

Tutorial 3

- LGS I Zee in Bending: Z 12 x 2.5 14g, $F_y = 50\text{ksi}$
- Objective
 - To model a typical Zee purlin or girt in bending and determine the elastic critical local buckling moment (M_{crl}) and elastic critical distortional buckling moment (M_{crd}).
- At the end of the tutorial you should be able to
 - enter material, nodes, elements, and lengths from scratch
 - OR use the C and Z template to enter a geometry
 - apply a reference load P , or M as desired
 - interpret a simple buckling curve
 - identify local and distortional buckling in a simple member
 - determine M_{crl} and M_{crd}

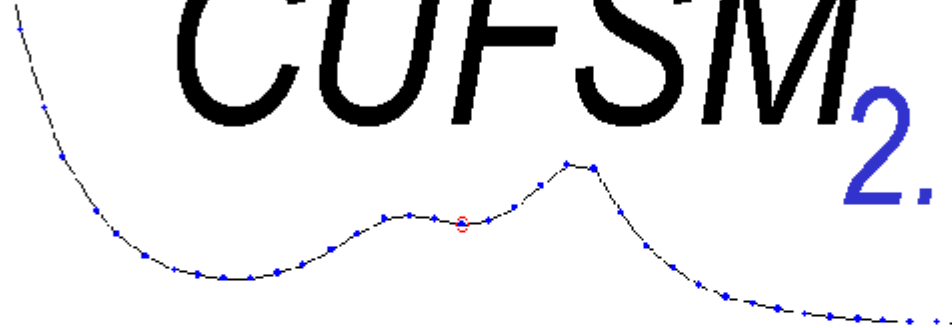


2. SELECT

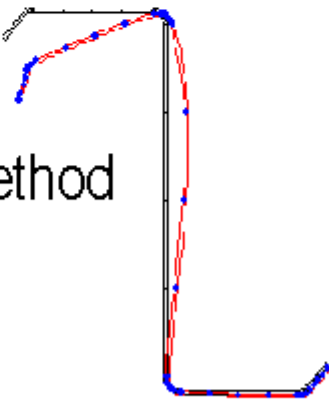
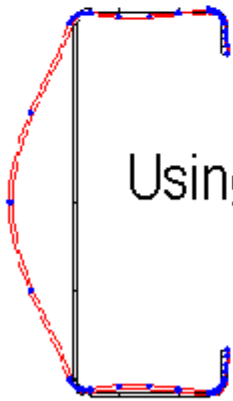


1. SELECT

CUFSM_{2.5}



Elastic Buckling Analysis of
Thin-Walled Members
Using the Classical Finite Strip Method



Ben Schafer
www.ce.jhu.edu/bschafer
version 2.5

Material Properties

mat# | Ex | Ey | vx | vy | Gxy

100 29500.00 29500.00 0.30 0.30 11346.15

Nodes

node# | x | z | xdof | zdof | ydof | qdof | stress

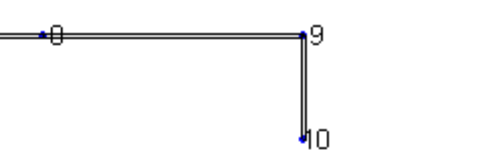
1	5.00	1.00	1	1	1	33.33
2	5.00	0.00	1	1	1	50.00
3	2.50	0.00	1	1	1	50.00
4	0.00	0.00	1	1	1	50.00
5	0.00	3.00	1	1	1	16.67
6	0.00	6.00	1	1	1	-16.67
7	0.00	9.00	1	1	1	-50.00
8	2.50	9.00	1	1	1	-50.00
9	5.00	9.00	1	1	1	-50.00
10	5.00	8.00	1	1	1	-33.33

Elements

elem# | nodei | nodej | thickness | mat#

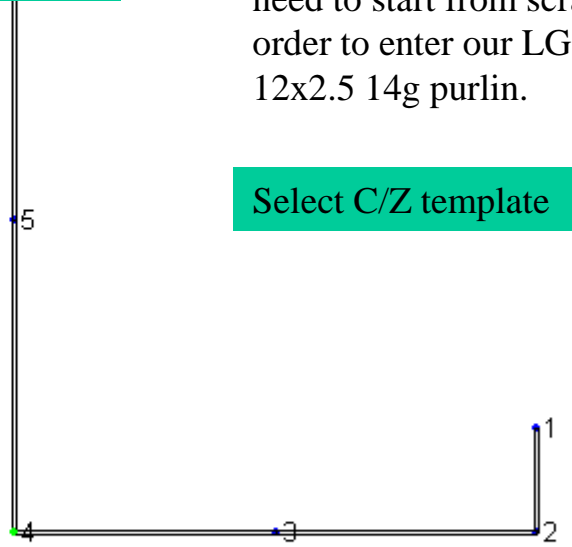
1	1	2	0.040000	100
2	2	3	0.040000	100
3	3	4	0.040000	100
4	4	5	0.040000	100
5	5	6	0.040000	100
6	6	7	0.040000	100
7	7	8	0.040000	100
8	8	9	0.040000	100
9	9	10	0.040000	100

Select
 Note, we could enter the geometry node by node as in Tutorial #2, but in this case, let's use the template instead.



This screen shows the default section that appears when you enter the Input screen for the first time. In our case we do not want to use this section so we need to start from scratch in order to enter our LGSIZ 12x2.5 14g purlin.

Select C/Z template



Lengths

1.0 2.0 3.0 4.0 5.0 6.0 7.0 8.0 9.0 10.0 11.0 12.0 13.0 14.0 15.0 20.0 30.0 40.0 50.0 60.0 70.0 80.0 90.0 100.0 200.0 300.0 400.0 500.0 600.0 700.0 800.0 900.0 1000.0

Springs

node# | DOF(x=1,z=2,y=3,theta=4) | kspring | kflag

0

Constraints

node#e | DOFe | coeff. | node#k | DOFk

0

This is the default template that comes up when you select the template button. Note that all dimensions are centerline dimensions - e.g., h is the flat distance, not the out-to-out distance as is typically listed in product catalogs, etc.

Enter in all the appropriate dimensions and select Submit to input.

Update Plot				<input checked="" type="radio"/> C <input type="radio"/> Z	
material (steel) units:				<input checked="" type="radio"/> kip&in. <input type="radio"/> N&mm	
h	6	t	0.06		
b1	2	b2	2		
d1	0.5	d2	0.5		
theta1	90	theta2	90		
r1	0.25	r3	0.25		
r2	0.25	r4	0.25		
Submit to Input		Close (don't submit)			
NOTE: ALL DIMENSIONS ARE CENTERLINE DIMENSIONS! (NOT OUT-TO-OUT)					

The centerline dimensions for an LGSI Z 12 x 2.5 14g member are shown to the right. Enter in these dimensions and then press Update Plot. When complete select Submit to Input.

Note, the material is assumed to be steel, but two units systems are supported. Geometry other than the typical Cee or Zee can be entered.

The template automatically selects an adequate number of elements.

The template automatically selects lengths to be analyzed as well.

Update Plot				<input type="radio"/> C <input checked="" type="radio"/> Z	
material (steel) units:				<input checked="" type="radio"/> kip&in. <input type="radio"/> N&mm	
h	11.344	t	0.07		
b1	1.6	b2	1.85		
d1	.697	d2	0.697		
theta1	50	theta2	50		
r1	0.258	r3	0.258		
r2	0.258	r4	0.258		
Submit to Input		Close (don't submit)			
NOTE: ALL DIMENSIONS ARE CENTERLINE DIMENSIONS! (NOT OUT-TO-OUT)					



Material Properties

mat# | Ex | Ey | vx | vy | Gxy

100 29500.00 29500.00 0.30 0.30 11346.15

C/Z template
Double Elem.
help

Nodes

node# | x | z | xdof | zdof | ydof | qdof | stress

1	2.50	0.63	1	1	1	1.00
2	2.39	0.49	1	1	1	1.00
3	2.28	0.36	1	1	1	1.00
4	2.17	0.23	1	1	1	1.00
5	2.06	0.09	1	1	1	1.00
6	2.02	0.05	1	1	1	1.00
7	1.97	0.02	1	1	1	1.00
8	1.91	0.01	1	1	1	1.00
9	1.86	0.00	1	1	1	1.00
10	1.46	0.00	1	1	1	1.00
11	1.06	0.00	1	1	1	1.00

Update Plot

Plot Options:

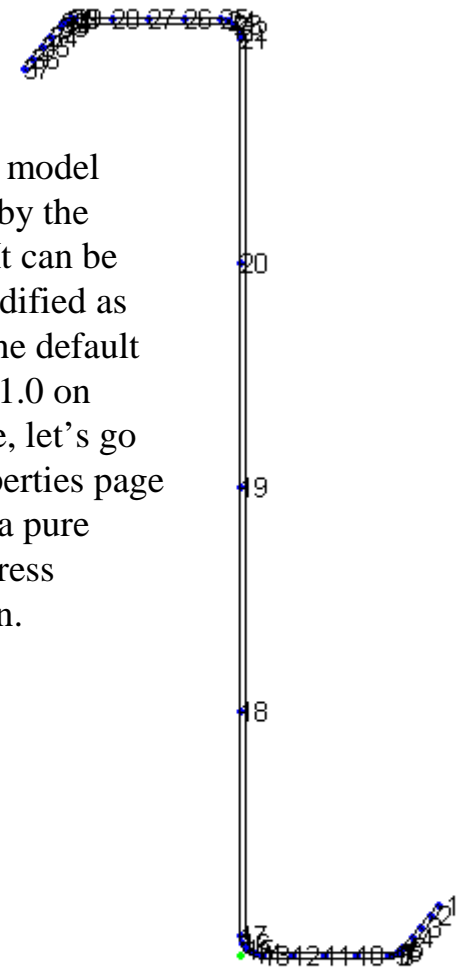
- node #
- element #
- material #
- stress mag.
- stress dist.
- coordinate:
- constraints
- springs
- origin

Elements

elem# | nodei | nodej | thickness | mat#

1	1	2	0.070000	100
2	2	3	0.070000	100
3	3	4	0.070000	100
4	4	5	0.070000	100
5	5	6	0.070000	100
6	6	7	0.070000	100
7	7	8	0.070000	100
8	8	9	0.070000	100
9	9	10	0.070000	100
10	10	11	0.070000	100

This is the model generated by the template. It can be still be modified as desired. The default loading is 1.0 on every node, let's go to the properties page and apply a pure bending stress distribution.



Lengths

1.1 1.4 1.7 2.0 2.4 2.9 3.5 4.2 5.1 6.2 7.4 9.0 10.8 13.1 15.8 19.0 23.0 27.7 33.4 40.3 48.7 58.8 70.9 85.6 103.3 124.6 150.4 181.5 219.0 264.3 319.0 384.9 464.5 560.6 676.5 8

Springs

node# | DOF(x=1,z=2,y=3,theta=4) | kspring | kflag

0

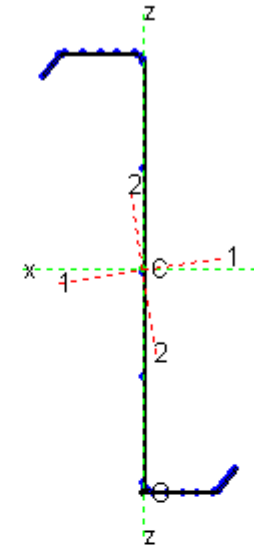
Constraints

node#e | DOFe | coeff. | node#k | DOFk

0

Calculated Section Properties

A = 1.2211	
xcg = -0.04168	zcg = 6.015
Ixx = 23.0469	Izz = 1.0744
Ixz = -3.3384	theta = 8.4511
I11 = 23.5429	I22 = 0.57838



Note that the principal coordinate system is not in line with the global x,z coordinate system, as expected.

Calculation of Loads and Moments for Generation of Stress on Member

Moment calculations should consider

Unsymmetric or Restrained Bending

fy = Calculate P and M ?

Loads and Moments

P =

Mxx =

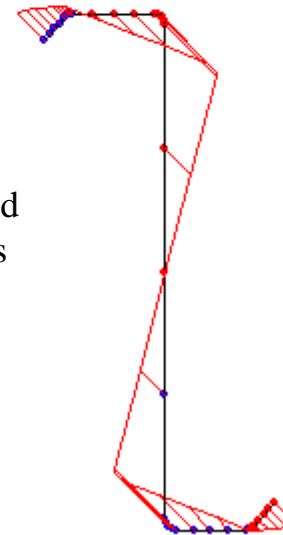
Mzz =

M11 =

M22 =

Generate Stress using checked P and M ?

Enter a yield stress, calculate P and M, uncheck P and examine the generated stress distribution. As shown to the right, it reflects unsymmetric bending.



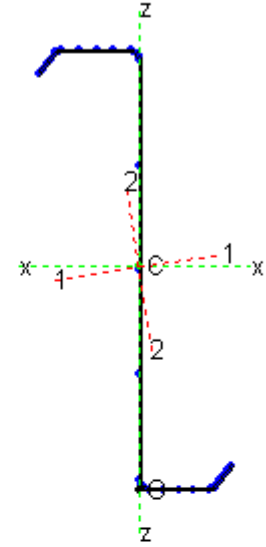
Switch to restrained bending and re-calculate the stress distribution.

Load Save Input Properties Analyze Post Compare ? Print Copy Reset EXIT

Calculated Section Properties

A = 1.2211
 xcg = -0.04168 zcg = 6.015
 Ixx = 23.0469 Izz = 1.0744
 Ixz = -3.3384 theta = 8.4511
 I11 = 23.5429 I22 = 0.57838

select 1, use the robust solver, analysis will proceed, then select 2



This is the yield moment, M_y , the buckling load factor results will be in terms of $M_y=192$ kip-in.

Calculation of Loads and Moments for Generation of Stress on Member

Moment calculations should consider

Unsymmetric or Restrained Bending

fy = 50 Calculate P and M ?

Loads and Moments

P = 61.0563

Mxx = 191.5794

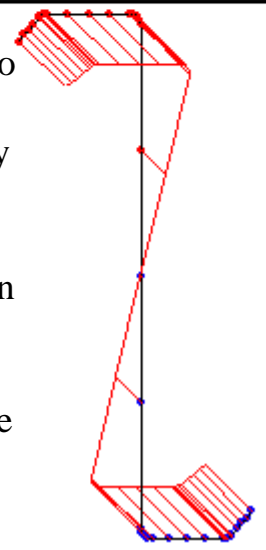
Mzz = 19.8349

M11 = 199.3363

M22 = 8.3936

Generate Stress using checked P and M ?

The stress distribution to the right would be applicable for a laterally braced beam, and is typically assumed in cold-formed steel design codes. Note that the flanges are different sizes and in this case the wider flange has been placed in compression.



Scale = 1 Max Comp. = 48.5872 Min Tens. = -50

half-wavelength = 1.1 load factor = 3.7656 mode = 1

Plot Mode ?

2D 3D Undef.

half-wavelength

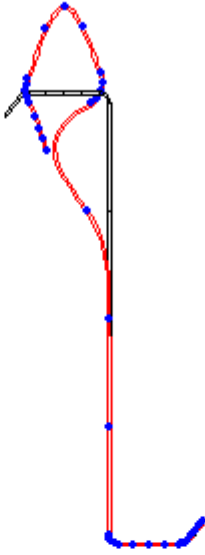
<-- 1.1 --> ?

Scale 1

mode <-- 1 --> ?

Stress Distribution ?

This screen shows the post-processing page that will come up when you select Post. Note, the two minima in the plot: local and distortional buckling.



Clean up the curve and change the half-wavelength to show local buckling.

Plot Curve ?

Min. Log X

xmin 0

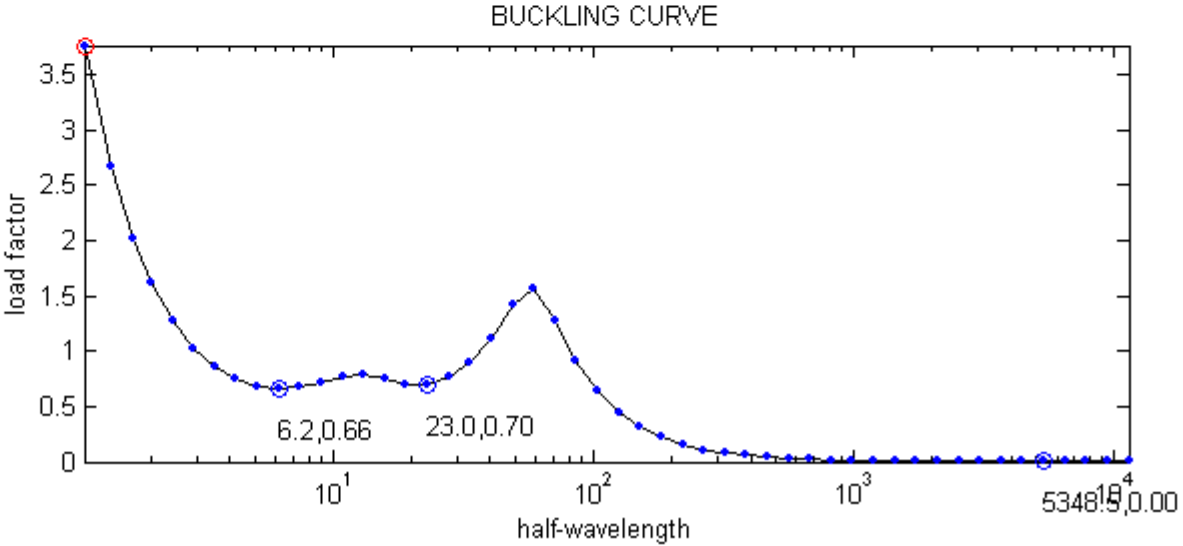
xmax 11344

ymin 0

ymax 3.7656

modes <-- 1 --> ?

Text Output ?



half-wavelength = 6.2 load factor = 0.65799 mode = 1

Plot Mode ?

2D 3D Undef.

half-wavelength

<-- 6.2 --> ?

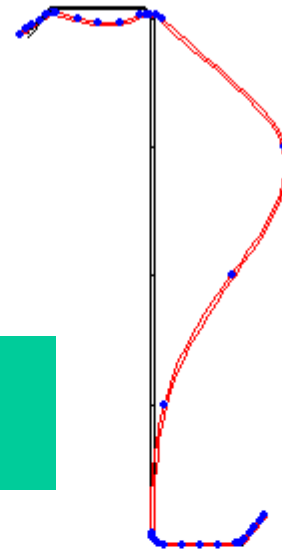
Scale 1

mode <-- 1 --> ?

Stress Distribution ?

Local buckling results are shown here. $M_{cr1}=0.66M_y$ and the buckling mode shape is as given to the right.

Change the half-wavelength to examine distortional buckling.



Plot Curve ?

Min. Log X

xmin 0

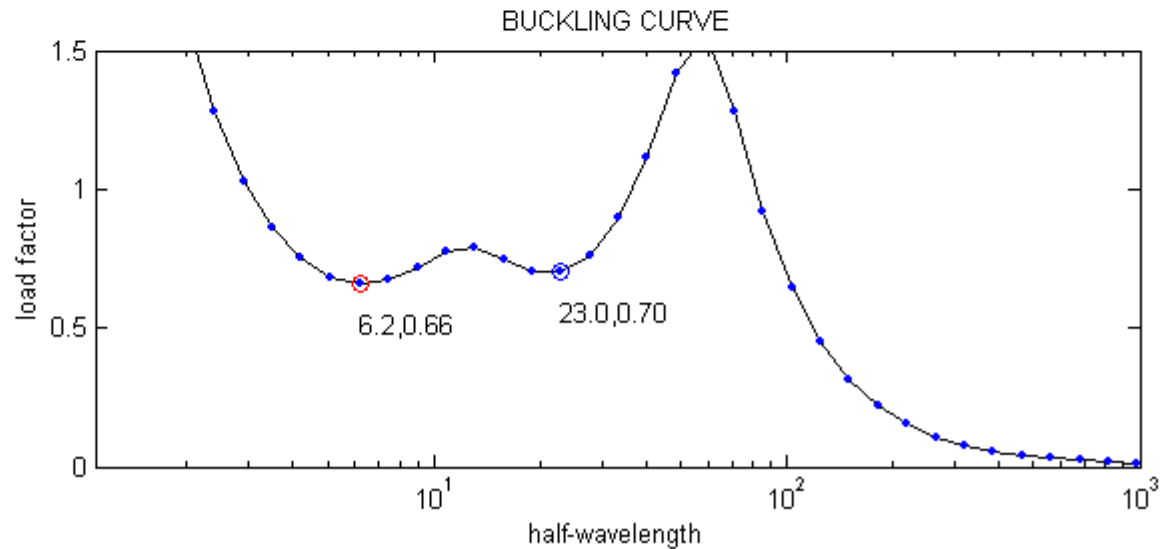
xmax 1000

ymin 0

ymax 1.5

modes <-- 1 --> ?

Text Output ?



half-wavelength = 23 load factor = 0.70106 mode = 1

Plot Mode ?

2D 3D Undef.

half-wavelength

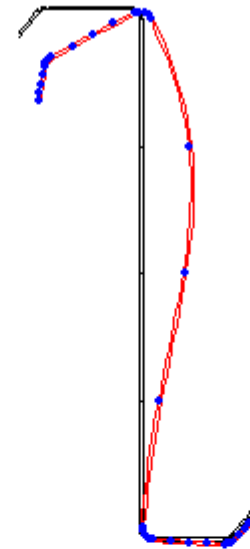
<-- 23 --> ?

Scale -0.5

mode <-- 1 --> ?

Stress Distribution ?

Distortional buckling results are shown here. $M_{crd} = 0.70M_y$ and the buckling mode shape is as given to the right.



Plot Curve ?

Min. Log X

xmin 0

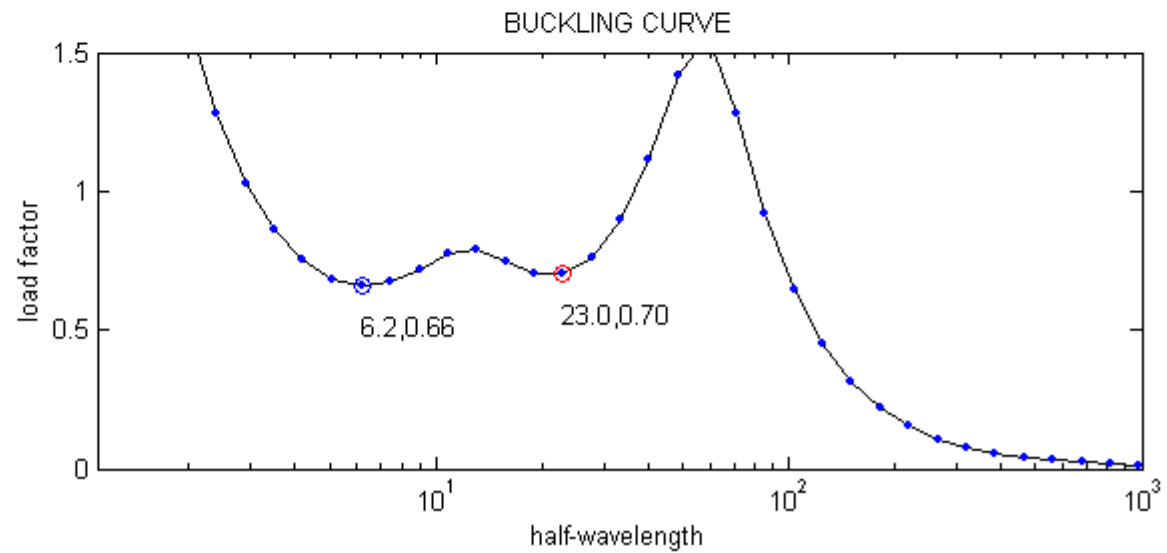
xmax 1000

ymin 0

ymax 1.5

modes <-- 1 --> ?

Text Output ?



Tutorial 3

- LGS I Zee in Bending: Z 12 x 2.5 14g $F_y = 50\text{ksi}$
- Objective
 - To model a typical Zee purlin or girt in bending and determine the elastic critical local buckling moment (M_{crl}) and elastic critical distortional buckling moment (M_{crd}).
- At the end of the tutorial you should be able to
 - enter material, nodes, elements, and lengths from scratch
 - OR use the C and Z template to enter a geometry
 - apply a reference load P , or M as desired
 - interpret a simple buckling curve
 - identify local and distortional buckling in a simple member
 - determine M_{crl} and M_{crd}