Engineering 5003 - Ship Structures I

Lab#5
Combined analysis types in ANSYS

By C. Daley

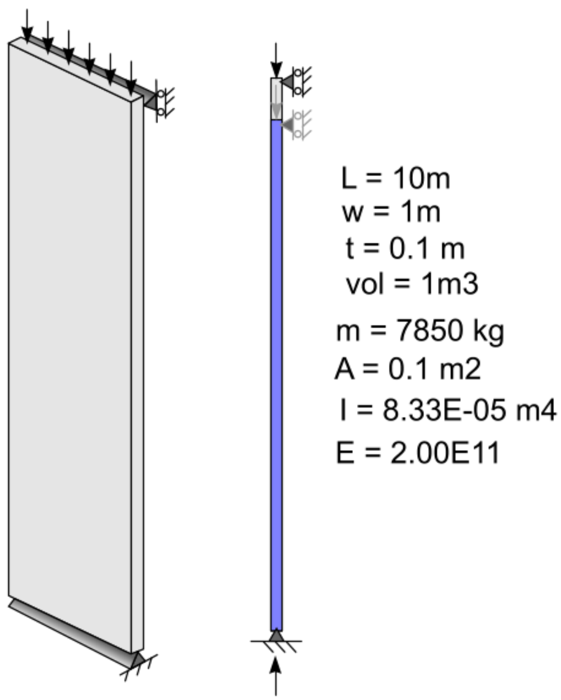
Overview

In this lab we will model a simple pinned column using shell elements. Once again, we will use SpaceClaim to create the geometry model of the problem, and then use ANSYS to model the structural behavior. We will explore the stress, buckling and vibration analysis features of ANSYS.

ANSYS Model #5 – Pinned Rectangular column

Step 1: describe and sketch the problem:

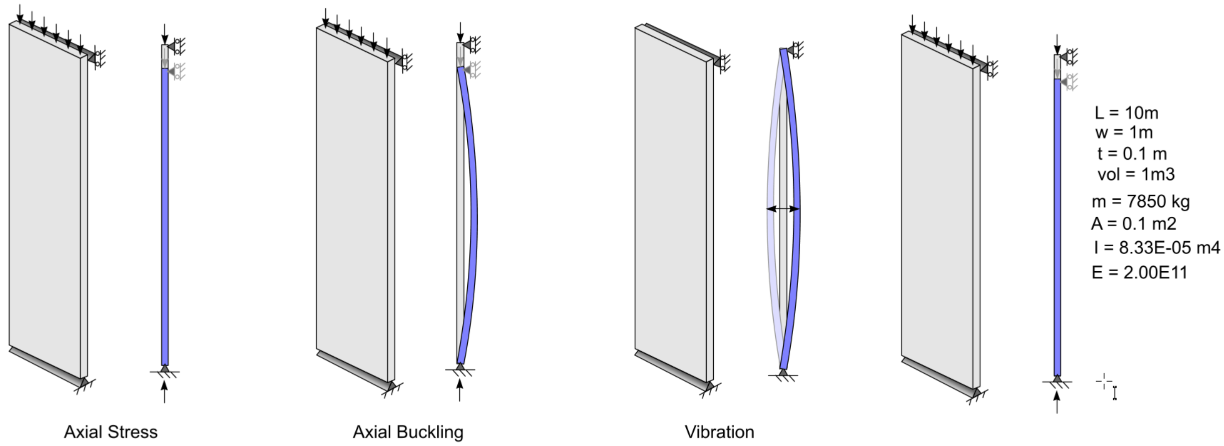
The model is a 10m x 1m x 0.1m plate with loads and parameters as shown;



The botton is a simple pin, while the top is a guided pin. In this way we still have a pinned-pinned beam even after it is compressed by an axial load.

Step 2: estimate expected results (analytically):

The analytical solutions for stress, buckling and natural vibration are shown below;



stress

F = 1 N

σ = 10 Pa

buckling

$$F_{cr_1} = \frac{\pi^2 EI}{L^2} \quad 1.645\text{ E6 N}$$

$$F_{cr_2} = \frac{\pi^2 EI}{(L/2)^2} \quad 6.580\text{ E6 N}$$

$$F_{cr_3} = \frac{\pi^2 EI}{(L/3)^2} \quad 1.483\text{ E7 N}$$

vibration

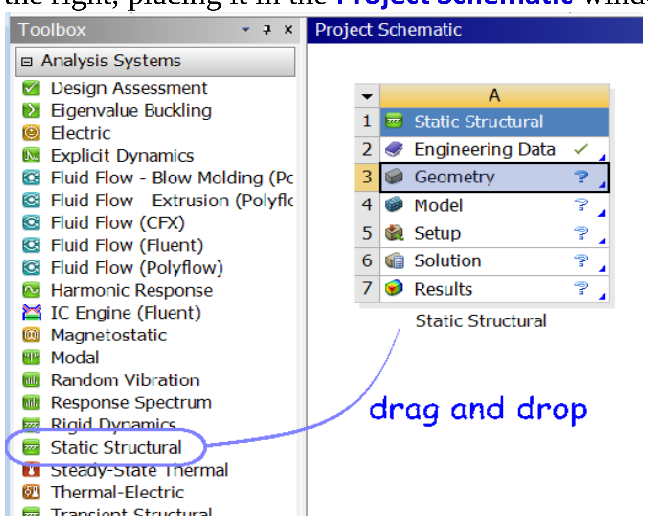
$$\omega_1 = \frac{9.87}{2\pi}\sqrt{\frac{EI}{m\,L^3}} \quad 2.29\text{ hz}$$

$$\omega_2 = \frac{39.5}{2\pi}\sqrt{\frac{EI}{m\,L^3}} \quad 9.16\text{ hz}$$

$$\omega_3 = \frac{88.8}{2\pi}\sqrt{\frac{EI}{m\,L^3}} \quad 20.6\text{ hz}$$

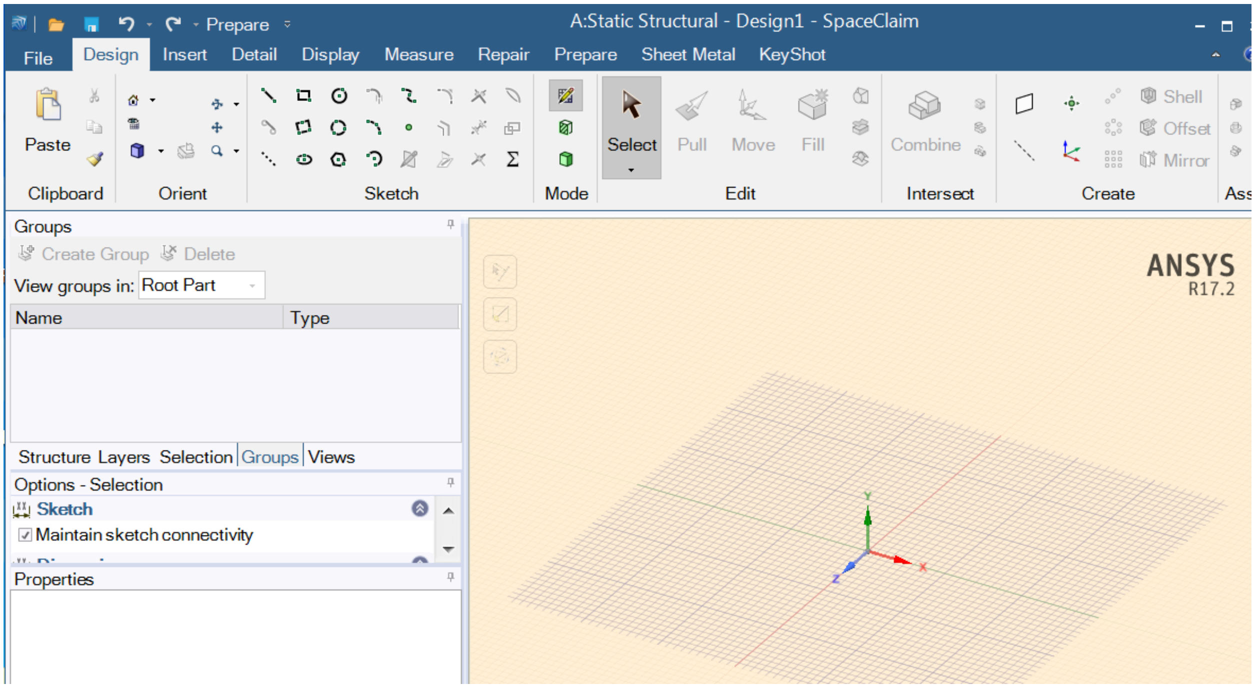
Step 3: open ANSYS Workbench 17.0

1) First, save the (empty) project as **Beam5.wbpj**
2) The left-hand window shows a set of analysis type options. Select **Static Structural** and drag the icon to the right, placing it in the **Project Schematic** window.

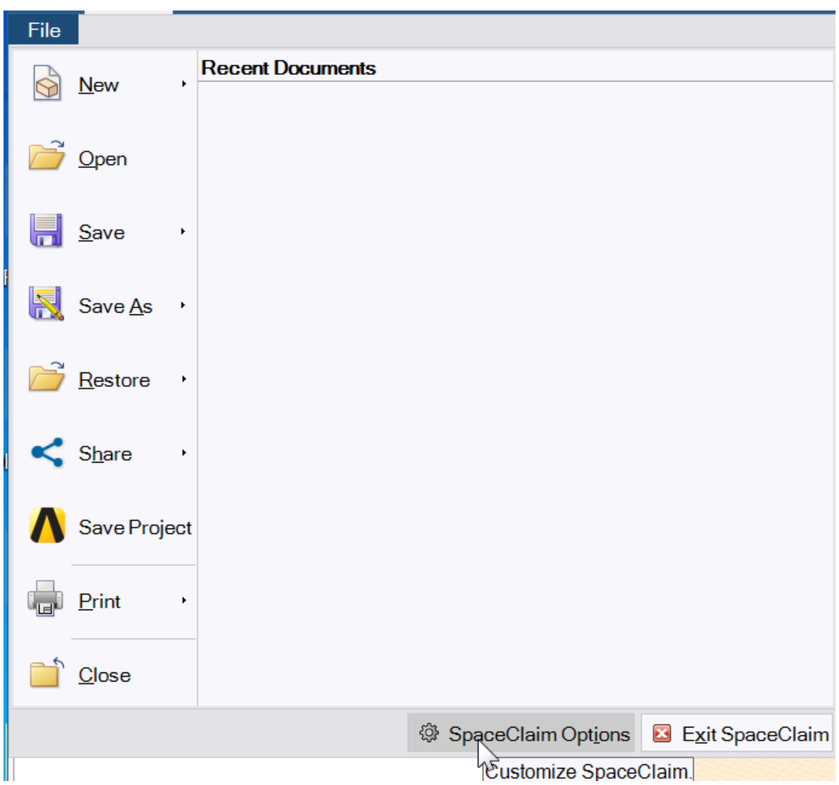


Step4: open Geometry and create the CAD model

1) By Clicking Geometry in the Project window, ANSYS will open a CAD modeling program called SpaceClaim. You will see a window like this;



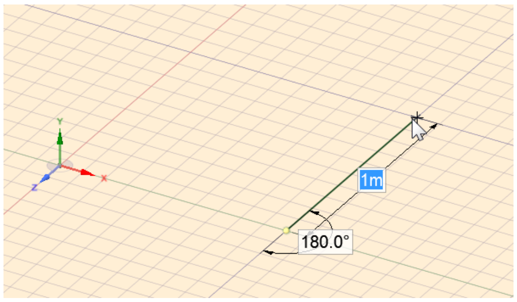
2) If you want to change options or units, you can go to the file tab and select SpaceClaim Options. This will let you change Units (mm are default) and many other options. Change units to meters, and .

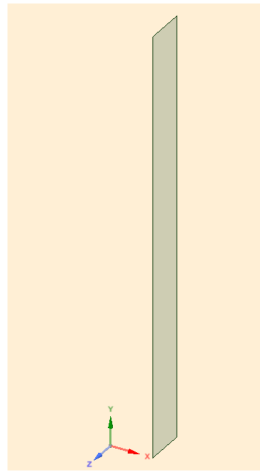


You now see the main window where the CAD model will be displayed, just as in Lab#1.

3) In the Design tab there are many drawing and editing tools. Let’s leave the sketch plane as a horizontal plane, so we can draw grillage

4) Draw a 1m line and pull it 10m.

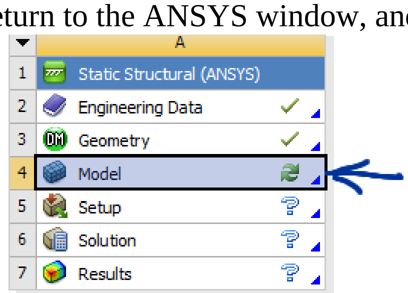




This will create a plane, which will be modelled with shell elements.

Step5: open Model and create the Finite Element model

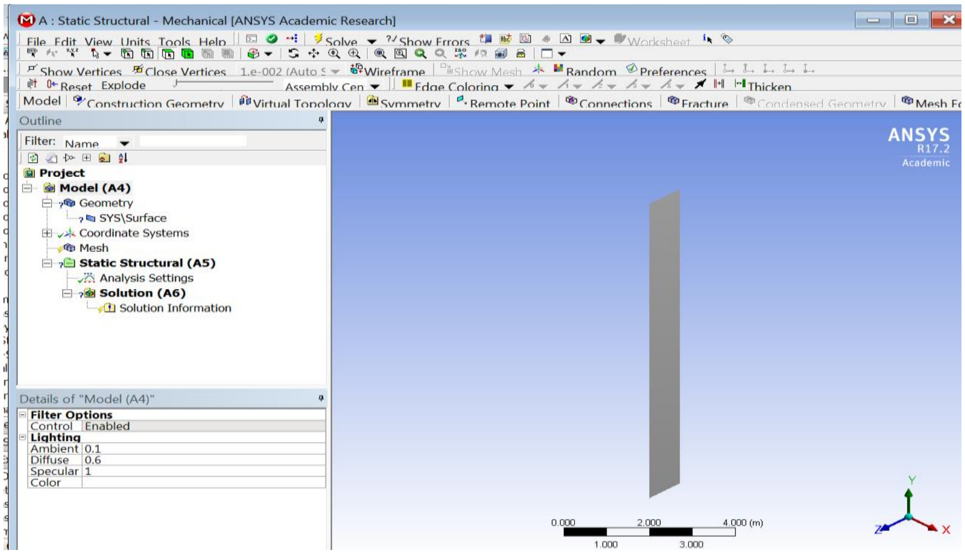
1) Return to the ANSYS window, and click on the Model feature in the Project window.



Static Structural (ANSYS)

This will start the ANSYS ‘Mechanical’ program, to setup the actual finite element model.

2) The Mechanical window shows the 1 component.
At first the model is shown with no mesh or loads yet. On the left is a list of the model features that have to be set. By default, the material to be used will be structural steel.



Note:

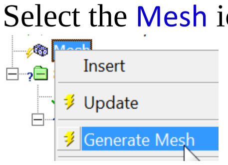
A green checkmark means that everything is OK

A yellow lightning bolt means that something hasn't been done, but its ready to be done.

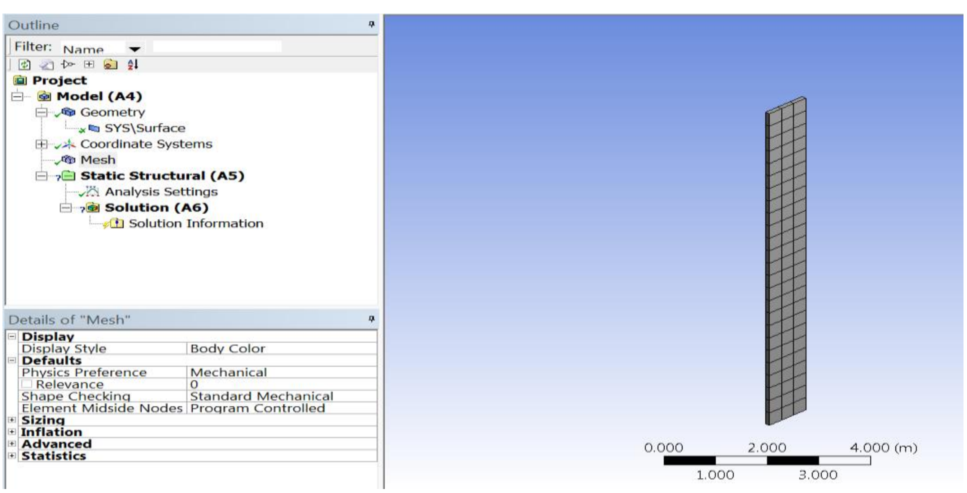
A question mark means that there is something missing, or not yet set. ANSYS can't solve the model if there are any question marks.

Edit the surface thickness to 0.1m. Now the Geometry should have all green check marks.

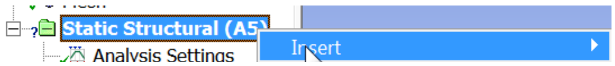
Select the Mesh icon in the Project and right-click Generate Mesh.

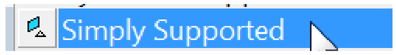


The mesh on the body is ;

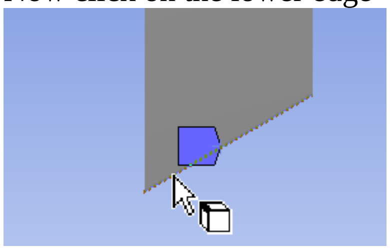


3) Now we will set the applied load and support conditions on the bar.
First, we pin the lower edge (line). Right-click on static structural, select Insert

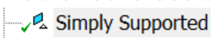


and select Simply Supported 

Now click on the lower edge

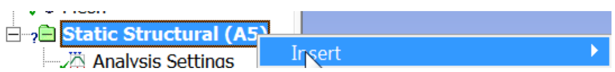


Now click Apply in the panel on the left that lists Details of Simply Supported.

You should see the face highlighted in purple and a green check by simply supported 

under Static Structural.

Right-click on static structural, select Insert



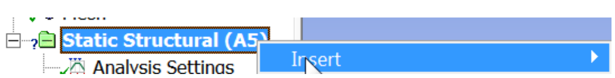
and select Remote Displacement, and add these conditions to the top of the beam;

Details of "Remote Displacement"

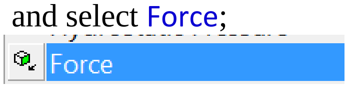
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Coordinate System	Global Coordinate System
X Coordinate	1. m
Y Coordinate	10. m
Z Coordinate	-0.5 m
Location	Click to Change
Definition	
Type	Remote Displacement
X Component	0. m (ramped)
Y Component	Free
Z Component	0. m (ramped)
Rotation X	0. ° (ramped)
Rotation Y	Free
Rotation Z	Free
Suppressed	No
Behavior	Deformable
Advanced	

This will allow the beam to compress but still act as a pin.

Right-click on static structural, select Insert

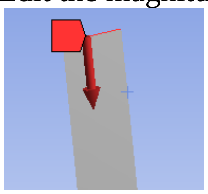


and select Force;



Now select the top of the bar
Now click Apply in the panel on the left that lists Details of Force.

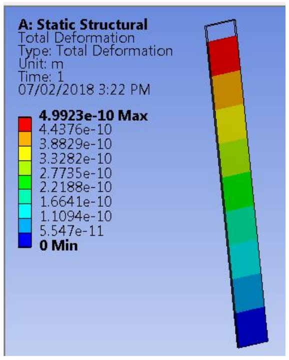
Edit the magnitude to be 1 N and have its direction downward (compression).

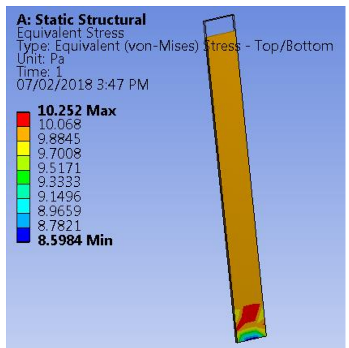


Now you can 

the system/

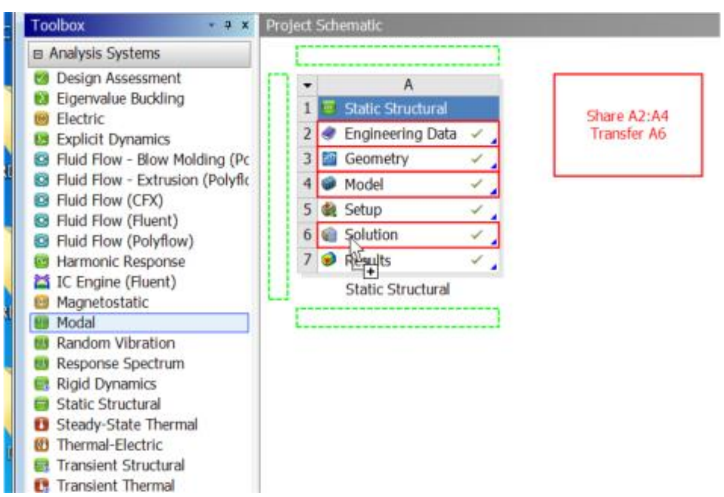
4) To specify output, right click on Solution in the tree, and select Insert, then Stress, then Equivalent Stress. Do the same to select Total Deformation.

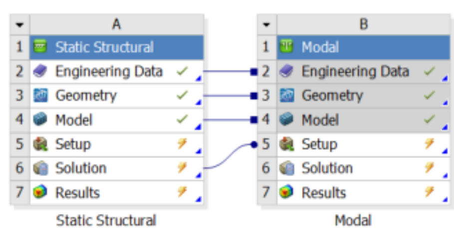




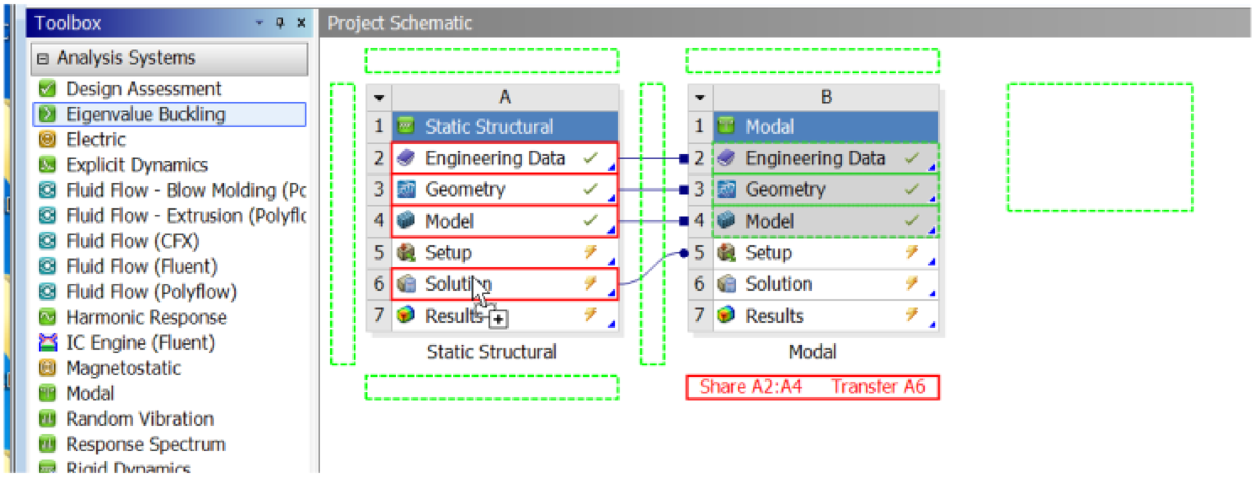
Step6: Add the Vibration (modal) and Buckling Analyses

1) The left-hand window shows a set of analysis type options. Select **Modal** and drag the icon to the right, on top of the **Solution** component of the **Static Structural** analysis.

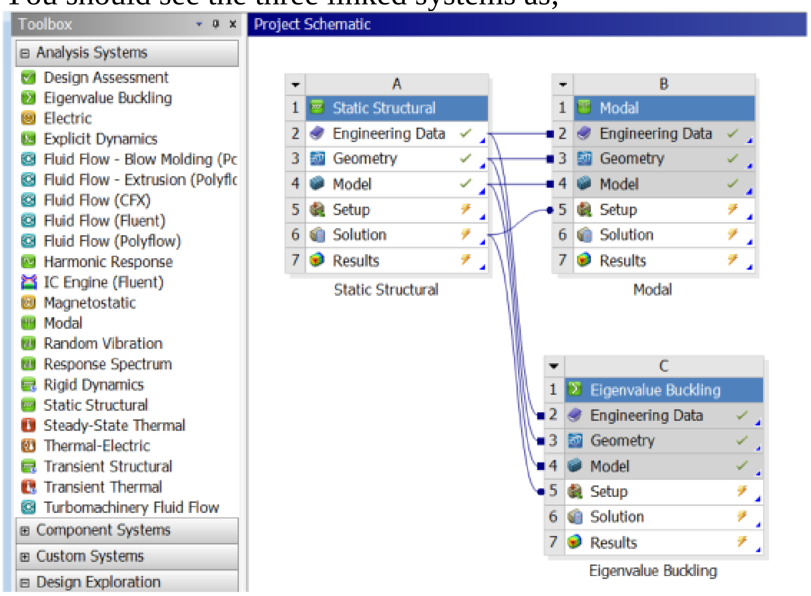




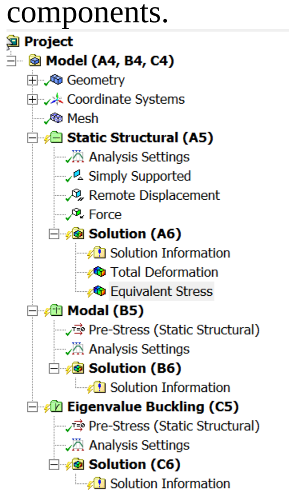
Now drag the **Eigenvalue Buckling** system on to the **Static Structural Solution**,



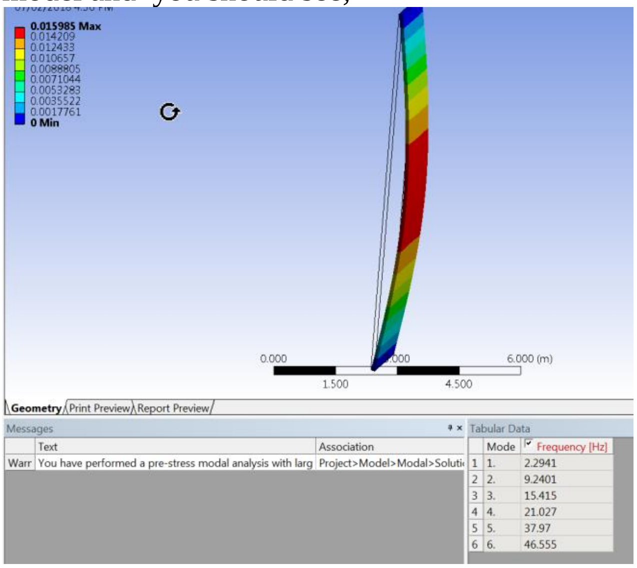
You should see the three linked systems as;



2) Now if you re-open the ANSYS **Mechanical** you will see additional analysis components.

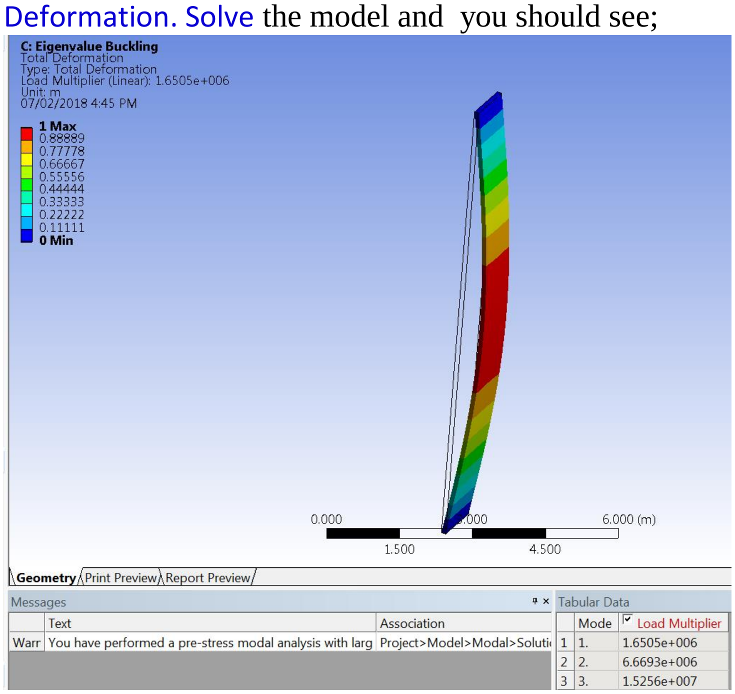


Now click on Solution under the Modal analysis and insert Total Deformation. Solve the model and you should see;



We expected frequencies of 2.29, 9.16 and 20.6. What is this 15.41? If you select the 3rd mode (in Details) and retrieve the result, you will see its actually a torsional mode. How can you prevent this mode?

Now under Analysis Settings for Eigenvalue Buckling, change the number of modes to find to 3. In Solution under the Eigenvalue Buckling analysis and insert Total Deformation. Solve the model and you should see;



Because we have applied a 1N load, the load multiplier to get to buckling is equivalent to out computed load. Now explore what happens if you apply a load of 1.65E6 N in Static structural and then examine the modal and buckling analyses.

Self Study Exercises: Student:_______________

For each of these exercises, show the instructor your results and make sure that it is recorded that you have completed the exercises.

Exercise #1 – Make a model of a guitar string vibrating at middle C (261.h hz). You choose the sizes etc.
A string of length *l* (m) and mass per unit length μ (kg/m), and tension T (N) has a frequency of;

$$v_f = \frac{1}{2l}\sqrt{\frac{T}{\mu}}$$

Ex#1
Initials of Instructor

____

Exercise #2 – What if the natural frequency of vibration of a 2m x 0.4m x 12mm steel plate? Assume pinned edges.

Ex#2
Initials of Instructor

____