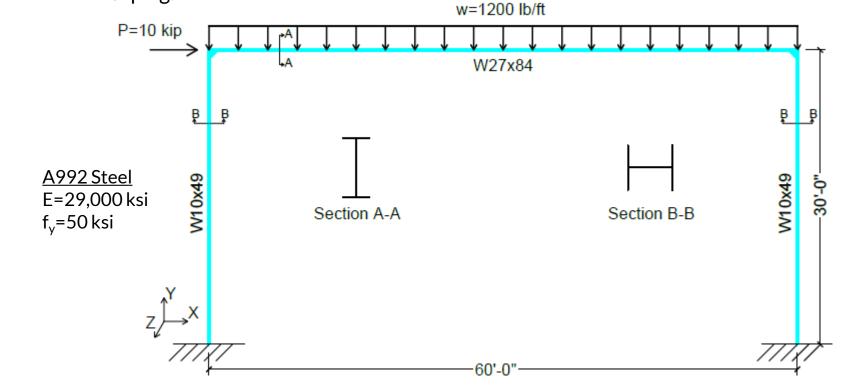


Section 3: 2-D Frame Analysis



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.

Y ______x

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

MASTAN2

₹

MASTA	N2									—	
<u>F</u> ile <u>V</u> ie	w <u>G</u> eometry	<u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults						
<u>N3</u>											
110					E2						
-											
E1											
Y											
N1 Z	×										
2											
_									_		
		2			s and dimensions of structur	e 360 1	✓ frames @	Status:		: Frame Generated.	
1	~ bays	(C)	720	1	 stories @ 	200	I frames @	0	Apply	Cou	ncel

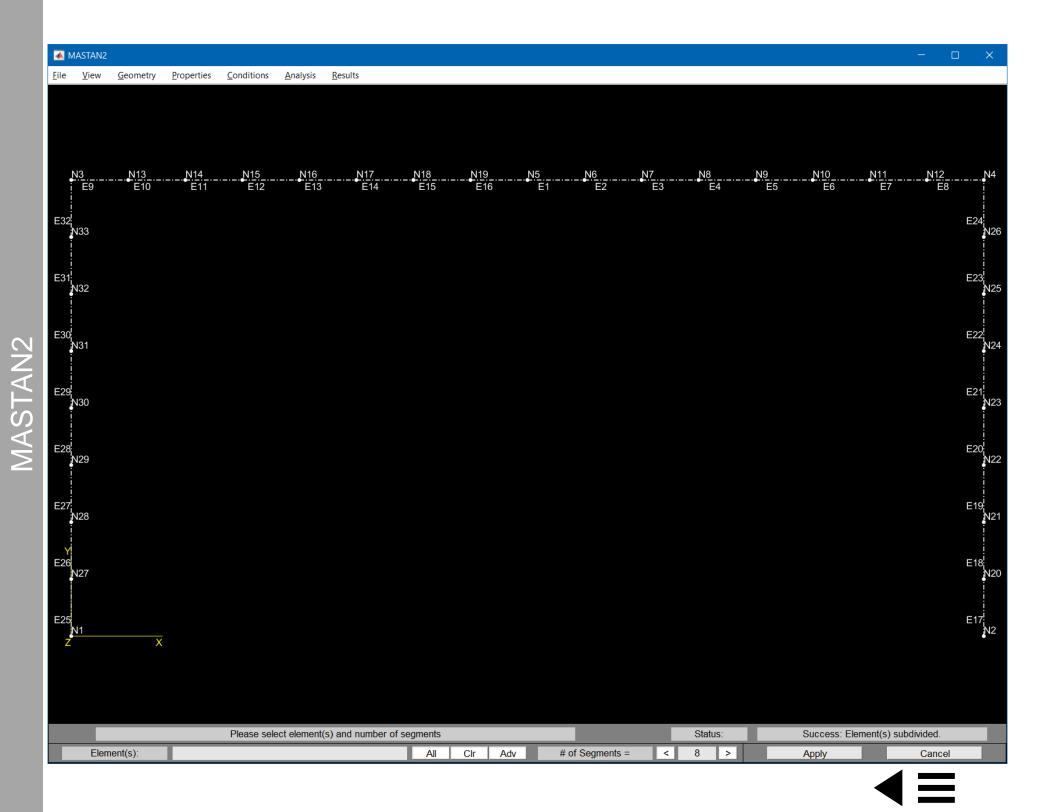
Element Modification

- 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							- 0
<u>File View G</u> eomet	y <u>P</u> roperties <u>C</u> onditions	s <u>A</u> nalysis <u>R</u> esults					
N3				N5			
	E3				E4		
E1							
Y							
N1							
Z	X						
	Please s	elect element(s) and number	of segments		Status:	Success: Element(s) s	ubdivided.
			All Cir Adv	# of Segments =	< 2 >	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		_													- 0	>
e <u>V</u> iew	<u>G</u> eometry	Properties	Conditions	<u>A</u> nalysis	<u>R</u> esults											
<u>N9</u> E9	<u>N10</u> E10	<u>N11</u> E11	<u>N12</u> E12	. <u>N13</u> E13	<u>N14</u> E14	<u>N15</u> E15	<u>N16</u> E16	<u>N17</u> E17	<u>N18</u> E18	<u>N19</u> E19	<u>N20</u> E20	<u>N21</u> E21	<u>N22</u> E22	N23 E23	<u>N24</u> E24	
N8																E32
																E31
N7																
N6																E30
16																E29
N5																
																E28
N4																E20
N3																E27
																E26
N2																
N1																E25
	Х															
		Please define	direction soo	wence for r	enumbering nod	20			Status			Comple	te: Bandwidth	Decreased		
X-Y-Z (2		X-Z-Y		X-Z (2D)	Y-Z-X		□ Z-X-Y		-Y-X	Rand		Comple	Apply	Decreased.	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.
 - Ļ
- 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.

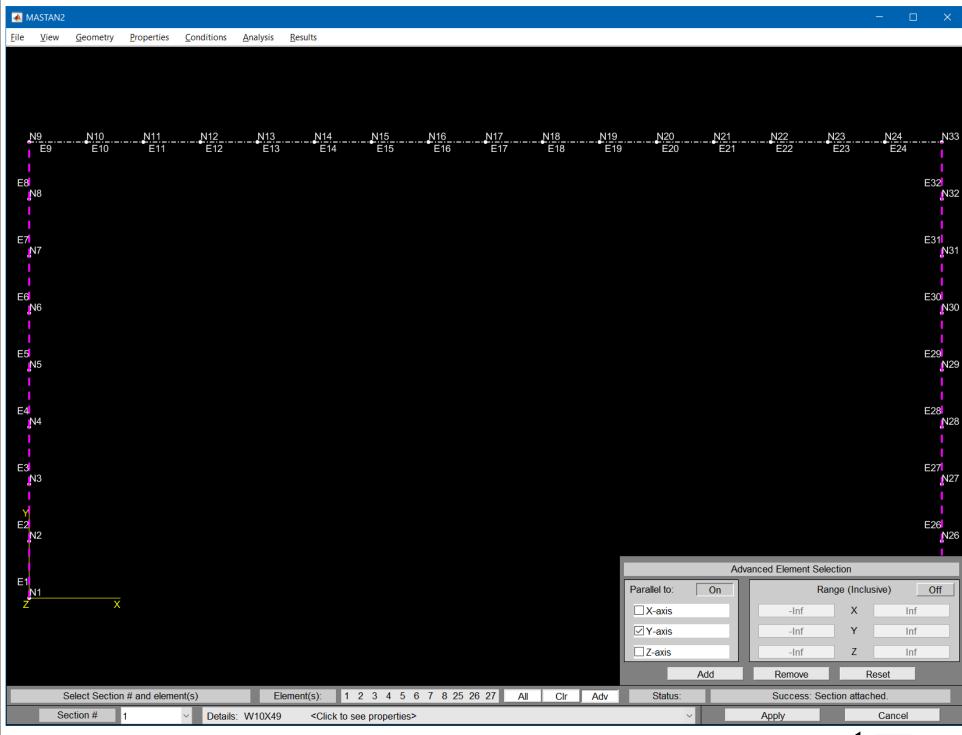
																_	
	ASTAN2	<u> </u>	D 11	C 1111		D 11										- 🗆	×
File	<u>V</u> iew	<u>G</u> eometry	<u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults											
	10	N10	N111	N12	N13	N11.4	N15	N16	N17	N18	N19	N20	NO1	N22	N23	ND4	N33
ļ	N9 E9	E10	<u>N11</u> E11	E12	E13		<u>N15</u> E15	E16	E17	E18	E19	E20	<u>N21</u> E21	E22	E23	N24 E24	
E8	N 8																E32 N32
																	504
E7	٨7																E31 N31
E6																	E30;
	N 6																N30
E5																	E29
	N5																N29
E4	14																E28 N28
																	120
												operties will be in		- 1			
E3	13									Database:		Sectior	1:	- 1			E27 N27
İ									AIS	C (in)	~ W12) W12)	X22 X19		^			Ĭ
Y E2											W122 W122						E26
	12									Type:	W10	X112					N26
									W S	hapes	~ W10) W10)						
E1											W102						E25
Z	N1	X								Shear Area	W10	X60					N25
									infin	ite	~ W10) W10)	x54 X49					
											W102	K45 K39					
										Clear	W10		_	~			
	_	_	section prop			Section 1	Name:		DX49	Databa		Status:	_				
	rea =	14.		z-z =	27		l y-y =	93.4	J =		1.39	Cw =	20	070	Basic	0-	~
2	Z-Z =	60	.4	Z y-y =	28.	.3	A y-y =	Inf	A z-z	-	Inf			Apply		Cancel	

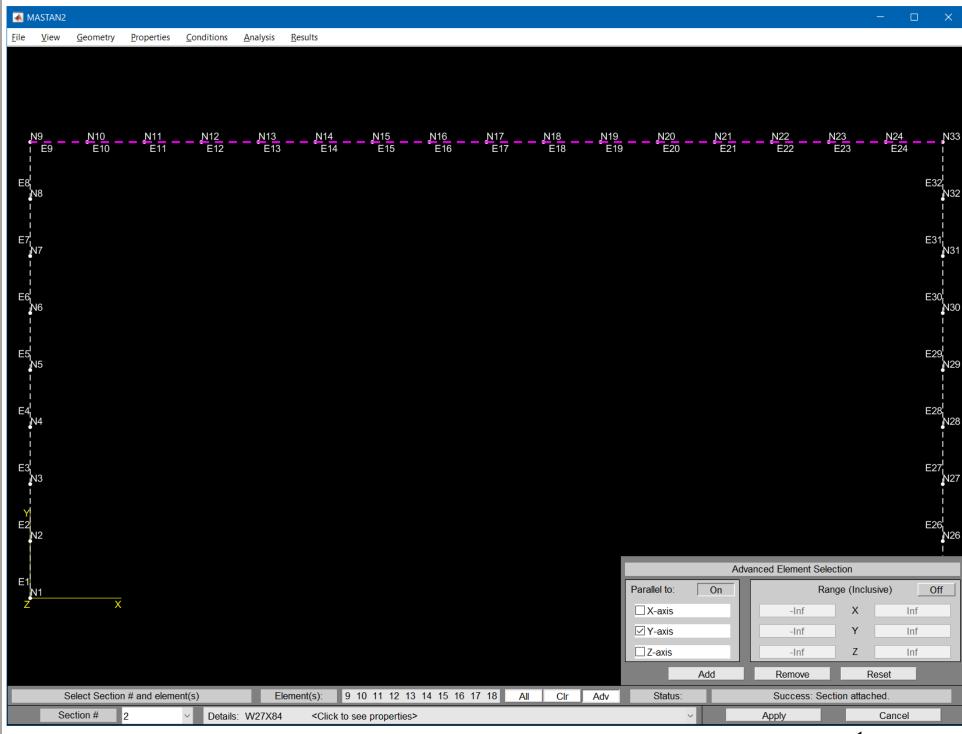
◀☰

<u>View Geometry</u> N9 N10 E9 E10 ⁸ N8 ⁷ N7 ⁶ N6 ⁵ N5 ⁵ N5 ⁴ N4 ³ N3 ² N2	<u>Properties</u>	<u>Conditions</u>	<u>Analysis</u> R <u>N13</u> E13	<u>N14</u> E14	<u>N15</u> E15	<u>N16</u> E16	<u>N17</u> E17		N19 N20 E19 E20		N22 E22	<u>N23N24</u> E23 E2	E32 N3 E31 N3 E30 N3 E29 N2 E28 N2
E9 E10 ⁸ N8 ⁷ N7 ⁵ N5 ⁵ N5 ⁴ N4 ³ N3	<u>N11</u> E11	<u>N12</u> E12	<u>N13</u> E13	<u>N14</u> E14	<u>N15</u> E15	<u>N16</u> E16					<u>N22</u> E22	<u>N23</u> N24 E23 E2	E32 N3 E31 N3 E30 N3 E29 N2 E28 N2
E9 E10 ⁸ N8 ⁷ N7 ⁵ N5 ⁵ N5 ⁴ N4 ³ N3	<mark>N11</mark>	<u>N12</u> E12	<u>N13</u> E13	<u>N14</u> E14	N15 E15	<u>N16</u> E16						<u>N23 N2</u> E23 E2	E32 E31 E30 E29 E28
E9 E10 ⁸ N8 ⁷ N7 ⁶ N6 ⁵ N5 ⁵ N5 ⁵ N5	N11 E11	N12 E12	N13 E13	<u>N14</u> E14	E15	<u>N16</u> E16					N22 E22	<u>N23 N2</u> E23 E2	E32 E31 E30 E29 E28
E9 E10 N8 N7 N6 N5 N4	N11 E11	N12 E12	<u>N13</u> E13	<u>N14</u> E14	<u>N15</u> E15	<u>N16</u> E16					N22 E22	<u>N23</u> E23 E2	E32 E31 E30 E29 E28
E9 E10 N8 N7 N6 N5 N4 N3	E11	Ē12	E13	E14	E15	E16					E22	E23 E2	E32 E31 E30 E29 E28
N8 N7 N6 N5 N4 N3								Select section	n and properties will	I be inserted			E31 E30 E29 E28
N7 N6 N5 N4								Select section	n and properties will	I be inserted			E31 E30 E29 E28
N7 N6 N5 N4								Select section	n and properties will	I be inserted			E30 E29 E28
N7 N6 N5 N4								Select section	n and properties will	I be inserted			E30 E29 E28
N6 N5 N4 N3								Select section	n and properties will	I be inserted			E30 E29 E28
N6 N5 N3								Select section	n and properties will	I be inserted			E29 E28
N6 N5 N3								Select section	n and properties will	l be inserted			E29 E28
N5 N4 N3								Select section	n and properties will	I be inserted			E28
N5 N4 N3								Select section	n and properties will	l be inserted			E28
N4 N3								Select section	n and properties will	I be inserted			E28
N4 N3								Select section	n and properties will	l be inserted			
N4 N3								Select section	on and properties will	l be inserted			
N3								Select section	n and properties will	l be inserted			
N3													
								Database:	S	ection:			E27
N2							AISC	(in) ~	W12X22 W12X19		^		
N2									W12X16				
								Type:	W12X14 W10X112				E26
							W Sha	apes ~	W10X100 W10X88				
									W10X77 W10X68				E25
N1 Z X								hear Areas:	W10X60				
							infinite	e ~	W10X49		- 18		
								Clear	W10X45 W10X39		~		
Diagon optor	agation proparti	tion	See	ction 2	Name:		_		Status:	_		Section 1 defined.	
Area = 0	section properti	l z-z =	0		l y-y =	0	J =	Database			0	Basic	
z-z = inf		Z y-y =	inf		A y-y =	inf	_	0			Apply	Canc	el
							A z-z =	= in	<u>ולר</u>				

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 popup menu with all section numbers. Click on 2 to select the W27x84 section.
- 8) Assign Section #2 properties by clicking the Apply button.





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.)

	MASTAN	2														- 0	×
<u>F</u> ile	<u>V</u> iew	<u>G</u> eometry	<u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults											
	N9	N10 E10	N11 E11	N12 E12	N13 E13	N14	N15 E15	N16 E16	N17 E17	N18	N19 E19	N20	N21 E21	N22	N23	N24	N33
	E9	E10	E11	E12	E13	E14	E15	E16	E17	E18	E19	E20	E21	E22	E23	E24	
E8	N8																E32 N32
	10																102
E7																	E31
	N7																N31
-																	500
E6	N6																E30 N30
E5	N5																E29 N29
																	Ĭ
E4	514																E28
	N4																N28
E3																	E27
	N3																N27
Y																	
E2	N2																E26 N26
E1	N1																E25 N25
z		×															6.20
		Coloret Meteri	al // and alarm					<u> </u>				Chatura		Guarante	Material atte	- had	_
		Select Materia Material #	ai # and eiem	_	s: A992	ement(s):	e properties>	All	Α	ll Clr	Adv	Status:	<u> </u>	Apply	Material atta	Cancel	-
			<u>'</u>	Dergi	3. NJYZ	SOUCK TO SE						Ť		түріу		Calicer	

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.

 M	ASTAN2	2															—	
<u>F</u> ile	<u>V</u> iew	<u>G</u> eomet	ry <u>P</u> roperti	ies <u>C</u> ondit	ions <u>A</u> naly	rsis <u>R</u> esults												
		N9 E9	N10 E10	N11 E11	N12 E12	N13 E13	N14 E14	N15 E15	<u>N16</u> E16	N17 E17	N18 E18	N19 E19	N20 E20	N21 E21	N22 E22	N23 E23	N24 E24	N33
	E8	N8																E32 N32
	E7	N7																E31 N31
	E6	N6																E30 N30
	E5	N5																E29 N29
	E4																	E28 N28
		N4																N28
	E3																	E27 N27
		N3																N27
	Y E2																	F26
		N2																E26 N26
	-1																	
	E1	N1	X															E25 N25
	F	Please sele	ct node(s) ar	nd fixity(s)		Node(s):		1 2	5	All	Clr	Adv	Status:		Success:	Node fixitie	s defined.	
⊻ X	disp		⊻ Y-dis	sp	∠ Z-	disp		X-rot	[✓ Y-rot		Z-rot			Apply		Cance	el

 \equiv

Loading

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.

ASTAN2															—	
<u>V</u> iew <u>G</u> eome	try <u>P</u> roperties	<u>C</u> ondition	ns <u>A</u> nalysi	s <u>R</u> esults												
NO	NIO		NICO	NICO	NI4.4	NAF	NIC	N147	NIAO	NICO	NOO	NO4	NICO	NOO	NO4	NO
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	N
E8																E32
N8																N
E7 N7																E31
E6 N6																E30
																Ī
E5 N5																E29
N5																
F4																E28
E4 N4																
E3 N3																E27
Y																
E2 N2																E26
																Ī
E1																E25
	X															E25
Please def	ïne node(s) and	forces		Node(s):		9		All	Clr	Adv	Status:		Success: Fo	orces at node	s defined.	
PX =	10	0		PY =		0		PZ =		0			Apply		Cance	el

📣 MASTAN2																- 0	×
ile <u>V</u> iew	<u>G</u> eometry	<u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults												
																	٦
\vdash	N9 E9	N10 V E10	N11 V 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
_																-	
E	8 N8															E3	32 N32
E	7 N7															E3	31 N31
E	6 N6															E3	30
	N6																N30
_	_															-	
E	5 N5															E2	²⁹ N29
E	4 N4															E2	28 N28
E	3															E2	27
	N3																N27
																F	20
E	2 N2															E2	N26
E	1 N1												Adv	anced Elemen	t Selection		
		X									Pa	rallel to:	On		Range (Inc	lusive)	Off
												X-axis	_	-Inf	Х	Inf	
												Y-axis	- 1	-Inf	Y	Inf	
] Z-axis	_	-Inf		Inf	
													Add	Remove		Reset	
		element(s) an		-	lement(s):			15 16 17 1			Adv	Status:	-		Element loads		
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.

- 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 filenameFrame and click Save. Note that the top of the window has now changed to include the file nameand directory as well as the time the file was last saved.

			RL\Desktop\Fra														- 0	×
<u>F</u> ile	<u>V</u> iew	<u>G</u> eometry	<u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults												
		N9 🔻	N10 V	N11 V	N12 V E12	N13 V E13	N14 V E14	N15 V E15	N16 🕎	N17 🕎	N18 🕎	<u>N19 ♥</u> E19	N20 🕎	N21 🕎	N22 🕎	N23 🖤	N24 🕎	N33
		E9	E10	E11	E12	E13	E14	E15	E16	E17	E18	E19	E20	E21	E22	E23	E24	
	E8	N8															E32	2 N32
	E7	N7															E3 [.]	1
																		N31
	E6	5															E30	0
		N6																N30
	5																E29	0
	H9	N5															EZ	.9 N29
	E4	N4															E28	N28
	E3	N3															E2	27 N27
	Y																	Ì
	E2	N2															E20	26 N26
		112																
	E1																E2	25 N25
	-{1	N1	X															N25
																		10
															-			
-	Title:		ease enter titl	e and selec	арріу			Frame					Status:		Apply	ss: Title defi	ned. Cancel	_
	1100.														түрту			
																•		

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.

esults:	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**.

			<pre> {L\Desktop\Fra </pre>		4:44]												- 🗆	×
<u>F</u> ile	<u>V</u> iew	<u>G</u> eometry	<u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults												
																		1
	\rightarrow	N9 🔽 E9	N10 🕎 E10	N11 🕎 E11	N12 🖞 E12	N13 🖞 E13	N14 V E14	N15 🕎 E15	N16 🕎 E16	N17 🕎 E17	N18 🕎 E18	N19 🕎 E19	N20 🕎 E20	N21 🕎 E21	N22 🕎 E22	N23 🕎 E23	N24 🕎 E24	N33
	-																	
	E8	N8															E32	N32
	E7	N7															E31	N31
	E6	N6															E30	N30
	E5	N5															E29	N29
	E4	N4															E28	N28
	E3	N3															E27	N27
	Y																	
	E2	N2															E26	N26
	E1	N1															E25	N25
		1	X															
			r Elastic Stati				atus:			Ap	plied Load R	Ratio = 1.000	> Succe	ess: Analysi				
		Analysi	s Type:		Planar Fran	ne (x-y)		~							Apply		Cancel	

	N2: C:\Users\SSI															- 0	×
<u>F</u> ile <u>V</u> iev	w <u>G</u> eometry	<u>P</u> roperties	<u>C</u> ondition	ns <u>A</u> nalysis		Deflected S	hape [,] 1 st -Or	rder Elastic	Incr # 1, App	ied Load Ra	tio = 1						
									шог <i>т</i> 1, дрр								
		N10 V E10	N11 V E11	N12 V E12	N13 V E13	N14 V E14	N15 V E15	N16 V E16	N17 🕎 E17	N18 V E18	N19 V E19	N20 V E20	N21 V E21	N22 🔽 E22	N23 🖤 E23	N24 V E24	N33
		EIU	EII	E 12	EIS	L 14	EIJ	EIO		ETO	E 19	E20	E21	LZZ	E23		
	E8 _{N8}															E	N32
	E7															E	31
	N7																N31
	E6 N6															E	30 N30
	Ĭ																Ĭ
	E5 N5															E	29 N29
	E4															E	29
	N4															Ξ.	N28
	E3 N3															E	27 N 2 7
	Y																NZ'
	E2 N2															E	26 N26
	E1 N1	X															25 N25
	A																Ø
	Define elemer		_		Element(s):		All		All		Adv	Status:			s: Deflection		
Defl	Line Type	Solid	~	Scale	10	#	of pts	10	Anima	ate	(1) 1.000	~		Apply		Cancel	

IASTAN2	: C:\Users\SSIF	\L\Desktop\Fr	ame.mat	[14:44]												- 0	×
<u>V</u> iew		<u>P</u> roperties			<u>R</u> esults												
						Deflected S	hape: 1 st -Or	der Elastic, In	ıcr # 1, Appli	ied Load Ra	tio = 1						
4	N9 V	N10 V	N11 🔽	N12 🔽	N13 🗸	N14 🔻	N15 🕎	N16 🕎	N17 🕎	N18 🔽	N19 🖤	N20 V	N21 🕎	N22 🔽	N23 🕎	N24 🕎	N33
	E9	E10	E11	E12	E13	E14	E15	E16	E17	E18	E19	E20	E21	E22	E23	E24	
E	8															E3	32 N22
	N8																N32
E	7															E3	81
	N7																N31
E	6															E3	30
	N6																N30
E	5															E2	29
	N5																N29
F	4															F 2	
Ξ4	4 <mark>1</mark> 4															E2	N28
_																	
E	3 <mark>.</mark> N3															E2	²⁷ N27
	ſ																
Eź	2 ,N2															E2	26 N26
	1 N1																25 N25
4		X															
ode:	33	Disp X		2.688	Disp Y		-0.03312	Disp Z				Status:			Disp. at ALR =		
Displace	ements	Rot X			Rot Y			Rot Z		0.01235	(1) 1.000	\sim		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.

1ASTAN2

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



Bile Yew Geometry Properties Conditions Analysis Besuits		- 🗆
	N21 V N22 V N23 V E21 E22 E23	N24 🖞 E24
E_{7}^{7} F_{6}^{7} F_{6}^{7} F_{6}^{7} F_{6}^{7} F_{7}^{7} F_{6}^{7} F_{7}^{7} F_{7		E32
		Ĭ
		E31
$E_{1}^{E_{1}}N_{2}^{E_{2}}$ $E_{1}^{E_{1}}N_{2}^{E_{2}}$ $E_{2}^{E_{2}}N_{2}^{E_{2}}$ $\sum_{i=1}^{n_{1}}N_{1}^{i}$ \times		
		E30
		Ĭ
		E29
		E28
		Ĭ
		E27
		E26
		E25
Second-Order Elastic Static Analysis Status: Incr # 10, Applied Load Ratio = 1.000> Suc		\prec
Second-Order Elastic Static Analysis Status: Incr # 10, Applied Load Ratio = 1.000> Su		
Second-Order Elastic Static Analysis Status: Incr # 10, Applied Load Ratio = 1.000> Sur		
Second-Order Elastic Static Analysis Status: Incr # 10, Applied Load Ratio = 1.000> Su		
	Success: Analysis Complete	
Solution Type: Predictor-Corrector ~ Incr Size: 0.1 Max. # of Incrs: 10		: 1
Analysis Type: Planar Frame (x-y) V [[Kff] Start New V		Cancel

MASTA	AN2: C:\Users\S	SIRL\Desktop\Fr	ame.mat ['	14:44]												- 0
<u>V</u> ie	ew <u>G</u> eometr	y <u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults											
	·															
	N9 V E9	N10 V E10	N11 V E11	N12 🕎 E12	N13 👽 E13	N14 🕎 E14	N15 👽 E15	N16 👽 E16	N17 🕎 E17	N18 🕎 E18	N19 🕎 E19	N20 V E20	N21 👽 E21	N22 👽 E22	N23 🕎 E23	N24 V
		210	211				210			LIO	210	LLO			220	
	E8 N8															E32
	E7 N7															E31
	E6 N6															E30
	E5 N5															E29 N
																Ĭ
	E4 N4															E28
																Ĭ
	E3 _{N3}															E27 N
	Y															
	E2 N2															E26
	E1 N1															E25
-		X														
lode:	33	Disp X		2.852	Disp Y		-0.05354	Disp Z				Status:		Success: D)isp. at ALR =	= 1.0000
Disp	lacements	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**.

<u>V</u> ie	w <u>G</u> eometry	<u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults											
	N9 V	N10 V	N11 🖤	N12 V	N13 🖤	N14 👽	N15 🕎	N16 👽	N17 V	N18 V	N19 🕎	N20 V	N21 👽	N22 🖤	N23 🗸	N24 V
	E9	E10	E11	E12	E13	E14	E15	E16	E17	E18	E19	E20	E21	E22	E23	E24
	E8 _{N8}															E32
	N8															
	E7 N7															E31
	E6 N6															E30
	-															E29
	E5 N5															E29
	E4 N4															E28
	F3															E27
	E3 N3															
	Y															
	E2 N2															E26
	Ĭ															
	E1 N1															E25
		X														\prec
	Please select	node(s) and f	ïxity(s)		Node(s):		9 17 33	3	All	Clr	Adv	Status:		Success:	Node fixities	defined.
-disp		Y-disp		Z-disp		X-r			Y-rot		Z-rot			Apply		Cancel

承 MASTAN2: C:	\Users\SSIRL\Desktop\Fra	me.mat [14:	44]									—	
<u>F</u> ile <u>V</u> iew <u>G</u>	Geometry <u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults									
	N9												
	E9	N101 E10	N11	112									
	E8 _{N8}			N12 E12									
	E7 _{N7}			E13	E14 N15	N16							
					EIS	E16	17	N18					
	E6 _{N6}							18		N20			
	E5 _{N5}									E20 N21 E21	N22 N23		
											E23 N24	3	
	E4 _{N4}												
	E3 _{N3}										E32 _{N32}	2	
	N3										E31 _{N3} ,		
	E2 _{N2}												
	F										E30 _{N30})	
											E20		
	z	×									E29 _{N29})	
	L										E28 _{N28}	3	
											E27 _{N27}	7	
											E26 _{N26}	6	
											E25 125	Ĵ.	
Place	se select node(s) and fi	vitu(s)	N	lode(s):	9 17 33		All	Clr	Adv	Status:	Success: Node fixiti	es defined	
X-disp	Y-disp	xity (3)	Z-disp		X-rot	Y-r			Z-rot	Oldius.	Apply	Canc	

MASTAN2

3-D Second-Order Elastic Analysis

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

MAS	TAN2: C:\Users\SSIR	RL\Desktop\Fran	me.mat [14	:44]								—		×
<u>F</u> ile <u>V</u>	/iew <u>G</u> eometry	<u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults									
		E9 E8 _{N8} E7 _{N7} E6 _{N6} E5 _{N5} E4 _{N4}	E10	N11 E11	N12 E12 E13	N14 E14 E15	N16 E16 E11	V Luc	N19 E19 E20	N21 E21 E22	N23 E23 E24 E24 E24 E32 N33 E32 N32			
		E3 _{N3}												
		E2 _{N2}									E31 _{N31}			
		-E1nt									E30 _{N30}			
			×								E29 _{N29}			
	Z										E28 _{N28}			
											E27 _{N27}			
											E26 _{N26}			
											E25 N25			
		er Elastic Stat			Status:			Incr # 7, A	pplied Load Ratio = 0.					
	Solution Type:		or-Corrector		~	Incr Size:	0.1		Max. # of Incrs:	10	Max. Appl. Ratio:		1	
	Analysis	s Type:	S	Space Fram	e	\sim		[Kff]	Start New	~	Apply	Car	ncel	

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\Fra	ime.mat [14	:44]										- 🗆	>
e <u>V</u> iew <u>(</u>	<u>G</u> eometry <u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults										
		N10												
	E9 E8 _{N8}	E10	N11 E11	N12										
	IN8		EII	E12 N13 E13	N14									
	E7 _{N7}				E14 N	15 N16								
	F6					E16	N17	N19						
	E6 _{N6}						E18	E19 N20	N21					
	E5 _{N5}							E20	E21	N22	N23			
	E4										E23 N24 E24	N33		
	E4 _{N4}											E32 _{N32}		
	E3 _{N3}											N32		
	Y E2											E31 _{N31}		
	E2 _{N2}											E30 _{N30}		
	- EXT													
	X45	×										E29 _{N29}		
	z											E28 _{N28}		
												N28		
												E27 _{N27}		
												E26 _{N26}		
												N26		
												E25 N25		
												BB		
	Elastic Critical Load A	Analysis		Status:			# of Mode	es Calculated = 1	> Succ	ess: Analy	sis Complete			
	Analysis Type:		Space Fram		~		lax. # of Modes:	< 1	>	sos. valuty:	Apply	-	Cancel	-

lie ver Generit Popertie Control Augus Bent Detect Stape: Eneir Critical Lauk Molé #1, Appled Lauk Rails = 0.08708	TAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44] — 🗆
	Deflected Shape: Elastic Critical Load, Mode # 1, Applied Load Ratio = 0.68709
E20 N30 E20 N29 Z E28 N28 E27 N27 E26 N26	$\begin{array}{c} 199 \\ \hline
z E ²⁸ _{N28} E ²⁷ _{N27} E ²⁶ _{N26}	E2 _{N2} E30 _{N30}
E27 _{N27} E26 _{N26}	z
E ²⁶ N26	E28 _{N28}
	E27 _{N27}
E25.165	E26 _{N26}
	E25 N25
Define element(s) and parameters Element(s): All All Cir Adv Status: Success: Deflection shown	Define element(s) and parameters Element(s); All All Cir Adv Status: Success: Deflection shown
Define element(s) and parameters Cline (s) Air Air Cline (s) Automatical (s) Outcome (s) Defi Line Type Solid V Scale 30 # of pts 10 Animate (1) 0.687 V 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 **CIr** 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 **CIr** 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]	— D	×
<u>File View G</u> eometry Properties Conditions Analysis Results		
F10 N11		
N8 E11 E12		
E7 _{N7}		
E6 _{N6}	E18 N19 E18 E19 N20 E20 N21	
E5 _{N5}	E20 N21 N22 E21 N23 E22 N23	
	E22 E23 N24 N33	
E4 _{N4}		
E3 _{N3}	E32 _{N32}	
Y	E31 _{N31}	
E2 _{N2}		
ET M	E30 _{N30}	
	E29 _{N29}	
z		
	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: C:\Users\SSIRL\Desktop\Frame.mat [14:44]	-	- 🗆 X
<u>Eile View Geometry Properties Conditions Analysis Results</u>		
N111		
E12		
E7 _{N7}	E14 N16 N16	
E6 _{N6}	E18 N200 N21	
E5 _{N5}		
*	E22 N23 N24 N33 E24 N33	
E ⁴ N4		
E3 _{N3}	E32 _{N32}	
	E31 _{N31}	
E2 _{N2}		
Et	E30 _{N30}	
ENT.	E29 _{N29}	
z x	N29	
_	E28 _{N28}	
	E27 _{N27}	
	E26 _{N26}	
	E25 N25	
Define element(s) and warping restraint Element(s):	1 25 9 All Clr Adv Status: Success: Warping Restraint d	lefined
Node i Warping Restraint Fixed		Cancel

\sim	
Ζ	
\triangleleft	
S)	
\triangleleft	
\geq	

₹

	MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]	- 🗆 X
	<u>File View Geometry Properties Conditions Analysis R</u> esults	
MASTAN2	$ \begin{array}{c} $	17 N18 N19 N20 N21 N22 N23 17 E18 E19 E20 E21 N22 N23 E31 N31 E30 N30 E28 N29 E28 N29 E28 N26 N27 N27 E28 N28 N29 N29 E28 N29 E28 N29 E29 N29 E28 N29 E29 N29 E28 N29 E29 N29 E29 N29 E29 N26 N26 N26 Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y
	Define element(s) and warping restraint Element(s): 8 24 32	All Clr Adv Status: Success: Warping Restraint defined.
	Node i Warping Restraint Continuous V Node j Warping Res	straint Fixed V 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: C:	\Users\SSIRL\Desktop\Fra	ame.mat [14	:44]	-	
e <u>V</u> iew <u>C</u>	<u>G</u> eometry <u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	Results	
	Na				
	E9	N10	N11		
	E8 _{N8}	210	E11	N12 E12 N13	
	F7			E_{13} E_{14} N_{15} E_{14} N_{15}	
	E7 _{N7}				
	E6 _{N6}				
	Ť			E19 H20 N21 E20	
	E5 _{N5}				
	E4			E23 N24 E23 E24 N33	
	E4 _{N4}			E22	
	E3 _{N3}			E32 _{N32}	
	Y			E31 _{N31}	
	E2 _{N2}				
	(F1)			E30 _{N30}	
				E29	
	z	×		E ²⁹ N29	
	L			E28 _{N28}	
				E27 _{N27}	
				E20	
				E26 _{N26}	
				E25 N25	
	Elastic Critical Load	_		Status: # of Modes Calculated = 1> Success: Analysis Complete	
	Analysis Type:	5	Space Fram	Max. # of Modes: < 1 > Apply Cancel	

/ASTAN2: C:\Users\SSIRL\Desktop\Frame.mat	
<u>V</u> iew <u>G</u> eometry <u>P</u> roperties <u>C</u> ondition	
	Deflected Shape: Elastic Critical Load, Mode # 1, Applied Load Ratio = 1.3106
N9	
E9 N10	N11
E8 _{N8}	E11 V12 E11 E12 V13
E7 ₂₁₇	
	E10 N18 E17 N18
E6 _{N6}	E18 N19 E19 N20
F	E10 H21 H22 H22 H22 H22 H22 H22 H22 H22 H22
E5 _{N5}	E22 H23
E4 N4	E23 E24 N33
- 184	
E3 N3	E32 N32
	E31 _{N31}
E2 _{N2}	
*	E30 _{N30}
FIN	
X	× E ²⁹ N29
z	
	E28 _{N28}
	E27 _{N27}
	E25.vc
Define element/s) and parameters	Element(s): All All Clr Adv Status: Success: Deflection shown
Define element(s) and parameters Defl Line Type Solid ~	Element(s): All Clr Adv Status: Success: Deflection shown Scale 30 # of pts 10 Animate (1) 1.311 V Apply Cancel
J. Jona	

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.

1ASTAN2

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.



MASTAN	2: C:\Users\SSIR	L\Desktop\Fra	me.mat [14	:44]								-		×
<u>F</u> ile <u>V</u> iew	<u>G</u> eometry	<u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults									
		E9 E8 7 E7 E7 E6 E5 55		E11	N12 E12 E13	N14 E14 E15	116 E16 E	17 V18 17 E18		N21 E21 N22	112			
		E4 _{N4}								E22	N23 E23 E24 N33			
		Ť									E32 _{N32}			
		E3 _{N3} Y E2 _{N2}									E31 _{N31}			
		Fin									E30 _{N30}			
	ź		X								E29 _{N29}			
											E28 _{N28}			
											E27 _{N27}			
											E26 _{N26}			
											E25,125			
		er Elastic Sta			Status:			Incr # 10,	Applied Load Ratio =					
So	olution Type:		or-Corrector		~	Incr Size:	0.1		Max. # of Incrs:	10	Max. Appl. Ratio:		1	
	Analysis	s Type:	5	Space Fram	e	\sim		🗌 [Kff]	Start New	~	Apply	Can	cel	

	MASTAN2:	C:\Users\SSIF	RL\Desktop\Fra	me.mat [14:	44]											—		×
<u>F</u> ile	<u>V</u> iew	<u>G</u> eometry	<u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults												
			29 E9 E8 N8 E7 E6 F F F F F F F F F F F F F		E11	N12 E12 E12 E1	E14			117 N18				N22 122 122 122 123 123 123 123 123 123 1	N24 E24			
			E ³ N3												E32 _{N32}			
			Y E2 _{N2}												E31 _{N31}			
			EINT												E30 _{N30}			
		ź	H	×											E29 _{N29}			
															E28 _{N28}			
															E27 _{N27}			
															E26 _{N26}			
															E25 225			
I	Node:	33	Disp X		2.852	Disp Y	-0.05	5354 D	isp Z		0		Status:	Succe	ss: Disp. at ALR	= 1.000	0	
	Displace	ements	Rot X		0	Rot Y		0 F	Rot Z	C	0.01243	(10) 1.000	\sim	Apply		Can	cel	

Section 5: Using MSASect

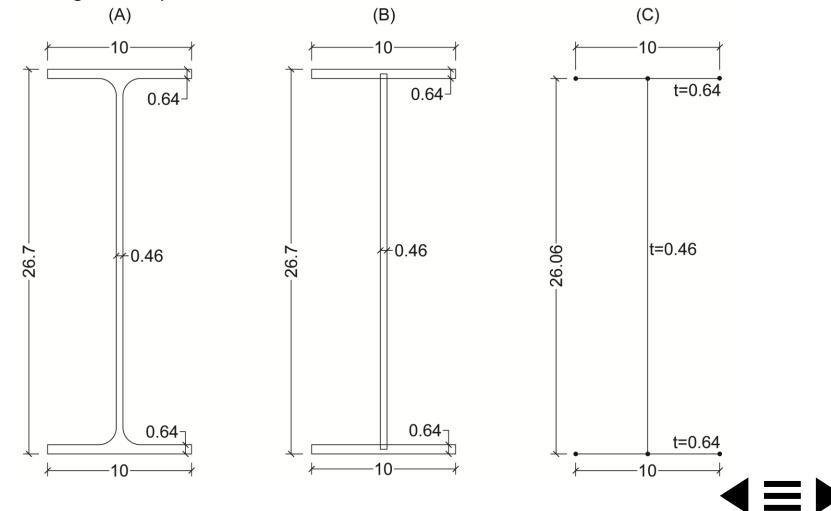


MSASect

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.

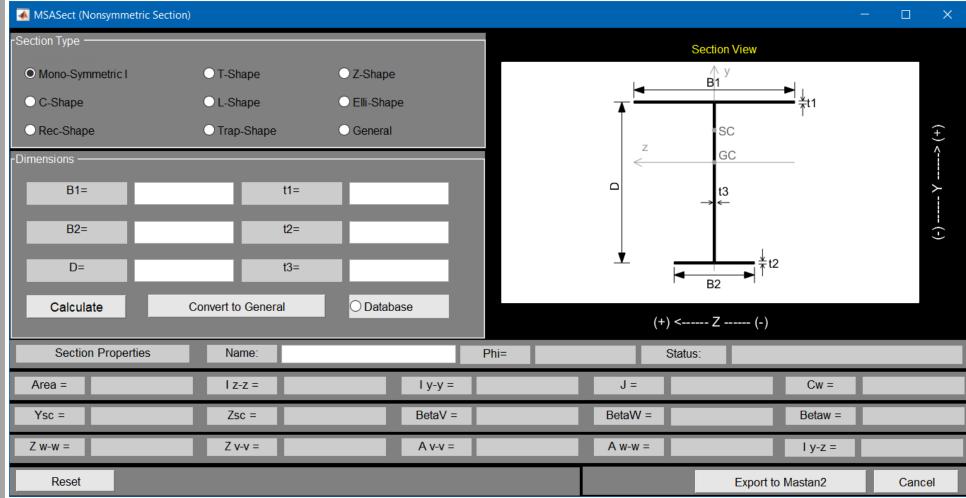
Note: The dimensions to enter in the template correspond to Figure B on the previous page. While the section property calculations need to be completed using the dimensions shown in Figure C, this information is automatically generated based on the assumption that the numbers provided followed Figure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

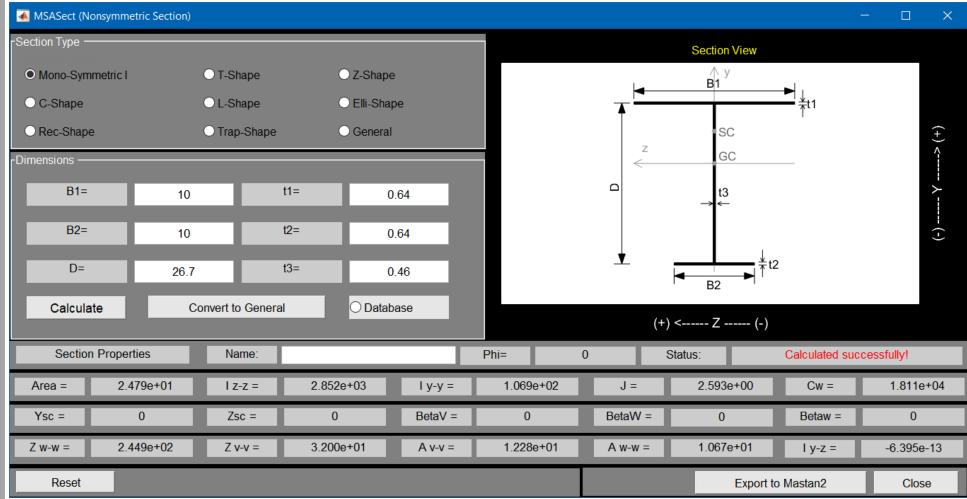




	rties <u>Conditions</u> <u>Anal</u>		N14 E14 E14 E15	N16	177 118 119 17 E18 E19		N22 E22 E22 E23 E24 E24	E32 _{N32}	
	E9 E10 N11 8 N8 E11 7 N7 6 N6 5 N5 4 N4 3 N3 4	N12 E12	E14 N15	N16	17 N18 N19	N20 E20 N21	N22 E22 E23 E23	E32 _{N32}	
	E9 E10 N11 8 N8 E11 7 N7 6 N6 5 N5 4 N4 3 N3 4	N12 E12	E14 N15	N16	17 N18 N19	N20 E20 N21	N22 E22 E23 E23	E32 _{N32}	
	E9 E10 N11 8 N8 E11 7 N7 6 N6 5 N5 4 N4 3 N3 4	N12 E12	E14 N15	N16	17 N18 N19	N20 E20 N21	N22 E22 E23 E23	E32 _{N32}	
	⁸ N8 E10 7N7 ⁶ N6 ⁵ N5 ⁴ N4 ³ N3	N12 E12	E14 N15	N16	17 N18 N19	N20 E20 N21	N22 E22 E23 E23	E32 _{N32}	
	6 N6 5 N5 4 N4 3 N3		E14 N15	N16	17 N18 N19	N20 E20 N21	N22 E22 E23 E23	E32 _{N32}	
E E E	6 N6 5 N5 4 N4 3 N3			N16	17 N18 N19	N20 E20 N21	N22 E22 E23 E23	E32 _{N32}	
E E E	5 N5 4 N4 3 N3			EI	E18 N19	N20 E20 N21	N22 E22 E23 E23	E32 _{N32}	
E E E E	5 N5 4 N4 3 N3					N20 E20 N21	N22 E22 E23 E23	E32 _{N32}	
E	4 N4 3 N3						N22 E22 E23 E23	E32 _{N32}	
E	3 N3						E23	E32 _{N32}	
E	3 N3							E32 _{N32}	
E	Y								
E	Y								
	2 _{N2}							E31	
	- NZ							E31 _{N31}	
								Е30 _{N30}	
								Ĩ	
t	×							E29 _{N29}	
Z								E28 _{N28}	
								N28	
								E27	
								E26 _{N26}	
								E25N25	
								25. 26	
Please enter section		Section 3	Name:		MSASect		_		
Area = 0	l z-z = Zsc =		· · ·		J = (0 Cw : 0 Betaw		l y-z = 0	_
Ysc = 0 Z w-w = inf		0 Be	etaV =	U B6				Advanced	\sim



MASTAN2



	MASTAN2:	C:\Users\SSIR	L\Desktop\Fra	ime.mat [13:	42]									- 🗆 ×
<u>F</u> ile	<u>V</u> iew	<u>G</u> eometry	<u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults								
			N9											
			E9		N11									
			E8 _{N8}	E10	E11	N12	N13							
							E13 N14	1145						
			E7 _{N7}				E14	N15 E15	11/7					
								E16	E17					
			E6 _{N6}						Ē18	E19				
			E5 _{N5}							E20	E21	N22		
			•									E22 N23 N2 E23 N2		
			E4 _{N4}									E2	4 N33	
													E32 _{N32}	
			E3 _{N3}											
			Y										E31 _{N31}	
			E2 _{N2}											
			FIN										E30 _{N30}	
													E29 _{N29}	
		ź		X									n29	
		2											E28 _{N28}	
													*	
													E27 _{N27}	
													F 00	
													E26 _{N26}	
													E25 N25	
													20. 20.	
		Please enter			_	Section 3	Name:	W27x84Ha		MSASect	Status:	10/10		0.00455.15
4	Area = Ysc =	24.7		l z-z = Zsc =		2.05 D	I y-y = BetaV =	106.878 0	J = BetaW =	2.59315 0	Cw = Betaw =	18110 0	l y-z = Advanced	-6.39488e-13
														\sim

◀☰

MASTAN2:	C:\Users\SSIRL\Deskto	op\Frame.mat [14	:44]							- 0	×
<u>F</u> ile <u>V</u> iew	<u>G</u> eometry <u>P</u> rope	rties <u>C</u> onditions	<u>A</u> nalysis <u>R</u> esults								
	<i>/</i>	Ma 1									
	_	E9 E10	N11								
	E	^B N8	E11 N12 E12	N13 N14							
	E	7 <mark>.</mark> N7		E13 E14	N15 E15						
		Ť			E15 E16	E17 N18					
	E	⁶ N6				E118	N19 E19 N20				
	E	5 <mark>N</mark> 5					E20		122 N23		
		T						E	E23 N24		
	E	⁴ N4									
	E	³ N3							E32 _{N32}		
									E31 _{N31}		
	E	2 _{N2}									
	Æ	T S							E30 _{N30}		
	The second se								E29 _{N29}		
	z	X							<u>N</u> 29		
	_								E28 _{N28}		
									F07		
									E27 _{N27}		
									E26 _{N26}		
									Ţ.		
									E25 1/25		
	Please enter section		Section 4				MSASect	Status:	Success: Section 3		
Area =	0	z-z =	0	l y-y =	0	J =	0	Cw =	0 I y-z =		_
Ysc =	0	Zsc =	0	BetaV =	0	BetaW =	0	Betaw =	0 Advance		~
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
А	in ²	24.7	24.79	0.4 %
lzz	in ⁴	2850	2852	0.1%
lyy	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 %

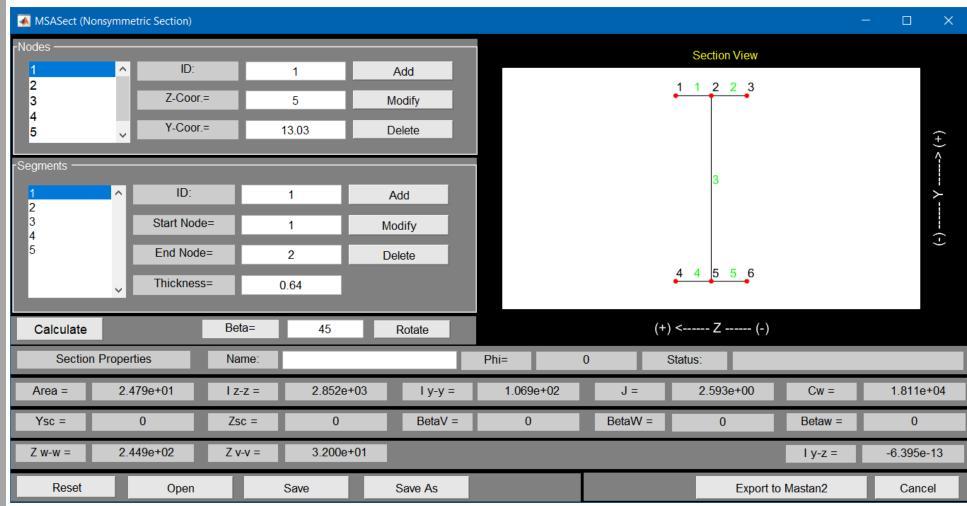
From the comparison of section properties from AISC and the values calculated by MSASect, most of the calculated properties match well. Take note that some of the template shapes calculate standard shear area values. To match the previous analysis, the **A v-v** and **A w-w** would need to be set to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

MASTAN2: (C:\Users\SSIRL\D) esktop\Fra	me.mat [13:	42]										- 🗆	×
<u>F</u> ile <u>V</u> iew	<u>G</u> eometry <u>P</u>	roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults										
<u>Tue</u> <u>T</u> iem		E^{9} E^{9} E^{7} E^{7} E^{6} E^{5} E^{5} E^{4}	E10	E11	N12 E12	N13 E13 E14	115 N16 E15 E16	N17 E17 E17 E1	8 N19 E19	N20 E20 E20 E2	. V I	2 2 2 E23 E23 E24			
		E3 _{N3} Y											E31 _{N31}		
		E2 _{N2}											Ť		
													E30 _{N30}		
	z		X										E29 _{N29}		
													E28 _{N28}		
													E27 _{N27}		
													E26 _{N26}		
													E25 N25.		
Select Section # for information			Section	# 3	 ✓ Nam 	we: W2	7x84Hand	Status		Success:	Section 3 displa	ayed			
Area =	24	.7876	z-z =		2852.05	l y-y =	106.878		2.5	932 Cv	v =	18110	l y-z =	-6.3	949e-13
Ysc =		0	Zsc =		0	BetaV =	0				aw =	0	Phi =		0
Z w-w =	24	4.883	Z v-v =		32	A v-v =	12.282	A w-w =	10.6	667			Cancel		

 \blacksquare

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.



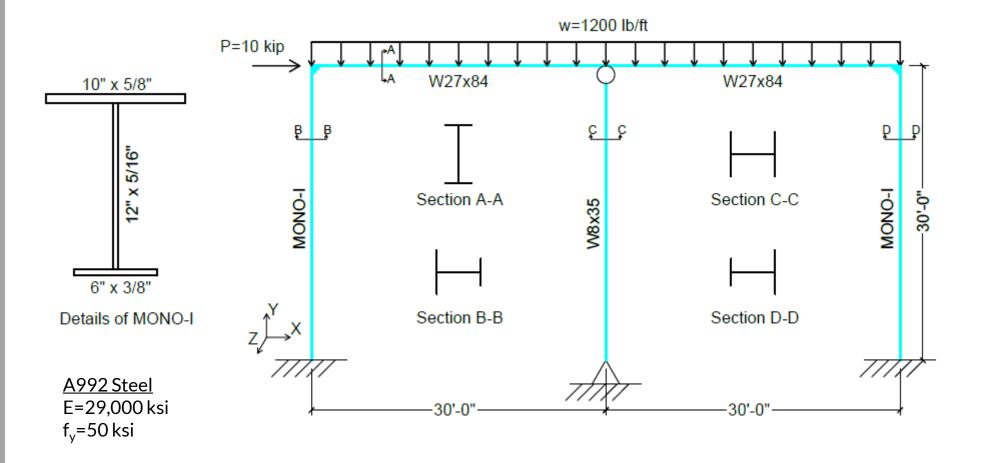
MASTAN2

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.



Adding Interior Column

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

承 Mast	TAN2: C:\L	Jsers\SSIRI	L\Desktop\Fra	me.mat [14:	:44]							-		×
<u>F</u> ile <u>V</u> i	iew <u>G</u> e	eometry	<u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults								_
			Ng	NIAO										
			E9	N10. E10	N11	N12								
			E8 _{N8}		E11	F12 N13	N14							
			E7 _{N7}				=14 N15	116						
							E	16 N17 N1 E17 N1						
			E6 _{N6}					E1	E19					
			E5 _{N5}						E2	E21	V22			
			Ť							E	E22 N23 N24 E23 E24	N 33		
			E4 _{N4}											
			E3 _{N3}									E32 _{N32}		
			Y									E31 _{N31}		
			E2 _{N2}											
			- Fint									E30 _{N30}		
												E29 _{N29}		
		z		X										
												E28 _{N28}		
								N34				E27		
												NZ7		
												E26 _{N26}		
												E25		
												25		
				Ple	ease enter	coordinates of node				Status:	Success:	Node 34 defined.		
	x =		360			/ =	0	z =	0		Apply	Can	cel	

MA	ASTAN2: (C:\Users\SSIR	RL\Desktop\Fra	me.mat [14:	:44]						—	
<u>F</u> ile	<u>V</u> iew	<u>G</u> eometry	<u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults						
			est a									
			E9	N10 E10	N11	LUS .						
			E8 _{N8}		E11	N12 E12 N13 N14						
			E7 _{N7}			E13 N14 E14	N15 E15 N16					
			Ť				E15 E16 E17	N18				
			E6 _{N6}					E18 E19	N20			
			E5 _{N5}						E20 N21	N22		
										E22 123 124 E23 124	N33	
			E4 _{N4}							E24	*	
			E3							E32	N32	
			E3 _{N3}							E3	l	
			E2 _{N2}							LU	N31	
							E33			E30	N30	
		_	4	X						Ezt	N29	
		Z								E28	3N28	
							N34					
							•			E2	N27	
										E26	N26	
										E2	N25.	
										F	*	
				Please sel	ect elemen	t end nodes and define beta			Status:	: Success: Elem	ent 33 defined	
	No	de i	۲			ode j	Beta (Deg)	0.0	Apply	Cance	el

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]		×
<u>F</u> ile <u>V</u> iew <u>G</u> eometry <u>P</u> roperties <u>C</u> onditions <u>A</u> nalysis <u>R</u> esults		
PM -		
E8 _{N8} E11 N12 N13 N14		
F7		
E16	N17 E17 N18	
	E17 E18 N19 N20 N20	
E5	E20 N21 N22 E21	
	E22 N23 N24	
E4 _{N4}	E24 N33	
	E32 _{N32}	
E3 _{N3}		
E2 _{N2}	E31 _{N31}	
E	E33 E30 N30	
E M		
×	E29 _{N29}	
Ź	E28 _{N28}	
	N34 E27	
	E26 _{N26}	
	E25 25	
Select Material # and element(s) Element(s): 33	All Clr Adv Status: Success: Material attached.	
Material # 1 ~ Details: A992 <click properties="" see="" to=""></click>	V Apply Cancel	

AASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]		- 🗆 X
<u>File View Geometry Properties Conditions Anal</u>	lysis <u>R</u> esults	
N9 E9 E10 E11 E8 N8 E7 N7 E6 N6 E5 N5 E4 N4 E3 N3 E2 N2		
E1 M	Select section and properties will be inserted	
X	Select section and properties will be inserted E29 Database: Section:	
Z	AISC (in) V10X22 W10X19 W10X17 W10X17 W10X15	
	W Shapes V W10X12 E27 N27 W8X67 W8X58	
	Shear Areas: W8X40 E20N26 W8X35	
	infinite W8X31 W8X28 W8X24	
	Clear W8X21	
Please enter section properties	Section 5 Name: O 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	2: C:\Users\SSIR	L\Desktop\Fra	me.mat [14:	44]											—		×
<u>F</u> ile <u>V</u> iew	<u>G</u> eometry	<u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults												
		29															
		E9	N10. E10	N11	N12												
		E8 _{N8}		E11	E12 N13												
		E7 _{N7}			É13	E14	N15 E15	N16									
		Ť						E16	17	18.							
		E6 _{N6}										N20					
		E5 _{N5}										E20 N21 E21	N22	N23			
													E22	E23 E24	N33		
		E4 _{N4}													*		
		E3 _{N3}												E	³² N32		
		Y												E	³¹ N31		
		E2 _{N2}						E33							Ĩ		
		FINT						E33						E	30 _{N30}		
														E	29 _{N29}		
	z		Х														
														E	28 _{N28}		
								Ņ	134					E	27 _{N27}		
														E	²⁶ N26		
															25 105		
														T T	20		
	Select Section	# and eleme	ent(s)	E	ement(s):		33		All	Clr	Adv	Status:		Success: S	ection attached.		
S	ection #	4	~ Details	: W8X35	<click td="" to<=""><td>see propert</td><td>ies></td><td></td><td></td><td></td><td></td><td></td><td>~</td><td>Apply</td><td>Ca</td><td>ncel</td><td></td></click>	see propert	ies>						~	Apply	Ca	ncel	

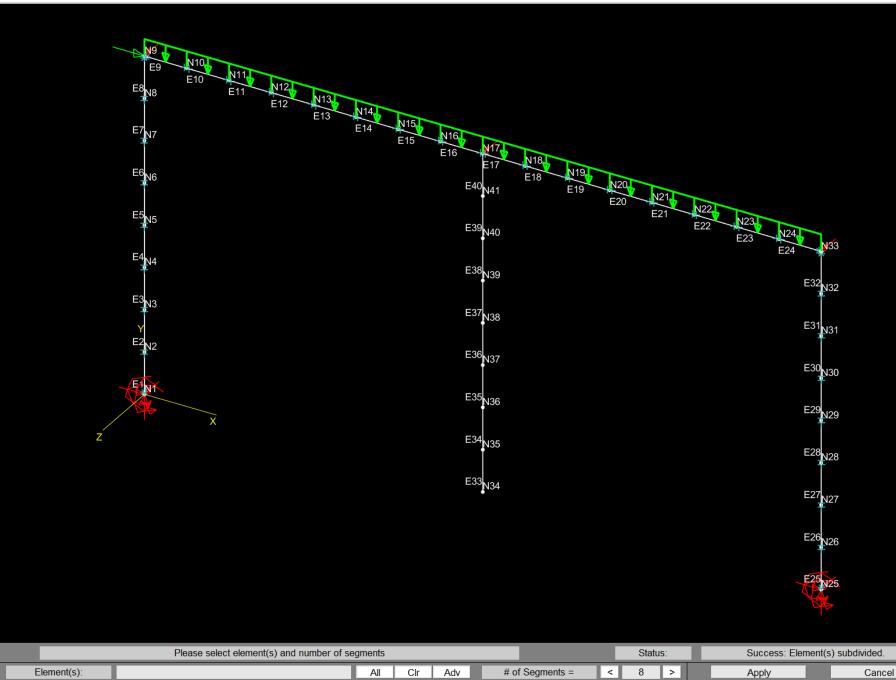
Element Modification

- 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**.
- At the bottom menu bar, define a pin support by clicking in the check boxes just to the left of Xdisp, 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

<u>File View Geometry Properties Conditions Analysis Results</u>





<u>V</u> iew <u>G</u> eometry <u>P</u> ro	operties <u>C</u> onditions	<u>A</u> nalysis <u>R</u> esults		
/	E8 18	N11 E11 N12 E12 N13	N14	
	E7 _{N7} E6 _{N6}	E13	E14 N15 N16 N17 N18 N19	N20
	E5 _{N5}		E ⁴⁰ N41 E ³⁹ N40	N20 N21 N22 N23 N24 N2 E21 E22 E23 N24 N2 E24 K
	E4 ₁₄ E3 ₁₁₃		E38 _{N39} E37 _{N38}	E32 ₀₁ E31 ₁₁
			E36 _{N37} E35 _{N36}	E30
z	X		E ³⁴ N35	E29 _N E28 _N
			E33 N34	E27 _{N2} E26 _{N2}
				E25

Please select node(s) and fixity(s)	Node(s):	34	All Cli	Adv St	tatus:	Success: Node t	fixities defined.
-disp	Z-disp	X-rot	Y-rot	Z-rot		Apply	Cancel
						•	

MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]

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: C:\Users\SSIRL\Desktop\Frame.mat [14:44]			×
<u>File V</u> iew <u>G</u> eometry <u>P</u> roperties <u>C</u> onditions <u>A</u> nalysis <u>R</u> esults			
E10 N11			
E13 $N14$ $N15$ $E14$ $E14$ $E14$ $E14$ $N16$			
E_{10} E			
E6. C			
	33		
	55		
E38 _{N39} E32 _N	32		
E37 _{N38}	31		
E36 _{N27}			
	30		
E35 _{N36}			
X E29	29		
z E34 _{N35} E38			
**************************************	28		
E33 _{N34}			
E27 N34	27		
Advanced Element Selection	n		
	(Inclusive)	0	Dn
X-axis 359		361	
✓ Y-axis -inf		+inf	
Z-axis -inf		+inf	الر
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 F			
Node i Warping Restraint Continuous V Node j Warping Restraint Continuous V Apply	Canc	el	

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.





<u>File View Geometry Properties Conditions Analysis Results</u>



Cancel



Node i

<u>File View G</u>eometry <u>P</u>roperties <u>C</u>onditions <u>A</u>nalysis <u>R</u>esults



Node j

Warping Restraint

Free

 \sim

Apply

Cancel

Node i

Warping Restraint

Continuous

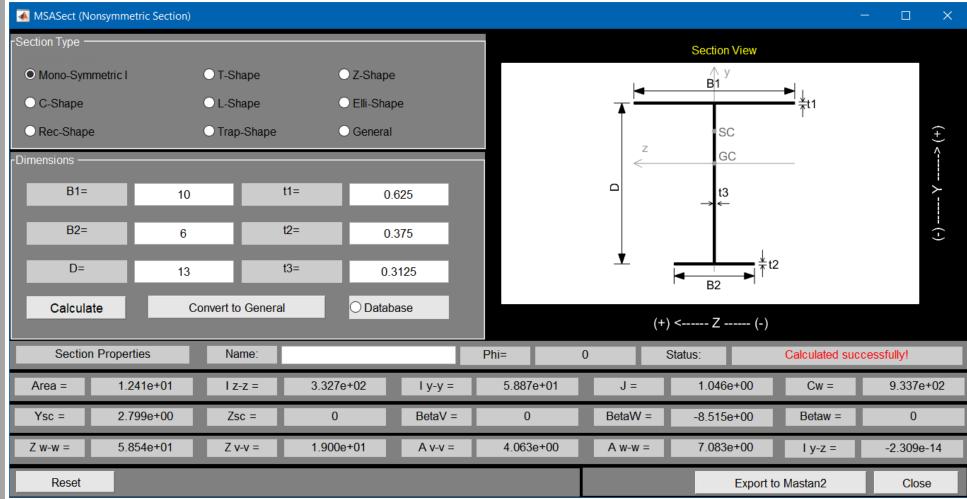
End Moment Release

- 1) From the **Geometry** menu select **Define Connections** and submenu option **Flexure**.
- 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]		– 🗆 X
<u>F</u> ile <u>V</u> iew <u>G</u> eometry <u>P</u> roperties <u>C</u> onditions <u>A</u> r	alysis <u>R</u> esults	
	1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	
	E ³³ N ³⁴ E ²⁷ N ²⁷	
	E26 _{N26}	
	E25,025	
Define element(s) and connections	Element(s): 40 All Cir Adv Status: Success: Connection(s)	defined.
Node i Type Rigid ~ kz	inf ky inf	
Node j Type Pinned v 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.



MASTAN2: (:\Users\SSIRL\Desk	top\Fra	me.mat [13:	:42]									—	
<u>F</u> ile <u>V</u> iew	<u>G</u> eometry <u>P</u> rop	erties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults									
<u>Eile ⊻</u> iew		erties 19 E9 E8 N8 E7 N7 E6 N6 E5 N5 E4 N3 E2 N2 E2 N		Analysis E11	N12	113 N14, 113 E14	E15	EX EX EX	N17 E17 E17 E1 39 N40 38 N39 37 N38 36 N37 35 N36		N21 E21	N22 E22 E23 E24 E32 E31 E30 E30	N31	
		B B	X						100			E29	N29	
	z								³⁴ N35			E28	N28	
								E	³³ N34			E27 3	N27	
												E26		
												E26	N25.	
Selec	t Section # and n	nodify p	properties	1	~	Name:		Mono I		MSASect	Status:	Success: Section	n 1 modified.	
Area =	12.40	_	z-z =		332.746	I y-y =		<mark>58.8651</mark>	J =	1.04643	Cw =		-z =	-2.30926e-14
Ysc =	2.799		Zsc =		0	BetaV =		0	BetaW =	-8.51467	Betaw =		anced	~
Z w-w =	58.54	105	Z v-v =		19	A v-v =		4.0625	A w-w =	7.08333		Apply	Can	cel

Column Orientation

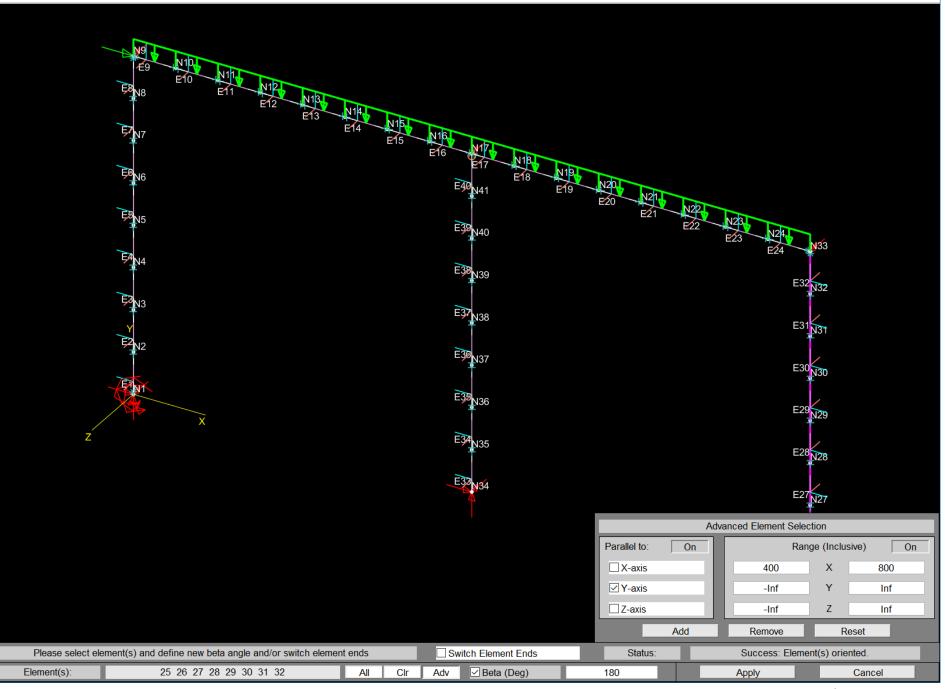
- 1) Since the section was modified, the exterior columns are already assigned the appropriate section. The orientation just needs to be verified.
- 2) From the **Geometry** menu select **Re-orient Element(s)**.
- 3) From the View menu select Labels and submenu option Element local x'-y'-z' axes. Each 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.

<u>File View Geometry Properties Conditions Analysis Results</u>



Please select ele	ement(s) and define new beta angle and/or switch element ends	Switch Element Ends	Status:		
Element(s):	All	Clr Adv ØBeta (Deg)	0.0	Apply	Cancel

<u>File View Geometry Properties Conditions Analysis Results</u>



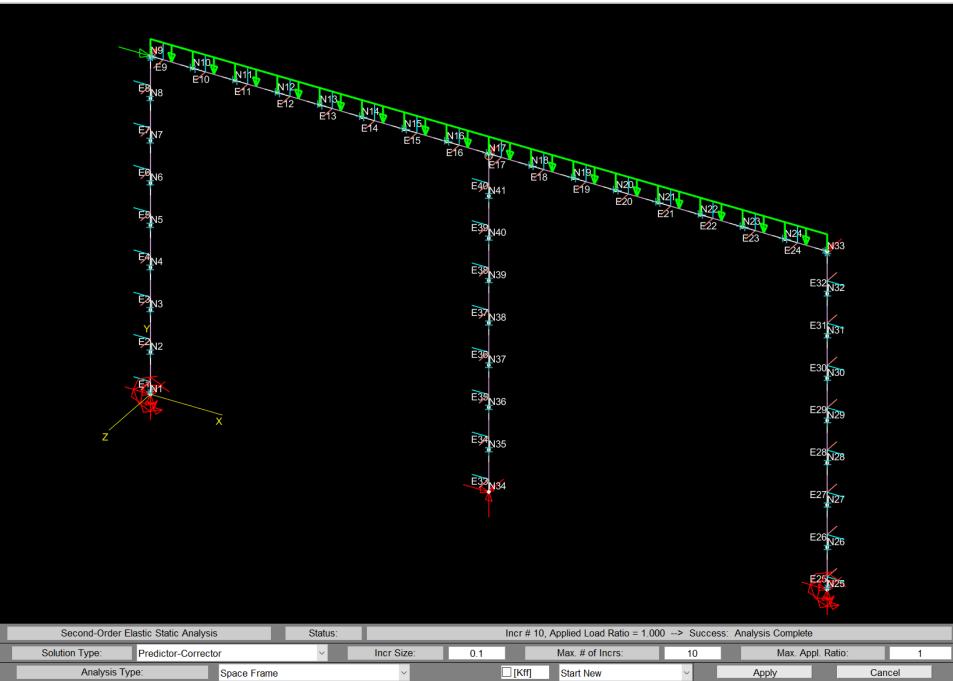
3-D Second-Order Elastic Analysis

- 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.

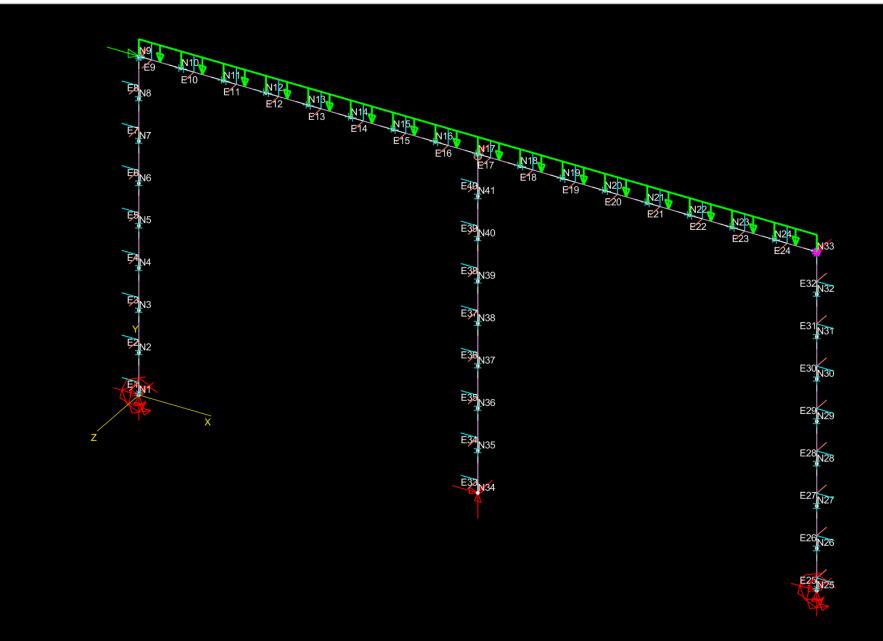
Results:

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





<u>File View Geometry Properties Conditions Analysis Results</u>



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]	- D	×
<u>File View G</u> eometry <u>P</u> roperties <u>C</u> onditions <u>A</u> nalysis <u>R</u> e	sults	
	Deflected Shape: Elastic Critical Load, Mode # 1, Applied Load Ratio = 2.1605	
Ng d		
F9 N10		
E8 _{N8} E10 N11		
	M13	
E7 _{N7}	E13 N14 N15 Luce	
<u>-</u> N7	E_{14} E_{15} N_{16} N_{17} N_{17} N_{17} N_{17} N_{17}	
F6		
E6 _{N6}	E4 N41 E19 N20	
	E20 N21 N22	
E5 _{N5}	E_{19} H_{23} E_{22} H_{23}^{23} H_{23}^{23}	
	E23 H24 N33	
E4 _{N4}	E 38 _{N39}	
=2	E32 _{N32}	
E3 _{N3}	E37 _{N38}	
Y	E31 _{N30}	
E2 _{N2}	=36 _{N37}	
EX.	E30 _{2N30}	
	E 35 _{N36}	
×	E29 _{N29}	
z		
	E28 _{N28}	
	E33 N34	
	E27 _{N27}	
	E26 _{N26}	
	E25 125	
Define element(s) and parameters Eleme	nt(s): All All Clr Adv Status: Success: Deflection shown	
	30 # of pts 10 Animate (1) 2.160 ✓ Apply Cancel	-15
Solite Solite Solite		

◀☰

	SIRL\Desktop\Frame.mat		D. II						-	
<u>V</u> iew <u>G</u> eometry	y <u>P</u> roperties <u>C</u> ondit	ions <u>A</u> nalysis	<u>R</u> esults							
			Deflected Sha	ape: Elastic Critica	al Load, Mode	# 2, Applied Load	Ratio = 2.8058			
	E9 E8 18 E7 N7 E6 N6	E11	12 N13 12 E13	N14 E14 E14 E15	N16 E16 E40	N17 E17 E18 N41	N19 E19 E20	N21 N22		
	E5 _{N5}				E39	N40		E21 N22 E22	N23	
					3	140			E23 N24 N33	
	E4 _{N4}				E38	N39			Ĩ	
	F3				3				E32 _{N32}	
	E3 _{N3}				E37	N38			501	
	E2 _{N2}								E31 _{N31}	
					E36	N37			F30	
	A CON								E30 32	
					E35	<u>N</u> 36			E29 ₁₀₂₉	
	7	X			E24					
	-				E34	N35			E28 _{N28}	
					E33	104			Ť	
						54			E27 N27	
									E26 _{N26}	
									E25.ver	
									¥25.	
		_	_							
Define eleme Defl Line Type	ent(s) and parameters		ment(s):	All	10	All Clr		Status:	Success: Deflection shown	
Dell Line Type	Solid ~	Scale	30	# of pts	10	Animate	(2) 2.806	~	Apply Canc	ei

▲ MASTAN2: C:\Users\S	SIRL\Desktop\Fra	me_End.mat	[14:27]									_	
ile <u>V</u> iew <u>G</u> eometr		<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults									
				Deflecte	ed Shape: E	lastic Critical	Load, Mode # 3, App	lied Load Ratio	= 4.9364				
	Na												
	1E9	N10	N11										
	E ⁸ N8	E10	E11	N1P N1	3								
				E12 E1	3 N14								
	₽ <mark>№7</mark>				E14	E15							
	E N6						E17	N18 E18					
	ND P						E40 _{N41}	E18 E19	N20	N21			
	E ^s N5									E21 N22	N2B		
							E39 _{N40}			EZZ	E23 N2 E24		
							E38 _{N39}				4 2-		
	E ² N3											E32 N32	
	- N3						E32N38					F-31	
	FZ _{N2}											N31 N31	
							E36 _{N37}					=30 N30	
							E35 _{N36}						
	4	×										29 N29	
	z						E34 _{N35}					E)8	
												E.28 N28	
							E338N34					E2. N27	
												E26 N26	
												E25	
												23.	
Define elem	ient(s) and para	meters	E	ement(s):		All	All	Clr Ad	tv Stat	tus:	Success	s: Deflection shown	
Defl Line Type	Solid	~	Scale	30	# of	pts	10 Anim	nate (3	3) 4.936	~	Apply	Ci	ancel

MASTAN2: C:\Users\SSIRL\Desktop\Frame.mat [14:44]	- D	×
<u>File View G</u> eometry <u>P</u> roperties <u>C</u> onditions <u>A</u> nalysis <u>R</u> e	sults	
	Deflected Shape: Elastic Critical Load, Mode # 1, Applied Load Ratio = 2.1605	
Ng d		
F9 N10		
E8 _{N8} E10 N11		
	M13	
E7 _{N7}	E13 N14 N15 Luce	
<u>-</u> N7	E_{14} E_{15} N_{16} N_{17} N_{17} N_{17} N_{17} N_{17}	
F6		
E6 _{N6}	E4 N41 E19 N20	
	E20 N21 N22	
E5 _{N5}	E_{19} H_{23} E_{22} H_{23}^{23} H_{23}^{23}	
	E23 H24 N33	
E4 _{N4}	E 38 _{N39}	
=2	E32 _{N32}	
E3 _{N3}	E37 _{N38}	
Y	E31 _{N30}	
E2 _{N2}	=36 _{N37}	
EX.	E30 _{2N30}	
	E 35 _{N36}	
×	E29 _{N29}	
z		
	E28 _{N28}	
	E33 N34	
	E27 _{N27}	
	E26 _{N26}	
	E25 N25	
Define element(s) and parameters Eleme	nt(s): All All Clr Adv Status: Success: Deflection shown	
	30 # of pts 10 Animate (1) 2.160 ✓ Apply Cancel	-15
Solite Solite Solite		

◀☰

MASTAN2: C:\Users\SSIRL\Desktop\Frame_End.mat [14:27]	- D X
<u>Eile View Geometry Properties Conditions Analysis Results</u>	
Deflected	I Shape: Elastic Critical Load, Mode # 2, Applied Load Ratio = 2.8052
N10	
E13	P11 NID
FR	
F ⁶ N6	E40 N41 E19 120 Line
	E20 E21 N22
Ę₽ _{N5}	E39 N40 E23 N24 N24
EA N4	E23 E24 N33
571N4 1	E38 _{N39}
- Faller	
Ę ² N3	E37 _{N38}
F ² N2	N ³¹
Z−N2	E30 _{N37}
For the second s	
	E35 _{N36}
X	_{N29}
Z	E34 _{N35}
	E33 N34 E27
	E27 V27
	526
	E26 N26
	E25
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 Y Apply Cancel

 \blacksquare

MASTAN2: C:\Users\!	SIRL\Desktop\Fra	ame_End.mat	[14:27]										- 🗆	×
e <u>V</u> iew <u>G</u> eomet	ry <u>P</u> roperties	<u>C</u> onditions	<u>A</u> nalysis	<u>R</u> esults										
				Deflecte	d Shape: Elasti	ic Critical Lo	ad, Mode # 3, Ap	plied Load Ra	atio = 4.0396					
	N9													
	Æ9	€ N1D	,N11											
	E ⁸ N8	E10	E11	N12 N11										
				E12 N1 E13	3 N14	N15								
	F N7				E14	F15								
						2	216 N17	N1B						
	Ę¢ _{N6}						E49 _{N41}	E18 E	19 N20	Not				
	Ę5 _{N5}								E20	N21 N E21				
							E39 _{N40}			E	22 N23 E23	N24		
	FA N4						-					E24 N33		
							E38 _{N39}					E32N32		
	F ³ N3						E37 _{N38}							
	Y A											E.31 N31		
	Ę2 _{N2}						E36 N37					- 20		
	-AN											≡30 N30		
							E35 _{N36}					129 N29		
	7	X					23					• • •		
							E34 _{N35}					E.28 N28		
							E33 N34							
							4					E2 N27		
												E26 N26		
												E25		
												1 20		
												40		
Define elen	nent(s) and para	ameters	E	lement(s):		All	All	Clr	Adv	Status:	Su	ccess: Deflection s	shown	
Defl Line Type	Solid	_	Scale	30	# of pts		10 Ani		(3) 4.040	~	Apply		Cancel	

 \blacksquare

This completes the tutorial.

