

# Tutorial 1

- Default Cee section in bending
- Objective
  - To introduce CUFSM and the finite strip method and gain a rudimentary understanding of how to perform an analysis and interpret the results.
- At the end of the tutorial you should be able to
  - enter simple geometry
  - enter loads (stresses)
  - perform a finite strip analysis
  - manipulate the post-processor
  - identify and understand different buckling modes

Load

Save

Input

Properties

Analyze

Post

Compare

?

Print

Copy

Reset

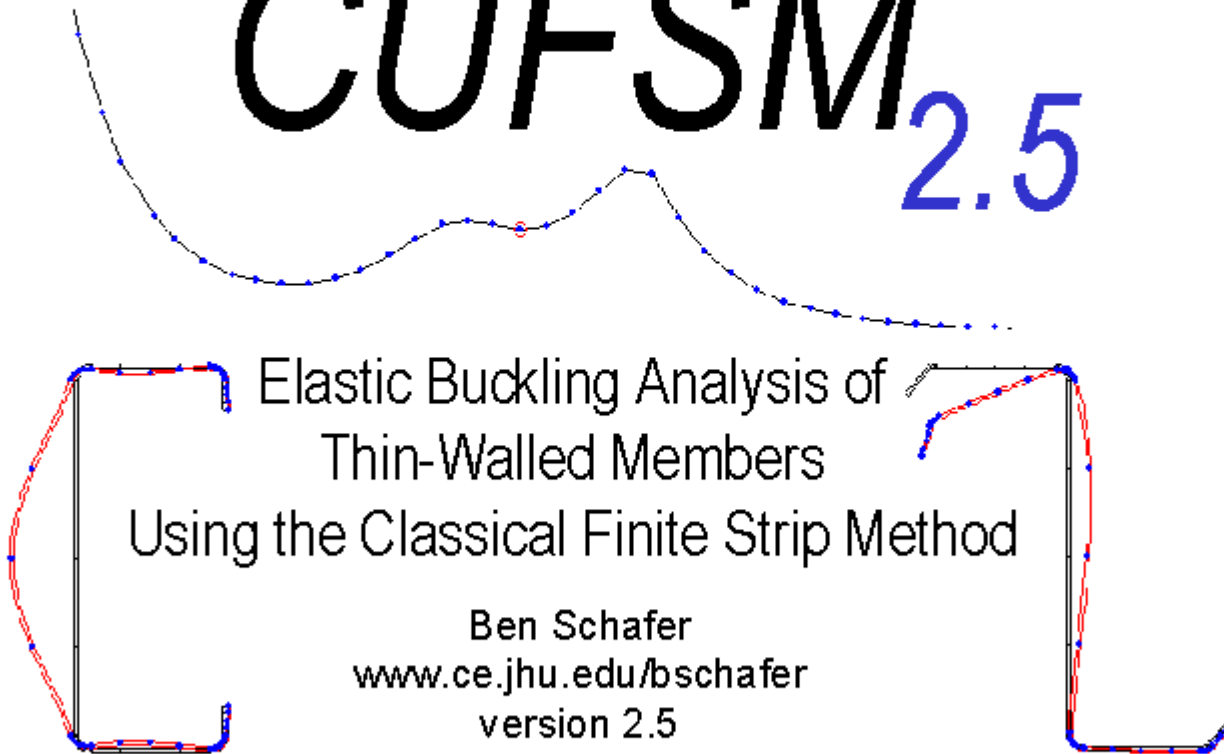
EXIT

SELECT

# CUFSM<sub>2.5</sub>

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

Ben Schafer  
[www.ce.jhu.edu/bschafer](http://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

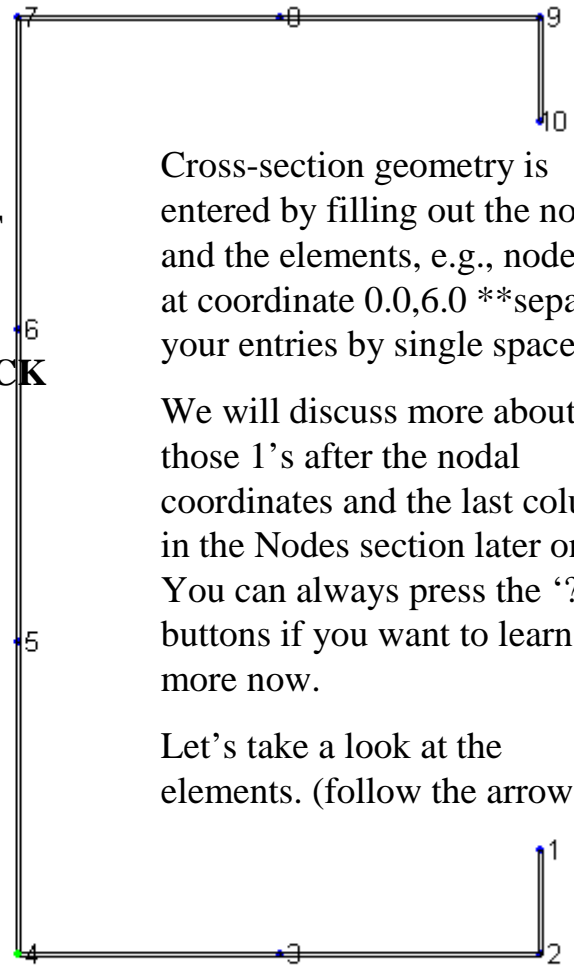
1	5.00	1.00	1	1	1	33.33
2	5.00	0.00	1	1	1	50.00
3	2.50	0.00	1	1	1	50.00
4	0.00	0.00	1	1	1	50.00
5	0.00	3.00	1	1	1	16.67
6	0.00	6.00	1	1	1	-16.67
7	0.00	9.00	1	1	1	-50.00
8	2.50	9.00	1	1	1	-50.00
9	5.00	9.00	1	1	1	-50.00
10	5.00	8.00	1	1	1	-33.33

**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

- C/Z template
- Double Elem.
- help
- Update Plot **3. SELECT**
- Plot Options:
- node # **1. UNCHECK**
- element # **2. CHECK**
- material #
- stress mag.
- stress dist.
- coordinate:
- constraints
- springs
- origin



Cross-section geometry is entered by filling out the nodes and the elements, e.g., node 6 is at coordinate 0.0,6.0 \*\*separate your entries by single spaces\*\*

We will discuss more about all those 1's after the nodal coordinates and the last column in the Nodes section later on. You can always press the '?' buttons if you want to learn more now.

Let's take a look at the elements. (follow the arrows)

**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

### 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	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

Update Plot **3. SELECT**

Plot Options:

node #

element # **1. UNCHECK**

material #

stress mag **2. CHECK**

stress dist.

coordinate:

constraints

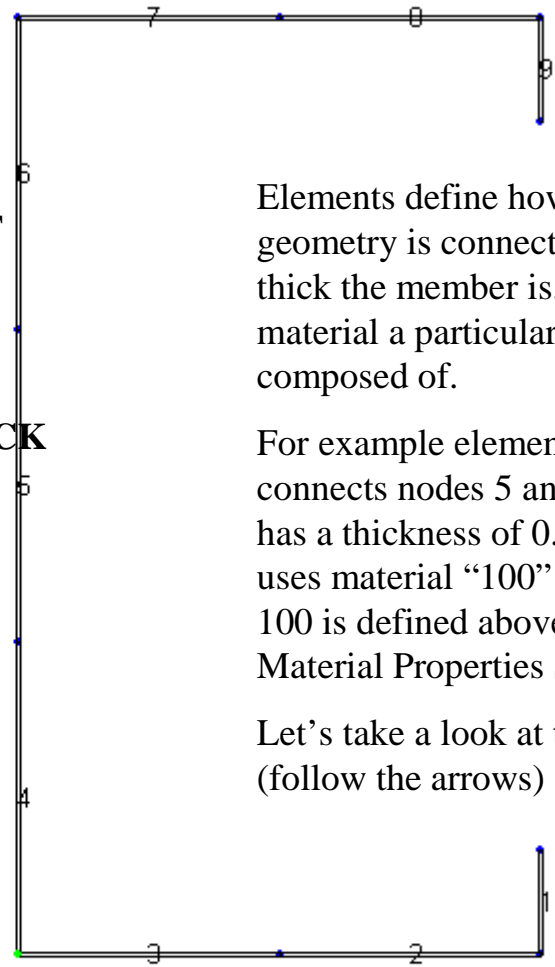
springs

origin

### 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



Elements define how the geometry is connected, how thick the member is, and what material a particular element is composed of.

For example element 5, connects nodes 5 and 6 together, has a thickness of 0.04 in. and uses material “100” - material 100 is defined above in the Material Properties Section.

Let’s take a look at the loading. (follow the arrows)

### 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
---

### 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	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

Update Plot **3. SELECT**

Plot Options:

node #

element #

material #

stress mag. **1. UNCHECK**

stress dist. **2. CHECK**

coordinate:

constraints

springs

origin

### 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

### 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

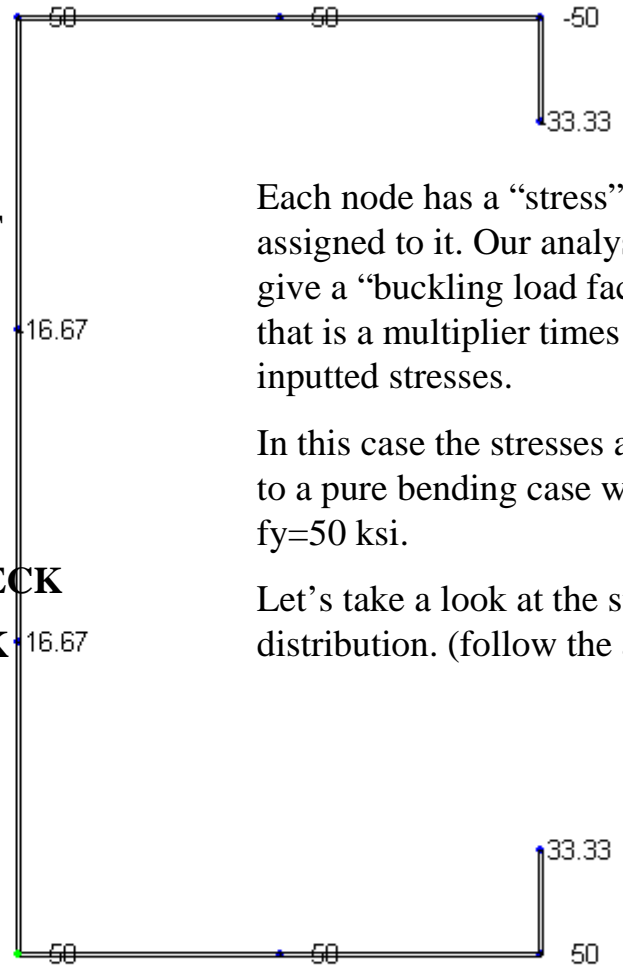
node# | DOF(x=1,z=2,y=3,theta=4) | kspring | kflag

0

### Constraints

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

0



Each node has a “stress” assigned to it. Our analysis will give a “buckling load factor” that is a multiplier times the inputted stresses.

In this case the stresses amount to a pure bending case with  $f_y=50$  ksi.

Let’s take a look at the stress distribution. (follow the arrows)

**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

C/Z temp

Double

help

Update

Plot Options:

node #

element #

material #

stress mag.

stress dist.

coordinate:

constraints

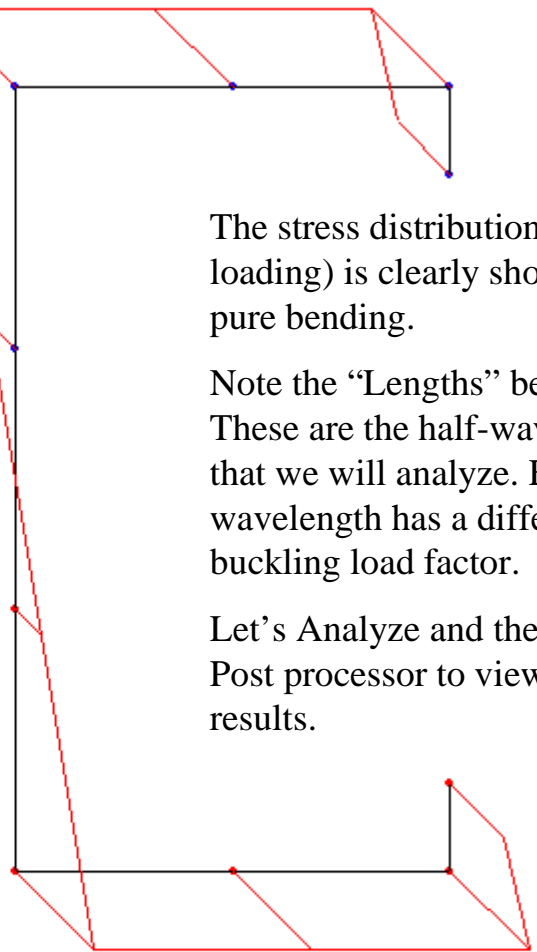
springs

origin

1

2

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



The stress distribution (the loading) is clearly shown to be pure bending.

Note the “Lengths” below. These are the half-wavelengths that we will analyze. Each half-wavelength has a different buckling load factor.

Let’s Analyze and then go to the Post processor to view the results.

**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** ?

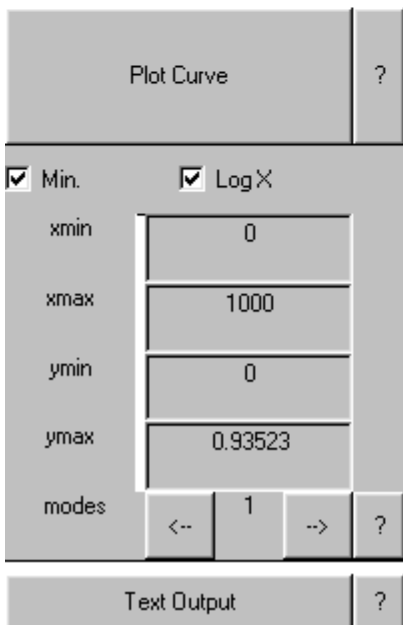
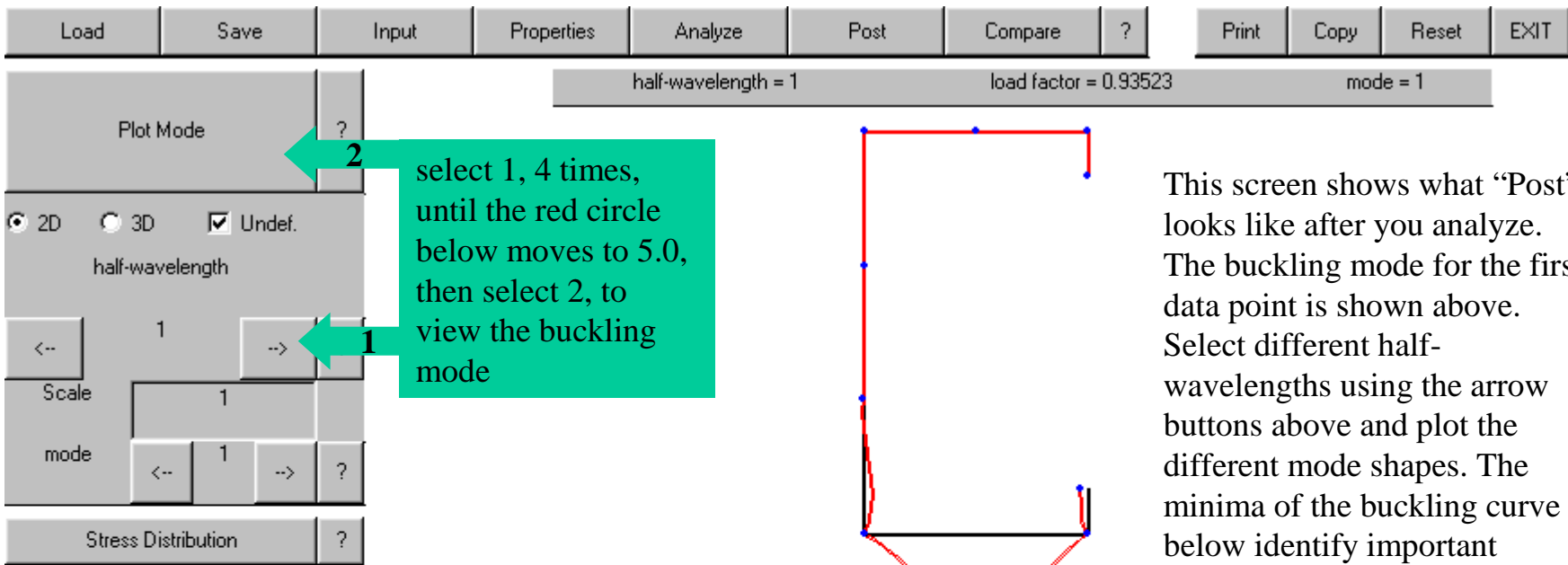
node# | DOF(x=1,z=2,y=3,theta=4) | kspring | kflag

0
---

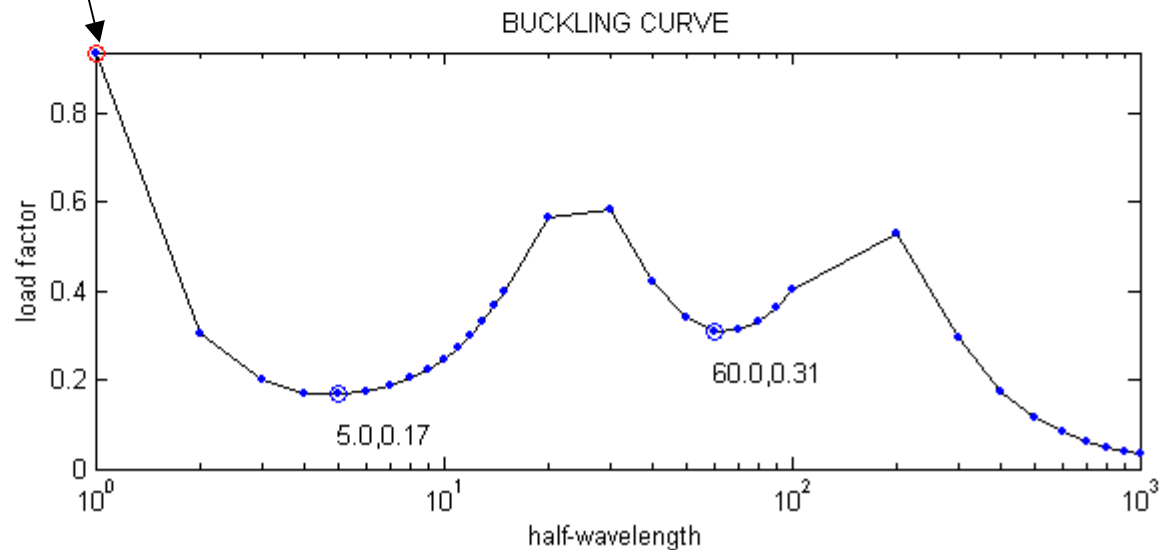
**Constraints** ?

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

0
---



“red circle” = where you are at



half-wavelength = 5 load factor = 0.1669 mode = 1

Plot Mode ?

2D 3D  Undef.

half-wavelength

5

Scale 1

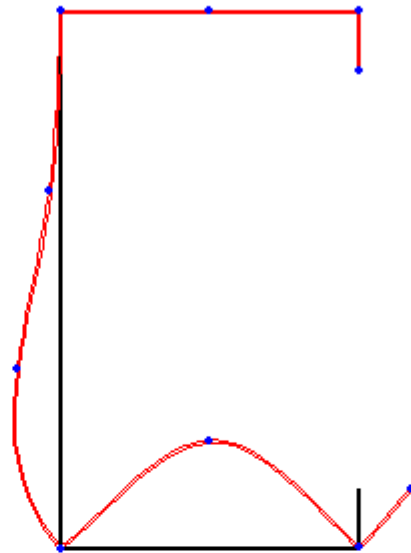
mode 1

Stress Distribution ?

2 select 1, until the red circle moves to 60.0, then select 2, to view the distortional buckling mode



note, the scale of the buckling mode is arbitrary! 1 or -1 are equally valid, as is 0.5 or 4, or any other convenient multiplier.



The local buckling mode is shown to the left. The mode repeats at a half-wavelength of 5.0 in. (See summary above plot and numbers below in the buckling curve). The buckling load factor is 0.17 for local buckling. This means elastic critical local buckling occurs at 0.17 times the stress distribution entered - remember the stress magnitudes from before?

Plot Curve ?

Min.  Log X

xmin 0

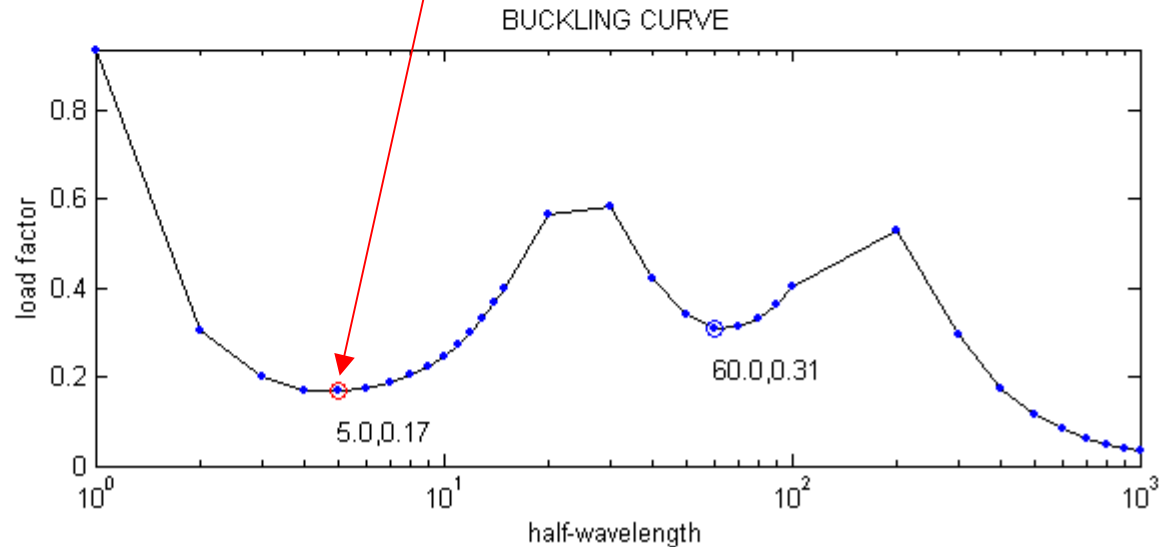
xmax 1000

ymin 0

ymax 0.93523

modes 1

Text Output ?



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

half-wavelength = 60 load factor = 0.30915 mode = 1

Plot Mode ?

3

1 2

2D 3D Undef.

60

Scale 1

mode 1

Stress Distribution ?

follow 1,2,3 to take a look at the buckling modes in 3D

The distortional buckling mode is shown to the left. The mode repeats at a half-wavelength of 60.0 in. The buckling load factor is 0.31. This means elastic critical distortional buckling occurs at 0.31 times the stress distribution entered.

Plot Curve ?

Min. Log X

xmin 0

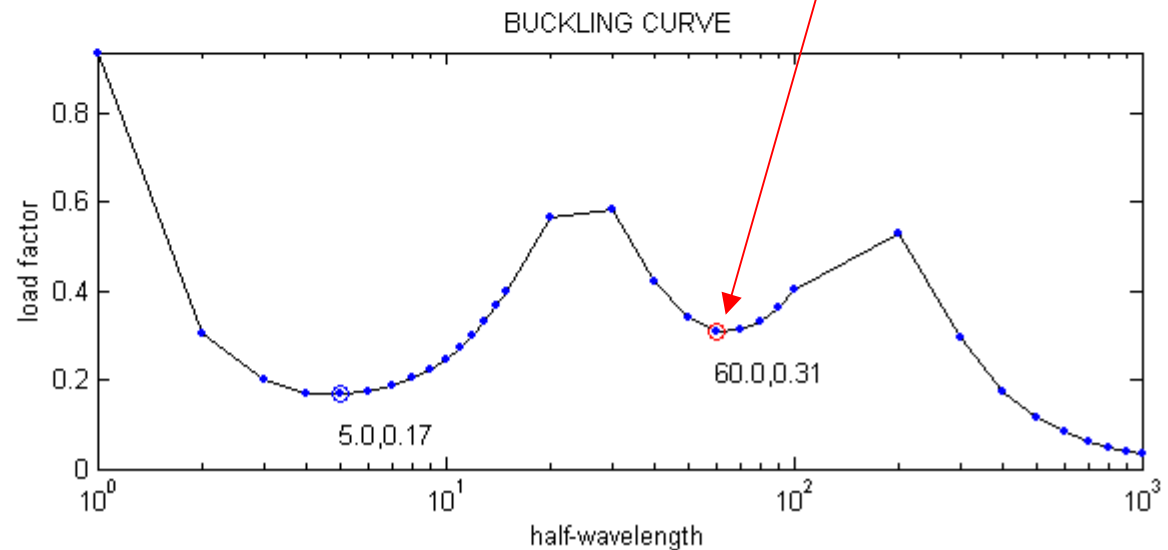
xmax 1000

ymin 0

ymax 0.93523

modes 1

Text Output ?



half-wavelength = 60 load factor = 0.30915 mode = 1

Plot Mode ?

2D  3D  Undef.

half-wavelength

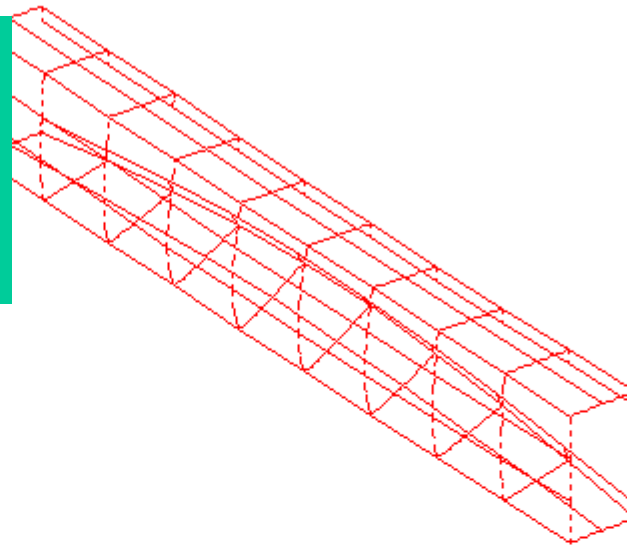
<-- 60 -->

Scale 1

mode <-- 1 --> ?

Stress Distribution ?

2 this is distortional buckling, select 1 and go back to 5.0, then select 2, let's look at local buckling first



Plot Curve ?

Min.  Log X

xmin 0

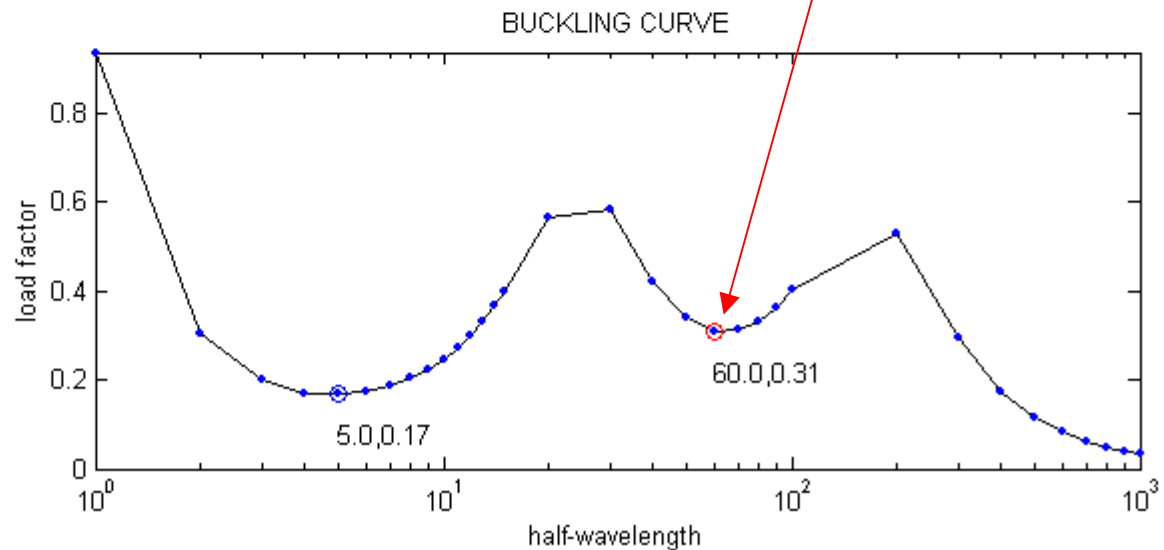
xmax 1000

ymin 0

ymax 0.93523

modes <-- 1 --> ?

Text Output ?



half-wavelength = 5 load factor = 0.1669 mode = 1

Plot Mode ?

2D  3D  Undef.

half-wavelength

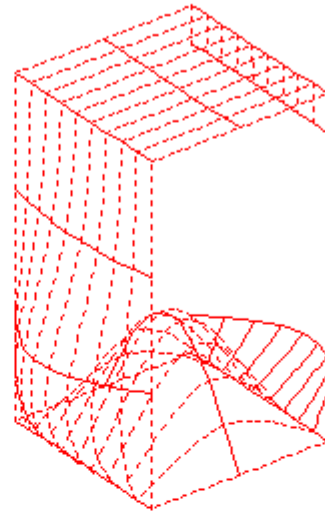
<-- 5 --> ?

Scale 1

mode <-- 1 --> ?

Stress Distribution ?

The lengths that are analyzed in the plot below are selected by the user. Let's add some points in the circled sections below to smooth out our plot. - Select input -



The 3D plot to the left shows local buckling at a half-wavelength of 5.0 in. Note, the 2D plot presented earlier shows the maximum cross-section deflected shape only.

CUFSM finite strip analysis assumes a single half sine wave in the longitudinal direction (as shown). Return to a half-wavelength of 60 in. to see the distortional buckling mode.

Plot Curve ?

Min.  Log X

xmin 0

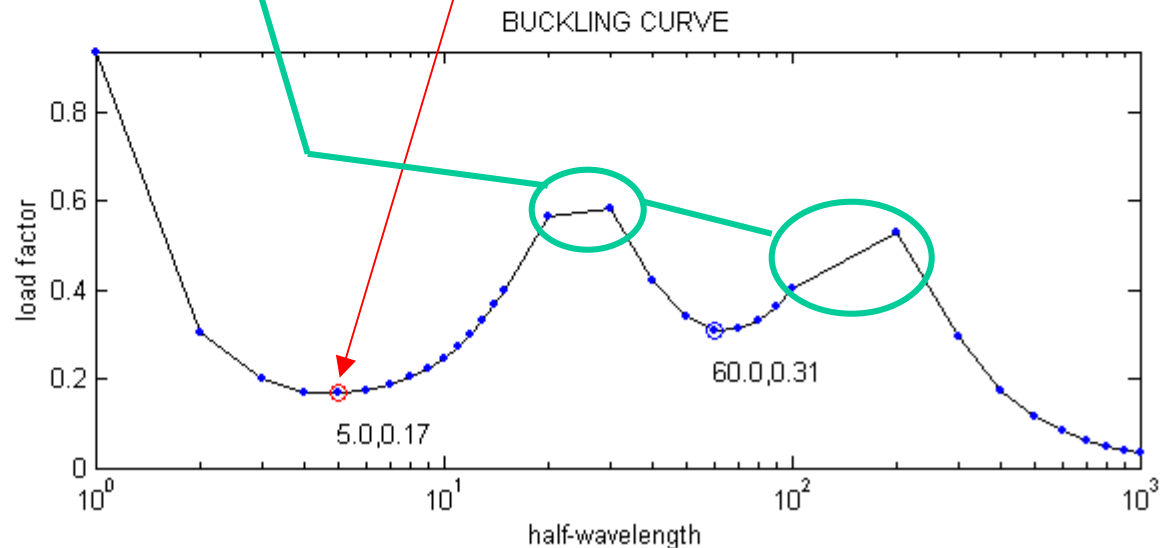
xmax 1000

ymin 0

ymax 0.93523

modes <-- 1 --> ?

Text Output ?



**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

C/Z temp

Double

help

Update

Plot Options:

node #

element #

material #

stress mag.

stress dist.

coordinate:

constraints

springs

origin

**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 22 24 26 28 30.0 40.0 50.0 60.0 70.0 80.0 90.0 100.0 120 140 160 180 200.0 300.0 400.0 500.0 600.0 700.

**Springs** ?

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

0

**Constraints** ?

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

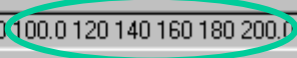
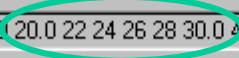
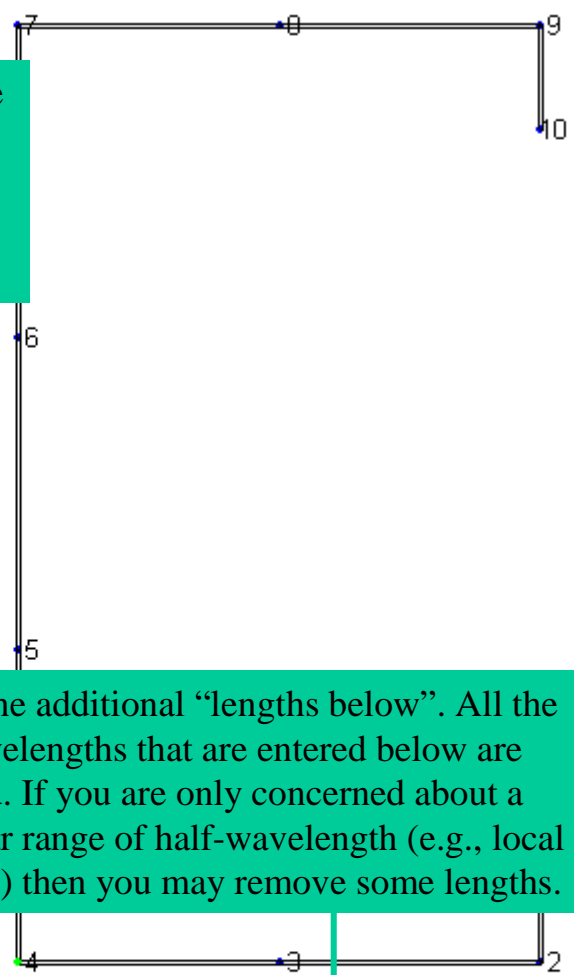
0

2

3

select 2, use the robust solver, analysis will proceed, then select 3

1. Add the additional "lengths below". All the half-wavelengths that are entered below are analyzed. If you are only concerned about a particular range of half-wavelength (e.g., local buckling) then you may remove some lengths.



Plot Mode ?

2D  3D  Undef.

half-wavelength

<-- 5 --> ?

Scale 1

mode <-- 1 --> ?

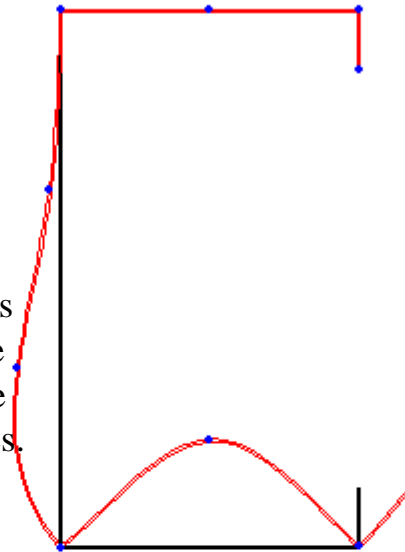
Stress Distribution ?

half-wavelength = 5 load factor = 0.1669 mode = 1

The analysis results for all of the lengths are shown below.

**Q:** Why is the load factor at a half-wavelength of 10 in. greater than at 5.0 in?

**A:** Because the analysis always assumes buckling is in a single half sine wave, this may not be how the actual member buckles.



**Q:** What would happen if I assumed the member always buckled in two half sine waves (i.e a full sine wave) instead of one?

**A:** You would see the same buckling curve, but it would be translated to the right. For example - local buckling has a minimum at 5.0 in. for one half sine wave, and will have a minimum at  $(2)(5.0) = 10.0$  in. for two half sine waves.

Plot Curve ?

Min.  Log X

xmin 0

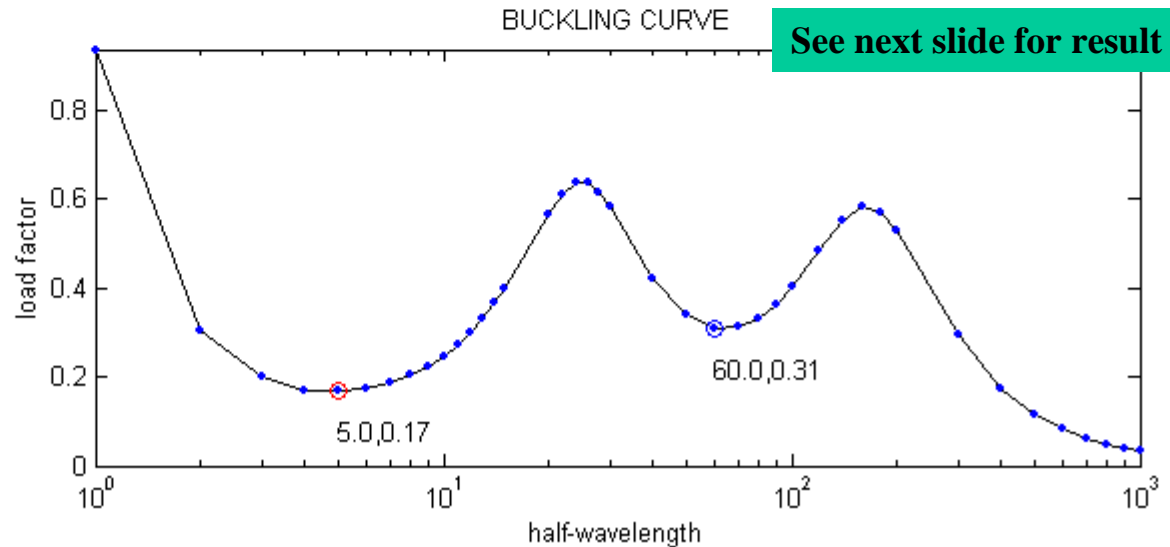
xmax 1000

ymin 0

ymax 0.93523

modes <-- 1 --> ?

Text Output ?



Plot Mode ?

2D  3D  Undef.

half-wavelength

<-- 10 --> ?

Scale 1 5

mode <-- 1 --> ?

file <-- 2 --> ?

loaded files:  
 1 = defaultC.mat  
 2 = defaultc2.mat  
 3 = defaultc3.mat

Plot Curve ?

Min.  Log X

xmin 2

xmax 1000

ymin 0

ymax 0.8

modes <-- 1 --> ?

plot files = 1 2 ?

(1)

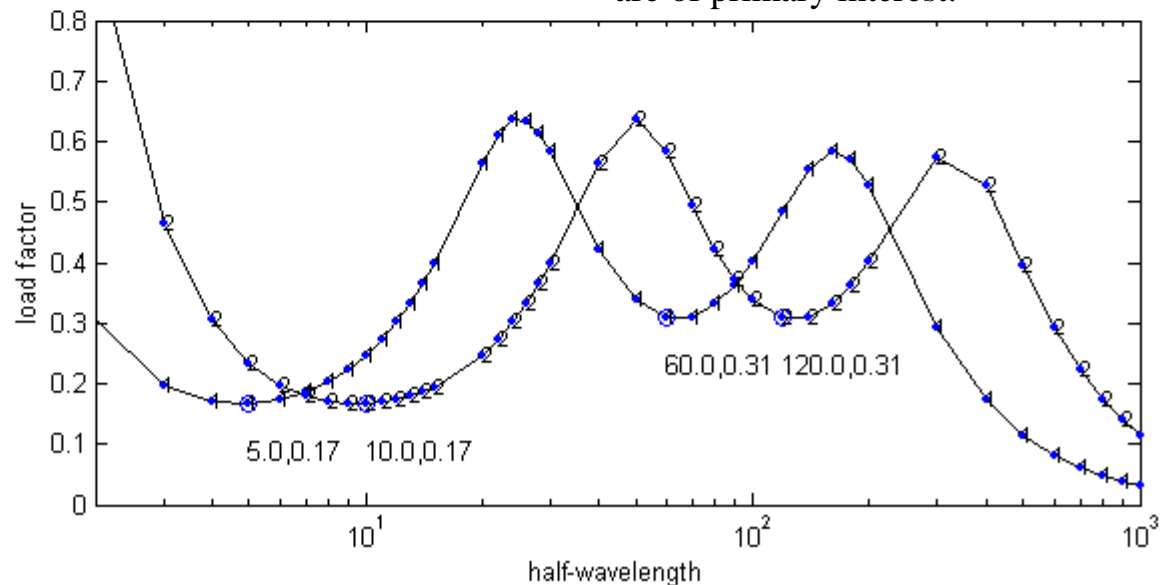
This is the same buckling curve we have been looking at for the Cee in bending. Note the familiar minimums: local at 5.0 in., and distortional at 60.0 in.

This curve, like all other finite strip analysis generated by CUFSM assumes a single half sine wave in the longitudinal direction.

(2)

This is a specially constructed curve, in which two half sine waves have been assumed throughout the analysis of the Cee in bending. Note now that local occurs at  $(2)(5.0) = 10.0$  in., and distortional at  $(2)(60.0) = 120.0$  in.

In a real structure the buckling mode is free to form any number of half sine waves - therefore only the minimums of the first curve are of primary interest.



Q: What would 3 half sine waves look like? 4?

# Tutorial 1: Conclusion

- Default Cee section in bending
- Objective
  - To introduce CUFSM and the finite strip method and gain a rudimentary understanding of how to perform an analysis and interpret results.
- At the end of the tutorial you should be able to
  - enter geometry
  - enter loads (stresses)
  - perform a finite strip analysis
  - manipulate the post-processor
  - identify and understand different buckling modes