

2011
Student Competition on Cold-Formed Steel Design
(1st CFS Design Student Competition)

Design Example

Goal

To design a optimal cold-formed steel cross-section shape. The optimal shape shall yield an as high as possible critical elastic buckling load for wavelengths equal to or less than 12 inches when uniform compression stresses are applied.

Given/Assumptions

- (1) The cross-section shape shall be a open section.
- (2) The total length of the cross section shall be 8.727 inches.
- (3) The thickness of the steel is 0.0451 inches.
- (4) Cold-formed steel properties: Elastic modulus = 29500 ksi, Possion's ratio = 0.3.
- (5) Shape corners (zero radius) are assumed.

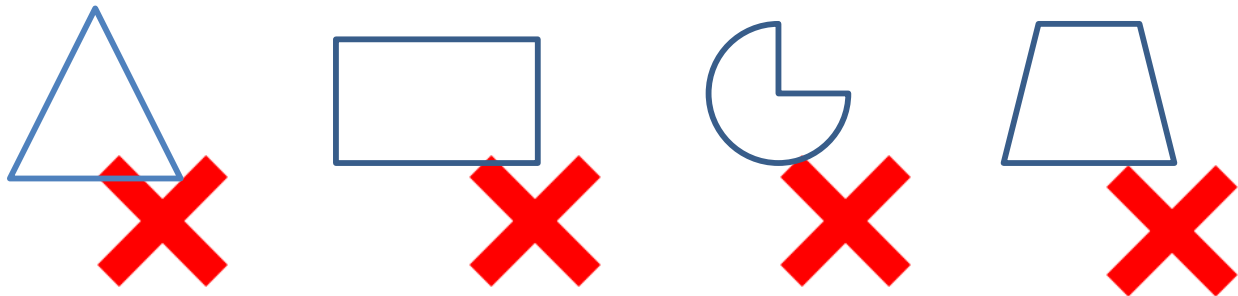
Step 1 – Design the cross-section shape

Basically you are given an 8.727 inches wide flat steel sheet, and you want to fold or bend the sheet to various shapes and pick the best shape which gives the highest possible elastic buckling load for any half-wave lengths up to 12 inches long.

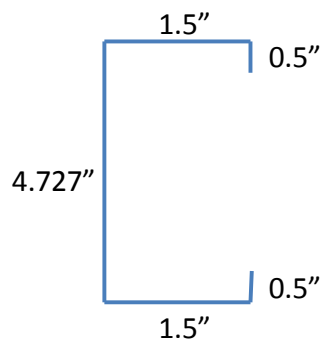
The shape has to be an open section. For example:



The shape cannot be closed sections.



We take a C section shape as an example.



Make sure the total length of the section = 8.727 inches.

Step 2 – Create the cross section model in CUFSM

2.1 Download and open the CUFSM software

Click “Input” to enter the geometric input modulus.

Note: the default units in CUFSM are inch, kip, and ksi.

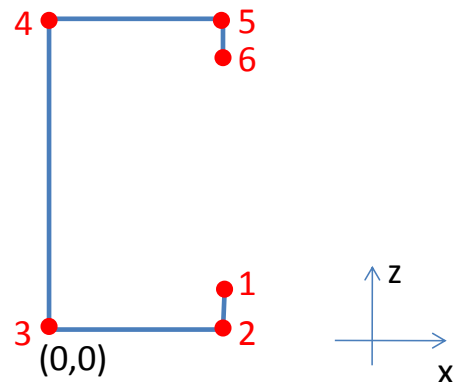
2.2 Define the nodes and update the “Nodes” entry

We need to assign nodes to all corners, intersections, and end points. For the C section, we can define 6 nodes and assume the origin of the Z-X coordinate system is at node #3.

According to the required Nodes entry format in CUFSM, the following node entry can be established.

```

1 1.5 0.5 1 1 1 1 1
2 1.5 0 1 1 1 1 1
3 0 0 1 1 1 1 1
4 0 4.727 1 1 1 1 1
5 1.5 4.727 1 1 1 1 1
6 1.5 4.227 1 1 1 1 1
    
```



(hint: you only need to specify the node #, x and z coordinates, leave 1 for the rest entries)

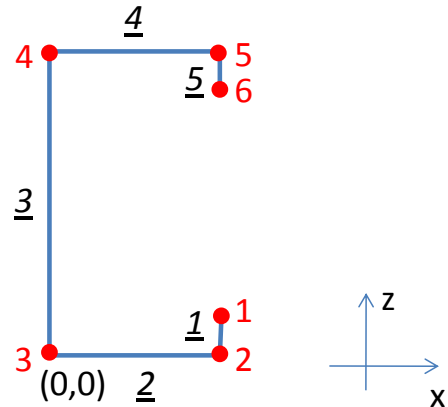
2.3 Define the elements and update the “Elements” entry

We will need to assign an element # to each flat portion which is defined by two end nodes.

According to the format, the element entry can be established as follows

```
1 1 2 0.0451 100
2 2 3 0.0451 100
3 3 4 0.0451 100
4 4 5 0.0451 100
5 5 6 0.0451 100
```

(hint: 0.0451 is the steel thickness required by the competition. 100 is the material #, the default material properties in CUF5M are accepted, no change is needed.)



2.4 Update the Nodes and Elements entries in CUF5M and click “Update Plot” button

Load Save Input Properties Analyze Post Z R Print Copy Reset ? X

Material Properties		Update Plot	
mat#	Ex Ey vx vy Gxy	Plot Options:	
100	29500.00 29500.00 0.30 0.30 11346.15	<input checked="" type="checkbox"/> node #	
		<input type="checkbox"/> element #	
		<input type="checkbox"/> material #	
		<input type="checkbox"/> stress m...	
		<input type="checkbox"/> stress dist.	
		<input type="checkbox"/> coordinat...	
		<input checked="" type="checkbox"/> constraints	
		<input checked="" type="checkbox"/> springs	
		<input checked="" type="checkbox"/> origin	
		C/Z Template	
		Double Elem.	
		Divide Elem.	
		Delete Elem.	
		Trans. Node	

Nodes	
node#	x z xdof zdof ydof qdof stress
1	1.50 0.50 1 1 1 1.00
2	1.50 0.00 1 1 1 1.00
3	0.00 0.00 1 1 1 1.00
4	0.00 4.73 1 1 1 1.00
5	1.50 4.73 1 1 1 1.00
6	1.50 4.23 1 1 1 1.00

Elements	
elem#	nodei nodej thickness mat#
1	1 2 0.0451 100
2	2 3 0.0451 100
3	3 4 0.0451 100
4	4 5 0.0451 100
5	5 6 0.0451 100

Lengths: 10.00 11.00 12.00 13.00 14.00 15.00 20.00 30.00 40.00 50.00 60.00 70.00 80.00 90.00 100.00 200.00 300.00 400.00 500.00 600.00 700.00 800.00 900.00 1000.00

Springs		General Constraints		cFSM	
node#	DOF(x=1,z=2,y=3,theta=4) kspring kflag	node#e	DOFe coeff. node#k DOFk	Basis for cFSM	View ?
0		0		<input type="radio"/> Natural modes <input checked="" type="radio"/> Axial mo... fully orthogonal Q.m...	
				<input type="checkbox"/> Global <input type="checkbox"/> Dist. <input type="checkbox"/> Local <input type="checkbox"/> Other	

2.5 Increase the element number by keeping clicking “Double Elem.” button.

(hint: more elements may result in accurate results, it is usually adequate to double the elements twice.)

The screenshot shows the software interface with the following components:

- 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
14 0.38 4.73 1 1 1 1 1.00
15 0.75 4.73 1 1 1 1 1.00
16 1.13 4.73 1 1 1 1 1.00
17 1.50 4.73 1 1 1 1 1.00
18 1.50 4.61 1 1 1 1 1.00
19 1.50 4.48 1 1 1 1 1.00
20 1.50 4.36 1 1 1 1 1.00
21 1.50 4.23 1 1 1 1 1.00
- Elements:** elem# | nodei | nodej | thickness | mat#
14 14 15 0.045100 100
15 15 16 0.045100 100
16 16 17 0.045100 100
17 17 18 0.045100 100
18 18 19 0.045100 100
19 19 20 0.045100 100
20 20 21 0.045100 100
- Plot Options:**
 - node #
 - element #
 - material #
 - stress m...
 - stress dist.
 - coordinat...
 - constraints
 - springs
 - origin
- Buttons:** C/Z Template, Double Elem., Divide Elem., Delete Elem., Trans. Node
- Lengths:** 1.00 2.00 3.00 4.00 5.00 6.00 7.00 8.00 9.00 10.00 11.00 12.00 13.00 14.00 15.00 20.00 30.00 40.00 50.00 60.00 70.00 80.00 90.00 100.00 200.00 300.00 400.00 500.00
- Springs:** node# | DOF(x=1,z=2,y=3,theta=4) | kspring | kflag
0
- General Constraints:** node#e | DOFe | coeff. | node#k | DOFk
0
- cFSM:** Basis for cFSM: Natural modes, Axial mo... fully orthogonal Q mo...
On/Off: Global 0, Dist. 0, Local 0, Other 0

2.6 Check the “Lengths” entry, and make sure number 1 through 12 are included.

The screenshot shows the software interface with the following components:

- Elements:** 16 16 17 0.045100 100
17 17 18 0.045100 100
18 18 19 0.045100 100
19 19 20 0.045100 100
20 20 21 0.045100 100
- Buttons:** Double Elem., Divide Elem., Delete Elem., Trans. Node
- Lengths:** 1.00 2.00 3.00 4.00 5.00 6.00 7.00 8.00 9.00 10.00 11.00 12.00 13.00 14.00 15.00 20.00 30.00 40.00 50.00 60.00 70.00 80.00 90.00 100.00 200.00 300.00 400.00 500.00
- Springs:** node# | DOF(x=1,z=2,y=3,theta=4) | kspring | kflag
0
- General Constraints:** node#e | DOFe | coeff. | node#k | DOFk
0
- cFSM:** Basis for cFSM: Natural modes, Axial mo... fully orthogonal Q mo...
On/Off: Global 0, Dist. 0, Local 0, Other 0

Step 3 – Perform elastic buckling analysis in CFUSM

3.1 Click “Properties” to define the compression load

Only check “P” (axial force), and define P =1 (initial uniform load = 1 kip). Then click “Generate Stress using checked P and M” to generate the initial compression stress.

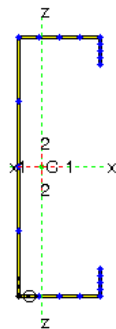
Load Save Input Properties Analyze Post Z R Print Copy Reset ? X

Calculated Section Properties

A = 0.39372	J = 0.00026695
$x_{cg} = 0.42955$	$z_{cg} = 2.365$
$I_{xx} = 1.3572$	$I_{zz} = 0.1303$
$I_{xz} = 0$	$\theta = 0$
$I_{11} = 1.3572$	$I_{22} = 0.1303$

Open Section Properties

$X_s = -0.69285$	$Z_s = 2.365$
$C_w = 0.61209$	
$\beta_1 = 0$	Basic Plot ω scale = 1
$\beta_2 = 4.9759$	warping text out



Calculation of Loads and Moments for Generation of Stress on Member

Moments consider: Unsymmetric Restrained Bending

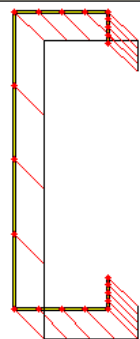
Generate P and M based on max (yield) stress = 0

Bimoment based on T = 0 L = 100 x = 50

Calculate P, M and B ? !

P =	1	<input checked="" type="checkbox"/>
$M_{xx} =$	0	<input type="checkbox"/>
$M_{zz} =$	0	<input type="checkbox"/>
$M_{11} =$	0	<input type="checkbox"/>
$M_{22} =$	0	<input type="checkbox"/>
B =	0	<input type="checkbox"/>

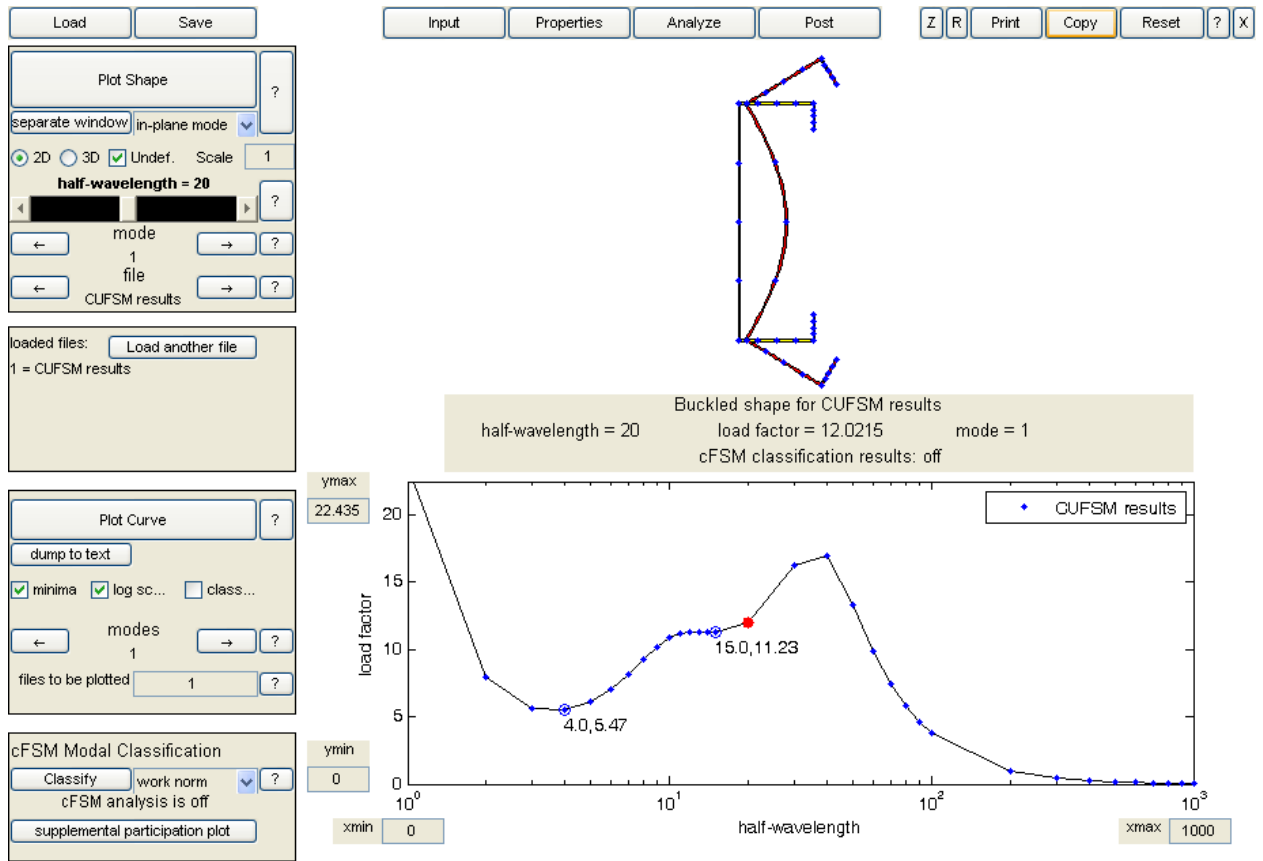
Generate Stress using checked P and M ?



Scale = 1 Max Comp. = 2.5399 Min Tens. = 0

3.2 Click "Analyze" to perform the analysis

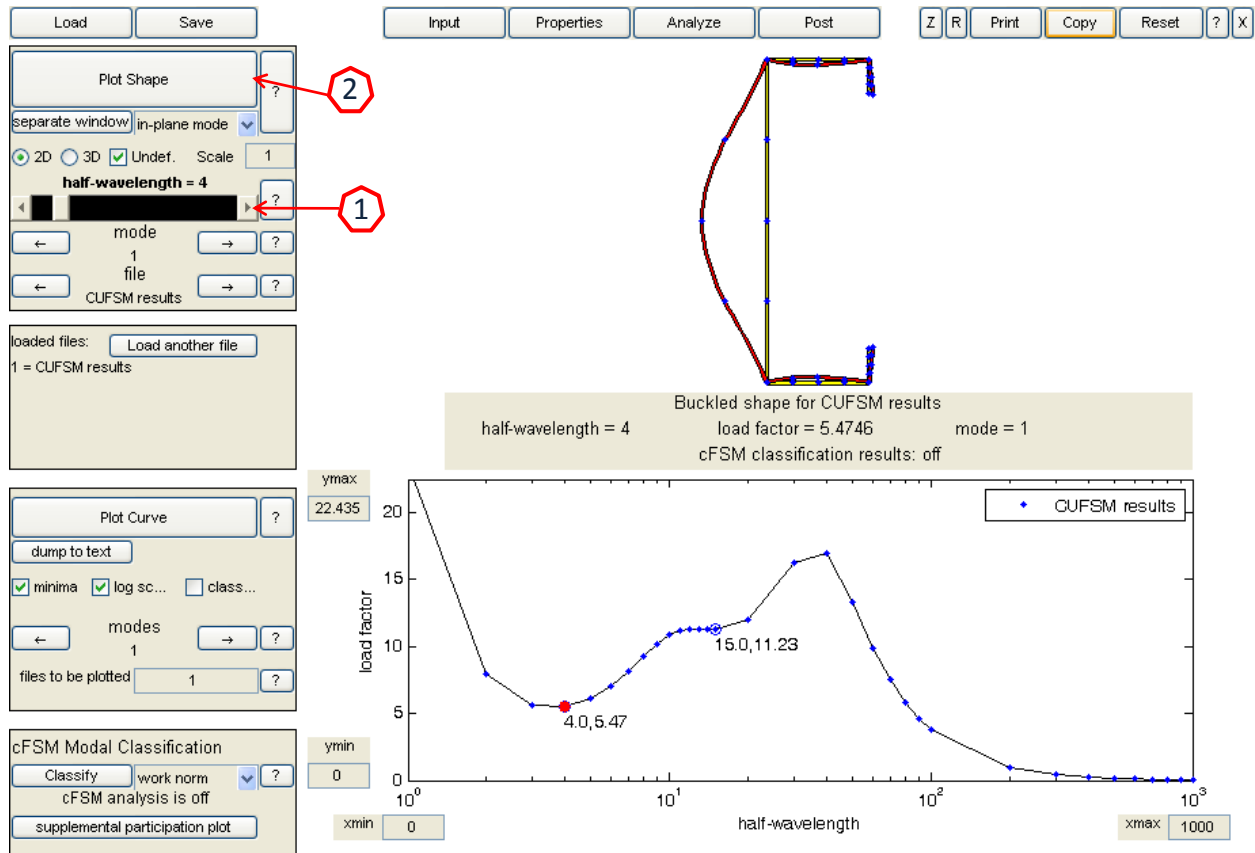
The result figure is automatically shown after the analysis.



Step 4 – Determine the critical elastic buckling load

The goal is to determine the critical elastic buckling load for the half-wavelength up to 12 inches. The result figure gives the elastic buckling load for entire half-wavelength range. We will determine the minimum load factor in the half-wavelength range between 1 and 12 inches.

For the C section, the minimum load factor is at half-wavelength 4 inches. We now need to change the half-wavelength to 4 (mark 1 in the figure below) and update the shape (mark 2 in the figure below).



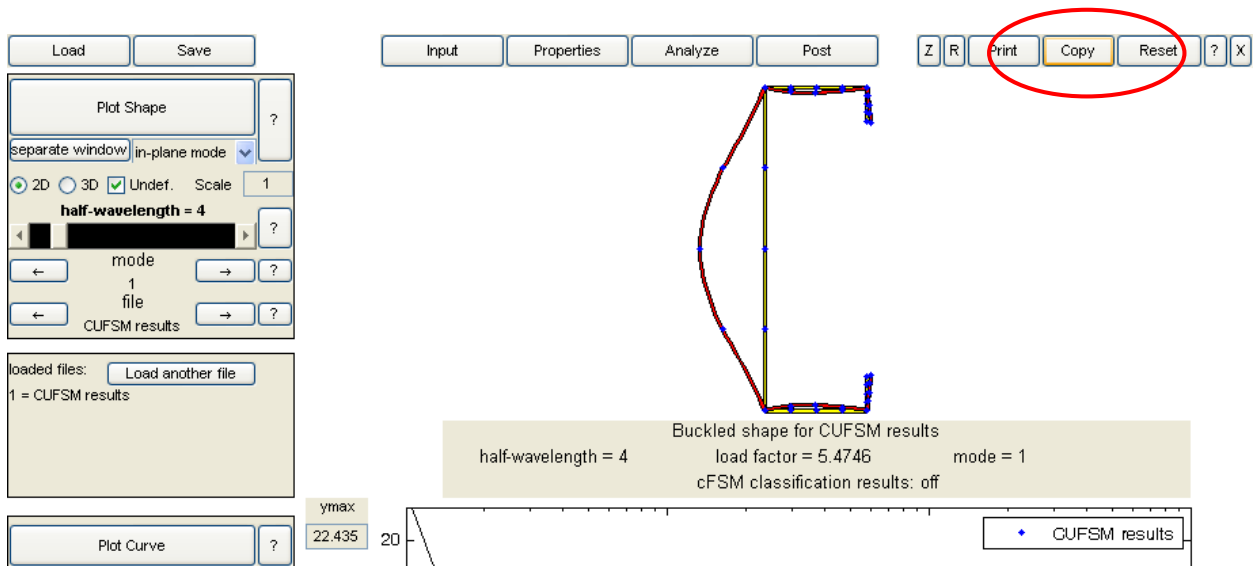
The load factor = 5.4746. Since we gave an initial load $P = 1$ kip,

the final result = load factor \times initial load $P = 5.4746 \times 1 = 5.4746$ kip.

The critical elastic buckling load for the C section is **5.4746 kip**.

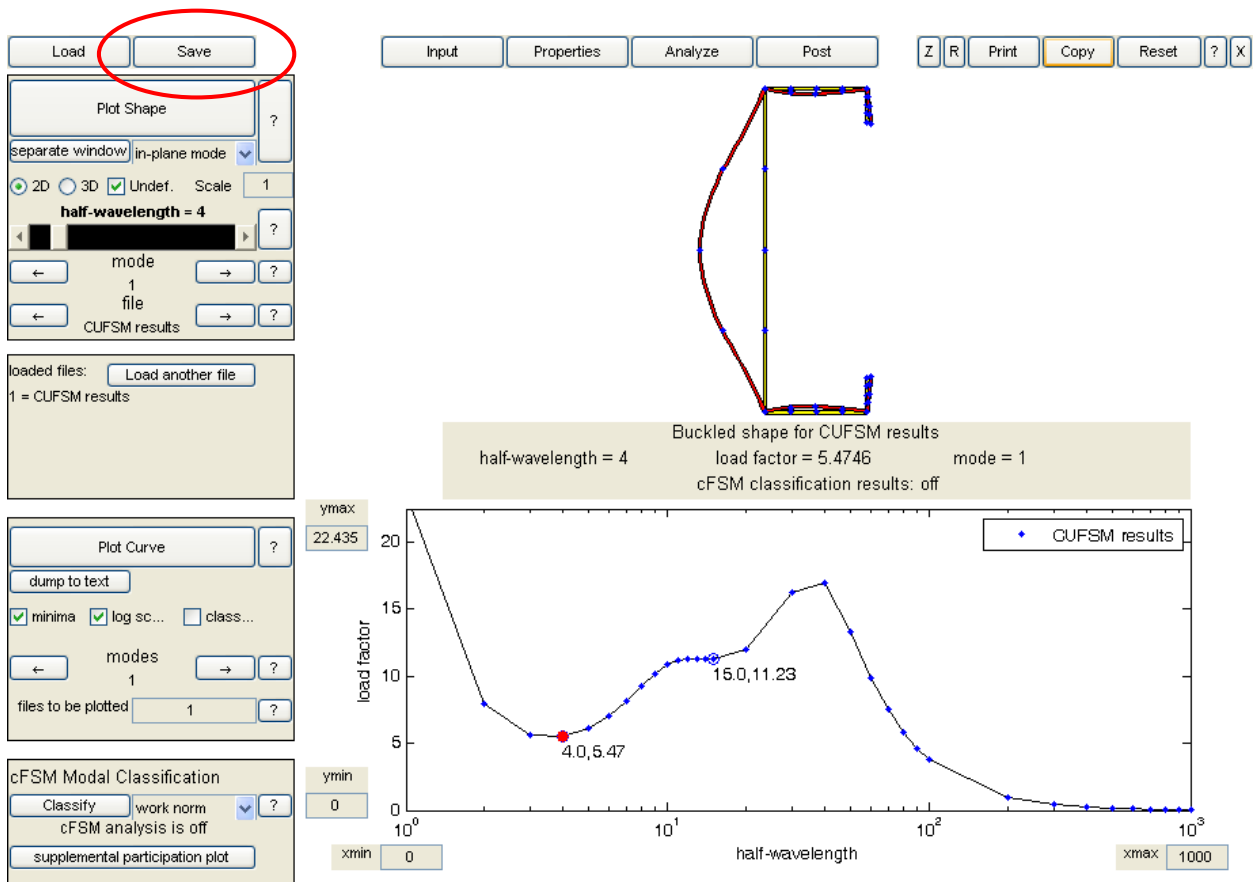
The critical half-wavelength is **4.0 inches**.

(hint: you can click “Copy” button to copy the CUFSM screen to clipboard and paste it to your design essay to show the buckling load and buckling shape)



Step 5 – Save CUFSM result file and start working on the Design Essay

Save the file name as first name _ last name.mat



Step 6 – Fill out the Information Form

2011 STUDENT COMPETITION ON COLD-FORMED STEEL DESIGN

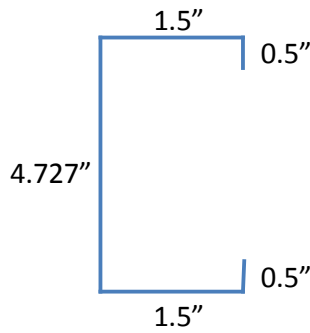
INFORMATION FORM

Student Information

Name: Design Example Email: Example@school.edu
School name: Example School location: Example
Student level: Graduate Undergraduate Other _____
Major: Civil Engineering
Mailing address: Example
(optional) Address for receiving awards

CFS Design Solution

Cross-section shape:



Critical Elastic Buckling Load: 5.4746 kip
Critical half-wavelength: 4.0 inches