

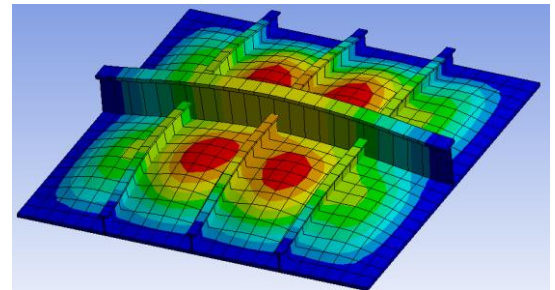
Lab#3

Beam modelling in ANSYS

By C. Daley

Overview

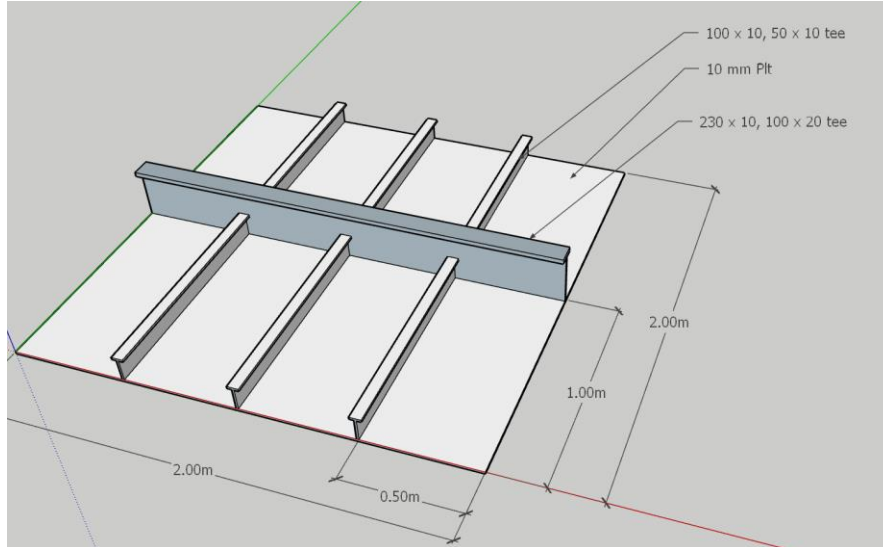
In this lab we will model a simple framework using beam elements. Once again, we will use [SpaceClaim](#) to create the geometry model of the problem, and then use [ANSYS](#) to model the structural behavior. We will explore some more features of both [SpaceClaim](#) and [ANSYS](#).



ANSYS Model #3 – T Frame Grillage

Step 1: describe and sketch the problem:

The aim is to model a simple 2m x 2m grillage with one central stringer and 4 transverse frames as shown;



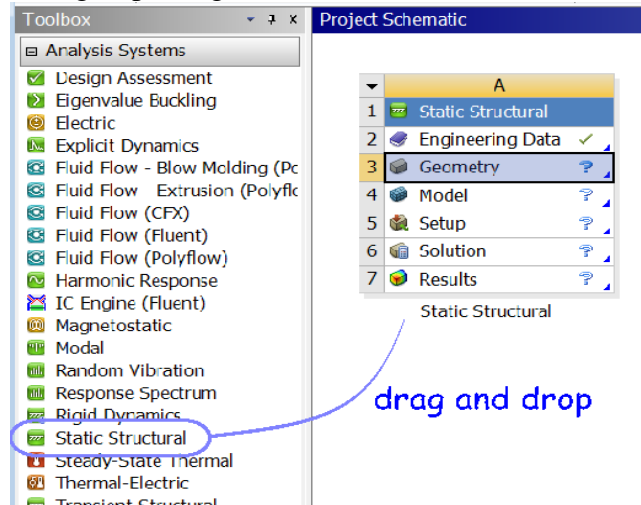
Apply a 100 kPa pressure to the whole grillage

Step 2: estimate expected results (analytically):

Not sure yet. My wild guesstimate is that this will cause a deflection is say 1mm and stresses less than half yield. No promises.

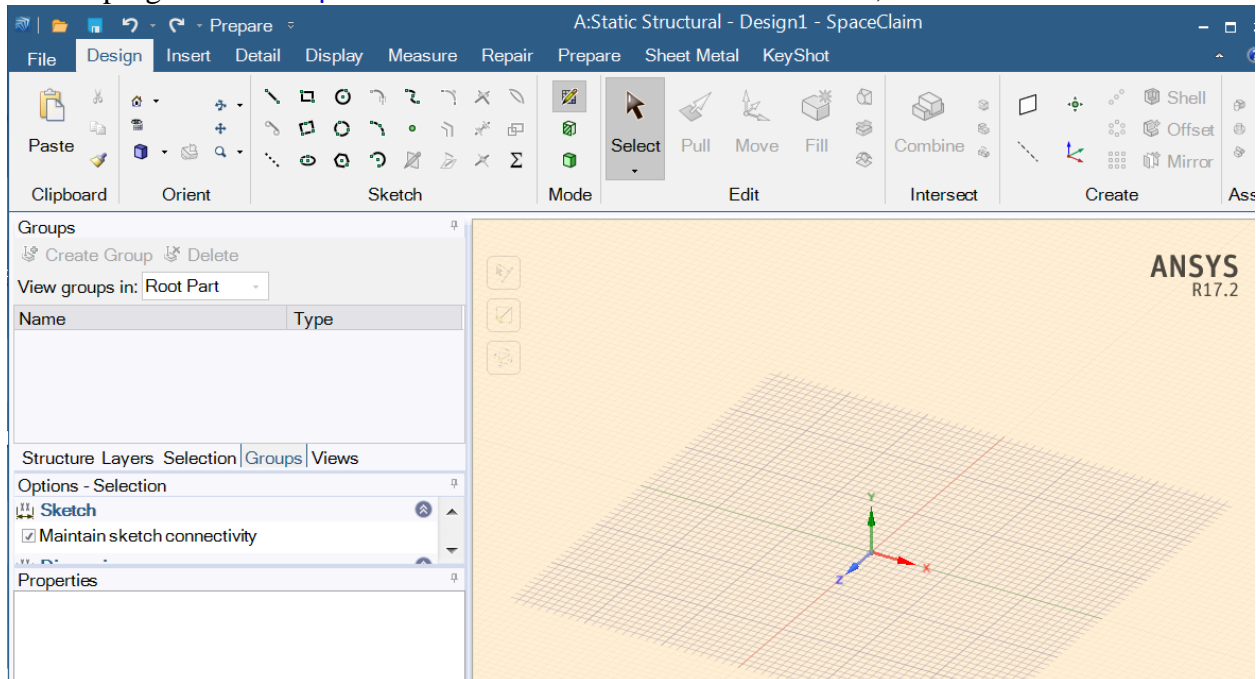
Step 3: open ANSYS Workbench 17.0

- 1) First, save the (empty) project as **Beam1.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.

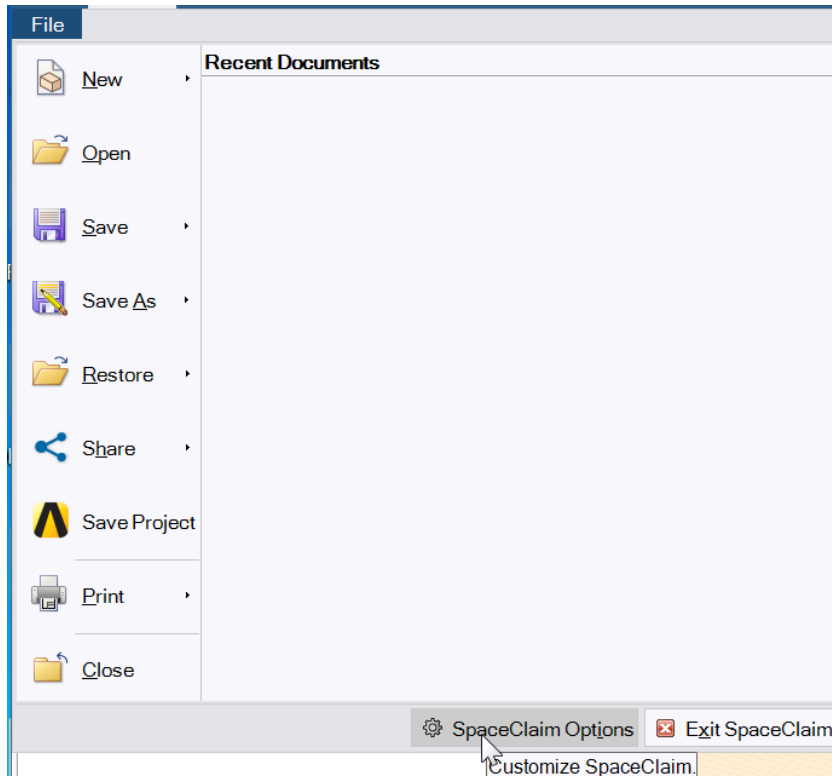


Step4: open Geometry and create the CAD model

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

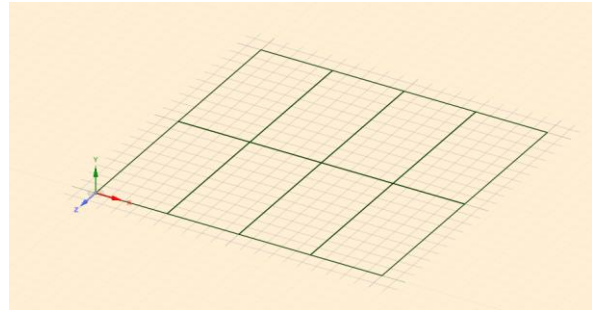
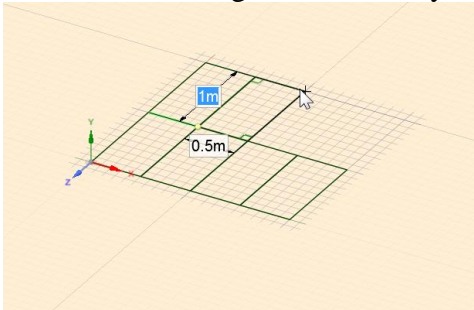


- 2) If you want to change options or units, you can go to the file tab and select SpaceClaim Options. This will let you change Units (mm are default) and many other options. **Change to meters.**



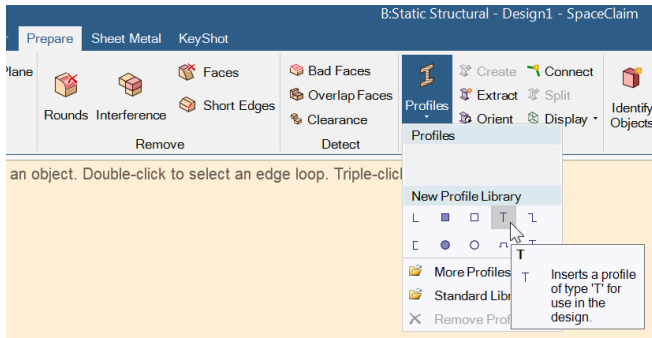
You now see the main window where the CAD model will be displayed, just as in Lab#1.

- 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 the 8 rectangles, each 1m by 0.5m.

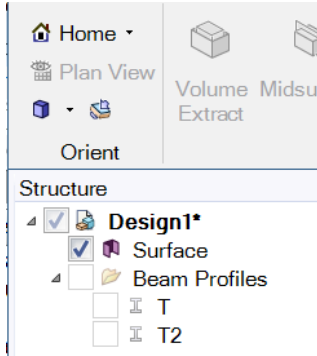


This will create 8 panels, which will eventually be the grillage shell, modelled with shell elements.

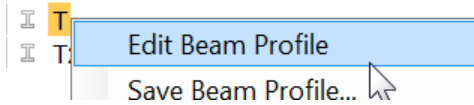
- 5) Create 2 tee frame profiles.
In the Prepare tab, select the beam profile and a tee section. Repeat.



You should now see a T and a T2 under Structure on the left window.

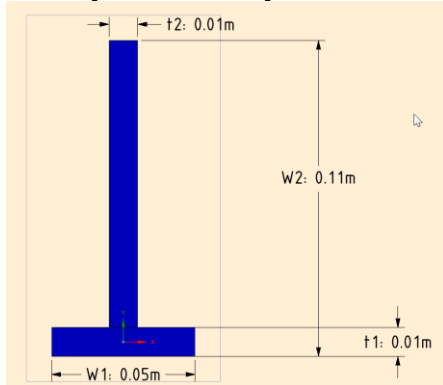


Select the T and click on Edit Beam profile.

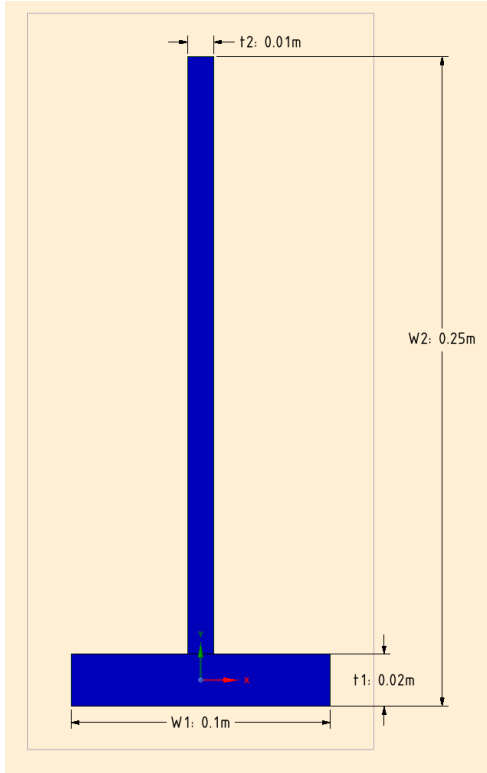


Edit the dimensions as follows $W1=0.050$, $W2=0.110$, $t1=0.010$, $t2=0.010$.

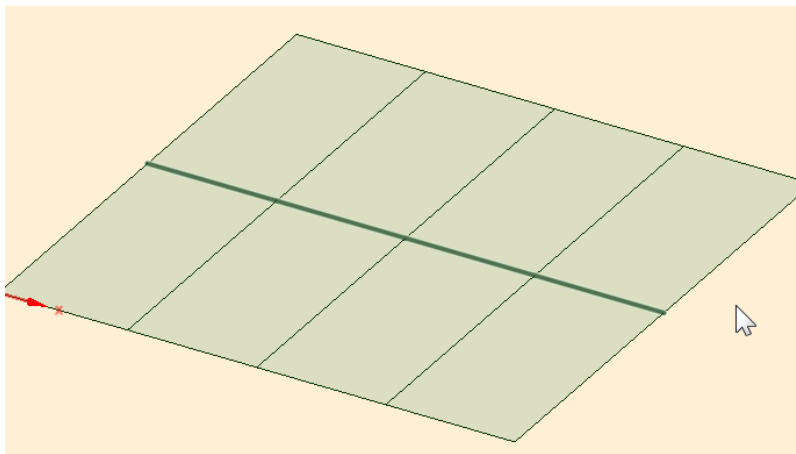
When you are done you should see;



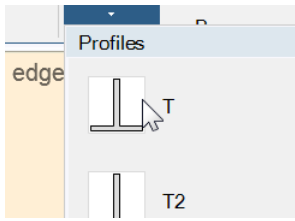
For the second beam (the stringer) you should see;



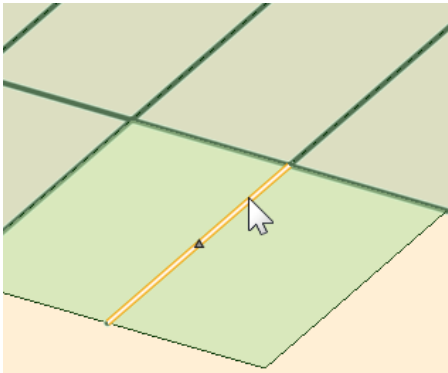
- 6) Apply the frames to the appropriate lines.
To do this, you need to use Create, in the Beams part of the Prepare tab. With T2 showing, click on Create, then select the lines across the middle of the grillage. You have to make sure the line is highlighted and double click. When done you should see;



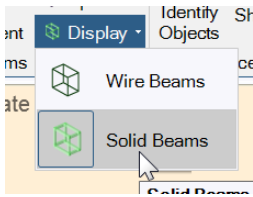
Now click on the frame type and select T.



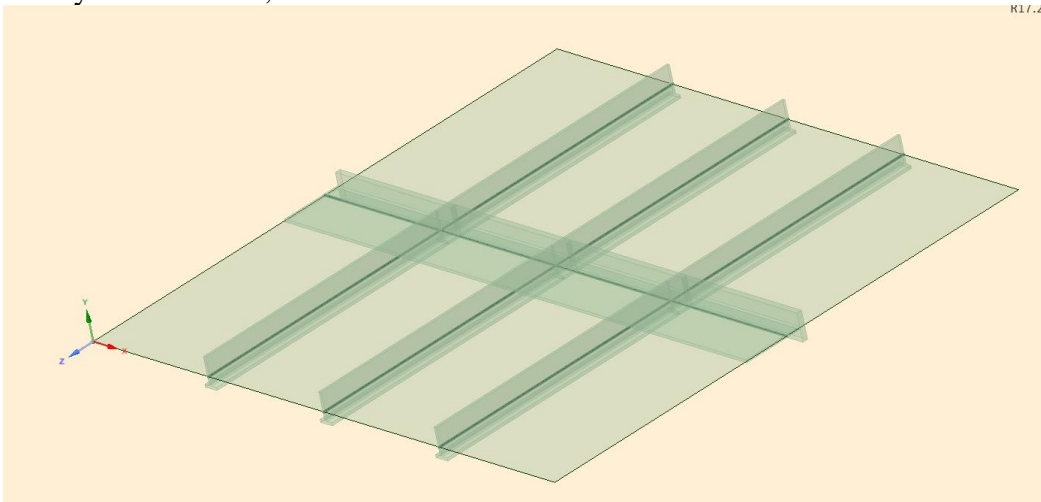
Now with Create still selected, select all the other (6) frame lines.



You can click on Display and select Solid Beams.



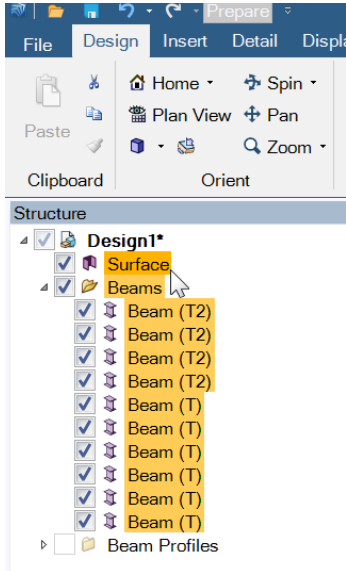
Now you should see;



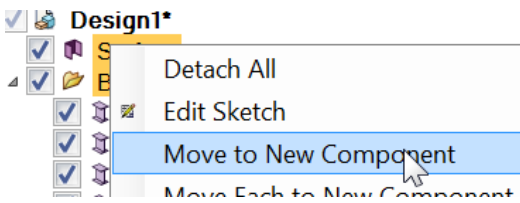
Note that the frames are not quite where they need to be. The frames are attached to the plate edges along their neutral axes, rather than at their ends as we want. Also the stringer is sideways. We will use Orient to fix all this.

7) Create a single component.

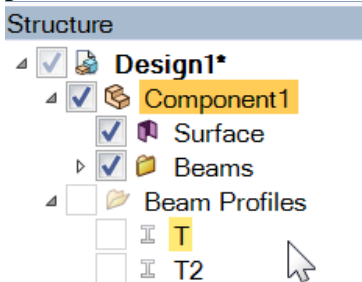
Before we edit our parts, we will merge everything into a single component, so that it all becomes a single weldment. Go to the design tab, select the Select arrow, and in the structure tree, select the Surface and all the Beams. It should look like;



Now right click and select Move to new Component.



The structure tree should show the new component and show the Surface and beams as parts ;

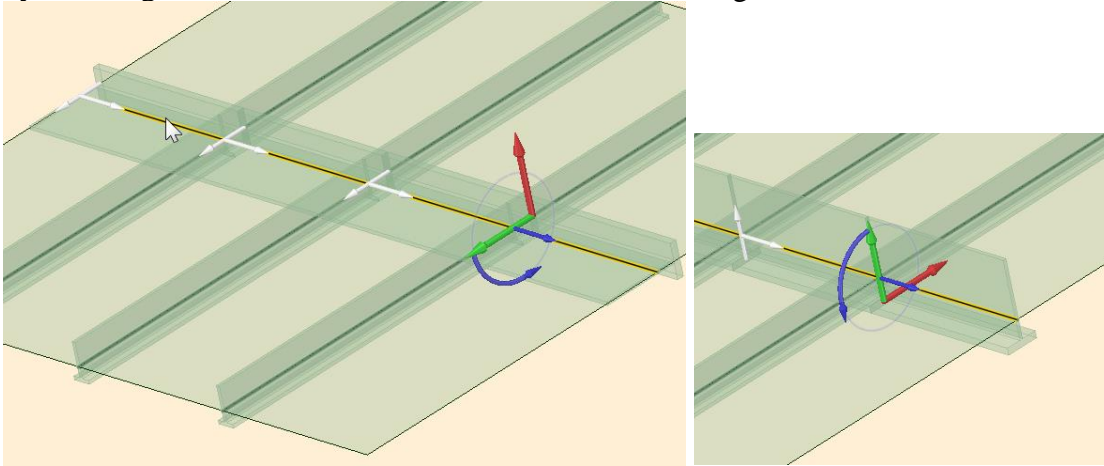


Down below, under the properties of the Component, Change the option of the Shared Topology to Merge. This will make everything a single weldment.

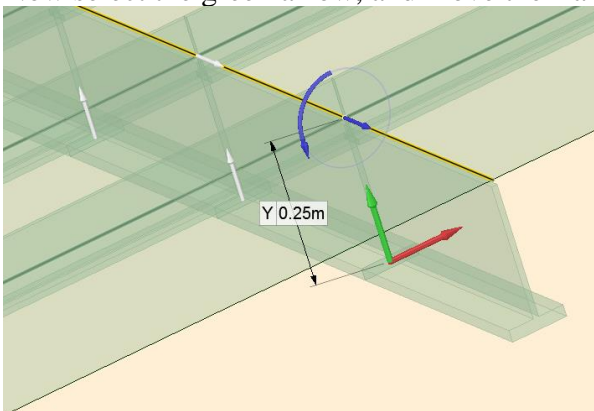
Properties	
4 Analysis	
Share Topology	None
4 File	
Category	Group
Content Status	Share
Content Type	Merge
Created	24/01/2019 10:30

8) Re Orient Frames.

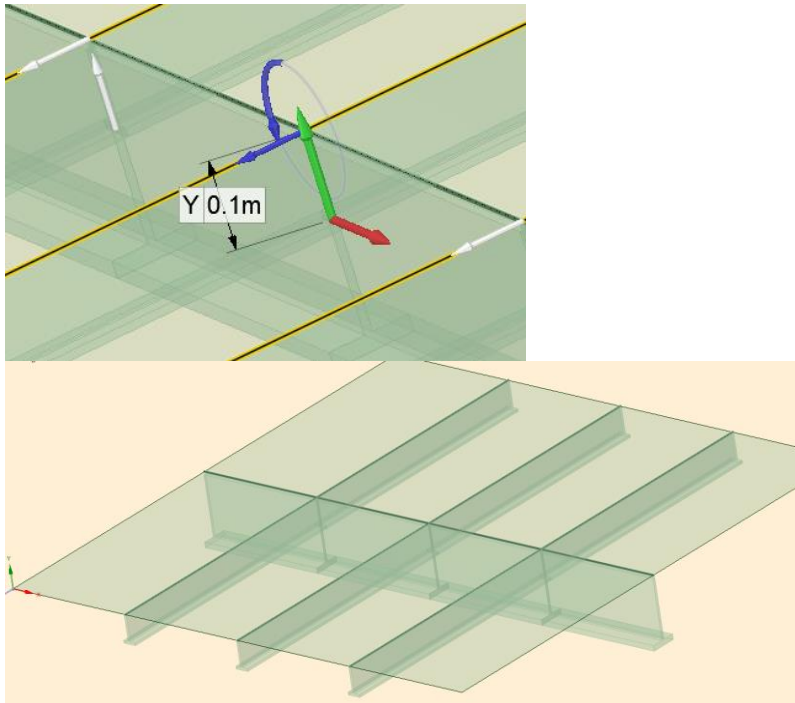
Return to Prepare. Select Orient in the Beams area. Select the 4 lines of the stringer, and by selecting the blue roll vector, rotate the frames 270 deg.



Now select the green arrow, and move the frame 0.18314m - you should see



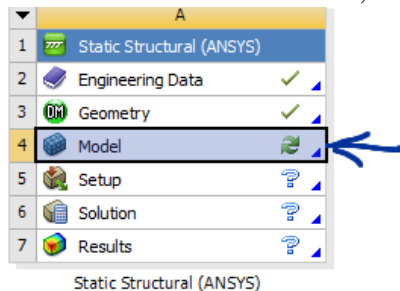
Now select all the main frames, the green arrow, and move the frames 0.06833m - you should see;



This completes all work in [SpaceClaim](#).

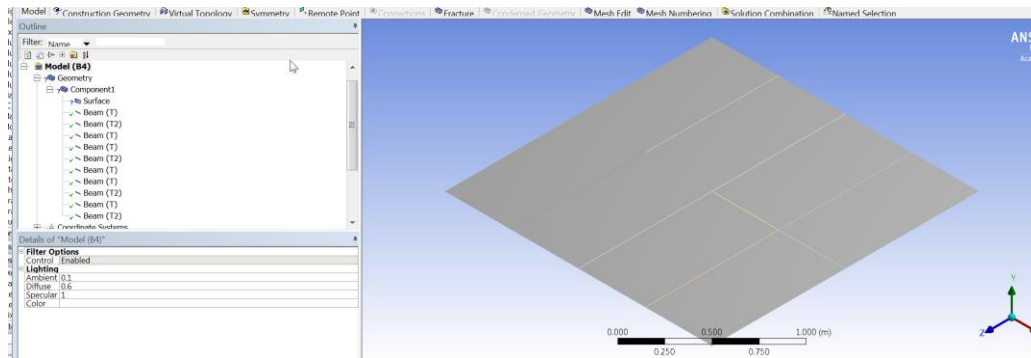
Step5: open Model and create the Finite Element model

- 1) Return to the ANSYS window, and click on the [Model](#) feature in the [Project](#) window.





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:

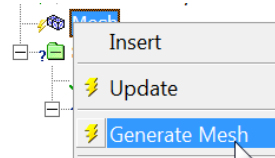
A green checkmark  means that everything is OK

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

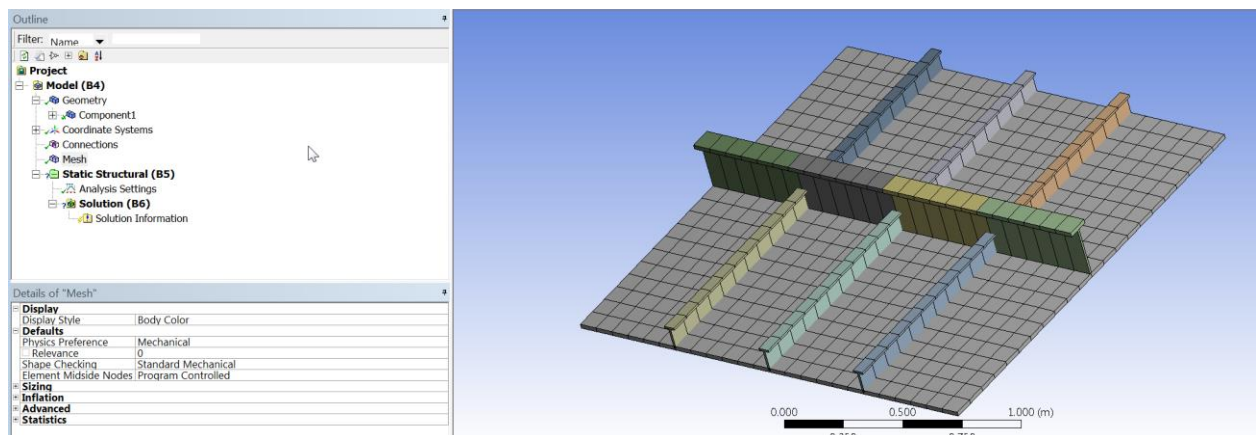
A question mark  means that there is something missing, or not yet set. ANSYS can't solve the model if there are any question marks.

Edit the surface thickness to 0.02m. Now the Geometry should have all green check marks.

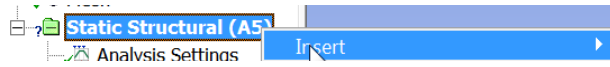
Select the **Mesh** icon in the Project and right-click **Generate Mesh**.



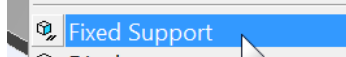
The mesh on the body is ;



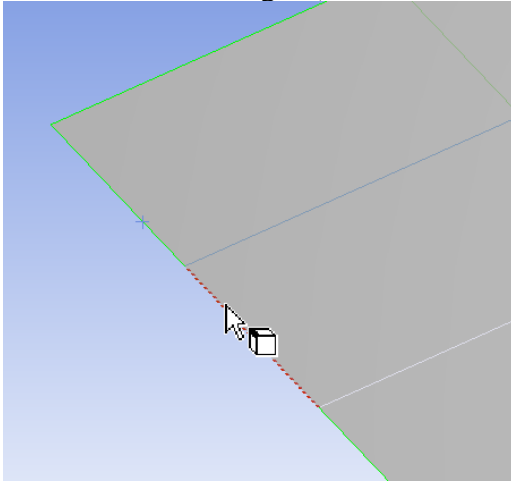
- 3) Now we will set the applied load and support condition on the grillage.
First we fix all the plate edges (lines). This will also fix all the beam ends. Right-click on **static structural**, select **Insert**



and select **Fixed Support**;



Now click on the edges (ensure that lines are selected)



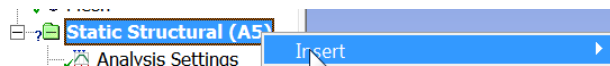
Add a second fixed support and

Now click **Apply** in the panel on the left that lists **Details of Fixed Support**.

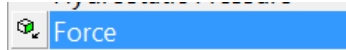
You should see the face highlighted in purple and a green check by fixed support

 **Fixed Support** under Outline.

Right-click on **static structural**, select **Insert**



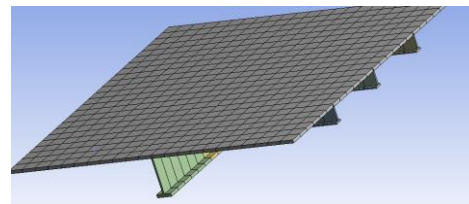
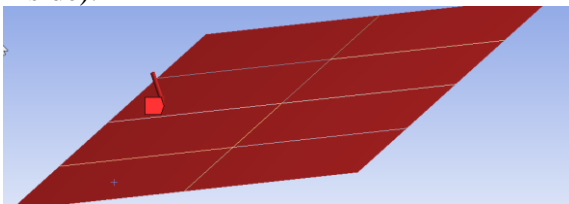
and select **Pressure**;



Now select the top face of the shell

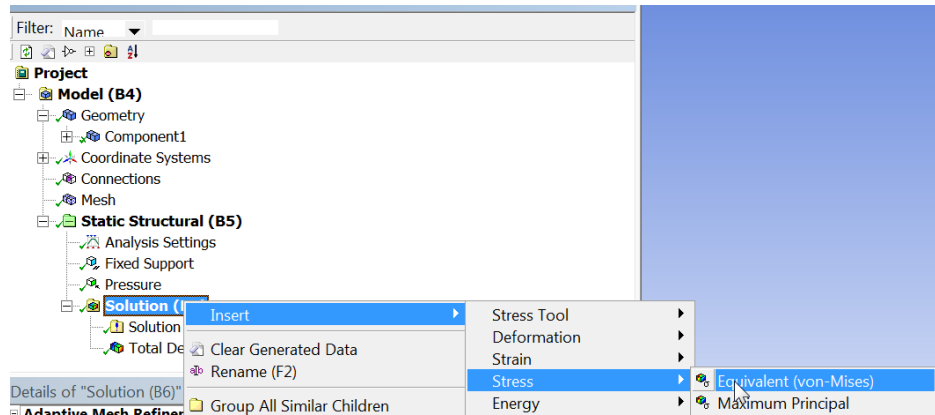
Now click **Apply** in the panel on the left that lists **Details of Pressure**.

Edit the magnitude to be 100,000 Pa and have its direction towards the frames (towards inside).

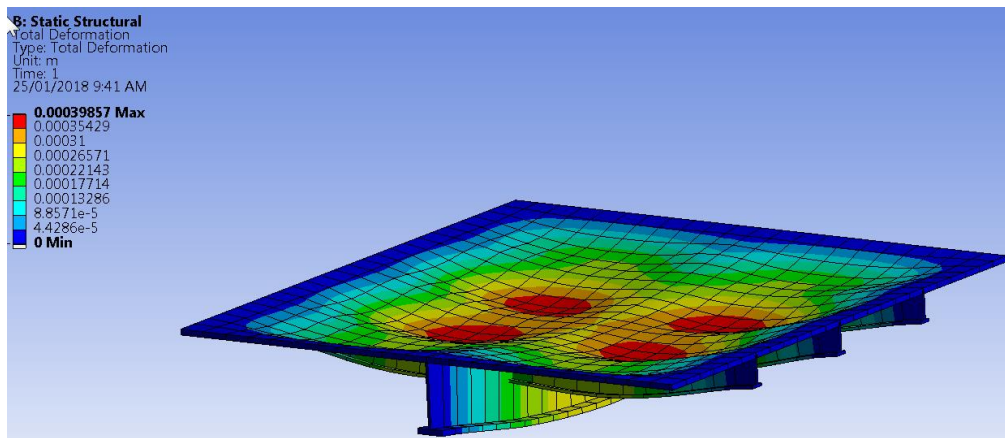


Now you can  **Solve** the system/

- 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**.



- 5) When you select the Equivalent Stress under Solution in the tree, the von-Mises equivalent stresses will be plotted on the deformed shape. The maximum stress in the plate is 2.3032×10^7 , which is in Pa. This is 23 MPa, and is even lower my simple guesstimate of half yield. If you click on Total Deformation, it shows a max value of $.00039857$, or 0.4 mm, compared with my guesstimate of about 1 mm. These values are close-ish to the guesstimates.



Getting the bending stress in the frames is a bit harder. Max stringer BM is 7542 Nm, with max frame BM of 1788 Nm. Unfortunately, these frames are not just simple beams. They act with the shell plate, which acts as a flange, changing their properties.

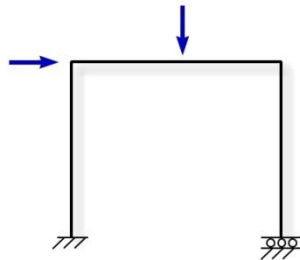
Self Study Exercises:

Student: _____

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

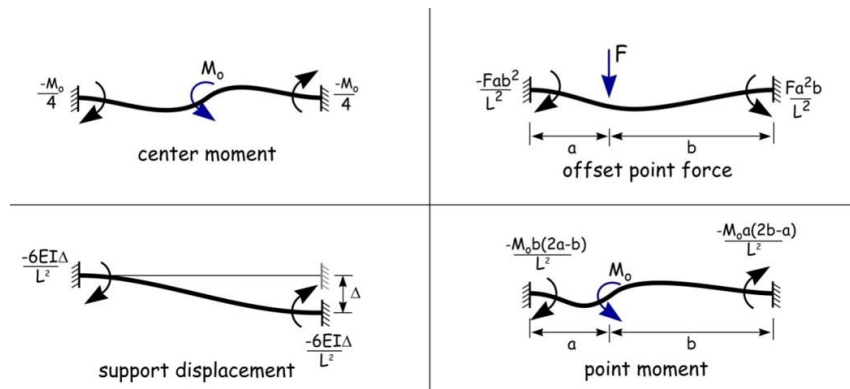
Exercise #1 – Make a simple frame model with I beams, similar to the problem below. You choose the sizes etc.

problem with loads along members



Ex#1
Initials of Instructor

Exercise #2 – Validate one of the cases shown, using a beam element model.



Ex#2
Initials of Instructor
