

# GETTING STARTED

## With EASYCIV<sup>®</sup>

Structural Analysis Of Three-Dimensional Structures

---



Calgary, AB, Canada

<https://www.easyciv.com>

Jan 2022

# Table Of Contents

OVERVIEW.....	3
Toolbar .....	4
Viewport Controls .....	5
Seletion palette.....	6
Copy / Move.....	7
Small Keyboard .....	7
Display Palette .....	7
Model Phases.....	8
A FRAME SAMPLE .....	9
RESULTS VERIFICATION.....	16
A SHELL SAMPLE .....	20

# OVERVIEW

Figure 1 shows the main window for the graphical user interface in a mobile phone. You can also operate EASYCIV on a desktop or tablet. Let's take a moment to explain what you have in front of you.

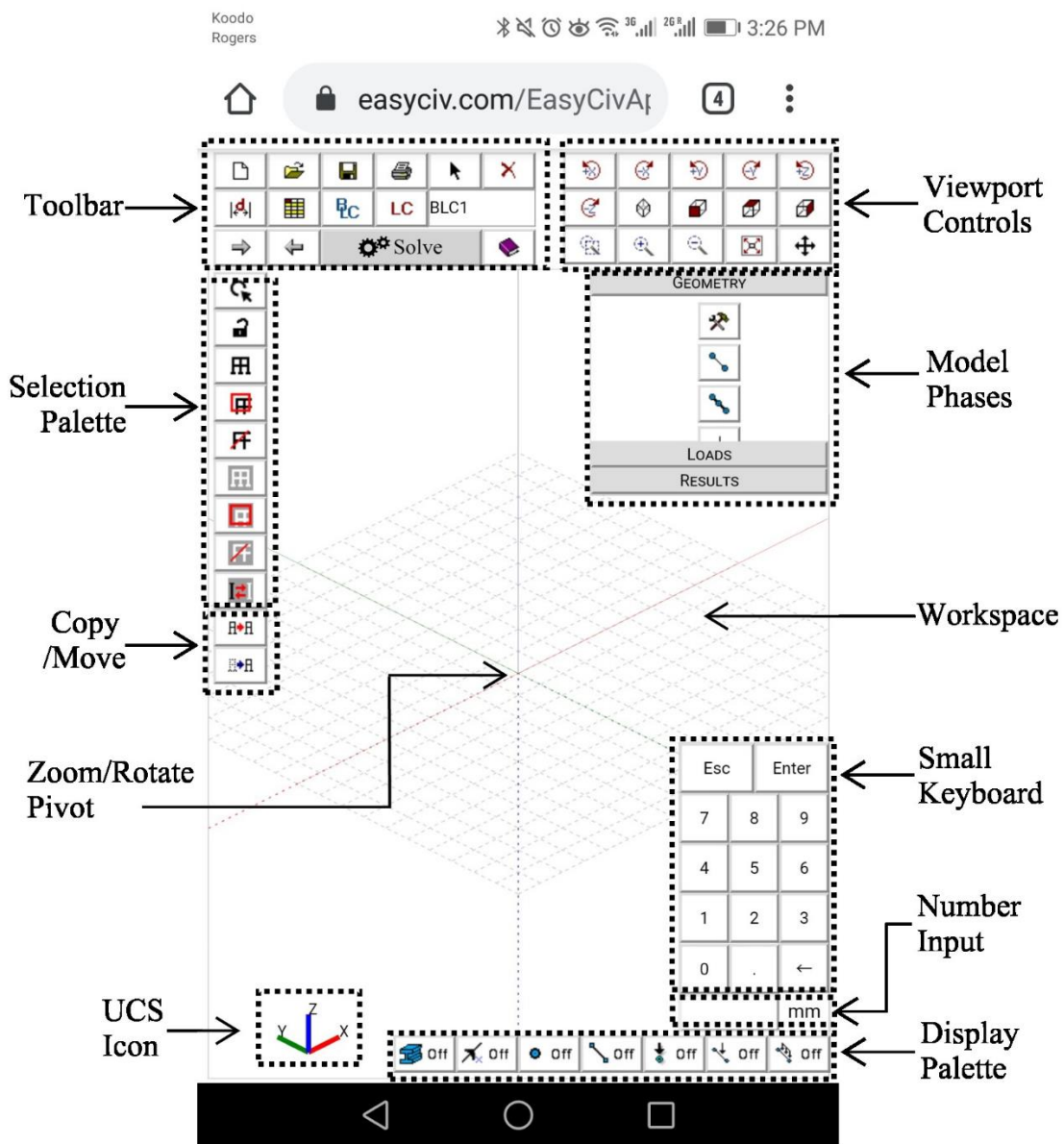




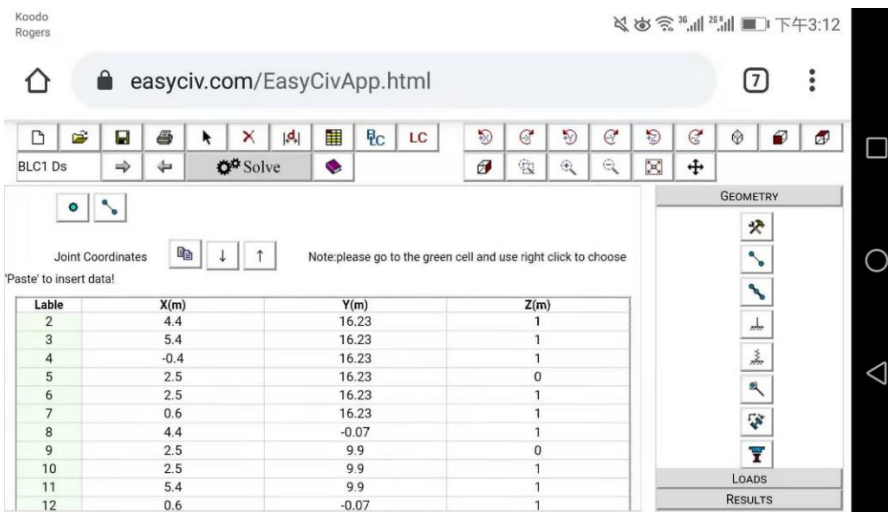


Figure 1 The Graphical User Interface Main Window

## Toolbar

The EASYCIV Toolbar is the upper left group of buttons. These buttons perform general actions such as opening and closing files, changing design parameters, printing, and solving the model.

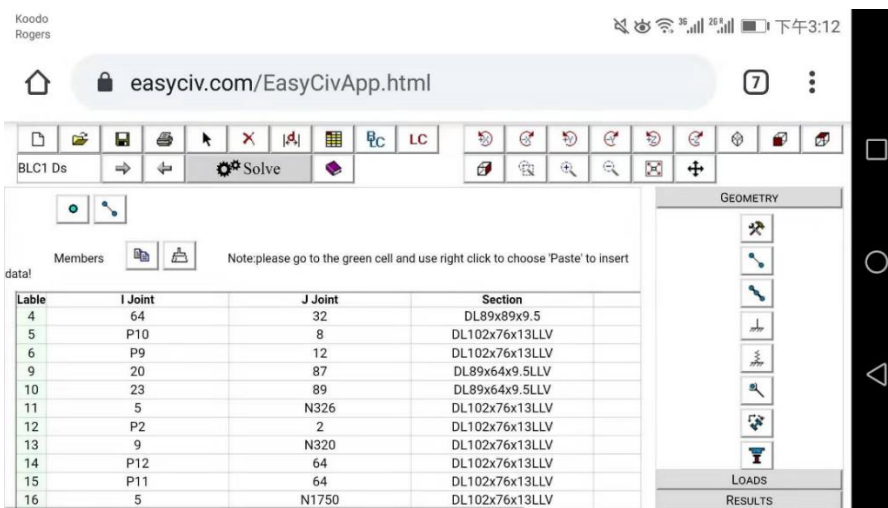
Also, point select , delete  and measure  are presented here as their widely used. Please also open the help  button to check the keyboard shortcut to greatly accelerate your work when having access to the keyboard.



The screenshot shows the EASYCIV application interface. The browser address bar displays `easyciv.com/EasyCivApp.html`. The toolbar includes icons for file operations, navigation, and solving. The main workspace features a table for point data with columns for Label, X(m), Y(m), and Z(m). The table contains 12 rows of data. To the right of the table is a vertical toolbar with icons for GEOMETRY, LOADS, and RESULTS.

Label	X(m)	Y(m)	Z(m)
2	4.4	16.23	1
3	5.4	16.23	1
4	-0.4	16.23	1
5	2.5	16.23	0
6	2.5	16.23	1
7	0.6	16.23	1
8	4.4	-0.07	1
9	2.5	9.9	0
10	2.5	9.9	1
11	5.4	9.9	1
12	0.6	-0.07	1







Figure 2 Point Data Sheet

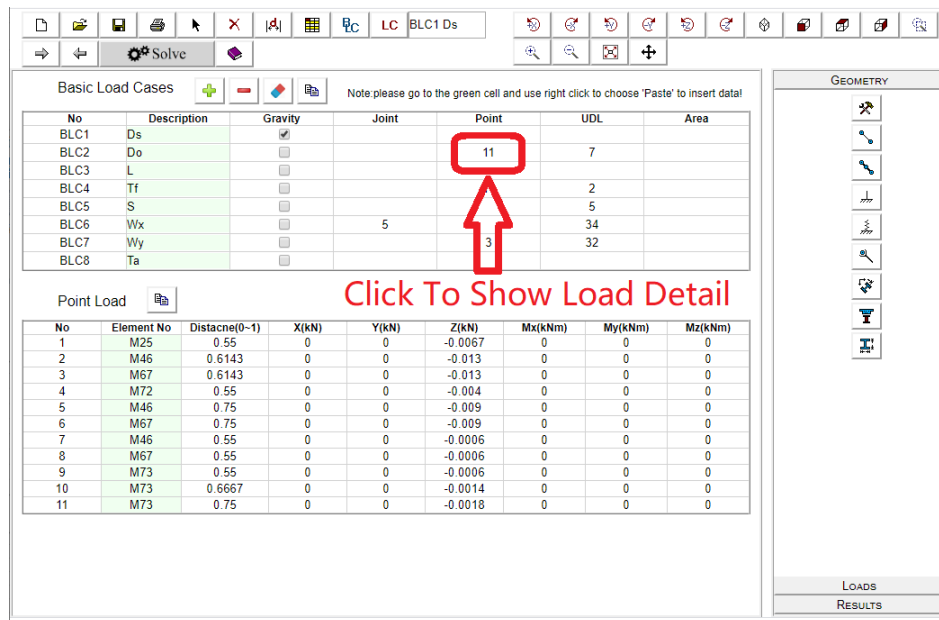


The screenshot shows the EASYCIV application interface. The browser address bar displays `easyciv.com/EasyCivApp.html`. The toolbar includes icons for file operations, navigation, and solving. The main workspace features a table for member data with columns for Label, I Joint, J Joint, and Section. The table contains 13 rows of data. To the right of the table is a vertical toolbar with icons for GEOMETRY, LOADS, and RESULTS.

Label	I Joint	J Joint	Section
4	64	32	DL89x89x9.5
5	P10	8	DL102x76x13LLV
6	P9	12	DL102x76x13LLV
9	20	87	DL89x64x9.5LLV
10	23	89	DL89x64x9.5LLV
11	5	N326	DL102x76x13LLV
12	P2	2	DL102x76x13LLV
13	9	N320	DL102x76x13LLV
14	P12	64	DL102x76x13LLV
15	P11	64	DL102x76x13LLV
16	5	N1750	DL102x76x13LLV

Figure 3 Member Data Sheet

In model , from  and  you can switch from Point and Member Data Sheet. It is recommended that copy  the data and paste to text editor or Excel and do the Editorial work in model , Basic Load Case **BLC** and Load Combination **LC** Sheet. Please make sure they are the same format when paste back. When copy some members from other program, it will be necessary to click clean up button , to make sure the local axis direction and the members are divided properly.



Basic Load Cases

Note: please go to the green cell and use right click to choose 'Paste' to insert data!

No	Description	Gravity	Joint	Point	UDL	Area
BLC1	Ds	<input checked="" type="checkbox"/>				
BLC2	Do	<input type="checkbox"/>		11	7	
BLC3	L	<input type="checkbox"/>				
BLC4	Tf	<input type="checkbox"/>			2	
BLC5	S	<input type="checkbox"/>			5	
BLC6	Wx	<input type="checkbox"/>	5		34	
BLC7	Wy	<input type="checkbox"/>		3	32	
BLC8	Ta	<input type="checkbox"/>				

Point Load

Click To Show Load Detail

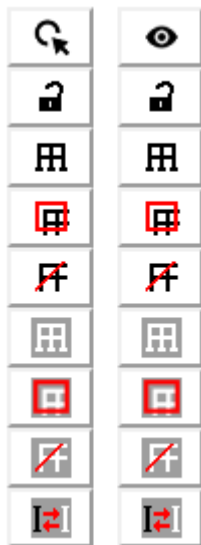
No	Element No	Distacne(0-1)	X(kN)	Y(kN)	Z(kN)	Mx(kNm)	My(kNm)	Mz(kNm)
1	M25	0.55	0	0	-0.0067	0	0	0
2	M46	0.6143	0	0	-0.013	0	0	0
3	M67	0.6143	0	0	-0.013	0	0	0
4	M72	0.55	0	0	-0.004	0	0	0
5	M46	0.75	0	0	-0.009	0	0	0
6	M67	0.75	0	0	-0.009	0	0	0
7	M46	0.55	0	0	-0.0006	0	0	0
8	M67	0.55	0	0	-0.0006	0	0	0
9	M73	0.55	0	0	-0.0006	0	0	0
10	M73	0.6667	0	0	-0.0014	0	0	0
11	M73	0.75	0	0	-0.0018	0	0	0

Figure 4 Basic Load Case Sheet

## Viewport Controls

When you are working in a graphic model, view the buttons above provide tools, such as rotate and zoom, to assist you with that view.

## Selection palette








Note: The Select To Operate  and Select To Show  button is controlling the behavior of lower buttons.

Figure 5 Selection Palett



Beside point select  on Toolbars, you may use these powerful tools to help you quickly achieve the selection that you want. They are located on the vertical Selection Palette on the left side of the screen.


The Box Select  and Box Unselect  tools work as you might expect; you use them to draw a box around the part of the model you wish to select or unselect. Only items that are entirely in the box will be affected.





To use these, first click the button then move the mouse to the model view. Click and hold the mouse button to establish an anchor point for the box; then move the mouse such that the box encloses that part of the model to be selected or unselected. Then release the mouse to perform the select or unselect operation.

The box cursor remains, and you may draw as many boxes as you need until you are finished.



When on **touch screen**, you might touch the screen and move along the screen to draw a red box in the same way.

With the Line Select  and Line Unselect  tools you simply draw a line through all the items that you want to select or unselect. Just click and hold the mouse at the start of the line and then draw the line until you release the mouse button.

The Invert Selected option  is used to invert the selected state of the model. All selected items are made unselected, and all unselected items are made selected.

The Unlock  button, when pressed it will become Lock  condition, under select to operate  condition, causes all currently unselected items to stay unselected, no matter what other selection buttons are pressed, until this button is pressed again. When under select to show  condition, Lock/Unlock condition will control the unselected member to show as dashed line or disappear completely.

## Copy / Move

EASYCIV allows you to copy/move selected portions of the model along a single direction. After selected proper portion of the model and click the copy  / move  button (Note that the copy button has a red arrow). Just click to choose a start point and input a number for the distance. Then move the mouse around to the desired direction (Color line connecting between start point and mouse tip shown accordingly). Finally, you can apply the copy/move by hitting an enter key.

## Small Keyboard

Small keyboard is designed for mobile or tablet with no keyboard attached with the same function of a physical keyboard.

## Display Palette

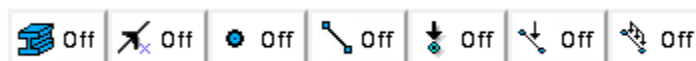


Figure 6 Display Palette

There are a series of toggle buttons located on the lower button corner of the screen. From left to right, they are the toggle for 3D member show, local axis, point label, member label/member size, node load, point load and line load.

## Model Phases

Model phases consists of three major stages of the design, GEOMETRY, LOADS AND RESULTS. Many important buttons and inputs are included inside, and we will get familiar with them through a quick sample below.

Note: EasyCiv is to assist you with your analysis and simulation and is not a substitute for your professional judgment or your own independent analysis, simulation or other activities, including those with respect to product stress, safety and utility.



## A FRAME SAMPLE

So, we have started EASYCIV and are ready to define our model. We should note here that your picture might not look like the screen in Figure 1. Your mobile, desktop or tablet screen resolution and font sizes can change the appearance of the menu and toolbars. This is simply cosmetic and should not affect your ability to go through this sample.

A plane frame calculation sample, Figure 6 and Figure 7, from the book: STRUCTURAL ANALYSIS--A UNIFIED CLASSICAL AND MATRIX APPROACH, Seventh Edition-Taylor and Francis (2017) by Ghali, Amin and Neville, Adam, is used here to demonstrate the modeling and serve as a verification example in next chapter.

### *EXAMPLE 5.3: PLANE FRAME WITH INCLINED MEMBER*

Obtain the bending moment diagrams for the plane frame in Figure 5.3a due to the separate effects of: (1) the loads shown; (2) a downward settlement  $\delta_D$  at support  $D$ . Consider  $EI = \text{constant}$  and neglect the change in length of members.

*Figure 7 A Plane Frame Example*

Case 1 will be considered here. Relevant data are listed as below:

Geometry:

Cross-section: square  $b = 100\text{mm}$

Span:  $l = 1\text{m}$

Material:  $E = 200\text{GPa}$

Loads:

$P = 1\text{kN}$

$q = 1\text{kN/m}$

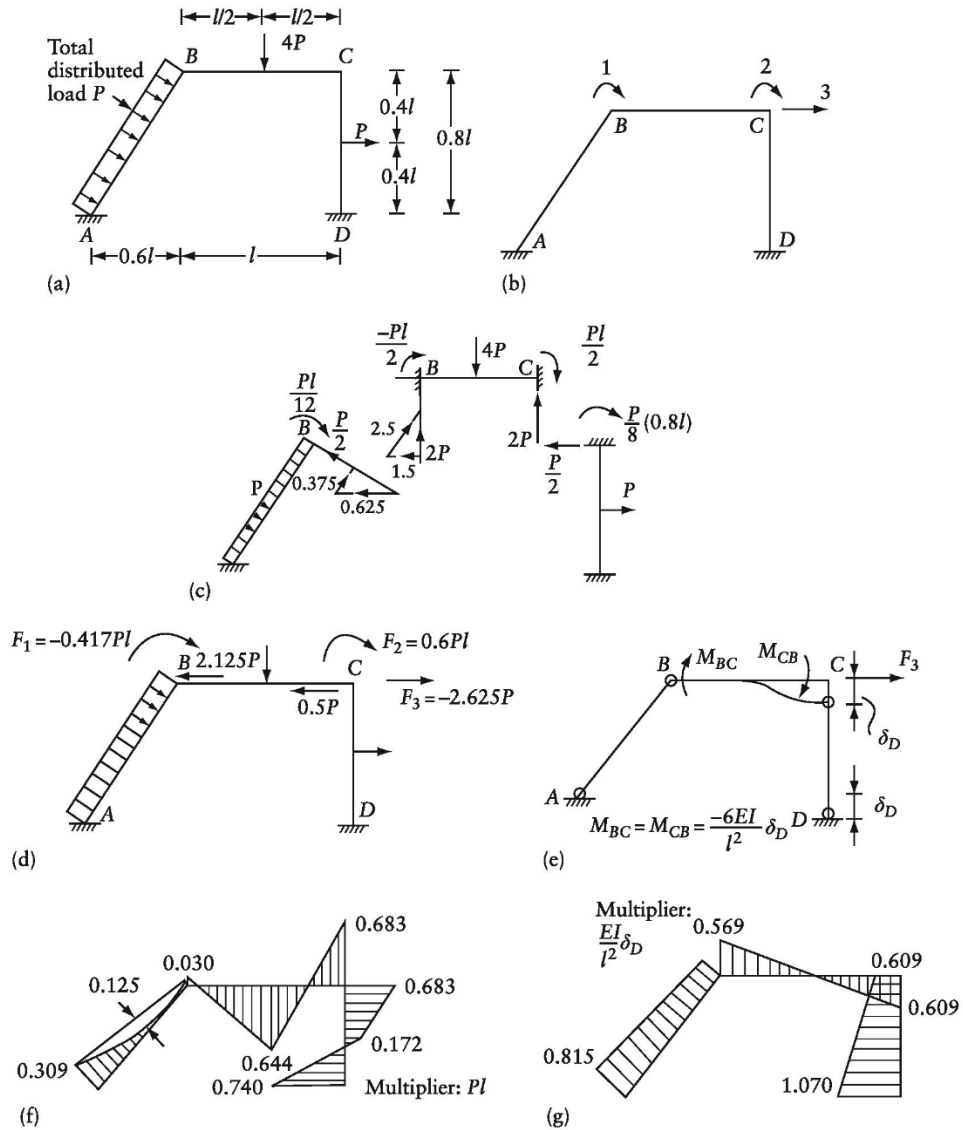







Figure 5.3 Frame analyzed in Example 5.3. (a) Frame dimensions and loading. (b) Coordinate system. (c) Fixed-end forces in case (1). (d) Restraining forces  $\{F\}$  in case (1). (e) Member-end moments with the displacements restrained at the coordinates in case (2). (f) Bending moment diagram in case (1). (g) Bending moment diagram in case (2).

for case (2), the shearing force at end B of member BC will be an upward force equal to  $(12EI\delta_D/l^3)$ . This force can be substituted by a component equal to  $(15EI\delta_D/l^3)$  in AB and a component equal to  $(-9EI\delta_D/l^3)$  in direction of coordinate 3. The latter component is equal to  $F_3$  in case (2).

The member-end moments when the displacements at the coordinates are prevented (Figure 5.4a and Figure 5.3e) are

## Step 1 Draw Elements

1. Click Draw Line  button.
2. Click on any point in workspace.
3. Click on solid light blue line above that point make a portion of line turn dark.
4. Input 800.
5. Click Enter.
6. Click Zoom All , Zoom In  button and Pan  button to get proper view.
7. Click Draw Line  button.
8. Click on top node of previous element.
9. Click on solid light red line.
10. Input 1000.
11. Click Enter.
12. Click on dash blue line.
13. Input 800.
14. Click Enter.
15. Click Esc, finish element drawing.

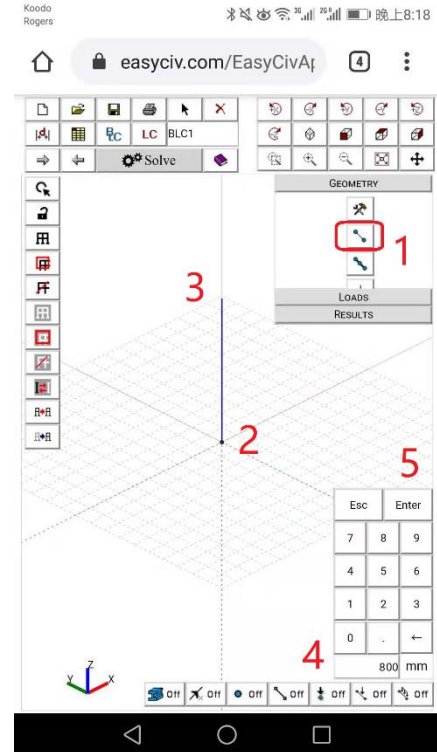


Figure 9

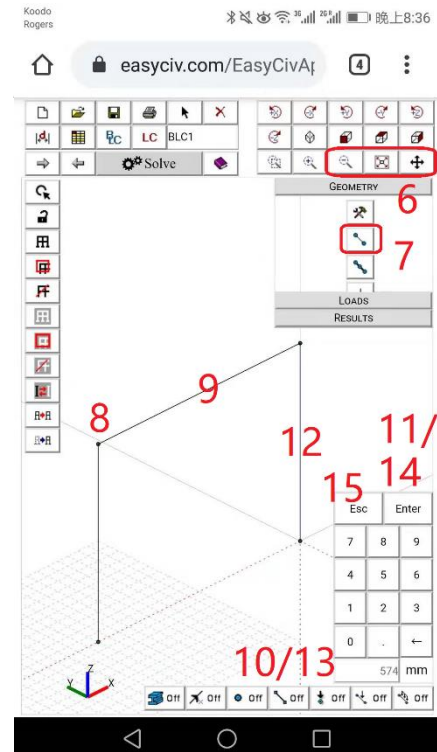






Figure 10

## Step 2 Modify Element

1. Click Box Select ;
2. Slice to create a box to select lower left corner point.
3. Click any point to start.
4. Slice to show a direction line.
5. Input 600.
6. Click Enter.
7. Click Box Select ;
8. Slice to create boxes to select two bottom points.
9. Click Support  button to expand the panel.
10. Click Apply  button to assign support.
11. Click Esc, finish this step.

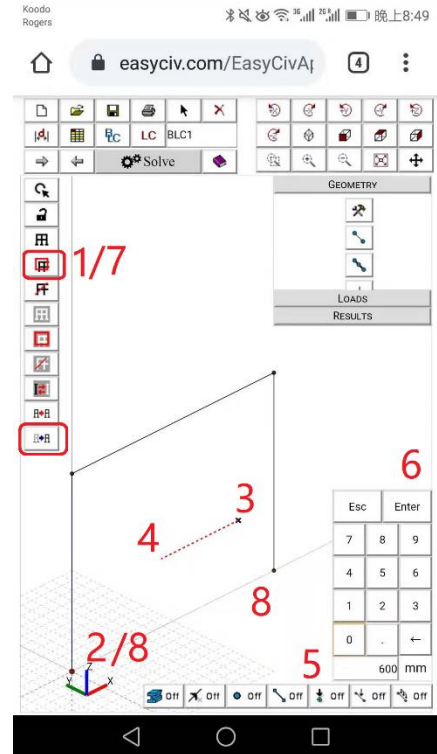


Figure 11

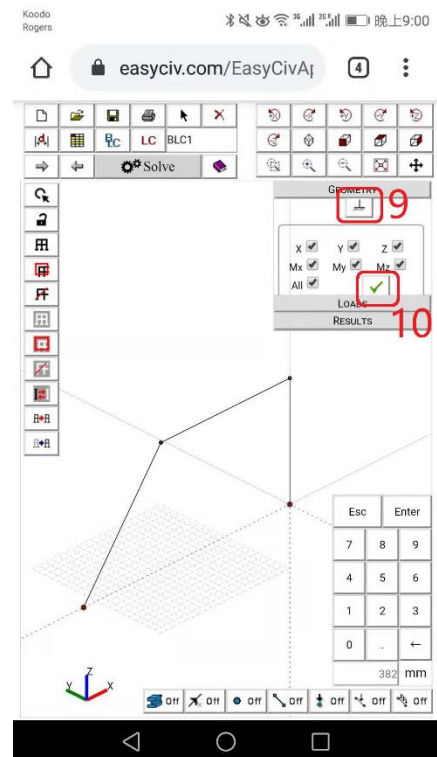









Figure 12

### Step 3 Define and Apply Cross Section

1. Click Cross Section  button to expand panel.
2. Click Self-Define  button to load dialog box.
3. Click Rectangle section.
4. Input sizes.
5. Click Apply  button.
6. Click Close X button to close dialog.
7. Select User-define and corresponding cross section.
8. Click Select All  button.
9. Click Apply  button.
10. Click Esc, finish this step.
11. Click Toggle 3D  On button to check.
12. Click Toggle 3D  On button again back to line show.

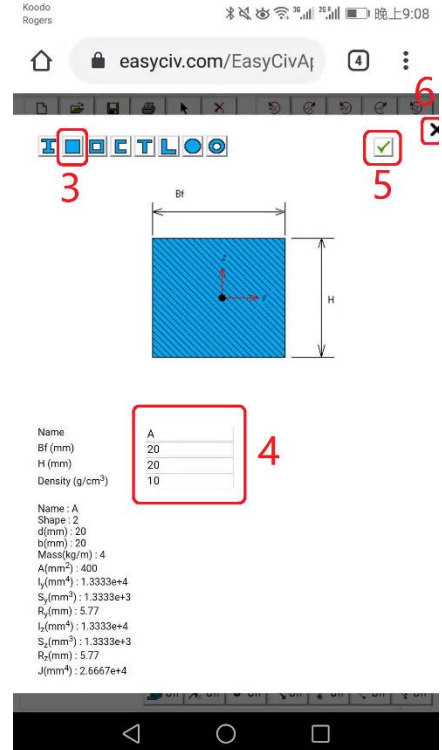


Figure 13

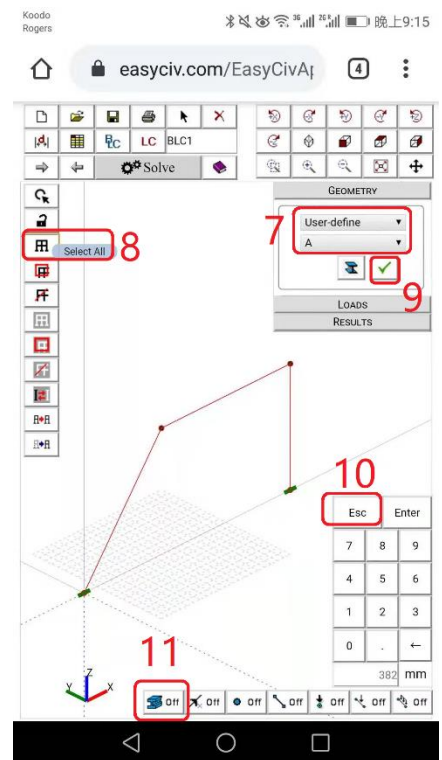









Figure 14



## Step 5 Calculate and Results

1. Click Solve  button.
2. Click Esc, clear the screen.
3. Click Bending Moment  button.
4. Click Apply  button.
5. Click Front View  button.
6. Click Parameter  button.
7. Click Toggle Detail/Max button to check the detail bending moment.
8. Click Select to Show  button.
9. Click Line Unselect  button.
10. Slice to unselect top beam to eliminate some

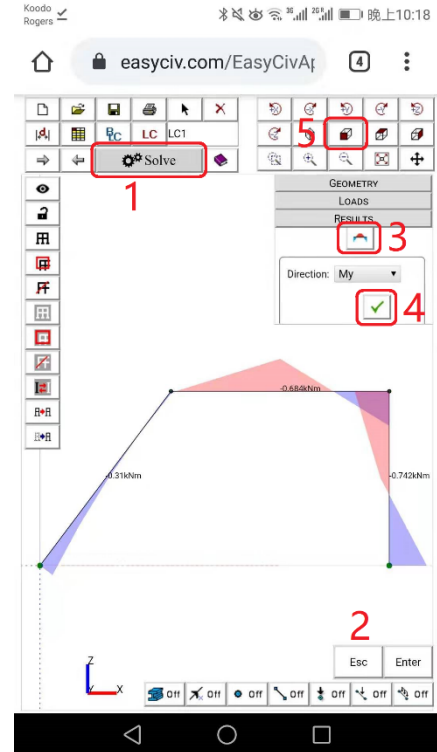



Figure 17

11. Click Displacement  button.
12. Adjust displacement scale in Parameter panel.
13. Check X direction to show the X direction displacement number.

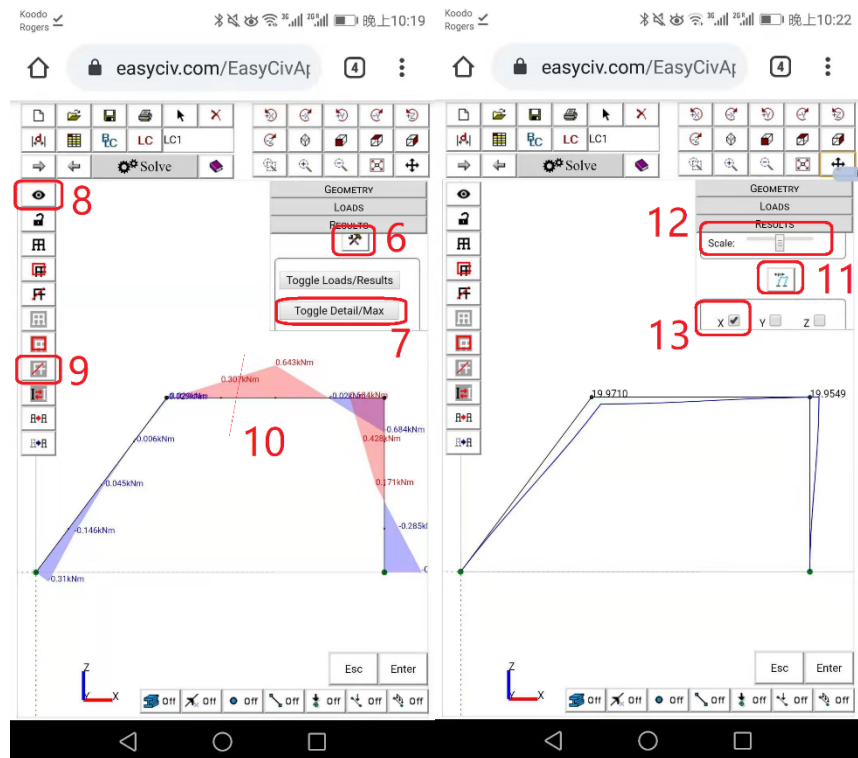


Figure 18

Figure 19

# RESULTS VERIFICATION

Theoretical results (STRUCTURAL ANALYSIS--A UNIFIED CLASSICAL AND MATRIX APPROACH, Seventh Edition-Taylor and Francis (2017) by Ghali, Amin and Neville, Adam,) Figure 19 / 20 are compared with the ones obtained from EASYCIV analysis.

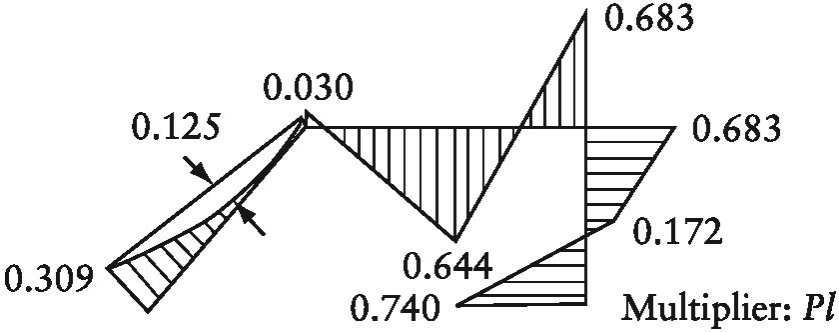


Figure 20 Bending Moment Diagram

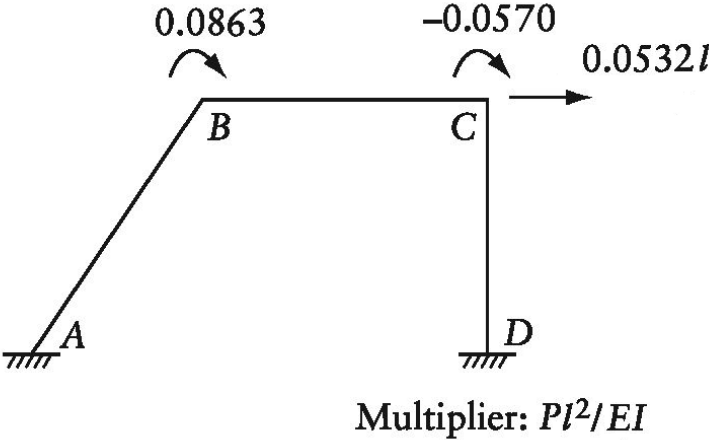



Figure 21 Displacement Diagram



# 1 Obtain Cross Section Properties from EASYCIV

## Step 1 Obtain Moments of Inertia

1.1 Click Cross Section  button to expand panel.

1.2 Click used cross section.

1.3 Obtain Moments of Inertia,  
 $I = 1.333e+4mm^4$ .

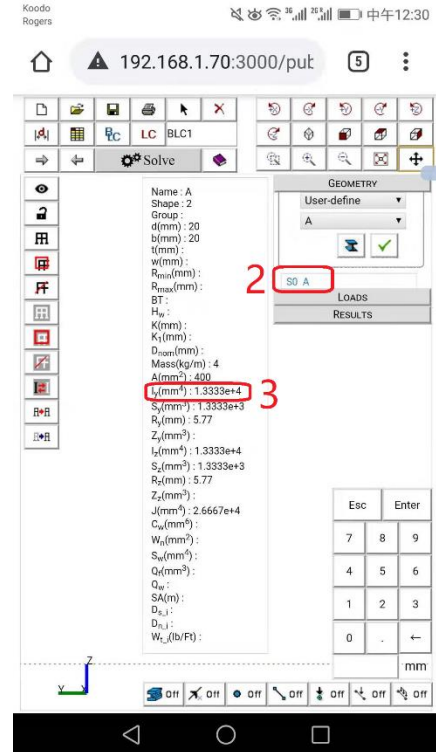



Figure 22

## Step 2 Obtain Young's Modulus

2.1 Click Material  button to expand panel.

2.2 Obtain Young's Modulus,

$E = 200GPa$

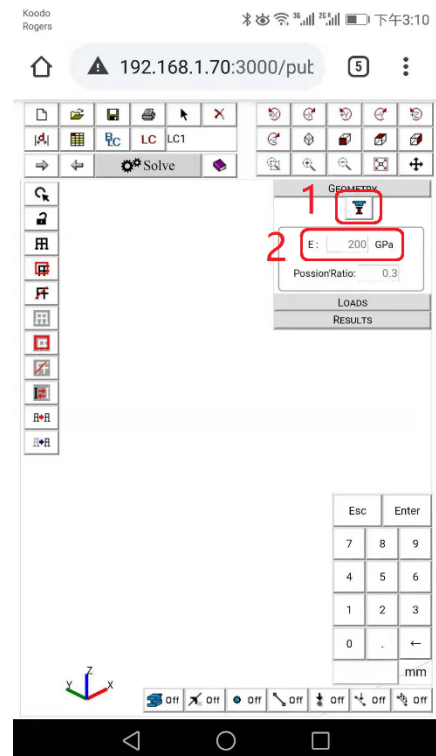


Figure 23

## 2 Diagrams from EASYCIV

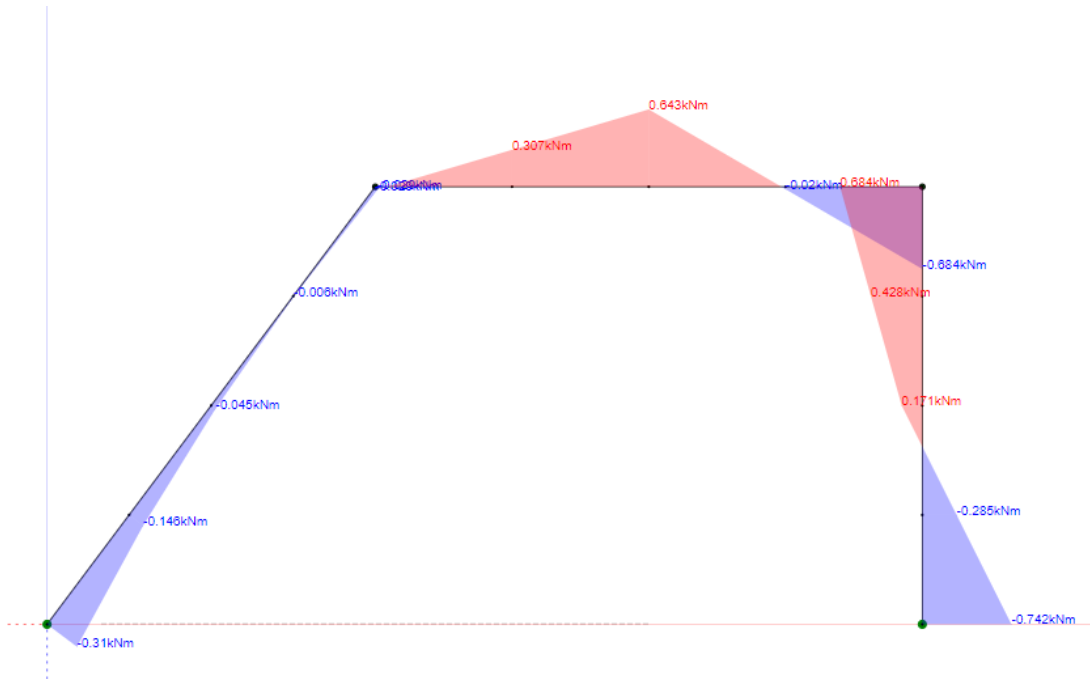


Figure 24

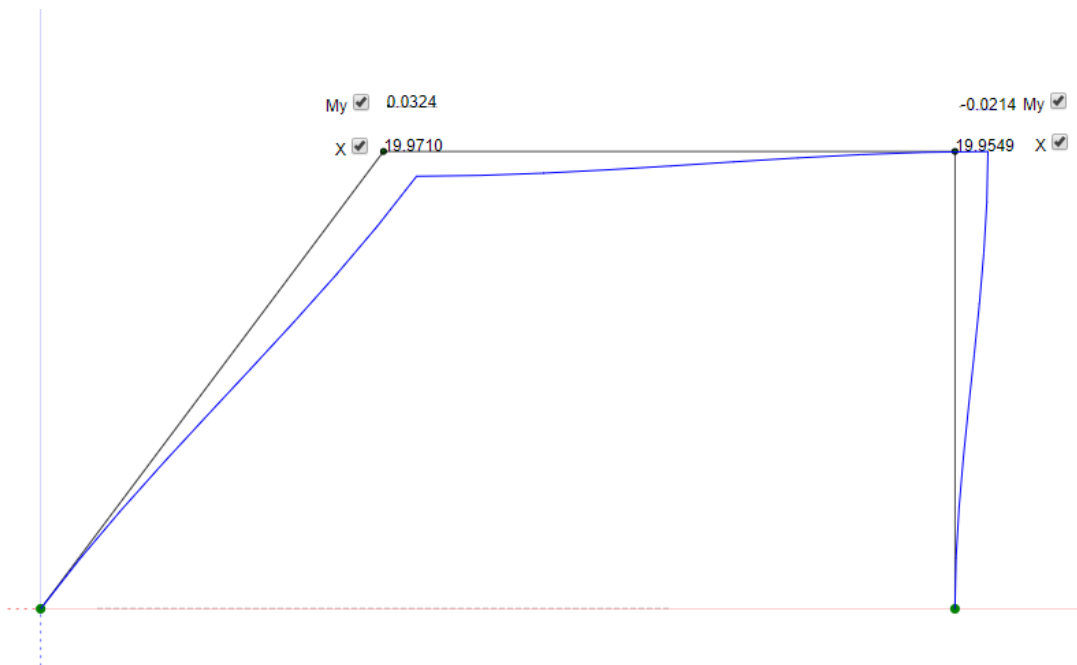


Figure 25

### 3 Results Comparison

Maximum values of displacements and bending moments at different positions obtained from EASYCIV analysis are presented in Table 1 and Table 2, respectively and compared with those obtained from theoretical results.

Table 1 Comparison of Results for Bending Moments (kN·m)

	$M_A$	$M_{MidAB}$	$M_B$	$M_{MidBC}$	$M_C$	$M_{MidCD}$	$M_D$
Theory	0.309	0.045	0.030	0.644	0.683	0.172	0.740
EASYCIV	0.310	0.045	0.029	0.643	0.684	0.171	0.742
Difference	0.32%	0.22%	0.68%	0.16%	0.15%	0.35%	0.27%






Table 2 Comparison of Results for Displacement (mm or rad)

	$M_B$	$M_C$	$X_B$	$X_C$
Theory	0.0324	0.0214	19.9500	19.9500
EASYCIV	0.0324	0.0214	19.9710	19.9549
Difference	0.12%	0.12%	0.11%	0.02%

#### a) Conclusion

The result comparison reflects acceptable agreement between EASYCIV software and theoretical results. In theoretical method, the assumption of neglecting the change in length of members might be the reason of the difference.






## A SHELL SAMPLE

Rectangular element with corner nodes (24 degrees of freedom) and thin (Kirchhoff) plates are considered in EasyCiv currently. In EasyCiv, 5 buttons are designed for finite element modeling specifically. Shell button, , is for creating shell element and modifying shell thickness and material. Combining with the display toggle button,  on, you can switch showing the thickness and material in model. Mesh button, , is for meshing selecting shell element while Mesh- select button, , is for turning frame element into shell element. Stress button, , is for showing the Von Mises stress in model.

Shell finite element analysis in EasyCiv is an excellent tool for looking into the detail of the distribution of stress over the element to avoid overstress locally.

Below is a very simple sample to demonstrate how to build and analyze the shell model in EasyCiv.

## Step 1 Create the First Element

1. Click Draw Line  button.
2. Click on any point in workspace.
3. Click on solid red line beside that point make a portion of line turn dark.
4. Input 1000.
5. Click Enter.
6. Click Esc, finish element drawing.
7. Click Select-all  button.
8. Click Cross Section  Button.
9. Select W-shapes and W200X36 from drop down list.
10. Click Apply  button.
11. Click Element Toggle  button to check if member size is right.

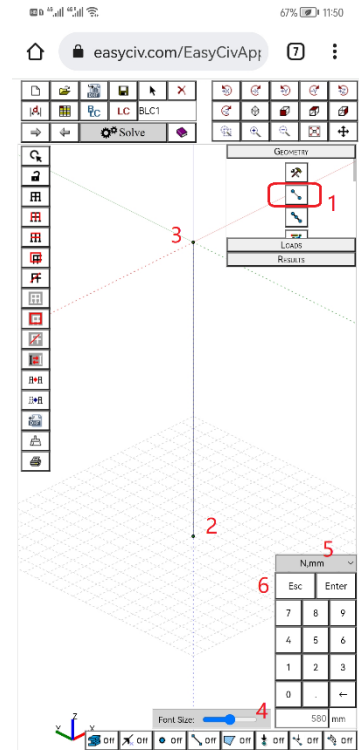


Figure 26

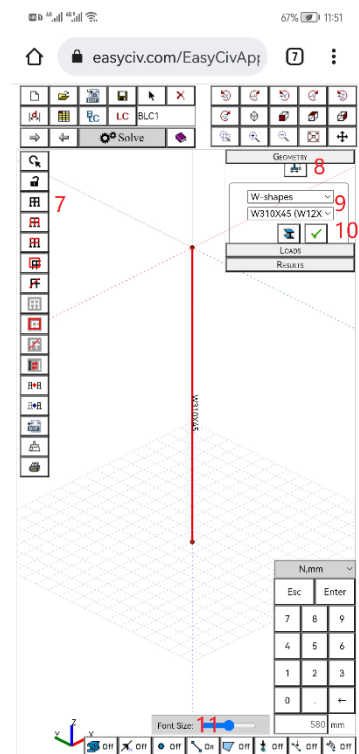












Figure 27

## Step 2 Mesh the First Element

1. Click Mesh- select  button.
2. Click Select-all  button.
3. Click Mesh  button.
4. Input 4/4 and click Apply  button.
5. Click Esc, finish element drawing.
6. Click Front View  button.
7. Click Select to Show  button.
8. Click Box Select .
9. Slice to create boxes to select web at center elevation. Click Lock  button to hide unselect.
10. Click ISO View  button. Click Select toggle  button to switch back to select mode.

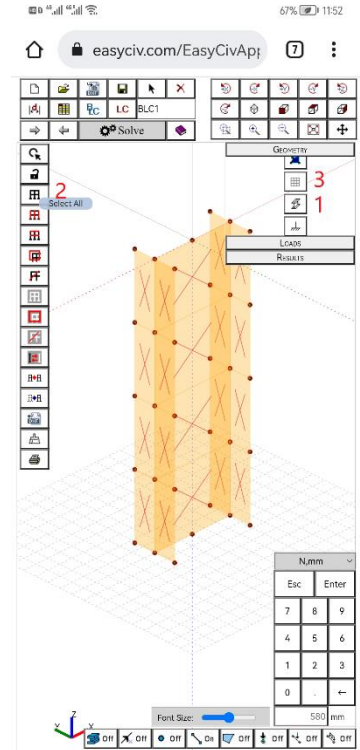


Figure 28

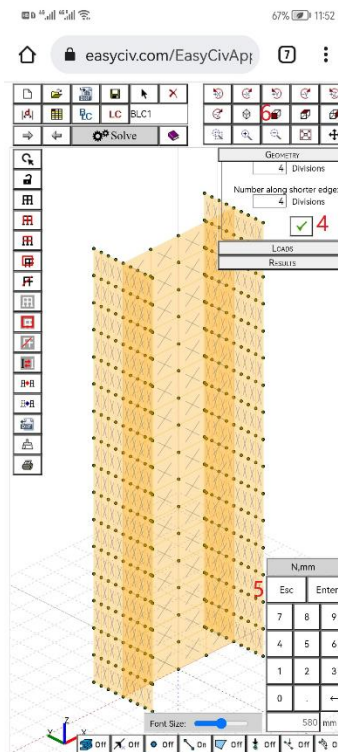


Figure 29

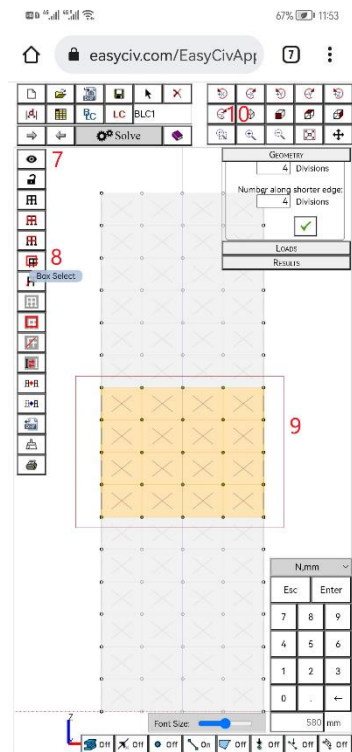







Figure 30

### Step 3 Create the Second Element

1. Click Draw Line  button.
2. Click on an existing point.
3. Click on dash green line beside that point make a portion of line turn dark.
4. Input 800.
5. Click Enter.
6. Click Esc, finish element drawing.
7. Click Select  button to select second element.
8. Click Cross Section  Button.
9. Select W-shapes and W200X36 from drop down list.
10. Click Apply  button.
11. Click Element Toggle  button to check if member size is right.

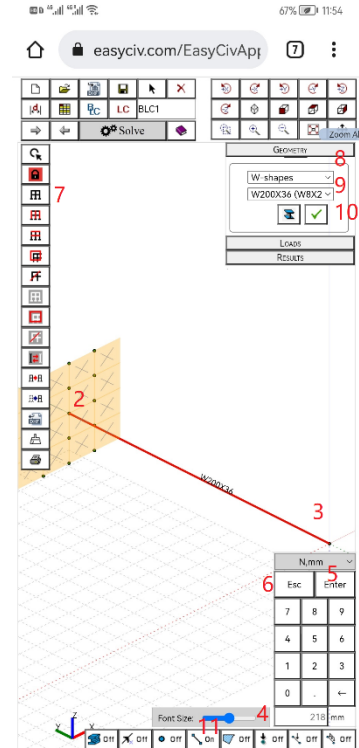






Figure 31

#### Step 4 Mesh the Second Element

1. Click Mesh- select  button.
2. Click Line-select  button.
3. Draw a line to select web elements.
4. Click Mesh  button.
5. Input 4/1 and click Apply  button.
6. Click Esc, finish element meshing.

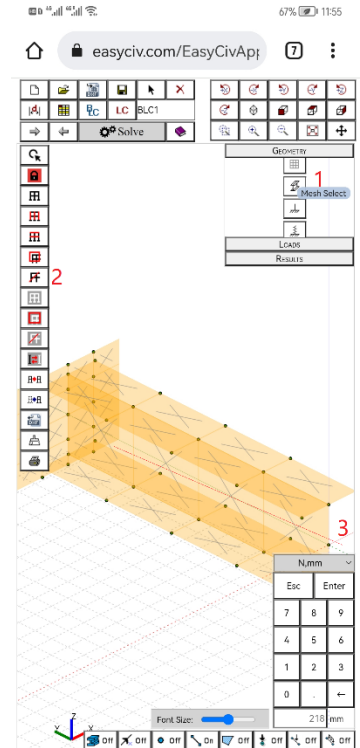


Figure 32

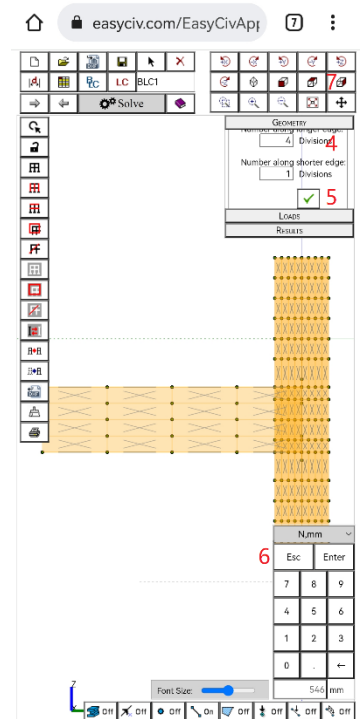








Figure 33



## Step 5 Fine Tune the Model and Add Supports

1. Click Front View  button.
2. Click Measure  button to get the gap between the nearby nodes from the conjunction of the two members.
3. Click Move  button to move exact distance to make sure two members share the nodes.
4. Click Support  button to expand the panel.
5. Click Box Select  select two end nodes.
6. Click Apply  button to assign support.
7. Click Esc, finish this step.

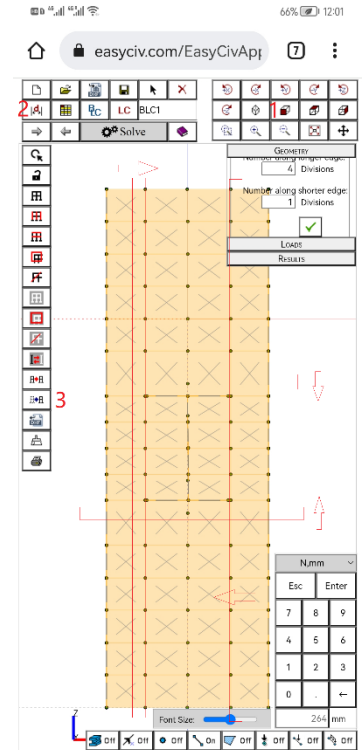


Figure 34

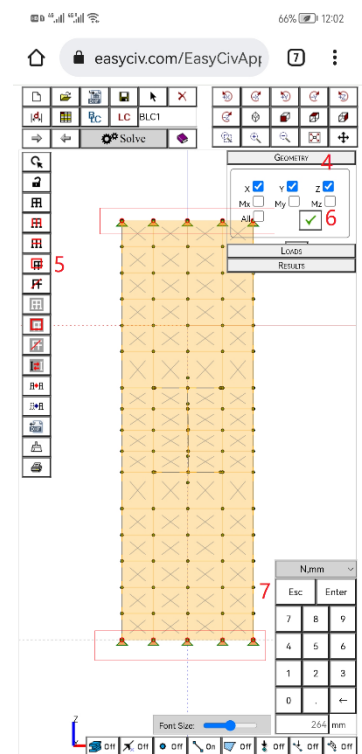



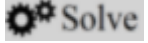



Figure 35



## Step 7 Apply Load and Analyze

1. Click ISO View  button.
2. Click Load on Model Phase to show load panel.
3. Click Node Load  button to expand panel.
4. Select force direction Y, 10 kN.
5. Click Apply  button.
6. Select center point at top flange.
7. Click Esc, finish this force apply.
8. Click Solve  button.
9. Click Results on Model Phase to show results panel.
10. Click Stress  button to show the stress contour.
11. Input material yielding strength to show yielding zone.

Note: the stress contour shows the maximum elastic stress exceed limit, so stiffeners are considered.

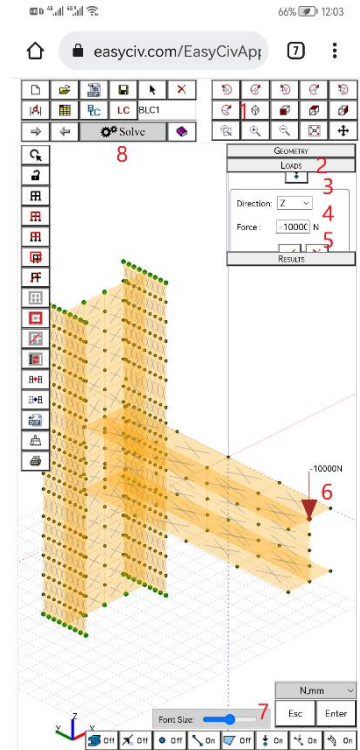


Figure 37

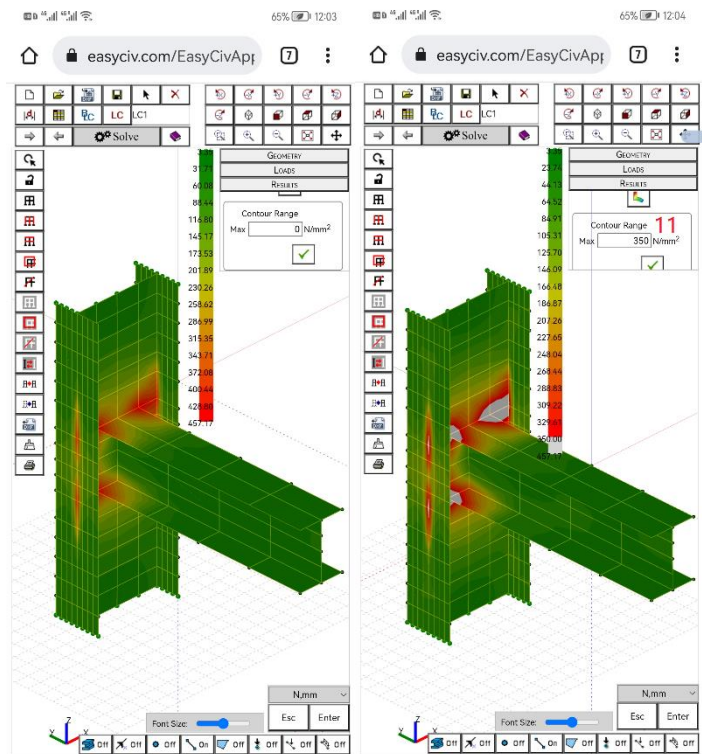





Figure 38

Figure 39

## Step 8 Add Stiffeners and Re-analyze

1. Show the joint only and click Shell  button to pick 4 points to create a single shell.
2. Select the shell, then click Mesh  button, Input 4/1 and click Apply  button.
3. Measure the distance between two flanges, show stiffener only and copy.

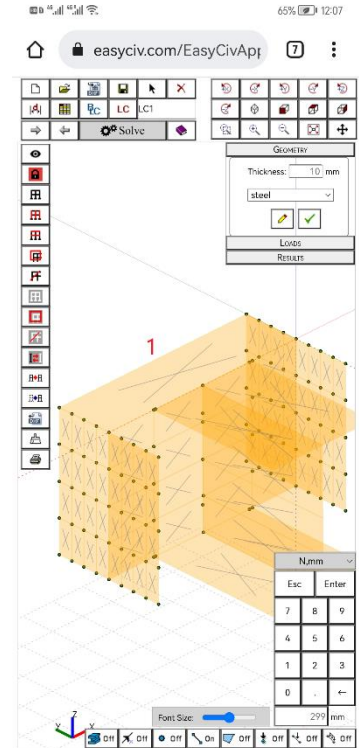


Figure 40

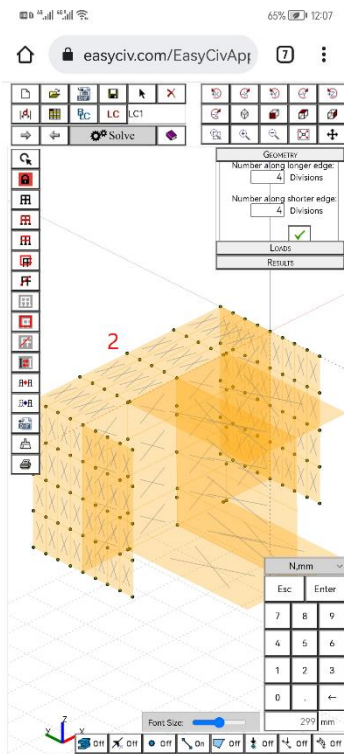


Figure 41

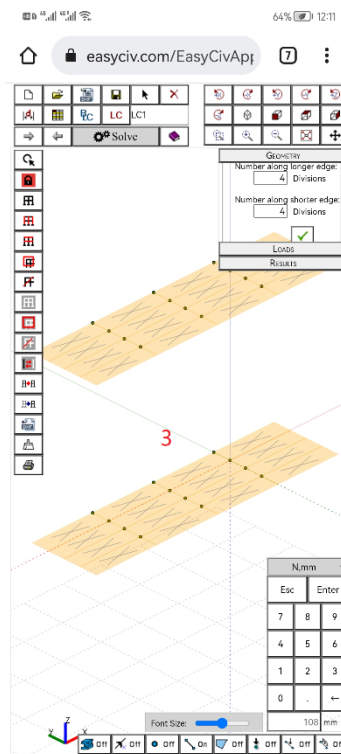


Figure 42

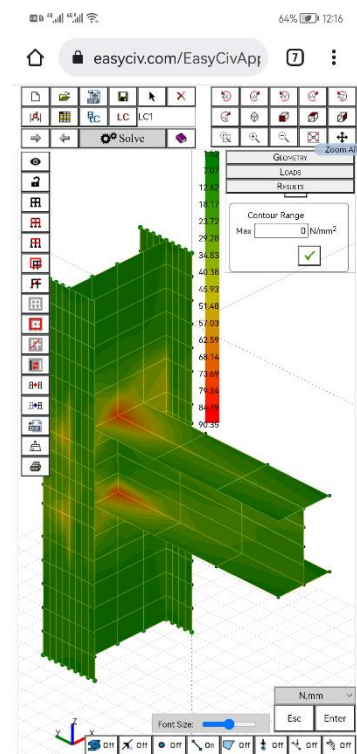


Figure 43