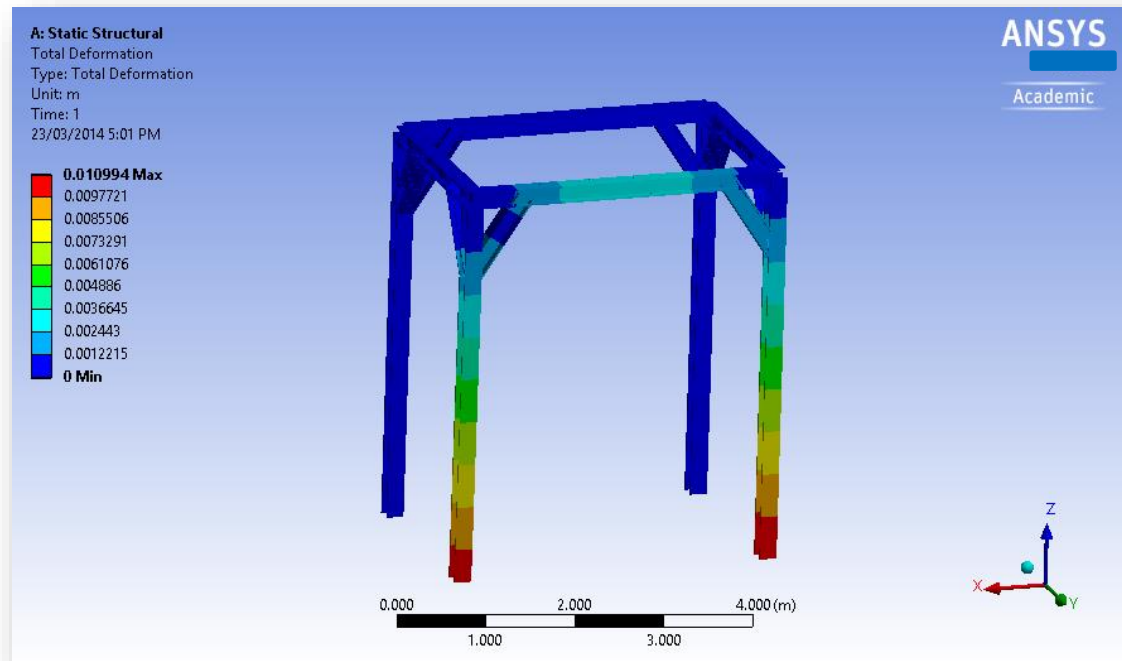


# Crane Tutorial



Mechanical Systems and Design

ENGN2217

Semester 1, 2021

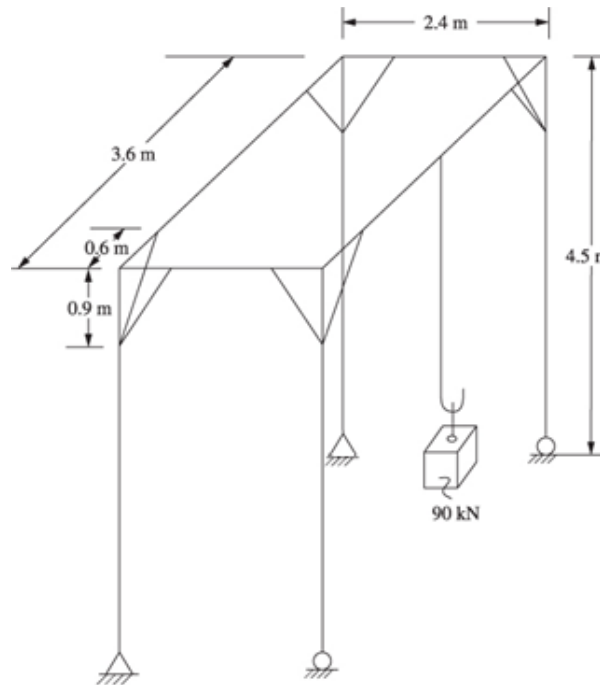
ANSYS v2020R2 – Windows 10

# Design, analysis and optimisation of a beam structure using finite element analysis (FEA)

- a) Create a space frame with line elements
- b) Create a new structural design simulation
- c) Create the geometry of the structure
- d) Assign cross sections to each line body
- e) Generate mesh and specify material data
- f) Apply loads and boundary conditions
- g) Solve the problem
- h) Perform post processing, obtain relevant structural performance indicators
- i) Optimising design using mechanics of materials principles
- j) Buckling analysis
- k) conclusion

**a) Create a space frame with line elements**

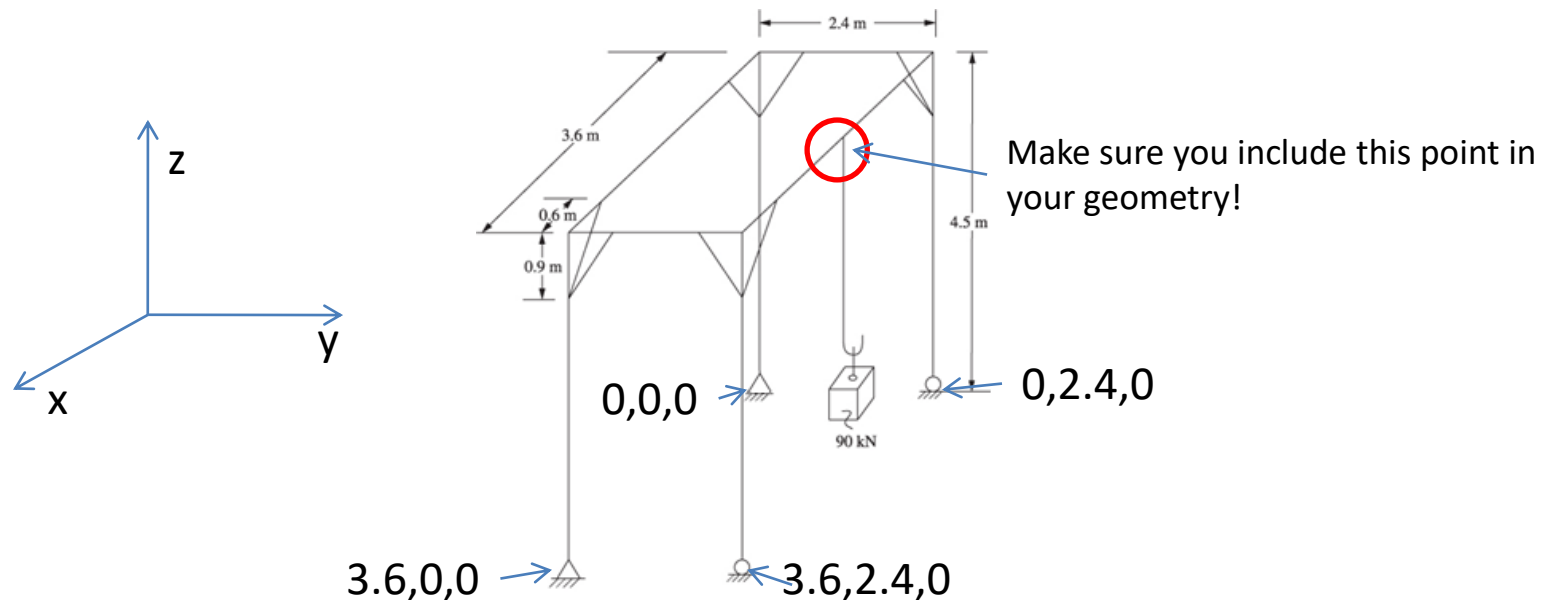
The objective of this tutorial is to perform a finite element analysis on the structure shown below:



This is a 3D structure, and each structure member is considered as a long and slender member. Therefore, in FEA, each structural member can numerically be approximated by a 1 dimensional (1D) line element.

### a) Create a space frame with line elements

First step: Create a set of points in ANSYS that represent the geometry of the crane. These points correspond to the joints of the structure including corners and points where the load is applied. Some points of interest are illustrated below:

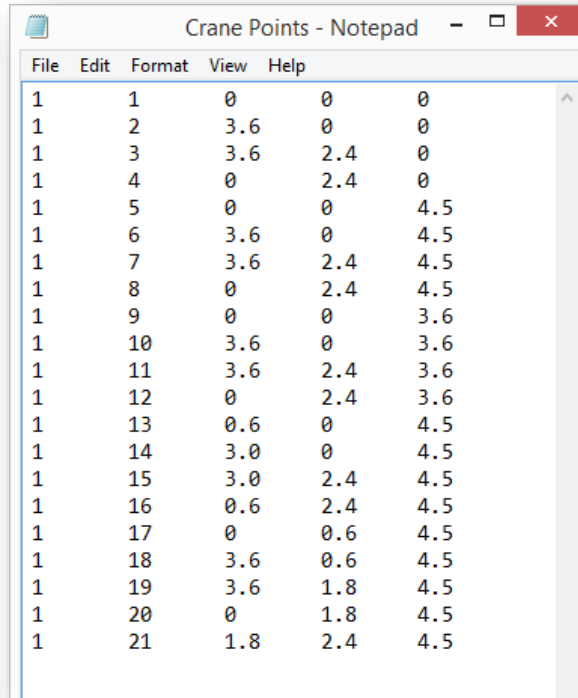


When drawing a structure from a set of given measurements, you will need to make an arbitrary choice about its positioning in space. You will choose an axis convention, and a reference point. Above, we have placed the far-left leg of the crane on the origin, i.e.,  $(x,y,z) = (0,0,0)$ , and defined all other points as adopting positively-signed coordinates.

### a) Create a points list for your structure

An easy way to create a structure is to feed ANSYS with a list of point coordinates for your design. Writing this up is a bit boring, but it ensure maximum accuracy in your points location.

The Import feature of ANSYS expects a tab-separated ASCII file such as the below. Additional instructions are provided in the next page.



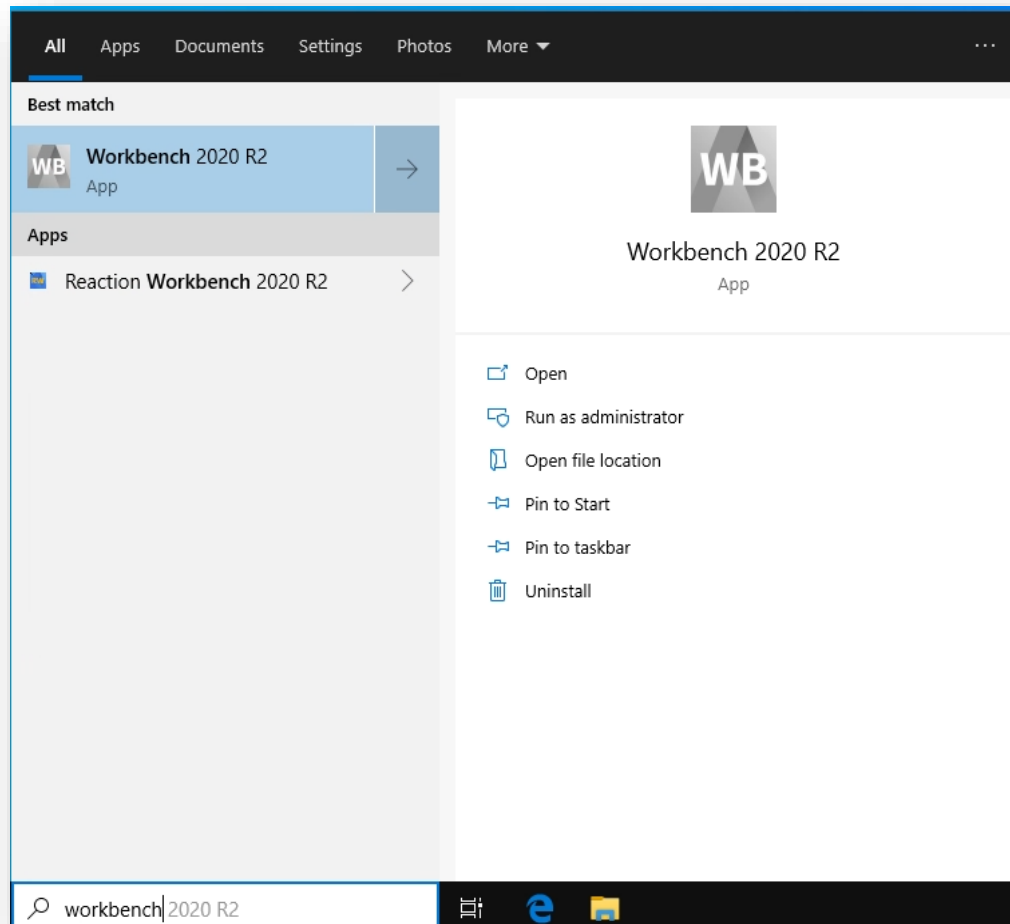
File	Edit	Format	View	Help
1	1	0	0	0
1	2	3.6	0	0
1	3	3.6	2.4	0
1	4	0	2.4	0
1	5	0	0	4.5
1	6	3.6	0	4.5
1	7	3.6	2.4	4.5
1	8	0	2.4	4.5
1	9	0	0	3.6
1	10	3.6	0	3.6
1	11	3.6	2.4	3.6
1	12	0	2.4	3.6
1	13	0.6	0	4.5
1	14	3.0	0	4.5
1	15	3.0	2.4	4.5
1	16	0.6	2.4	4.5
1	17	0	0.6	4.5
1	18	3.6	0.6	4.5
1	19	3.6	1.8	4.5
1	20	0	1.8	4.5
1	21	1.8	2.4	4.5

### **a) Create a points list for your structure**

- Create a text file (with the extension of .txt) using notepad.
- First type '1' into notepad, then press 'Tab' key (important!) on the keyboard to move to a new column.
- Every line of the file contains information about one point. The format for each line is as follows:
  - the first column ALWAYS contain 1,
  - the second column defines the point number, i.e. point 1, point 2, etc.
  - Columns 3, 4, and 5 define the x, y, z coordinates of the points.
- There are 21 points in total, representing 4 supports at the base, 4 double-hinged joints midway and 12 single-hinged joints on the top frame, plus the point where the force is applied (see previous slide, this point is circled in red).
- Save the .txt file, make sure you know the directory path where it is saved.

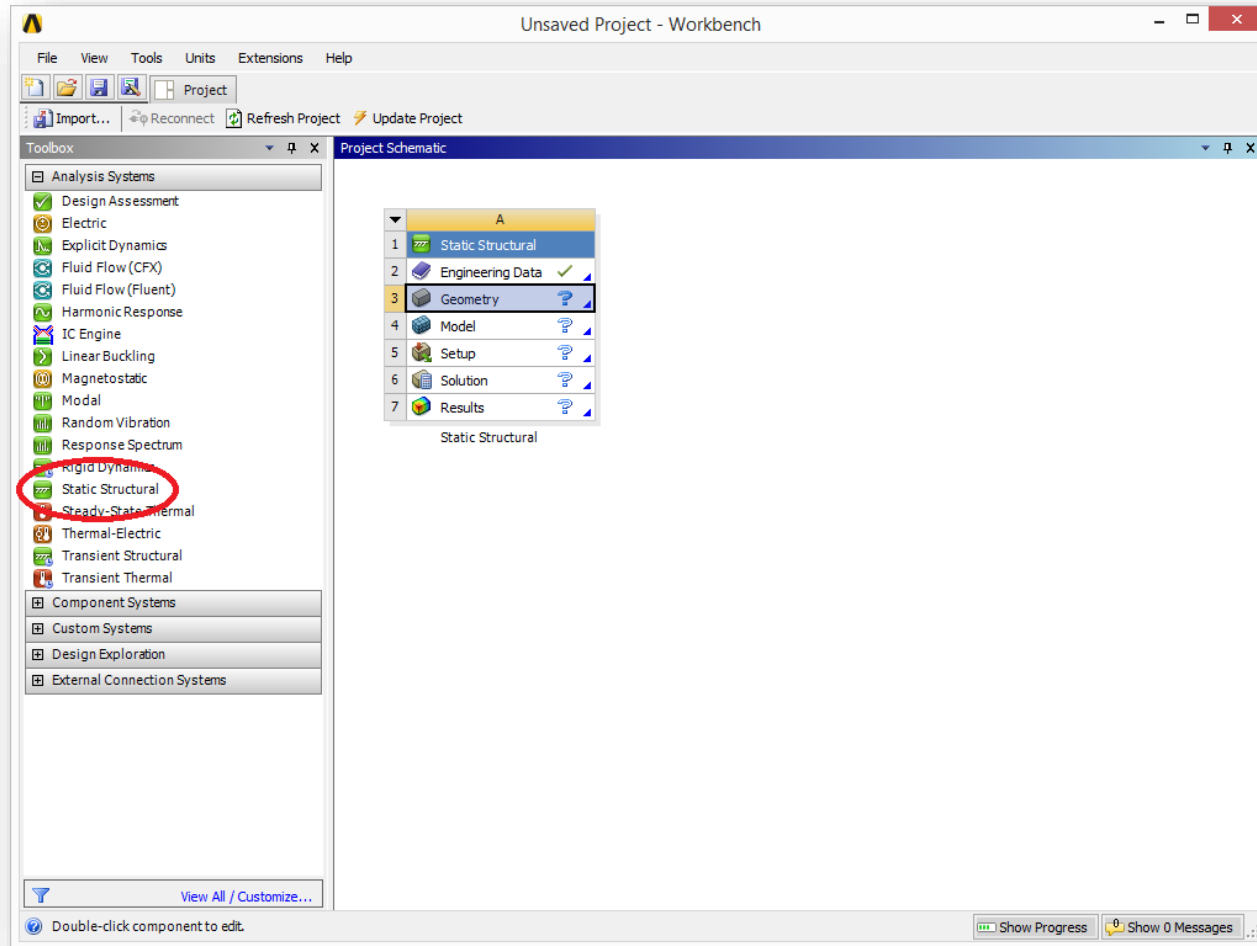
## b) Create a new structural design simulation

Open 'Workbench 2020 R2 '



## b) Create a new structural design simulation

Double click on **Static Structural**.

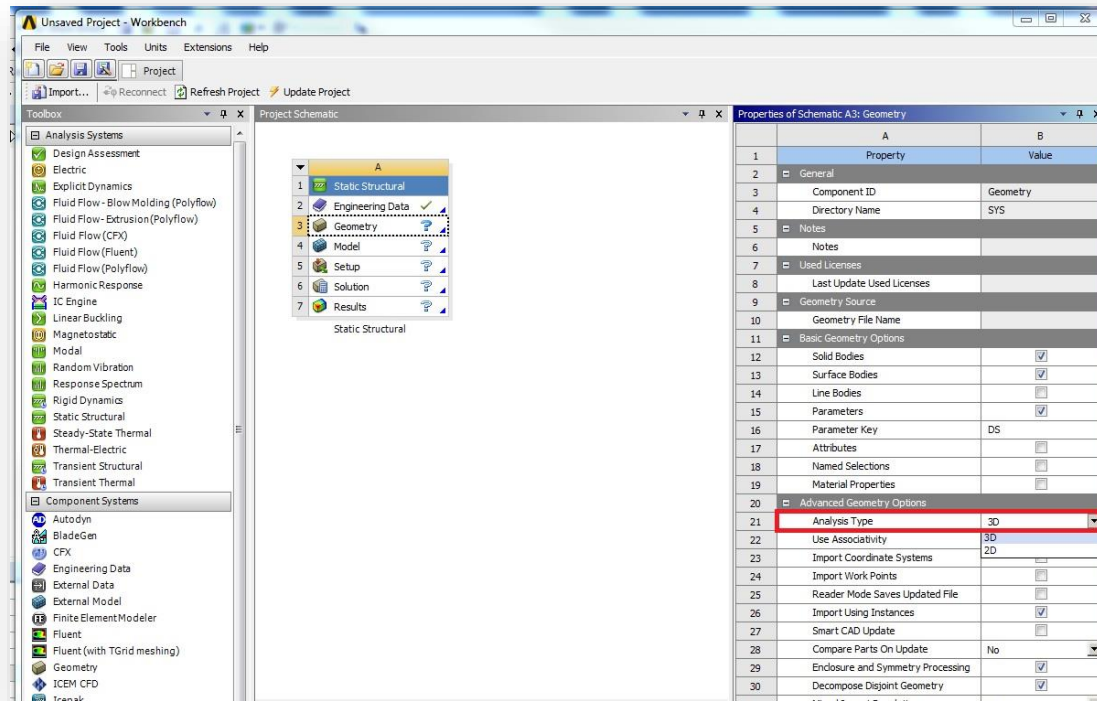




## b) Create a new structural design simulation

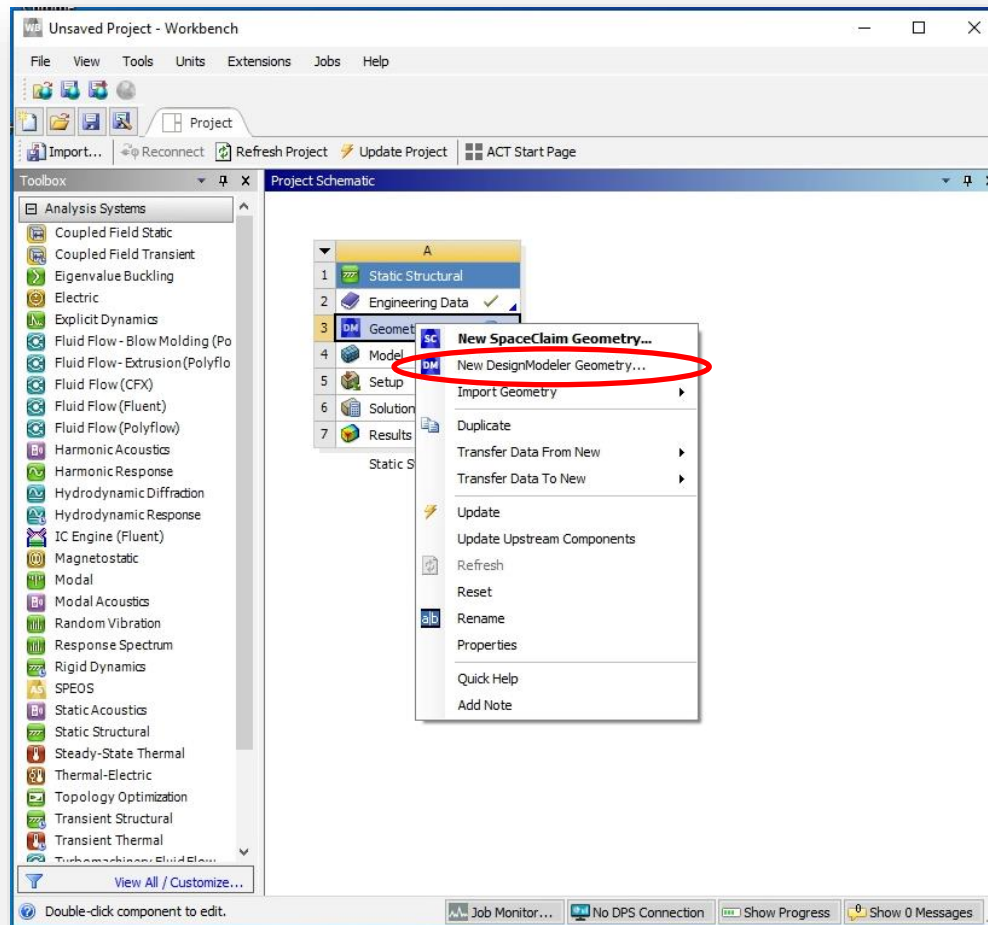
The next important step is to specify whether the analysis is 2D or 3D:

1. Right click on **Geometry** and choose **Properties**.
2. A **Properties of Schematic** pane will appear on the right side of the workbench.
3. In **Advanced Geometry Options -> Analysis type** (red box below) you can specify 2D or 3D.
4. The default setting is **2D**. For the current spaceframe you need to change it to **3D**.



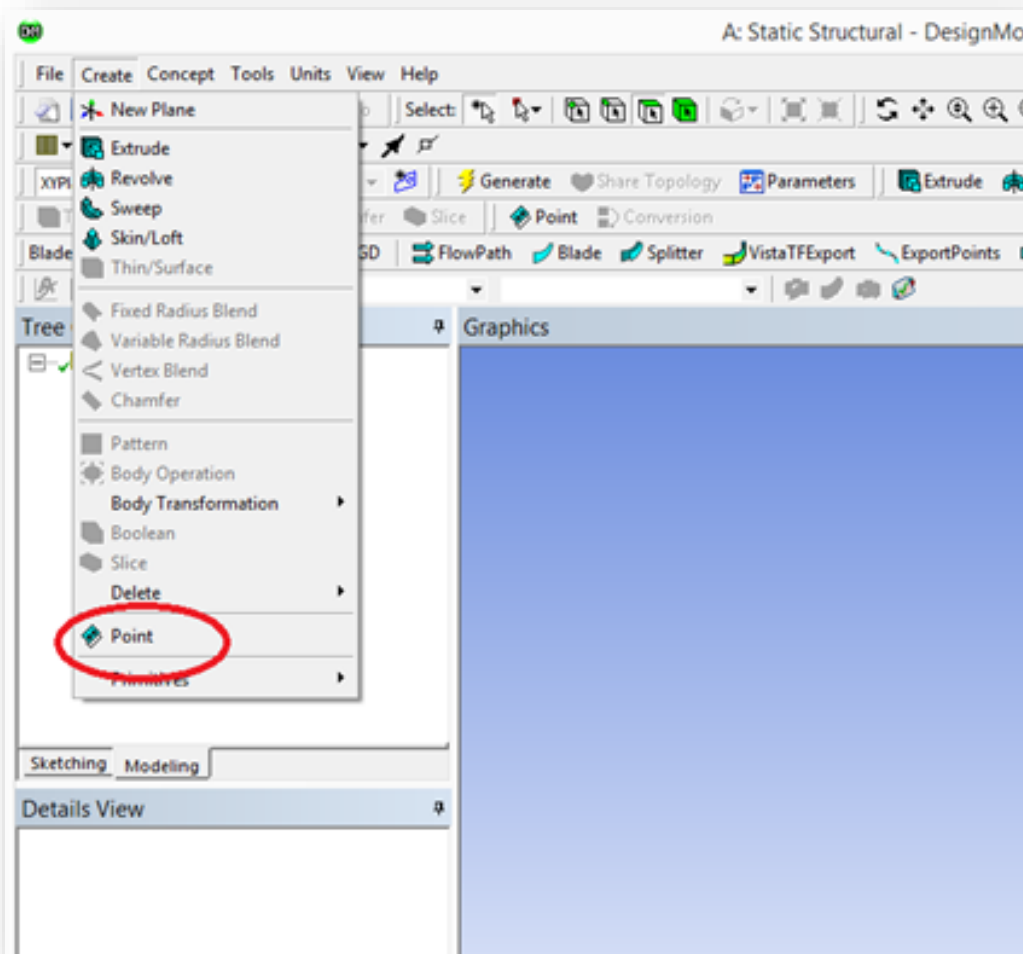
### c) Create the geometry of the structure

Right click on **Geometry** and choose 'New DesignModeler Geometry'



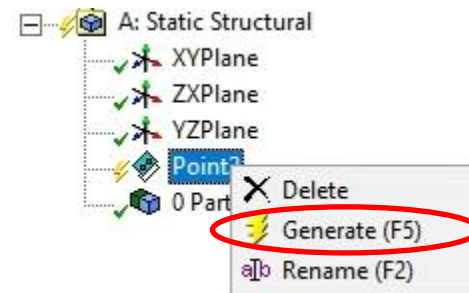
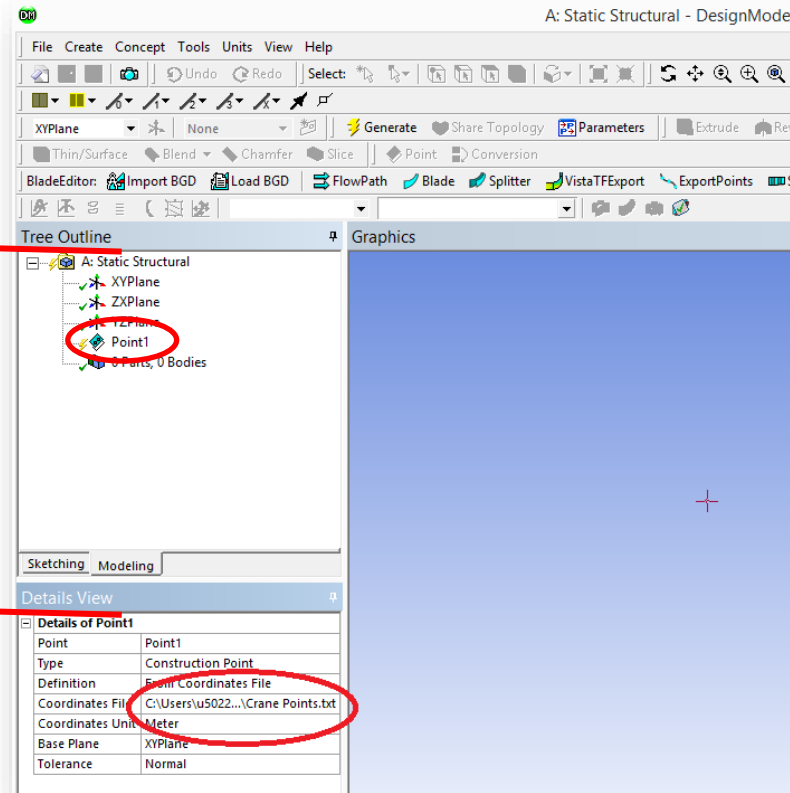
### c) Create the geometry of the structure

On the top right menu, click **Create**, then in the drop down menu click on **Point**.

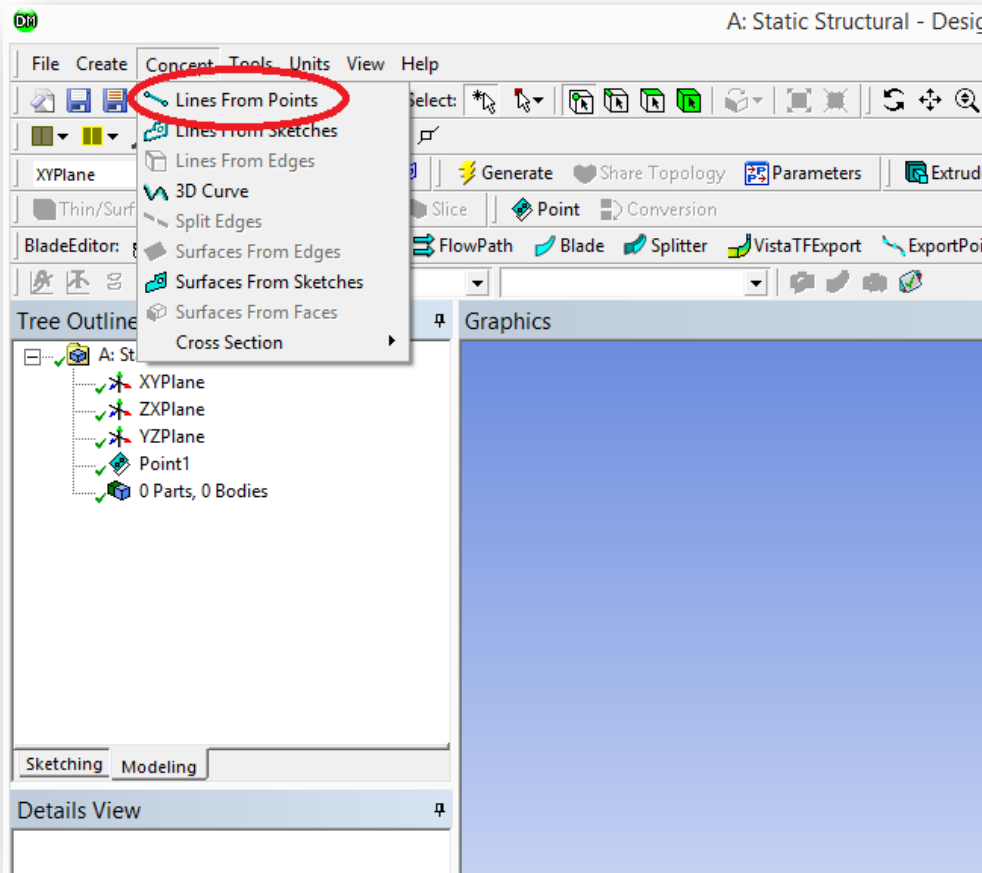


## c) Create the geometry of the structure

1. You will see a new item (Point 1) appear in the **Tree Outline** pane.
2. Under **Details View**, there is an entry dialogue box that asks for a **Coordinates File**.
3. Edit the path shown to point ANSYS to the file '**Crane Points.txt**' (created by you or downloaded from Wattle).
4. Finally, **right click on 'Point 1'** and click **Generate** to create all 24 points from the coordinates file.



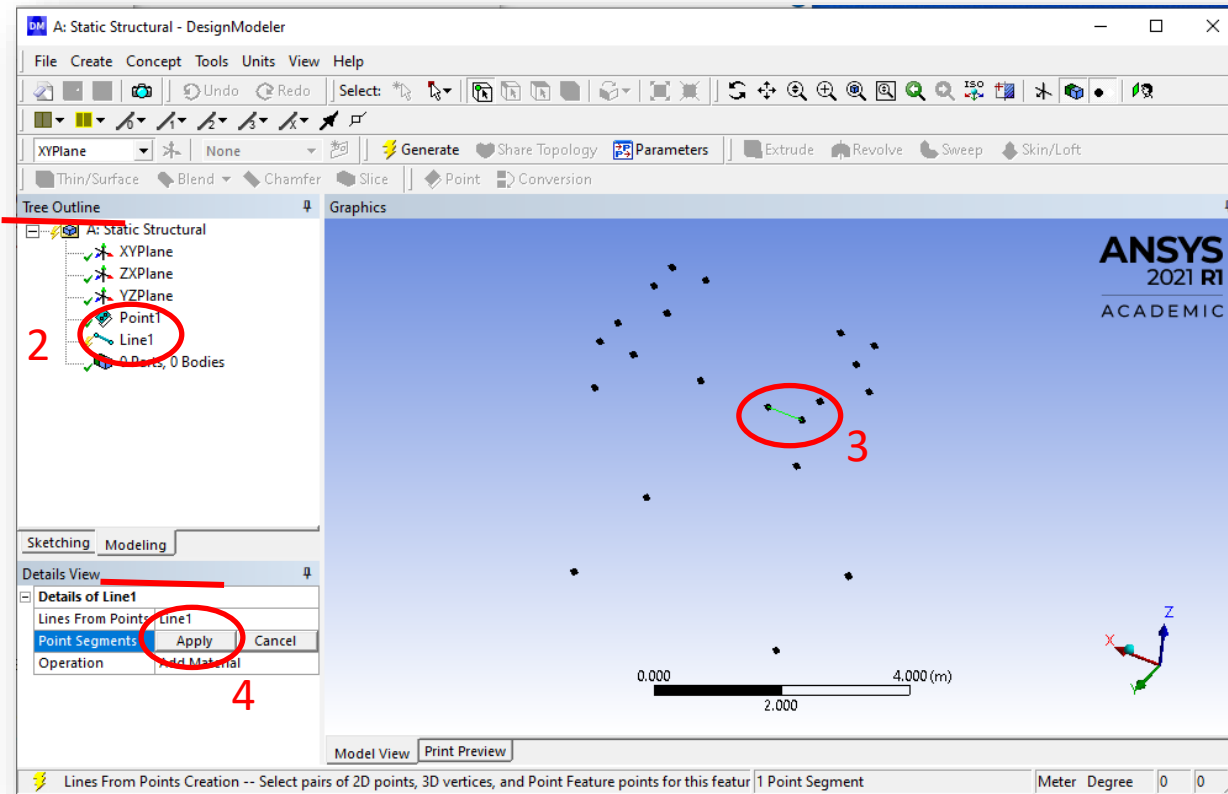
### c) Create the geometry of the structure



In the next few slides, we will be focusing on linking points together to create a space frame made of line elements.

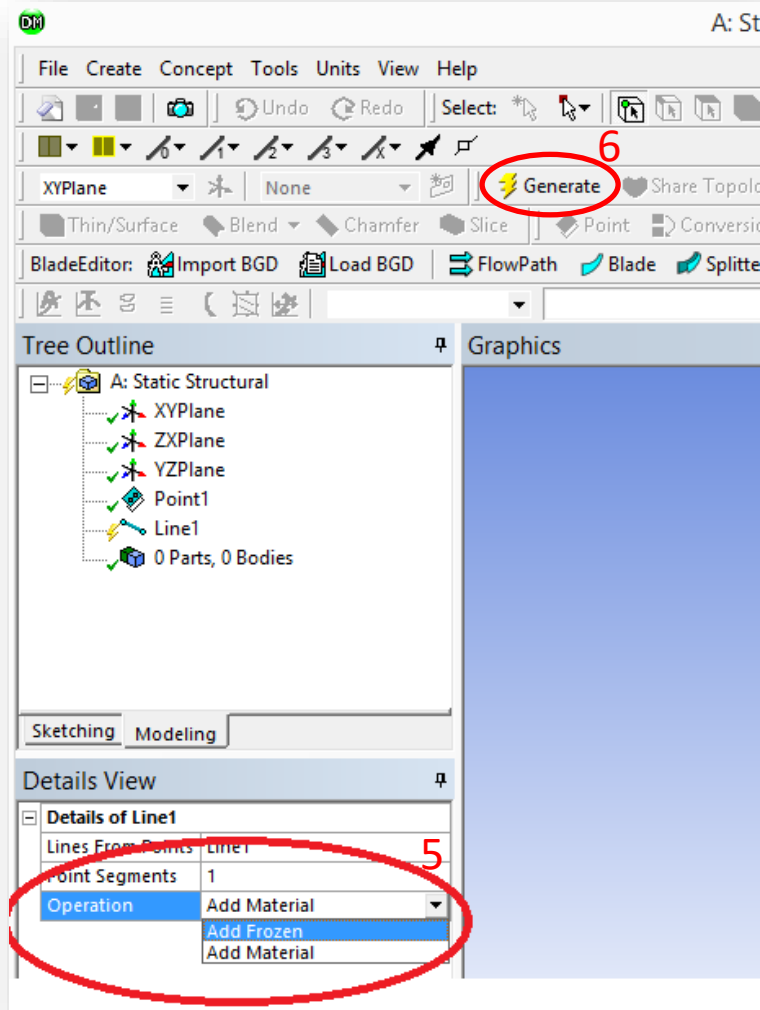
1. Click on **Concept**, then from the drop-down menu, click on **Lines from Points** to join the points.

### c) Create the geometry of the structure



2. Note that 'Line 1' has appeared in the **Tree Outline**. Under Details View, ensure Point Segment is selected. This allows us to select points of interest to be joined.
3. Left click on a point, as the start point. Then HOLD 'Control' button and left click on another point that you would like to connect it to the previous point (start point, in this case). A green segment (line) now connects the two points.
4. Click "**Apply**" to save the line segment.

### c) Create the geometry of the structure



5. In the **Details View** pane, open the **Operation** drop down menu and select '**Add Frozen**'.

#### *For your understanding*

*"Add Frozen" allows each line in a set to exist as a separate body. If two separate lines were created and are geometrically connected, then two bodies will exist (one body for each line).*

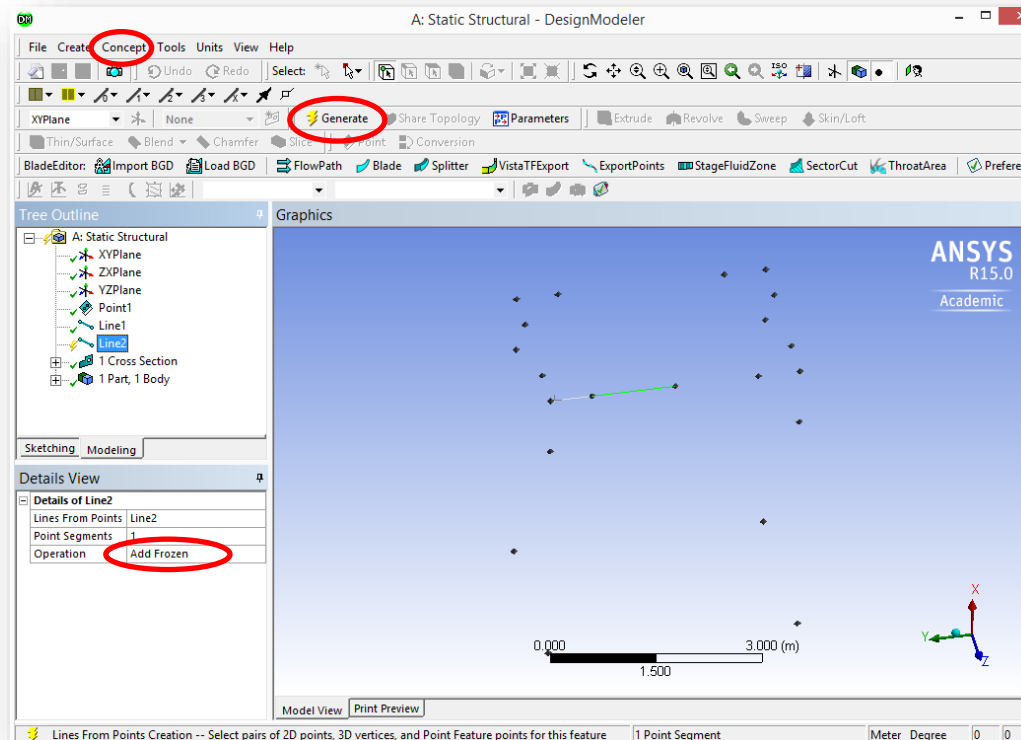
*Selecting the other option 'Add Material', would lead to the creation of a single body out of multiple lines. The purpose of defining the structure as separate bodies (lines) instead of one common body is to allow different properties to be assigned to each line segment, if required.*

6. Click **Generate** and the first line segment will be created.

### c) Create the geometry of the structure

Repeat the steps from the previous slides to create new lines until you have drawn the whole structure:

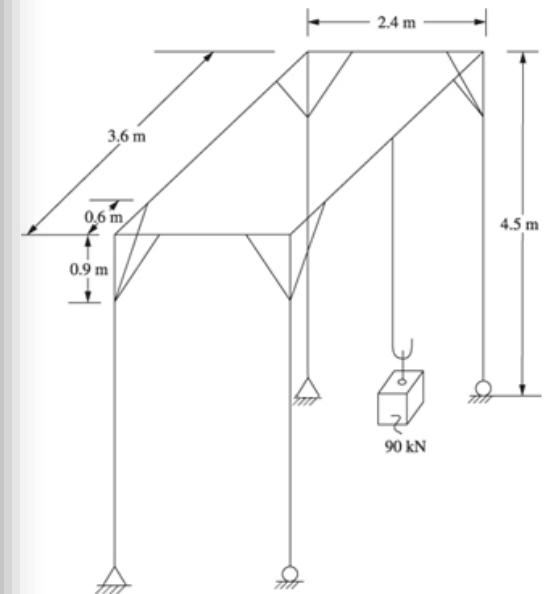
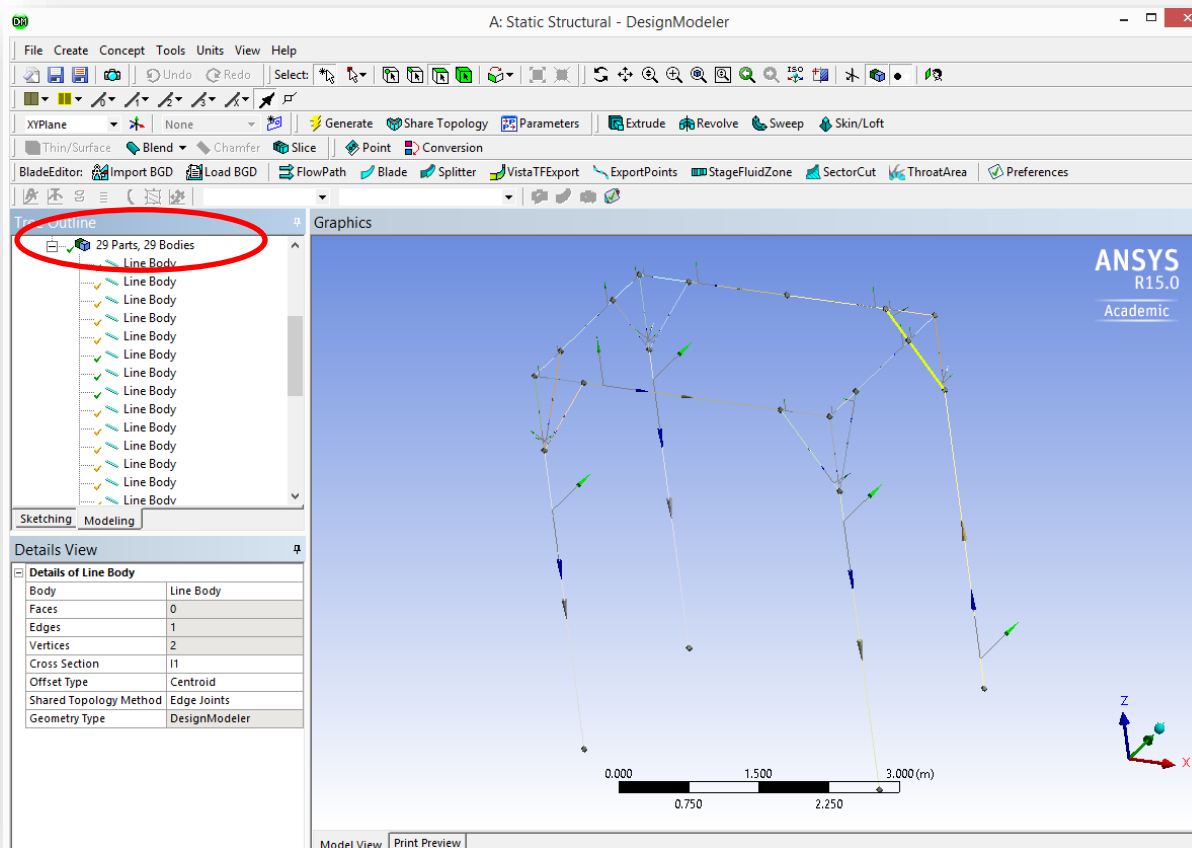
- Click on **Concept**, then **Line from Points** to create a new line element in the tree.
- **Select another pair of points** from the second segment and click "**Apply**" for "Point Segment".
- Change Operation to **Add Frozen** and click **Generate** to for the new line to appear.





## c) Create the geometry of the structure

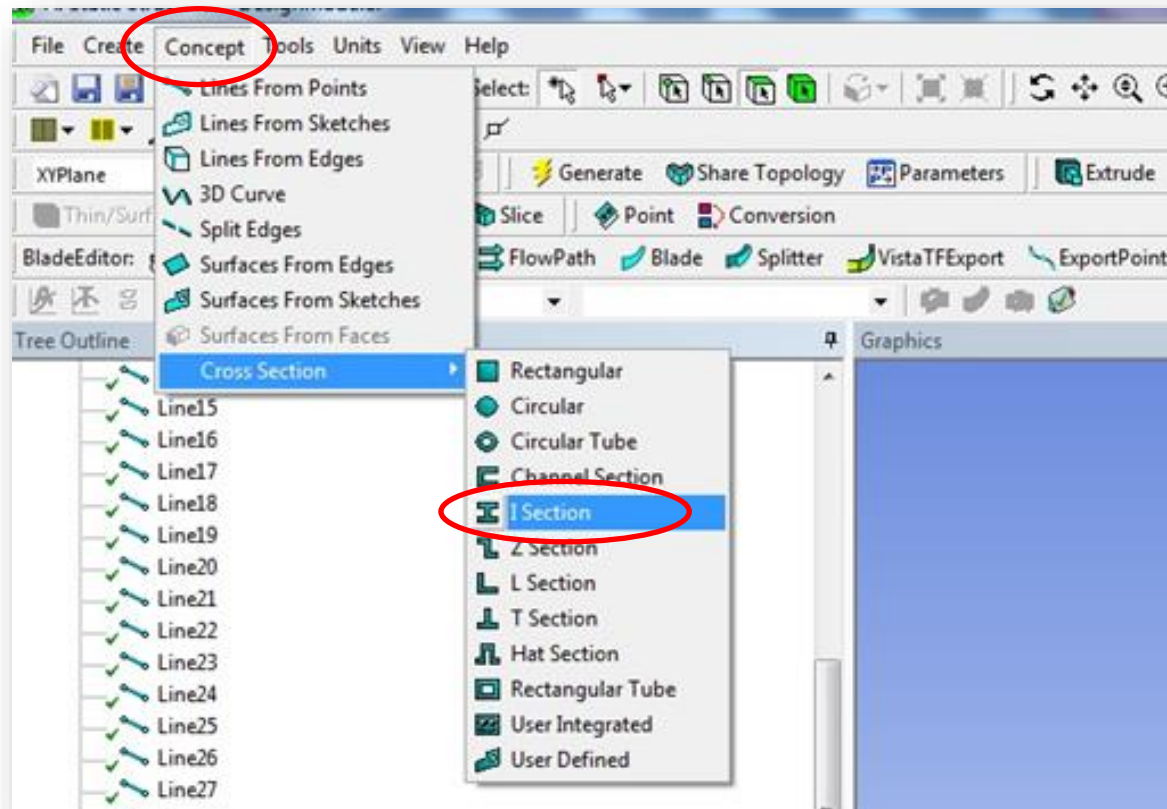
At the end of the process, you will have created 29 parts / 29 bodies, as shown in the design tree pane (red below). Your structure should visually match the required design (minus the load)



#### d) Assign cross sections to each line body

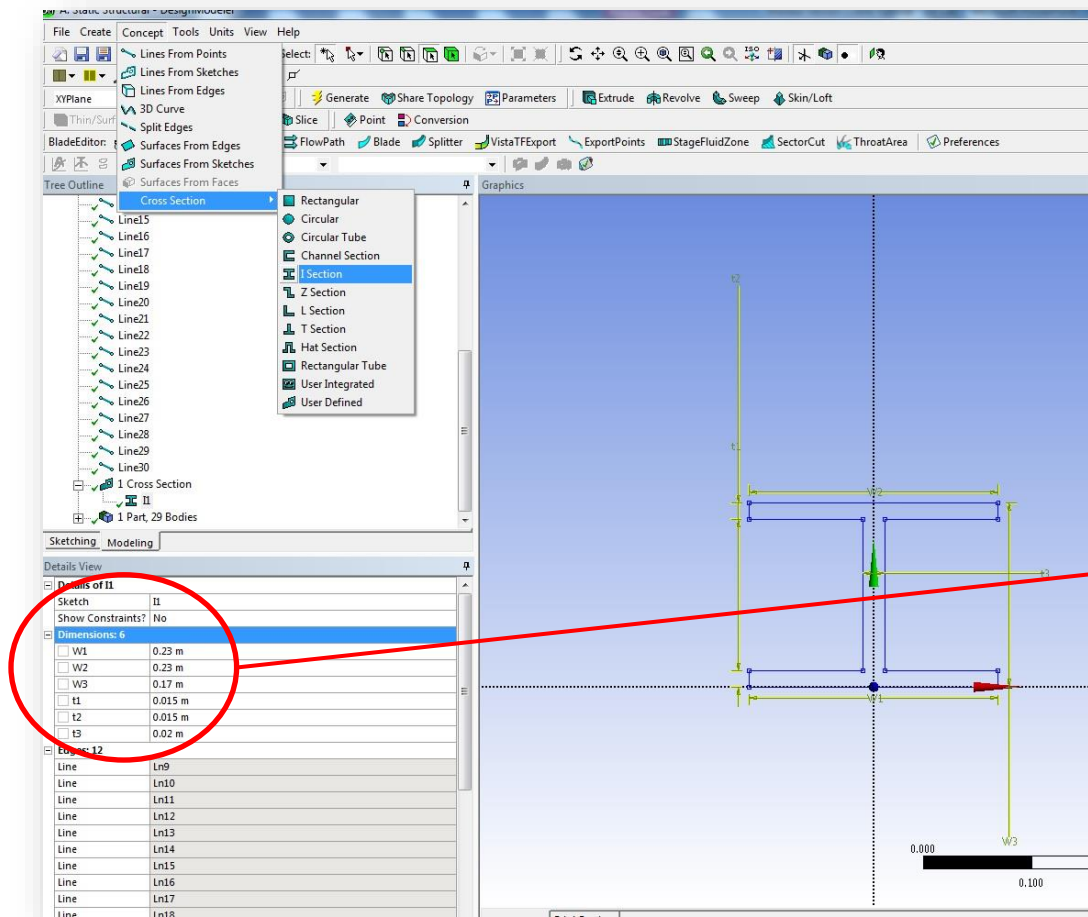
In the next few slides, we focus on creating the cross section profile and assign it to the crane. In this tutorial, we assign “I” cross sections to members.

Click on Concept, from the drop down menu, select cross section, then “I section”



### d) Assign cross sections to each line body

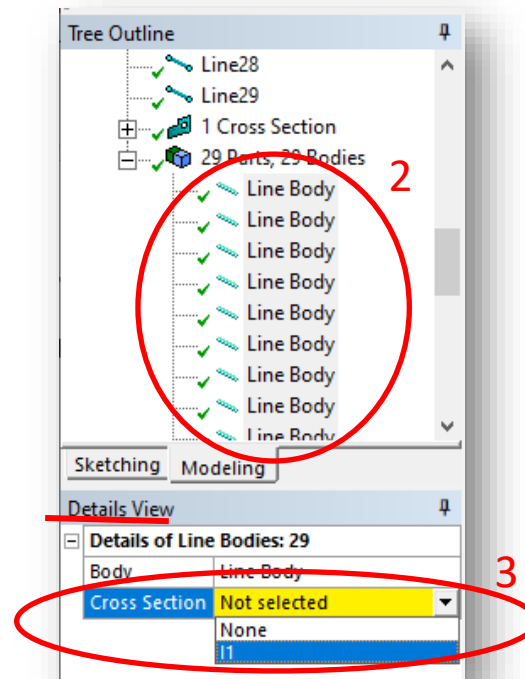
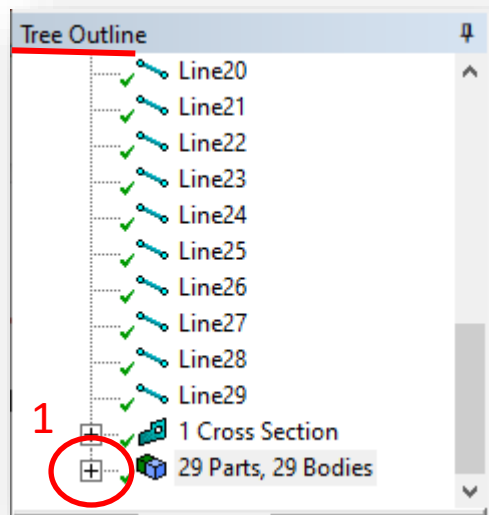
Then, we specify the geometric parameters (Dimension) of the I Section by entering the values given below.



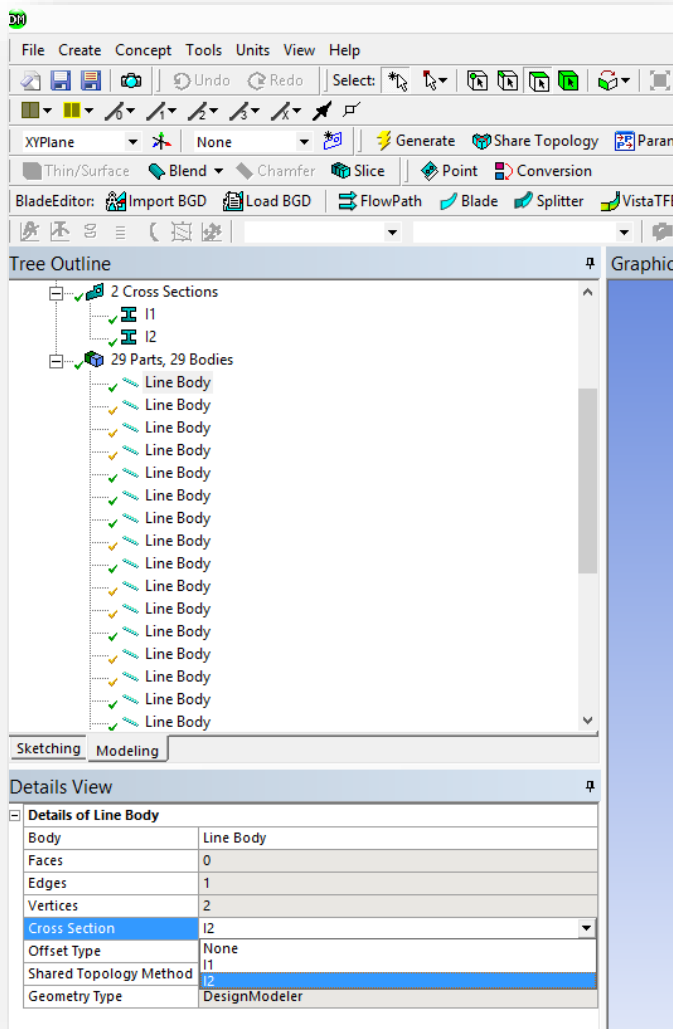
Details View	
[-] Details of I1	
Sketch	I1
Show Constraints?	No
[-] Dimensions: 6	
<input type="checkbox"/> W1	0.23 m
<input type="checkbox"/> W2	0.23 m
<input type="checkbox"/> W3	0.17 m
<input type="checkbox"/> t1	0.015 m
<input type="checkbox"/> t2	0.015 m
<input type="checkbox"/> t3	0.02 m

**d) Assign cross sections to each line body**

1. In the **Tree Outline** pane, **expand** the **'29 Parts, 29 Bodies'** category to display 29 entries named "Line Body".
2. Select all 29 line bodies.
3. In the **Details View**, you can use the drop-down to assign cross sections to these lines.
4. In this tutorial, for simplicity, all bodies have the same cross section ("I1").
5. The choice of cross section for members of a structure will depend on the type of loading applied to that member, (axial, bending, torsional loads or a combination thereof). Different cross-section profiles are better at resisting against different type of loading.



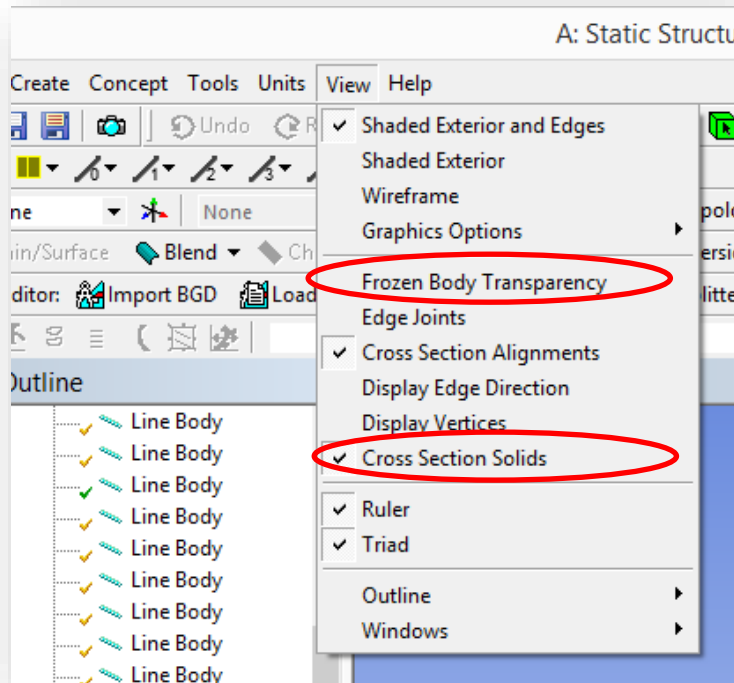
### d) Assign cross sections to each line body (Optional in this tutorial)



1. Assignment of cross-sections is done on a member-by-member basis, so it is possible to assign different cross sections to different members of the structure.
2. To modify a cross-section assignment, you can select the line body you wish to modify, then under **Details View**, change the cross section by selecting a different item in the drop down list.
3. The figure on the left shows an example where the first “Line Body” in the list is assigned an alternative cross section named “I2”.

*Using multiple different cross sections is not required for this lab exercise. However, you are welcome to experiment with this in your own time*

#### d) Assign cross sections to each line body



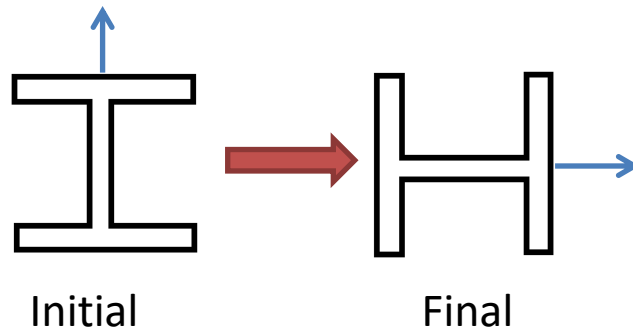
You can visualise the space frame (crane) now, by selecting **View**, then select **Cross Section Solids** from the drop down menu.

Note: if you have difficulty in visualising the alignments of I beams in 3D space, you can deselect Frozen Body Transparency, from the menu shown in the left.

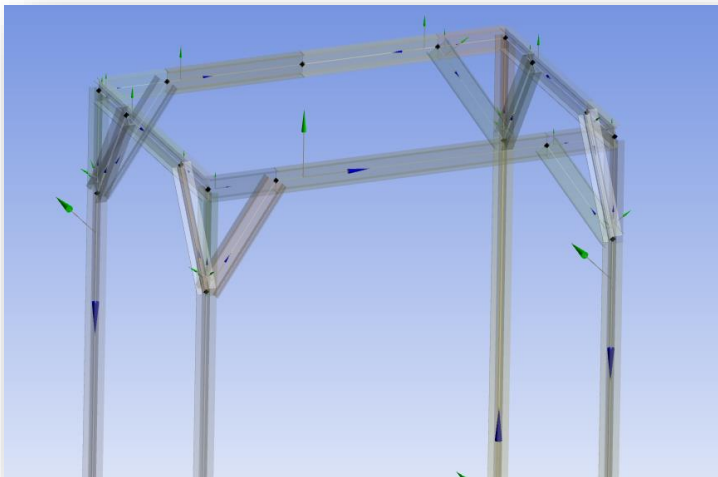
## d\*) Changing cross section orientation (optional)

**IMPORTANT: this is optional. Any orientation changes will change your final solution! If you experiment with this, please revert to the original assignment of cross-sections before having your results marked.**

The last technique we wish to explore is orientation of cross sections. Just like changing cross-section geometry, a change in orientation can vary the mechanical response of a structural member.



1. The following slides will teach you how to perform a 90° rotation on a cross section.
2. The blue arrows on the left and right hand figures show the alignment direction of cross sections.

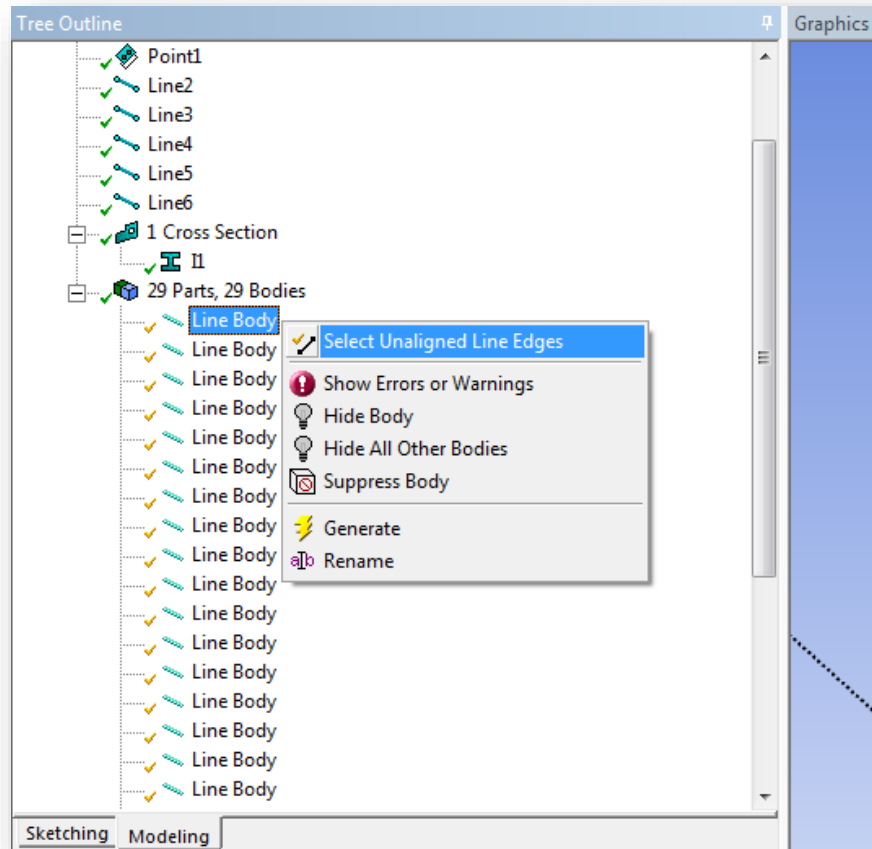


First, we need to define the current alignment (directions) of cross sections. These alignments are shown by green arrows (appearing on the structure in Graphics view).

OPTIONAL



## d\*) Changing cross section orientation (optional)



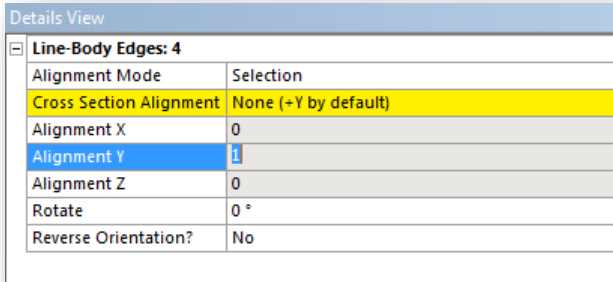
To change the alignment of a member:  
Right click on a line body, which you  
would like to change its alignment, and  
click on '**Select Unaligned Line Edges**'.

Note: your selection will be highlighted  
in the Graphics view

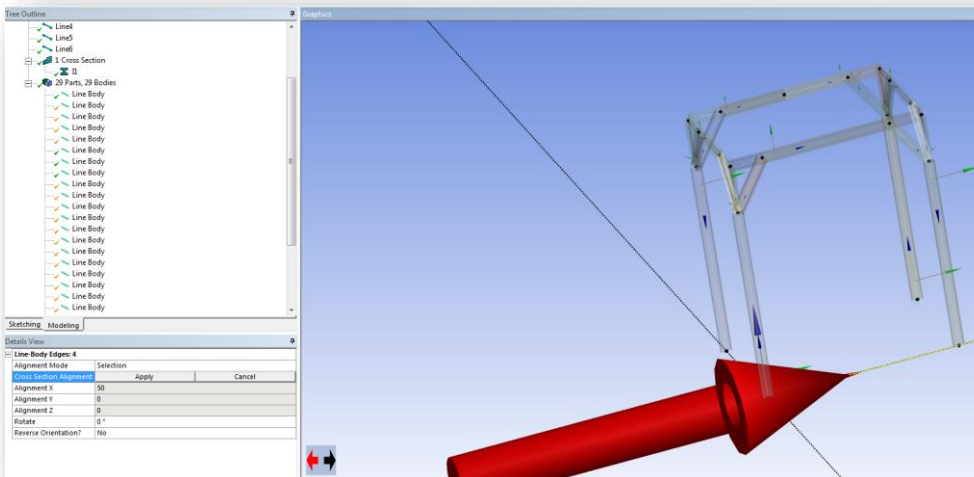
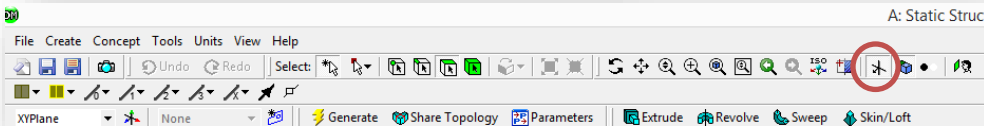
OPTIONAL



## d\*) Changing cross section orientation (optional)



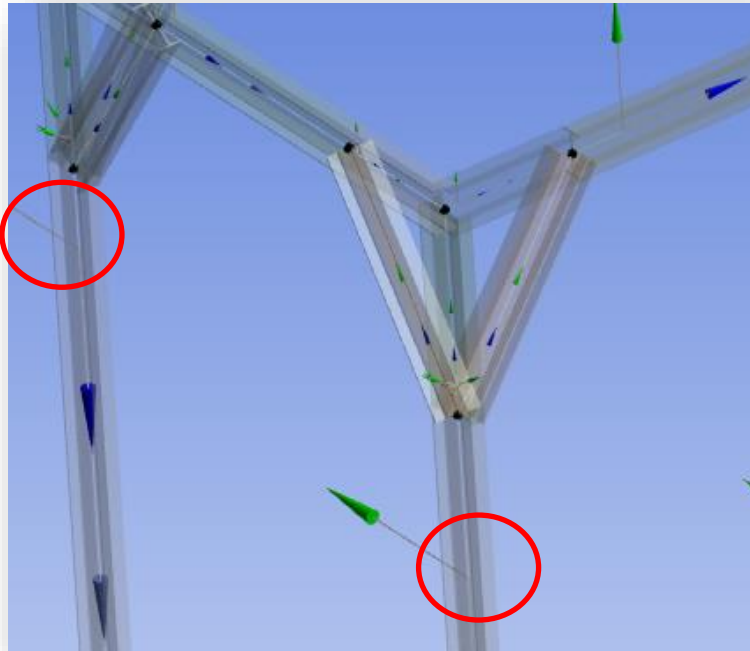
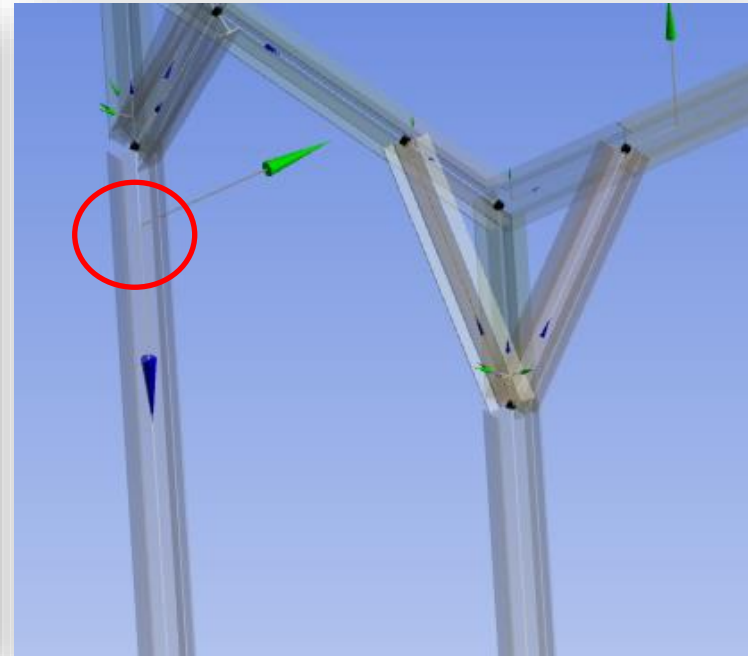
1. After clicking 'Select Unaligned Line Edges', under '**Details View**' you can choose between different options available.
2. Click on '**Cross Section Assignment**' and click on the **X Axis** (at the origin, please keep track which axis it is, and simply click on any part of it)



Note: if you cannot find X Axis, click on the icon '**Display plane**'

You will see a large red arrow appearing over the X Axis, then click **Apply**.

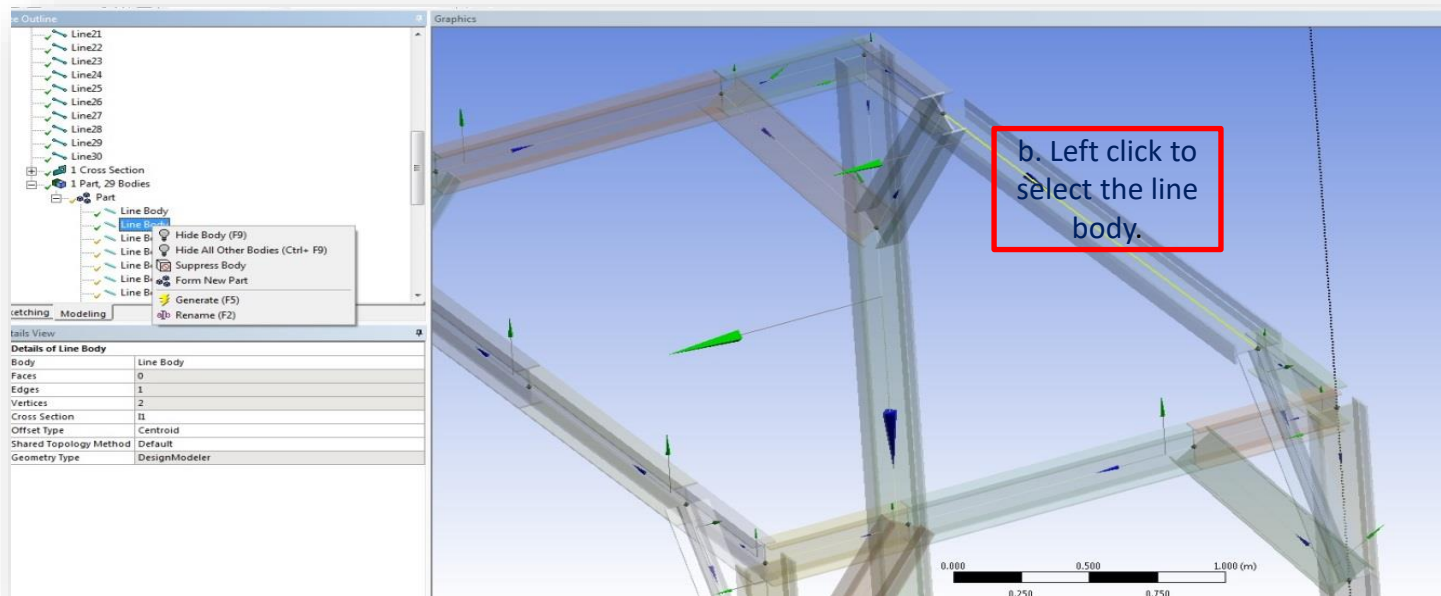
OPTIONAL

**d\*) Changing cross section orientation (optional)****OPTIONAL****Before****After**

In figures shown above , two bodies have been realigned (After), which were originally facing toward the same direction (Before). Note the difference in the direction of the green arrows in both of images. and now you can visualise a rotation in the structural member.

## d\*) Changing cross section orientation (optional)

a. After Aligning the cross section, the tick beside the line body becomes green and by right click on the line body, it is not possible to access its cross section alignment option under detail view specified in slide 23.

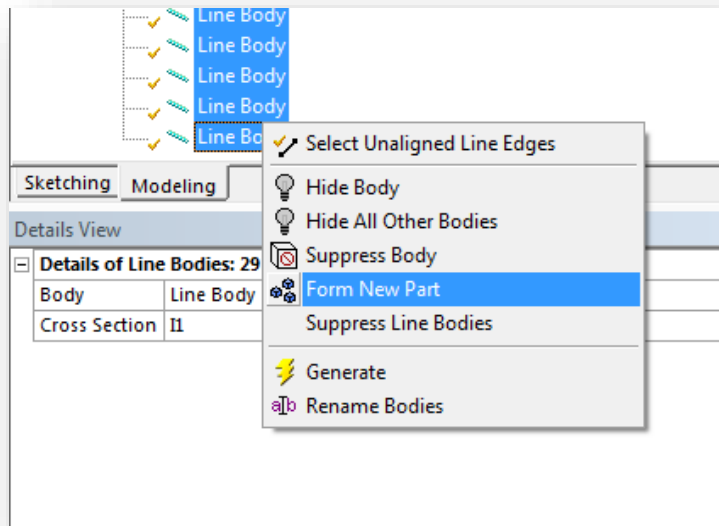


b. In order to change the alignment of this cross section again, this line body has to be directly selected by left clicking on the structure in the graphical area. After accessing cross section alignment under detail view, like the procedure explained in page 23, it is possible to change the direction of the cross section again.

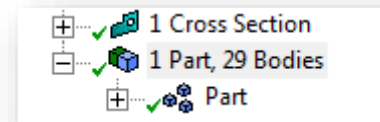
OPTIONAL

d) Assign cross sections to each line body

**Now the really important bit!**

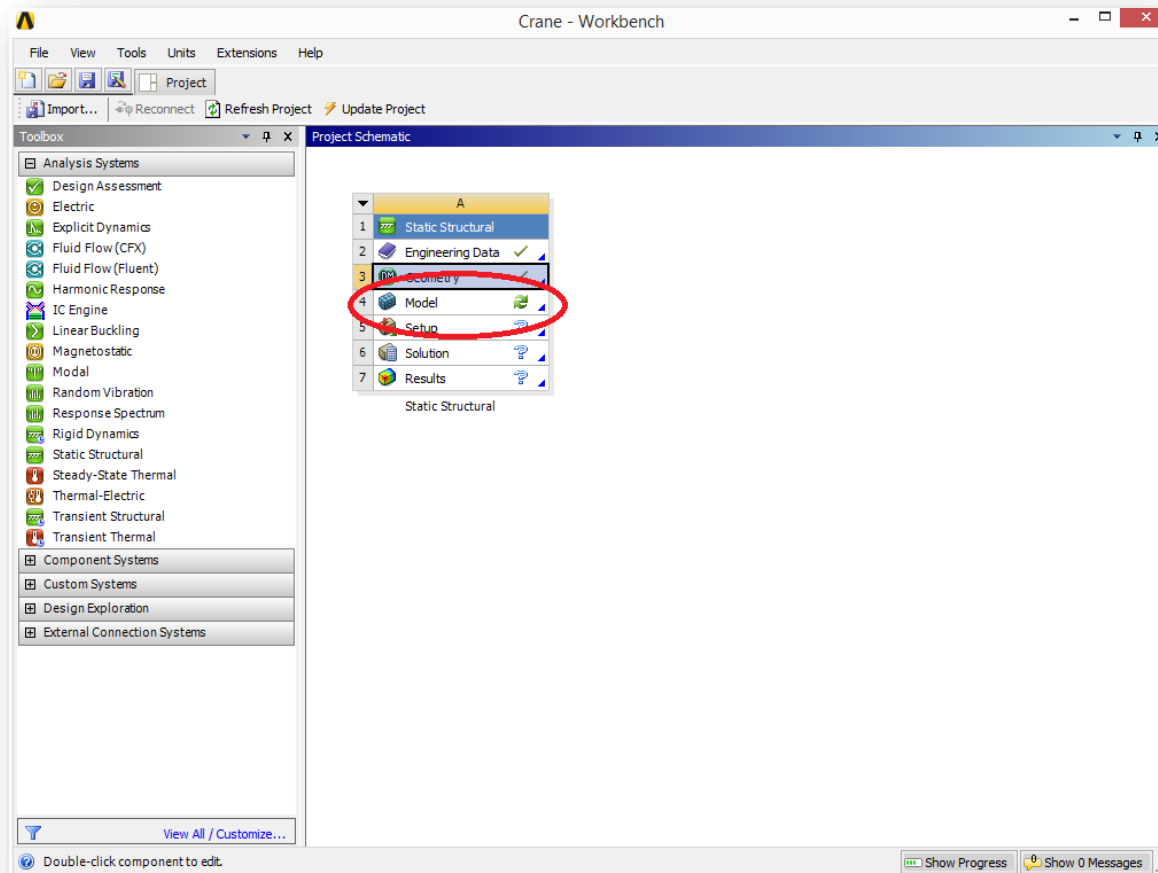


Select ALL line bodies, then right click on one of them, select '**Form New Part**' option. This is ABSOLUTELY necessary for your model to work. You will see '1 Part, 29 Bodies' displaying in the tree outline.



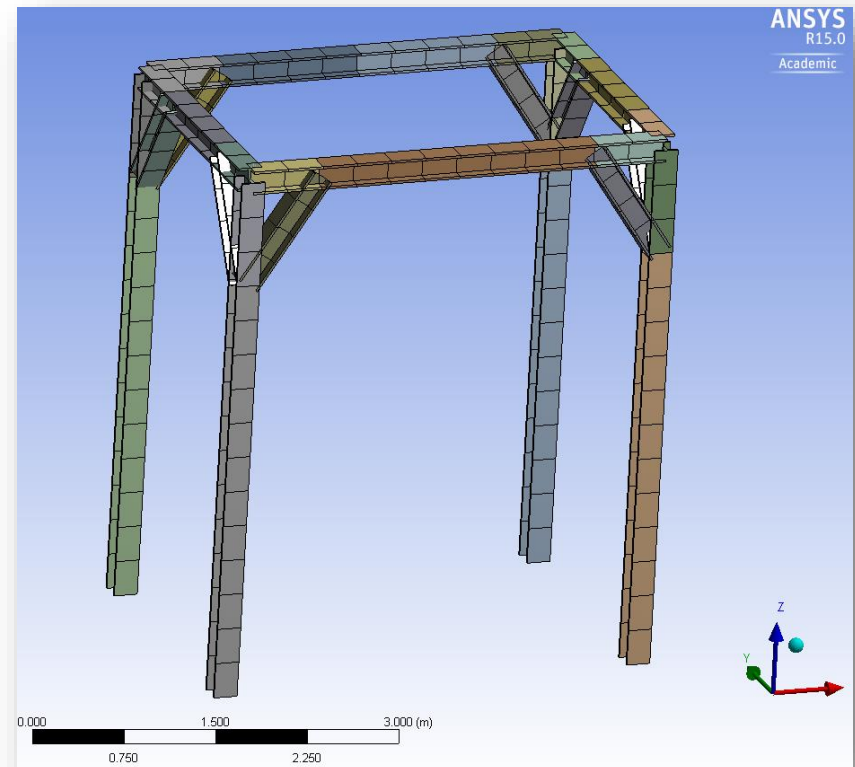
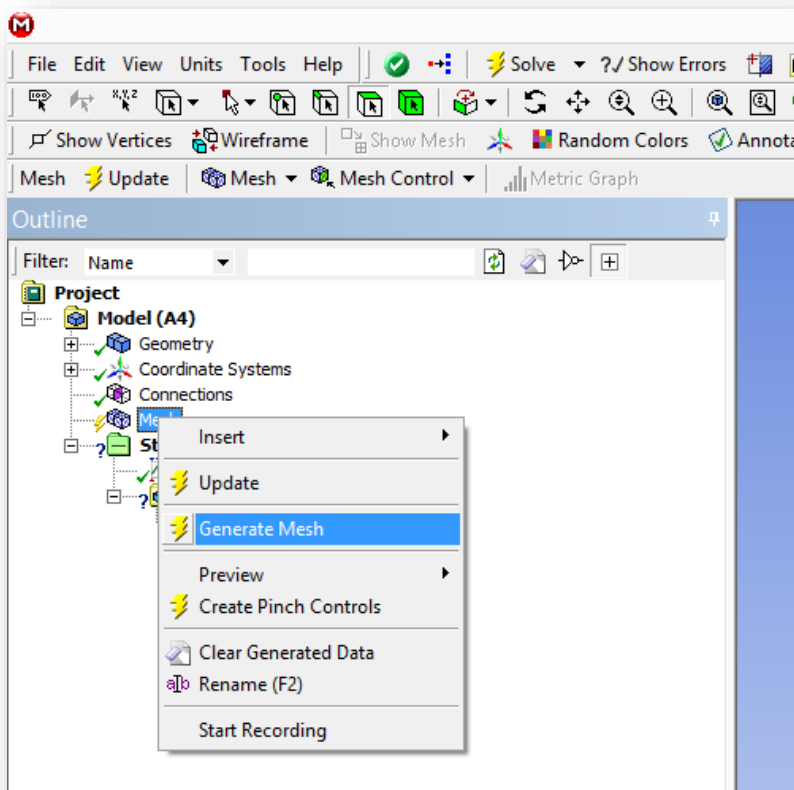
### e) Generate mesh and specify material data

Close the Geometry modeller and enter Mechanical modeller, by double clicking on the **Model** (Workbench area).



### e) Generate mesh and specify material data

Right click on **Mesh** and select **Generate Mesh** from the drop down menu and observe the crane now is partitioned into several segments.



## e) Generate mesh and specify material data

What is a **mesh**?

In simple words, each cell you see in the (meshed) crane is called a finite element, and a network (group) of finite elements is called a mesh. A mesh is a discretisation of the real geometry into small elements.

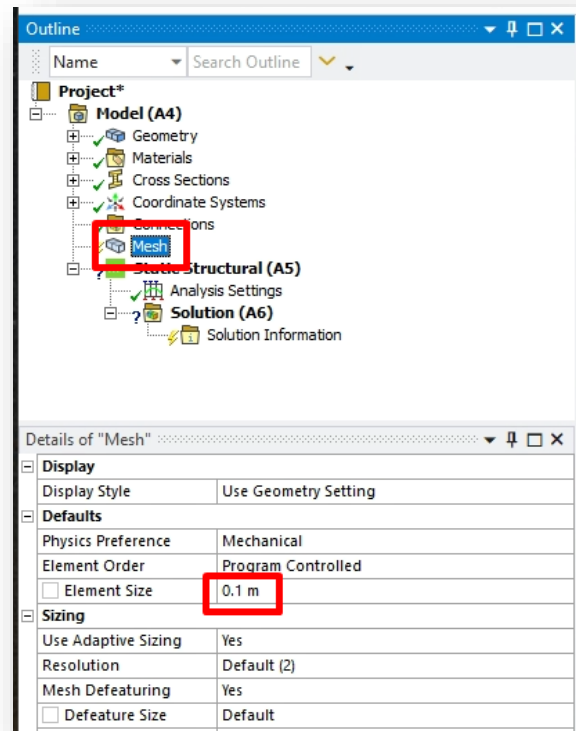
The FEA program applies governing mathematical equations to calculate displacements and forces in each node of the elements.

You can control mesh sizing- i.e. make the cells (elements) smaller or larger.

- What happens when the elements become smaller? – The analysis results become more accurate, but the computational burden increases.
- Should I go for a finer mesh? – Not in scope of this course, but you will need it if you want to solve real engineering problems by FEA. For the simple problem presented in this tutorial, the automated mesh is good enough to obtain accurate results.

### e) Generate mesh and specify material data

When you click on **Mesh**, you can manually set the **Element Size** from default to 0.1. Then click **Generate** and you see the elements are now much smaller than before. This process is called '**mesh refinement**'.





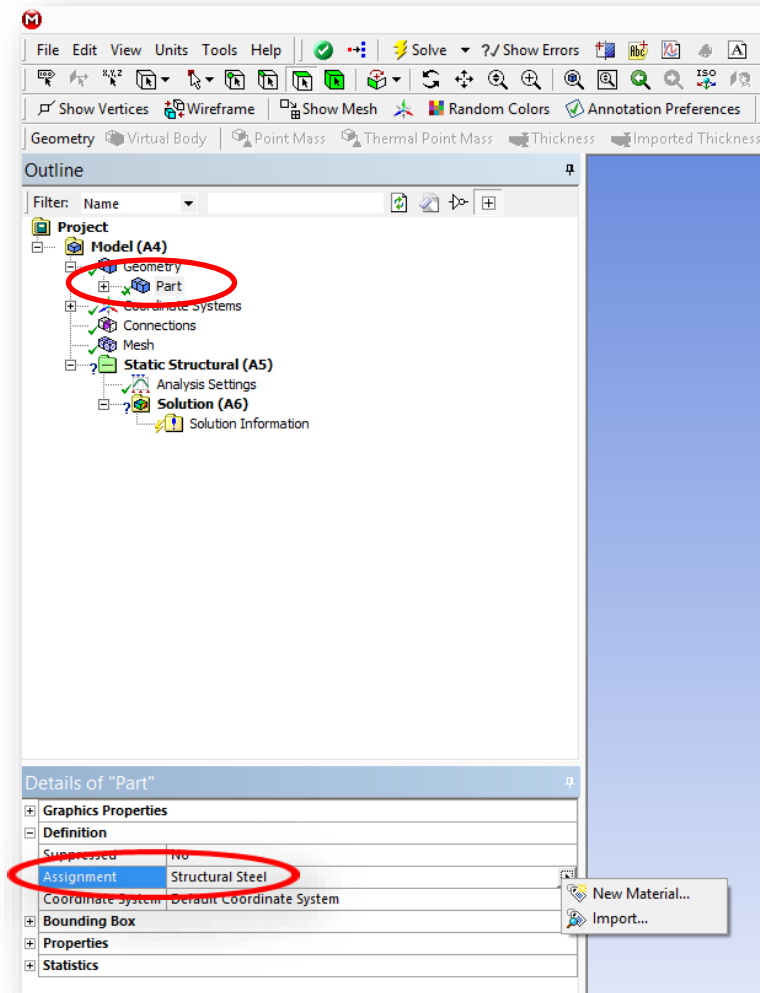
### **e) Generate mesh and specify material data**

Is a finer mesh always better?

No! There is always a trade-off between accuracy and computational time (costs). If you make the mesh finer, your results become more accurate but the analysis takes much longer to solve.

At some point, increasing the number of elements will gain little improvement in result accuracy but will significantly increase the analysis time.

## e) Generate mesh and specify material data



The next step is to specify material data: click on **Part** (under **Geometry**), then in **Assignment** box you will see **Structural Steel** is selected by default.

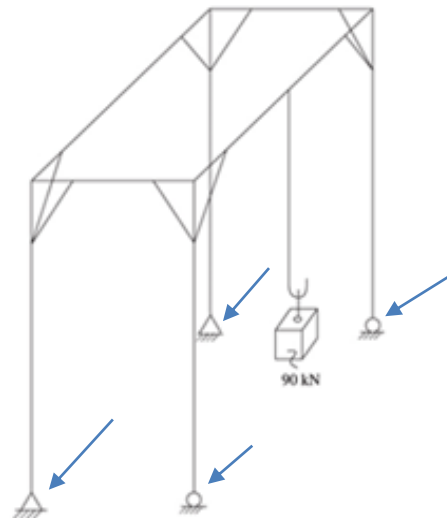
It is possible to select other materials, but you will have to provide material data (density, Poisson's ratio, Young Modulus, etc.). For this particular problem, we select the structural steel as the material of our frame.

## f) Introduce load and boundary conditions

A *Boundary Condition* (BC) is a constraint on the model. It specifies the *Degrees Of Freedom* (DOF) of different points on the model (spaceframe), i.e., what sort of kinematic behaviour that point is allowed to have.

*For example: a fixed support imposes that a structural member cannot translate nor rotate; a slider support allows a point to translate within one spatial plane but not rotate, etc.*

BCs are most often use to describe the external supports applied to the structure.



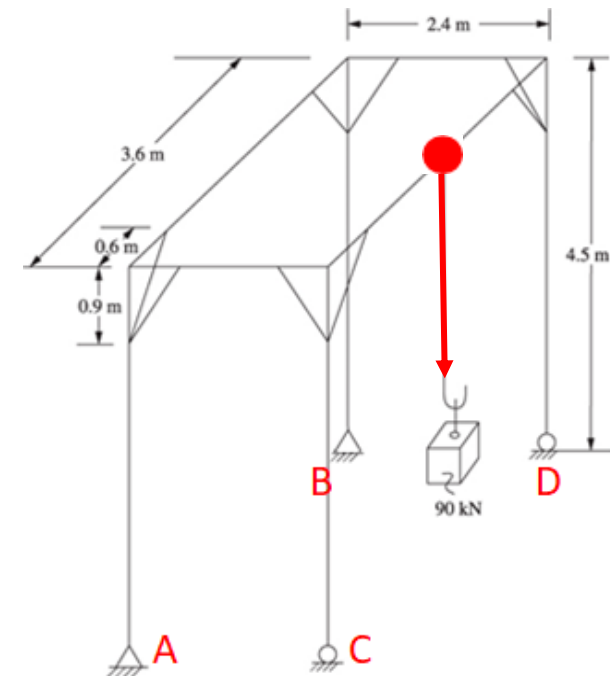
## f) Introduce load and boundary conditions

We will add a force of 90'000N (90kN) at the point highlighted in the figure below. This simulates the weight that the crane is lifting.

We will use roller supports on vertexes C and D (assume the crane has wheels). This means these two points are fixed vertically but are allowed free horizontal displacement (on the xy plane) and free rotation in all three axes.

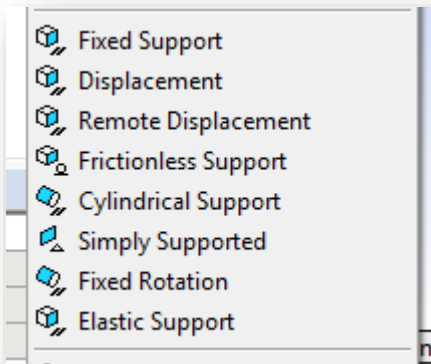
We will use ball (socket) supports on vertexes A and B. This means that displacements along all axes are restricted ( $x, y, z = 0,0,0$ ), but rotations are free.

**Important:** You *MUST* constrain all displacements on vertex A and B. The analysis **will fail** if the structure remains statically under-constrained overall, i.e., if the problem definition allows the structure to translate/rotate in space as a rigid body. (The reason for this is that the inverse of the stiffness matrix cannot be calculated by FEA.)



## f) Introduce load and boundary conditions

ANSYS has a number of fancy looking supports, but 'Displacement' and 'Fixed Rotation' are two of the principal ones. Note that Ansys program offers 8 different types of BC, from fixed support to elastic support, as shown in the figure.

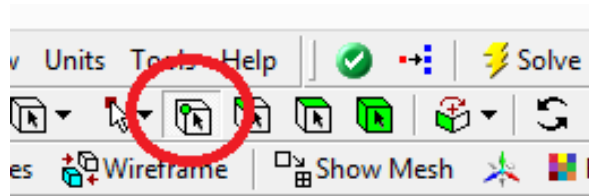


Remember from ENGN1217 that ANY kind of structural support can be expressed in terms of degrees of freedom. In 3D space, we have 6 Degree of freedoms (DOFs), 3 DOFs for *displacement* and 3 DOFs for *rotation*.

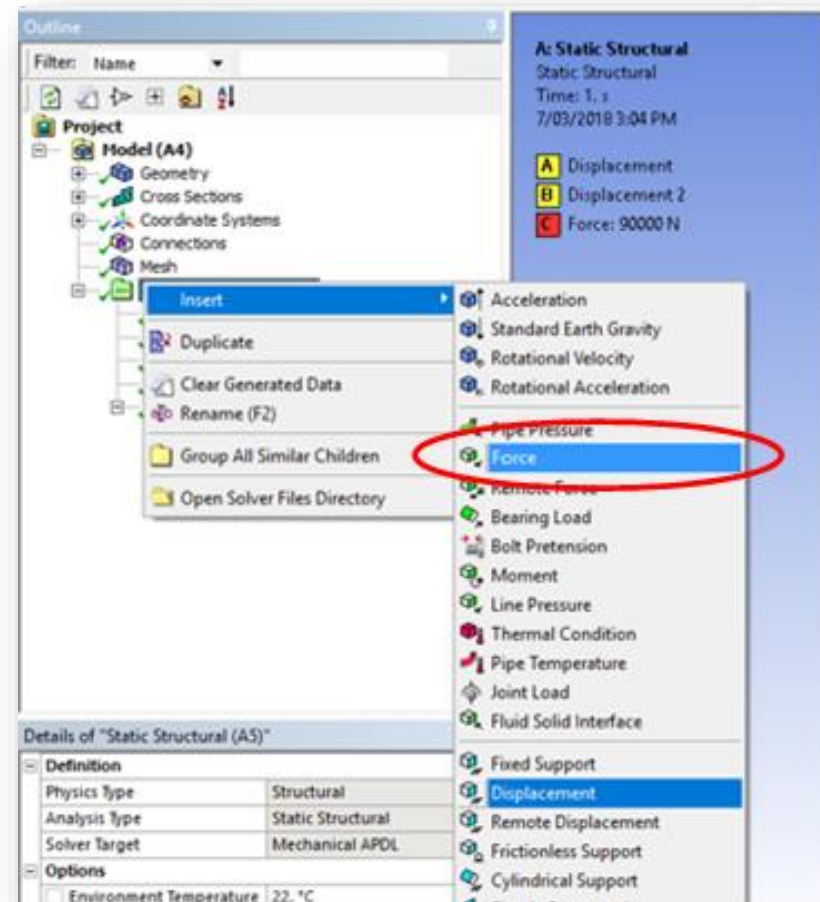
For example, a ball and socket support allows free rotation, but no displacement is allowed. In this case, displacements along all axes are restricted ( $x, y, z = 0,0,0$ ), but rotations are free. A 'Fixed Support' can be created by applying both a 'Displacement' and a 'Fixed Rotation' to the same location and then setting their values to zero.

## f) Introduce load and boundary conditions

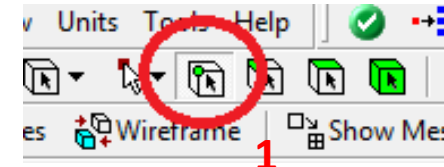
As the loading and supports are applied to points, to select these points you need to make sure you are using the vertex selection option for your mouse cursor.



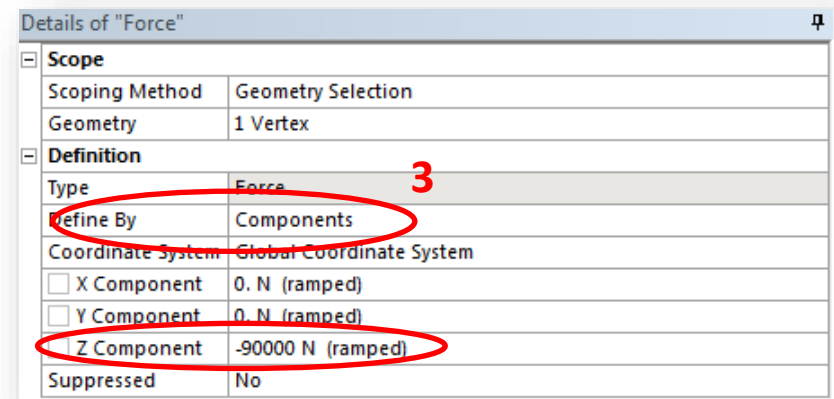
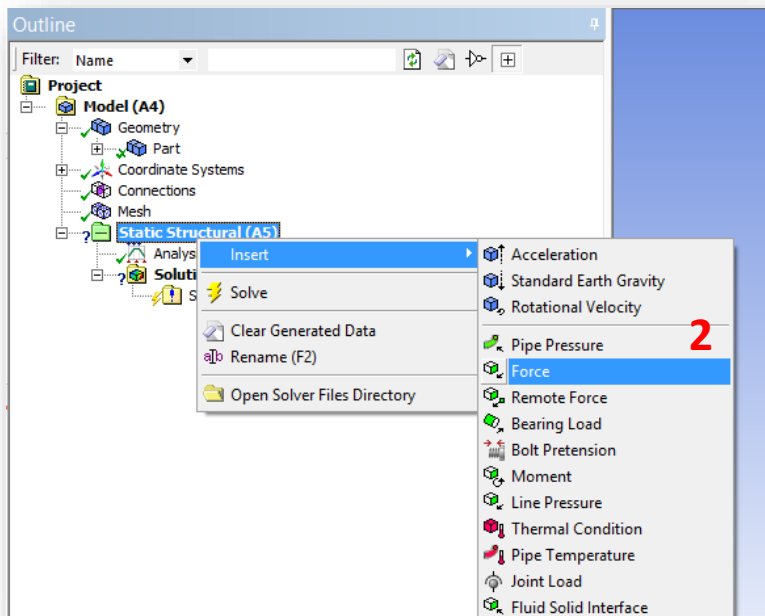
To define 'Force' and 'Displacement' conditions, you need to right click on **Static Structural**, go to **insert** and then you can find loading and support options.



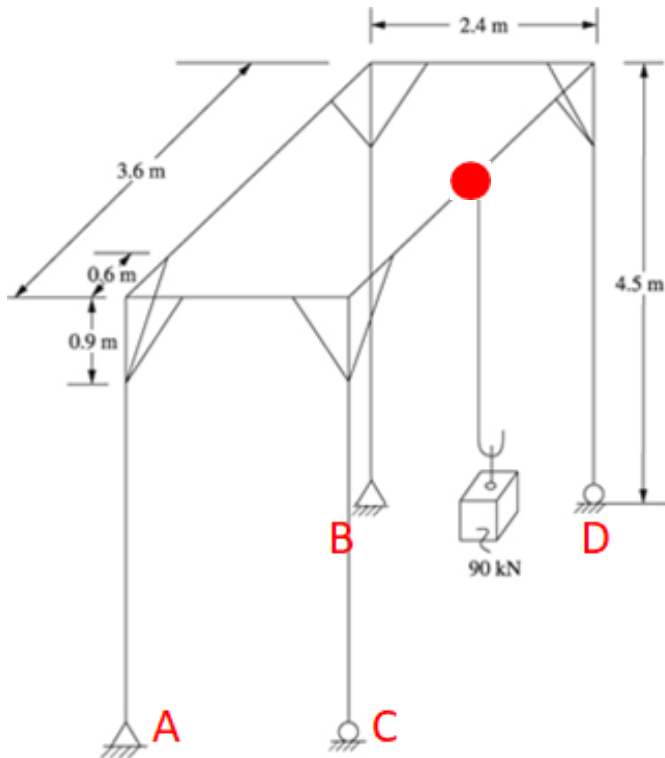
## f) Introduce load and boundary conditions



1. Choose **vertex filter** (to enable point selection) and click the point that the force is applied to. (force position is shown in Page 36)
2. Right click on **Static Structural**, so to **insert** and choose **Force**.
3. Under **Details of Force**, change '**Define By**' to Components, and specify a force of -90'000 N (-90 kN) for Z component (x and y components of the force are zero).



## f) Introduce load and boundary conditions



(right click on Static Structural, go to insert and then find Displacement)

Select 'Displacement' for vertex A and B. Then, assign '0' displacement to all 3 axes.

Select 'Displacement' for vertex C and D. Then, set '0' displacement for vertical direction (vertical direction may not be Z axis in your model.) and 'Free' for the other two.

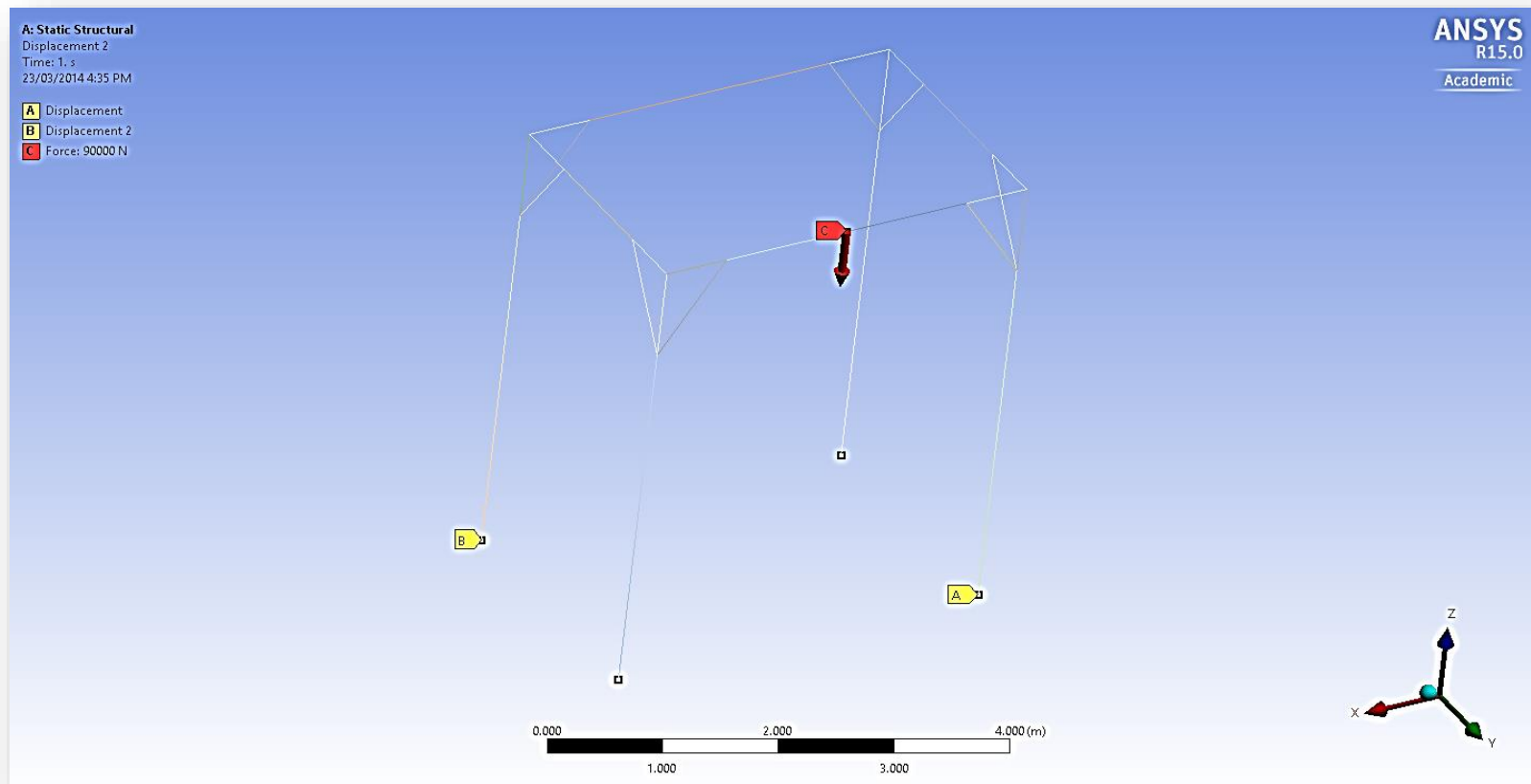
Details of "Displacement 2"	
Scope	
Scoping Method	Geometry Selection
Geometry	2 Vertices
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
<input type="checkbox"/> X Component	0. m (ramped)
<input type="checkbox"/> Y Component	0. m (ramped)
<input type="checkbox"/> Z Component	0. m (ramped)
Suppressed	No

Details of "Displacement"	
Scope	
Scoping Method	Geometry Selection
Geometry	2 Vertices
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
Y Component	Free
<input type="checkbox"/> Z Component	0. m (ramped)
Suppressed	No



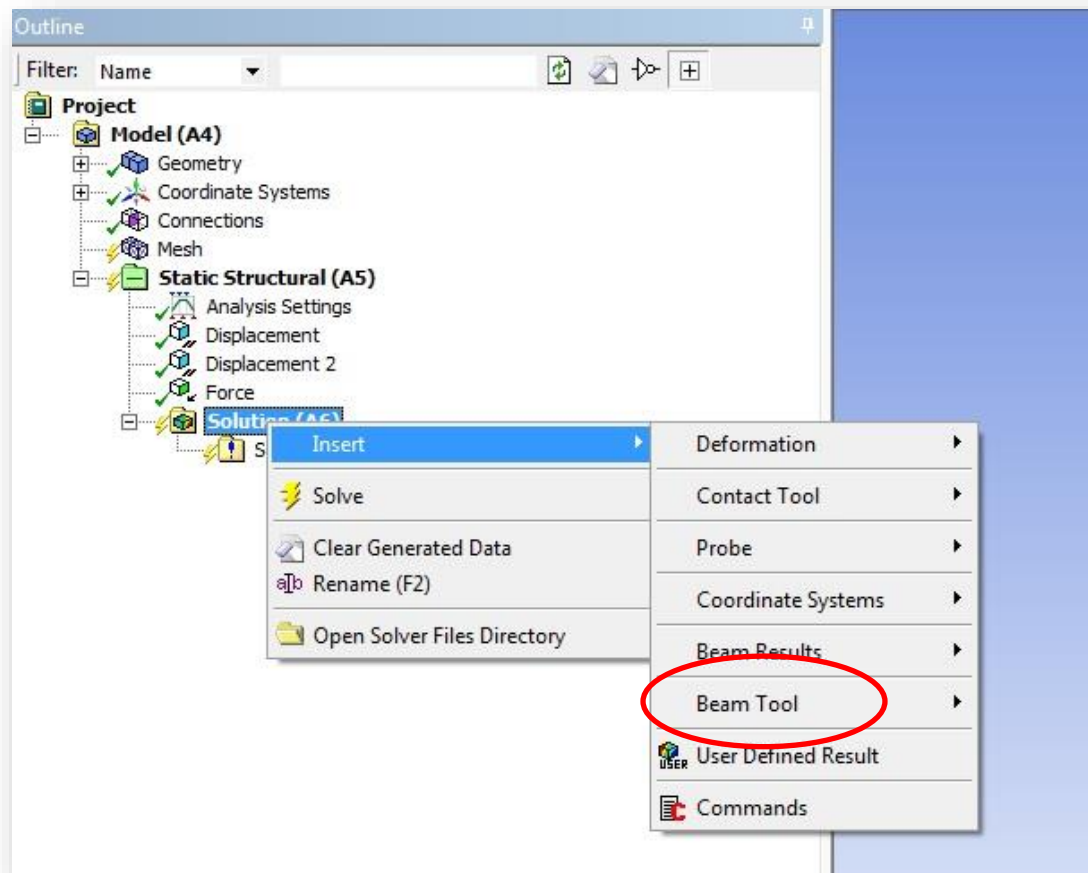
## f) Introduce load and boundary conditions

This is how it should look like after the load is applied:



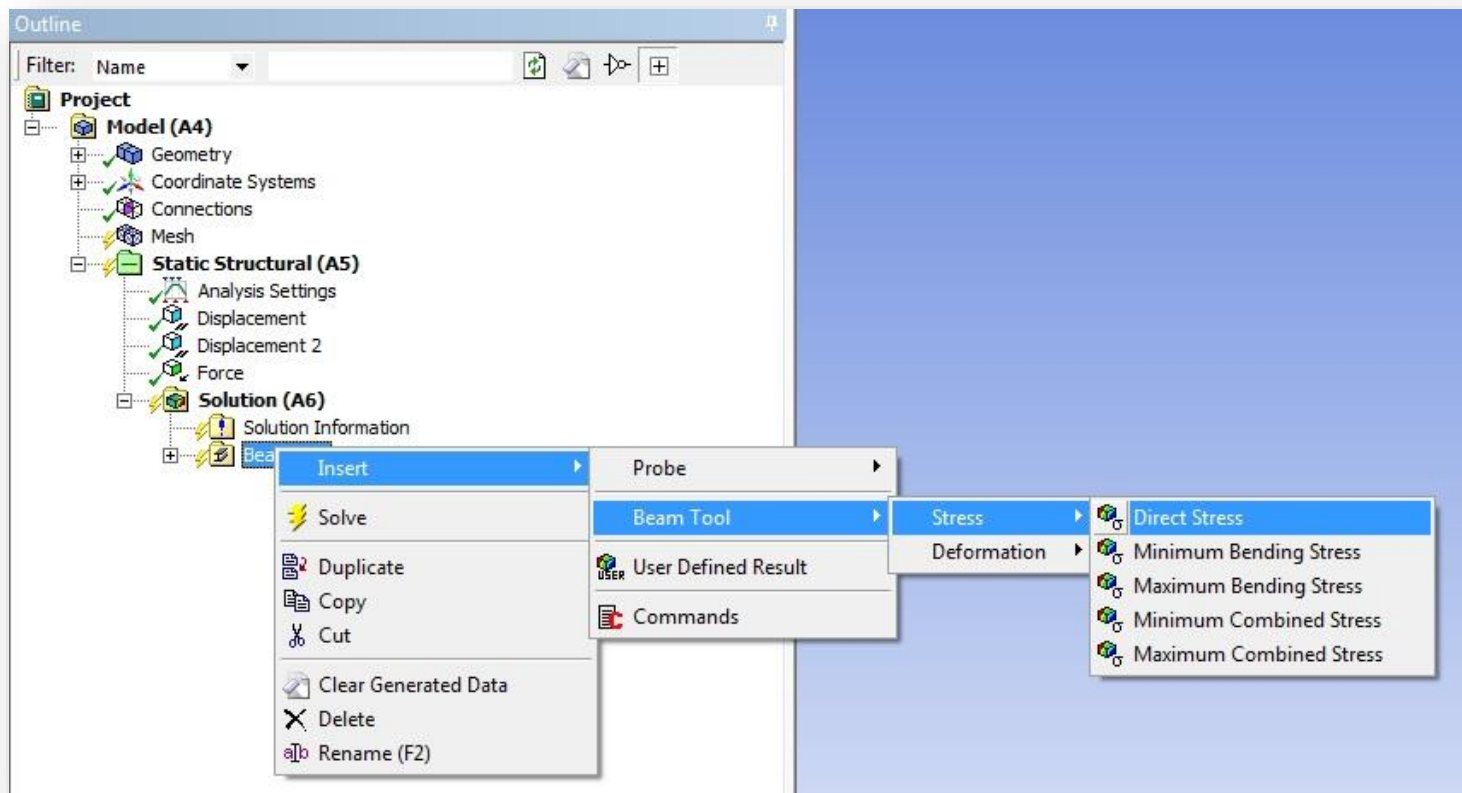
### g) Solve the problem

The information you would like to see in the solution can be chosen by right clicking on **Solution** and then **Insert**: beam tools will give us stress values in each structural member



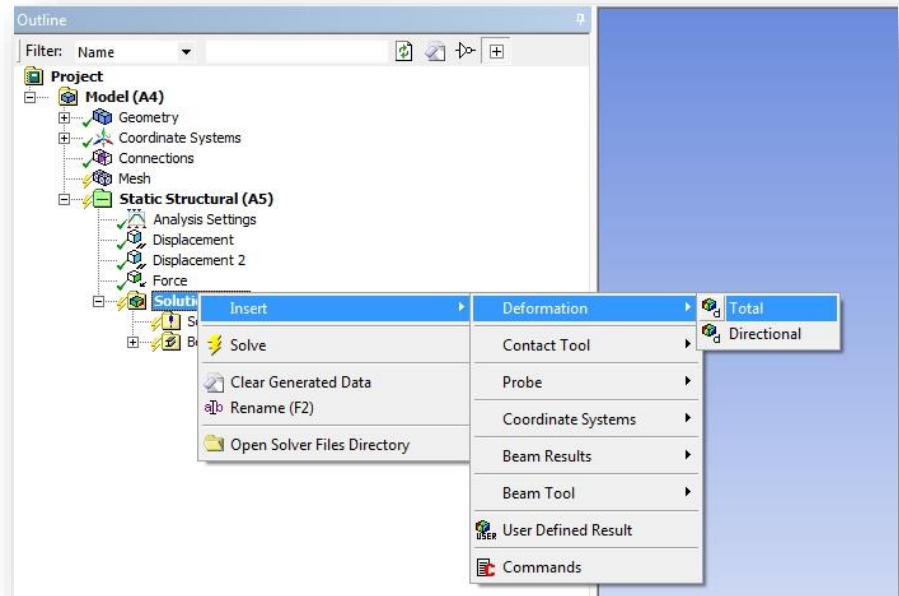
## g) Solve the problem

Right click on 'Beam Tool' and manually select *all* of the the stress measures available in the list.

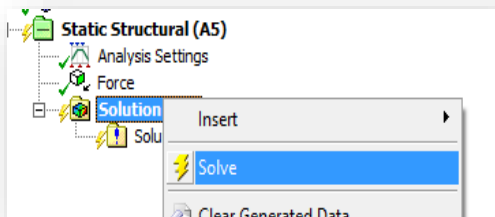


## g) Solve the problem

1. Right click on '**Solution**' and then select '**Total**' deformation.

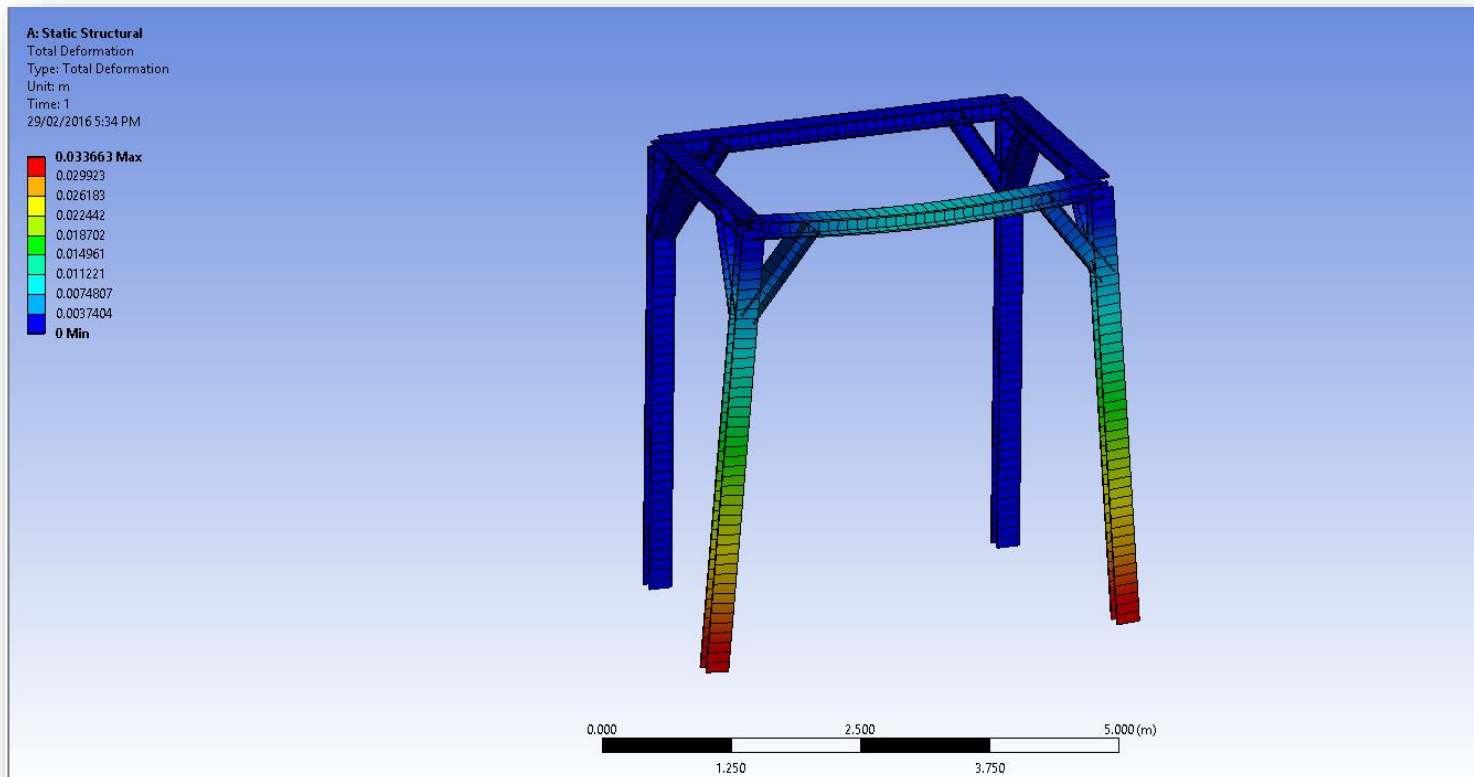


2. Click **Solve**!



### g) Solve the problem

After the analysis is completed, you can visualise the deformations at different locations on the space frame. Please do note that the results view often exaggerates the deformation for easier visualisation. Think about it: if the frame distorted this much, our all-important small strain assumptions would not be applicable! See if you can find a scale selector in the toolbar which will allow you to see the true deformation with no scaling.



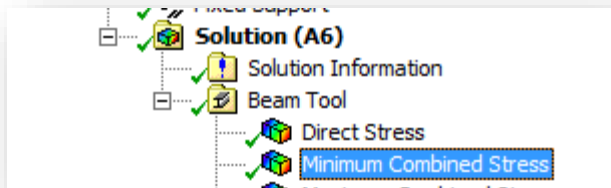
## g) Solve the problem

### Common solver errors:

- **Pivot error** – this error occurs when you have not set boundary conditions properly and thus some structural members are not constrained perfectly (the member experiences rigid body motion).
- **Solver quits unexpectedly** – this error might occur when you do not have enough disk space on your hard drive. Please move your file to a USB with sufficient free space and then analyse and solve the FEA problem.

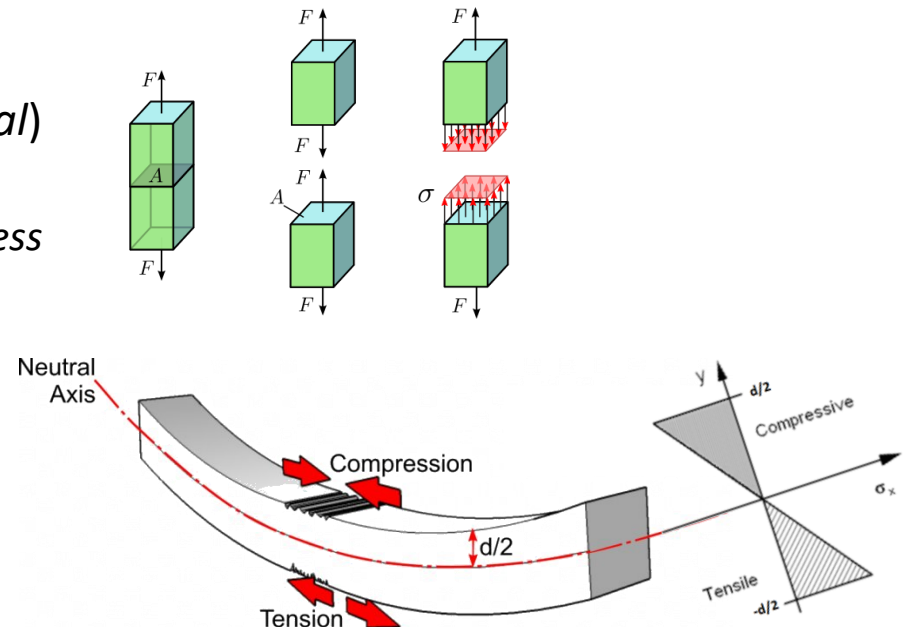
## h) Perform post processing, obtain relevant structural performance indicators

Click on the type of stress you wish to view under Beam Tool

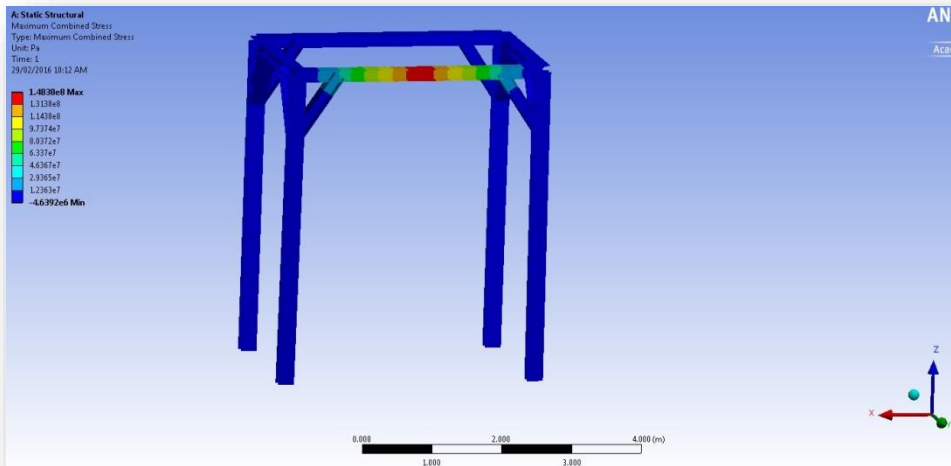


### Types of stress :

- \* Direct stress (also known as *normal or axial*) due to axial forces  
*in a member experiencing an axial load, the stress can be deemed uniform in the cross-section*
- \* Bending stress due to bending moments  
*in a bent member, the concave side of the beam is in compression and the convex side is in tension, with a neutral region mid-way.*
- \* Combined – direct and bending stresses are combined (magnitudes are superimposed)



## h) Perform post processing to obtain relevant structural performance indicators



Review the results for “maximum combined stress” and “maximum bending stress”. Note that these values are very close.

**Now answer the following questions**

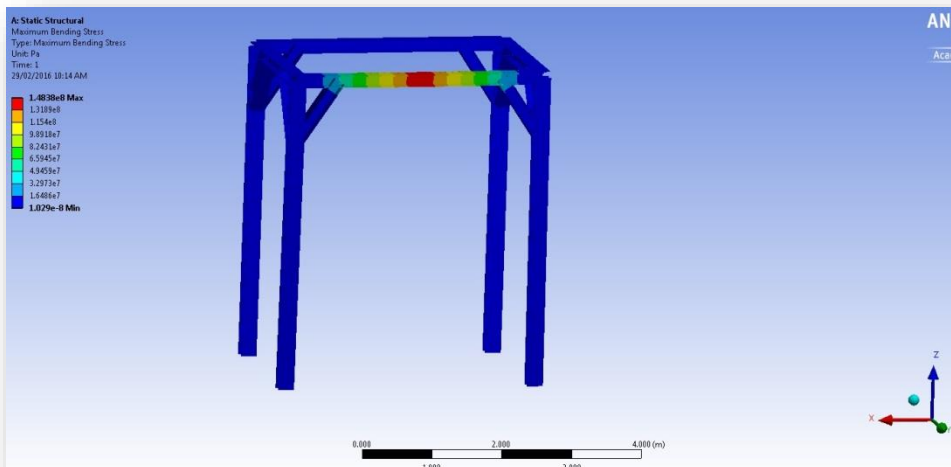
**Q1. What is the highest value of “maximum combined stress” in your loaded structure?**

**Q2. Why are the values of combined stress and bending stress very close?**

*Hint: what type of loading is the crane experiencing?*

**Q3. Compare the maximum stress result from the simulation with the failure stress for steel. What is the factor of safety for this structure?**

**Q4. What could you change in the design to increase the factor of safety?**



**(see some hints in slides 49-51)**



## i) Design improvements

Once you have obtained the max stress value in your application, you can compare it with the failure stress of the material and determine the safety margin for the crane structure.

- Steel is a ductile material so the relevant failure stress will be **yield stress** ( $\sigma_y = 250 \text{ Mpa}$  for steel)
- If the safety margin is below 1, the structure is found to be unviable. If the result is  $>1$ , the structure will nominally resist the load. However, a safety margin above 1 but too close to 1 might still be unacceptable on risk considerations (some engineering structures, such as bridges, require safety factors of 5 or even more!).

The safety margin result provides you with a metric to inform any re-design efforts. Design is always a compromise among safety, cost and weight factors. Reflect on how these are related.

## Stresses and cross-sections

$$\sigma_{axial} = \frac{F}{A}$$

- When considering axial stress, for a given force, the cross-sectional area can be manipulated to control stress levels.

$$\sigma_{bending} = \frac{My}{I}$$

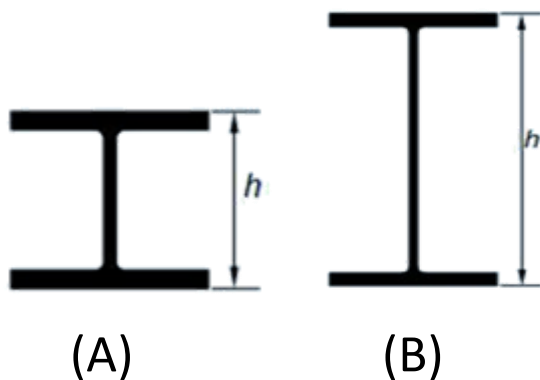
- Reducing bending stress is tricky! For a given moment  $M$ , reducing 'y' (the "thickness" of the cross-section) will also cause a decrease in 'I' (moment of inertia of the cross-section) as these two are geometrically related. Still, there is a solution for this. Try to apply your problem solving skills.

## i) Design improvements

A common design improvement might involve a change in cross-section to try and increase a structure's safety factor. Instead of blindly trialling a large number of alternative geometries (VERY time consuming!!), you might want to estimate the factor by which the combined stress will be *approximately* reduced (compared to the current geometry) when a new cross section is assigned.

Let's consider the below situation where two alternative cross-sections A and B are being evaluated. The combined stress from axial and bending effects is given by

$$\sigma_{combined} = \sigma_{axial} + \sigma_{bending} = \frac{F}{A} + \frac{My}{I}$$



Assume that the geometric relationships between the two cross-sections are as below

$$A_B = 0.8 A_A \quad y_B = 1.5 y_A \quad I_B = 2 I_A$$

Can you figure out  $x$ , i.e., the factor by which combined stress will be reduced by using the new cross-section?

$$x = \frac{\sigma_B}{\sigma_A}$$

**Hint:** to calculate this **exactly**, you will need to use geometry data (see table on slide 52) and some results from your simulation

## i) Design improvements

Developing sound intuition around structural problems can help you optimise your design much more quickly!

Consider the axial and bending contributors to the maximum combined stress in your structure. What is their relative magnitude? Could one of the contributions be neglected, as a rough approximation? If so, which one?

$$\sigma_{combined} = \sigma_{axial} + \sigma_{bending} = \frac{F}{A} + \frac{My}{I} \qquad x = \frac{\sigma_B}{\sigma_A}$$

Try to omit one of the two terms and re-assess the value of  $x$  for the two cross-sections A and B. Your calculations will be much simplified as you will not need all 'A', 'y' and 'I' parameters to calculate how much the stress will be reduced by (and, hence, how much the safety factor varies by).

Reflect: Is your simplified result far off the original one?

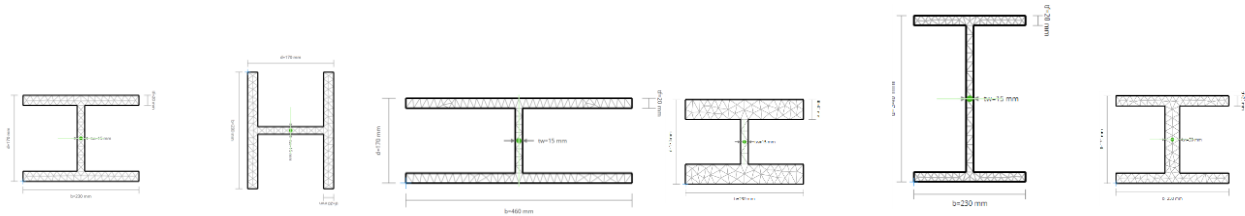
*When possible, engineers will try to design structures so that, in members that experience combined loading, one loading mode will dominate over others. This can make design troubleshooting and optimisation of a structure so much easier!*

## i) Design improvements

Let's now extend the exercise from the previous page. In the table below, you will find different variations of the I-beam cross-section. Figure out  $x$  (exact or approximate) for each option and then **answer the questions**:

**Q5: which member cross-section option would make your structure the strongest with respect to the loads considered in this tutorial?**

**Q6: which would make it the most expensive? (steel cost is a function of weight)**



	unit	I beam tested this tutorial	Rotated section (H beam)	Double Width	Double flange thickness	Double total depth	Double web thickness
Total depth (W1)	mm	170	170	170	170	340	170
Width (W2)	mm	230	230	460	230	230	230
Flange thickness (t1)	mm	20	20	20	40	20	20
Web thickness (t2)	mm	15	15	15	15	15	30
Area A	mm <sup>2</sup>	11,100	11,100	20,600	19,700	13,700	13,100
y (centroid distance)	mm	85	115	85	85	170	85
I (moment of inertia)	mm <sup>4</sup>	54 800 000	40 600 000	108 000 000	81 100 000	270 000 000	57 500 000

## j) Buckling analysis

The last step of the analysis is linear buckling analysis

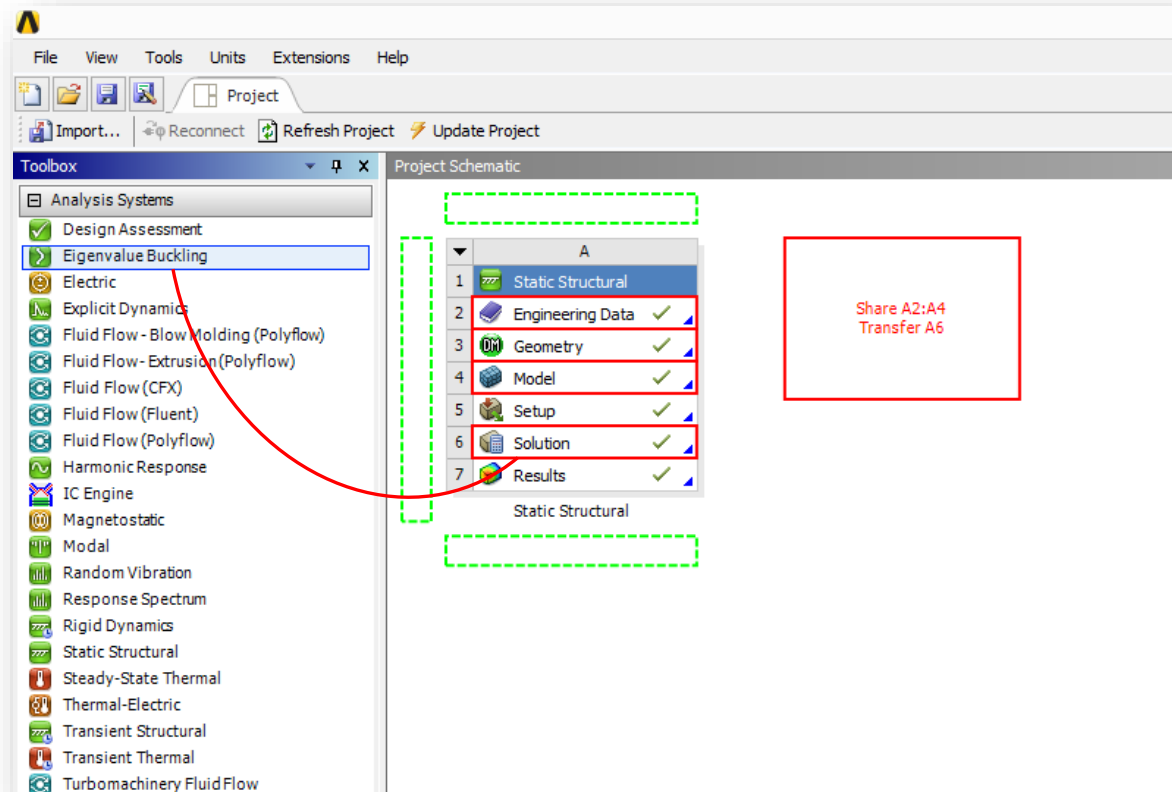
Buckling is the instability of the structure under compression. Very slender members, when compressed, may undergo major transversal deformation and fail under buckling effects (instead of pure compressive effects). See picture below:



The theory of buckling analysis is covered in chapter 13 of the Hibbeler textbook.

ANSYS provides a built-in buckling analysis module (called “**eigenvalue**” analysis).

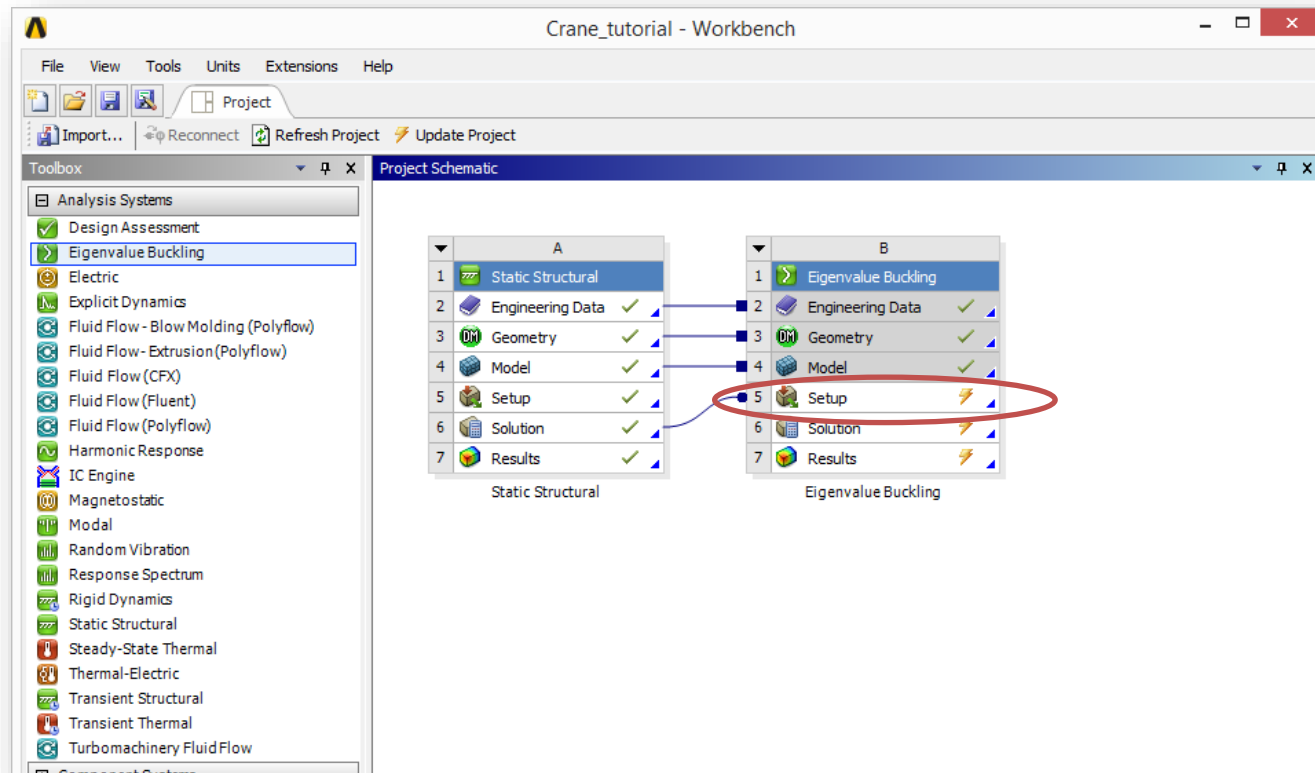
## j) Buckling analysis



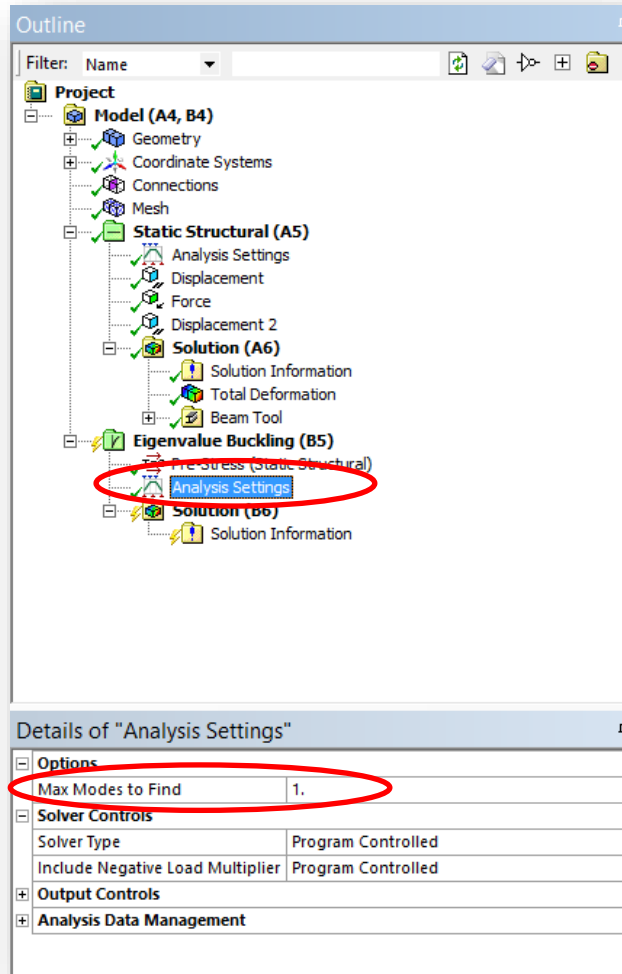
This step is quite tricky! Return to the workbench window and drag a new '**Eigenvalue Buckling**' module and place it on the '**Solution**' cell of the "Static Structural" module. This will automatically create connections (information sharing) between the two modules – See next slide.

## j) Buckling analysis

You should see some connections being created automatically between the two analysis modules. If this does not happen, you can create manual connections by clicking and dragging connection lines across the two modules.  
Now double click on the **'Setup'** box in the buckling analysis module.



## j) Buckling analysis



Click on the **Analysis Settings** and change **Max Modes to Find** from 2 (default) to 1.

*Note: "Max Modes to Find" indicates how many buckling modes (the different ways the structural member can deform) we want the software to identify.*

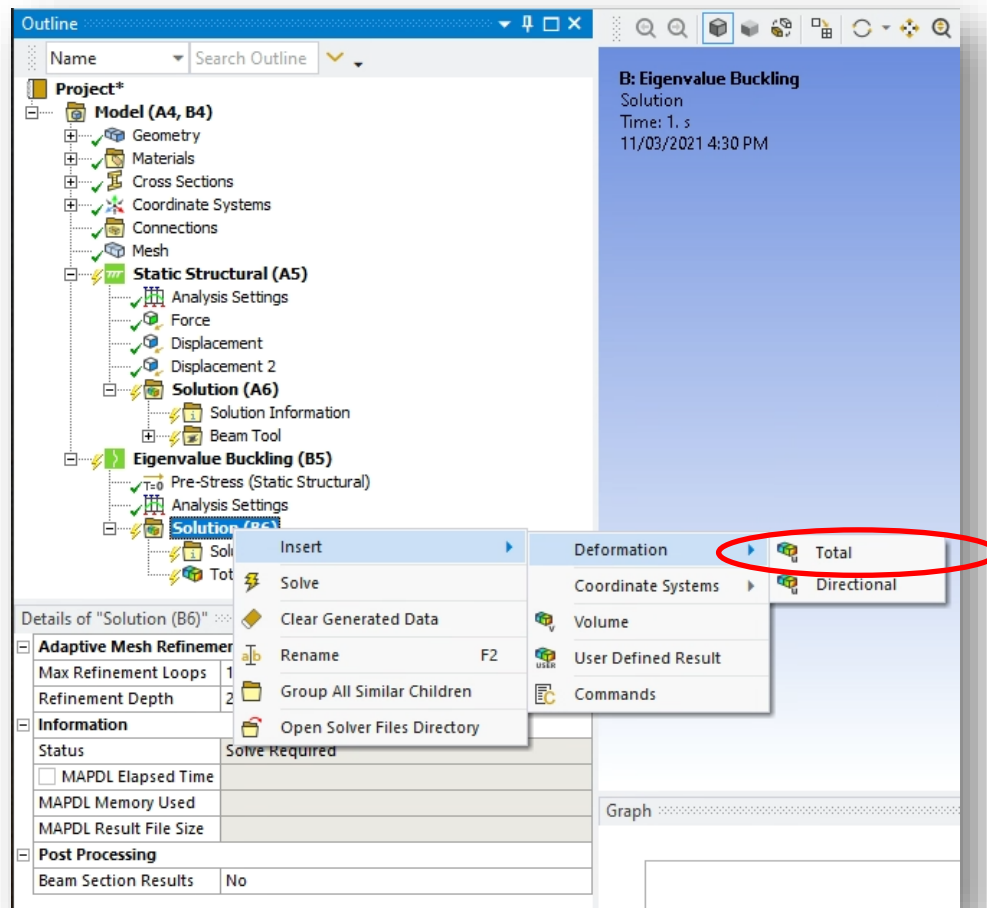
*e.g., if we set "Max Modes to Find=4", the model will return 4 results of different buckling shapes. These 4 shapes are triggered at different critical buckling stresses*

For simplicity, we will only test 1 buckling shape in this lab. If you are curious about buckling, you are always welcome to discuss with your tutor.



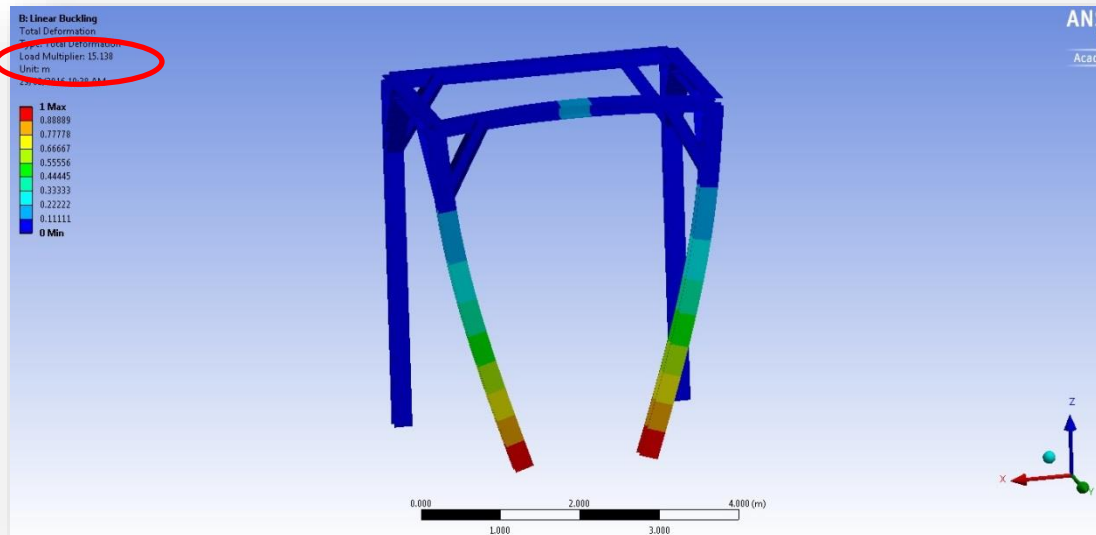
## j) Buckling analysis

Under **Eigenvalue Buckling**, right click on “**Solution**”, then select: **Insert, Deformation**, and choose **Total**. Then click **Solve** to start running the buckling computations.



## j) Buckling analysis

Once the computations are complete, you should see the buckled shape!



*Take note of the value of 'Load Multiplier' shown in the window. This is effectively a safety margin with respect to buckling.*

*The result is showing that the structure will buckle—and assume the deformed shape shown—for a load that is a multiple of the one simulated here. If the multiplier is bigger than 1, the structure will not experience buckling*

### Answer the following questions

**Q7. What is the value of load multiplier you have computed in your simulation?**

**Q8. In terms of safety, how does the load multiplier compare with the safety margin you calculated earlier on? What type of failure will this structure exhibit most likely?**

**Q9. What design changes could the structure undergo so that its cost may be reduced, without compromising safety?**

## k) Conclusion

This tutorial has **introduced you to the key steps to analyse a simple structure using ANSYS:**

- Geometry and cross-sections definition
- Boundary conditions definition
- Mesh generation
- Solution computation
- Results review

A key takeaway of this activity is that **the choice of cross-section in structural members is a fundamental aspect of structural design**: different choices of cross-sections will result in different mechanical behaviour of the structure. Cross-section geometry should be tailored to the specific loading problem at hand. The 'H' beam used here was recommended based on existing knowledge.

What should you do first when you have no idea about the nature of a structural problem?

- Consider: will members experience mostly bending or axial stress?
  - In some cases, you can intuitively guess this; otherwise, you can use ANSYS to find out!
- Analyse: spend some words/equations to frame the problem using mechanical theory; consider free body diagrams and cross section geometries.
  - Guiding your approach with theory and calculations will lead to a better initial choice.
- Remember that design is often an iterative process and the first solution is rarely the best one

A good engineer **can always logically justify their design choices.**

## k) Conclusion

The theory and equations illustrated in the previous pages allow a design engineer to evaluate trade-offs when making choices between stronger vs weaker beams, as appropriate to ensure that their design fits specific requirement (e.g., cost, safety, strength).

Beams come in standard sizes and there are technical catalogues available online providing essential information for engineers.

- For an example, you can check out the Liberty Steel catalogue by [following this link](#).

When considering structures, there are three main things a designer needs to be concerned about: **stress**, **deflection** and **buckling**.

Which aspect will your design be the weakest against? Once the weakness of your design has been exposed (in this tutorial, you have seen how ANSYS can assist in determining this), you will be able to modify your design choices, if/as required, to address that particular issue.

## I) Recommendations

- It is much more efficient to solve ANSYS problems in person rather than via e-mail. Please contact your tutors and arrange to attend a lab session so that you can sit down and figure things out together.
- If you are unsure of something, ask early! This is a learning exercise, not an assessment!

**Good luck!**

# Deliverables

To obtain the marks for this lab you need to do the following:

- ✓ 1. Provide answers to the nine questions  
(you are allowed to ask the tutors for help if you are having difficulties)
- ✓ 2. Show your ANSYS work to the tutors during the lab.