MAEG 5130 GUIDE CASE

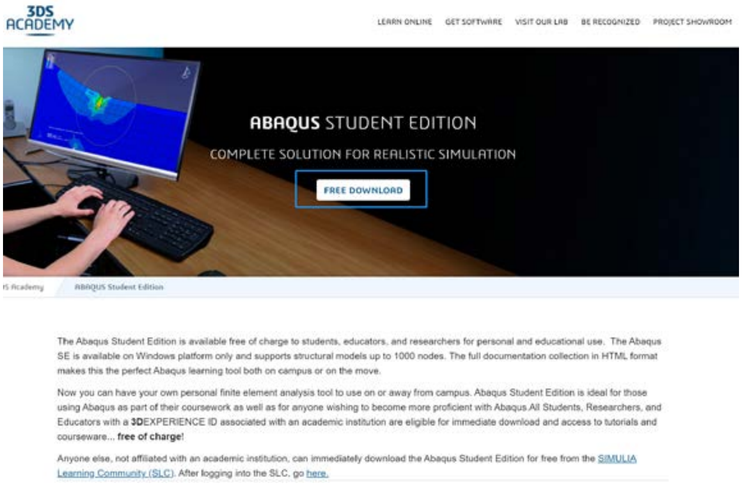
Deflection of Beam Simulated by Abaqus

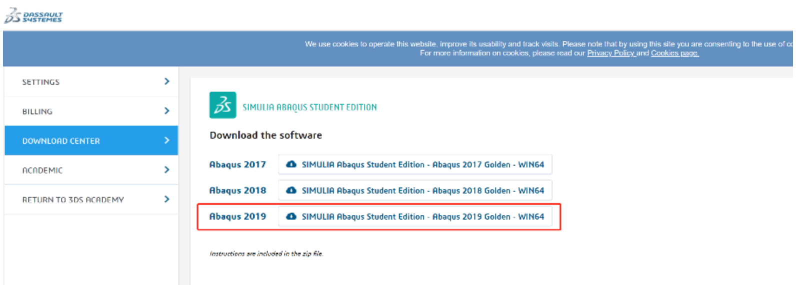
This is a guidance for lab 1 based on Windows 10. Other OS is similar. The report requirement is on the end of this guidance.

1 Preparation

1.1 Download Abaqus student version

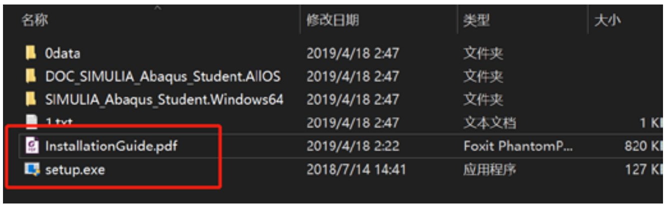
Register at https://academy.3ds.com/en/software/abaqus-student-edition, and then download the software of 2019 version





1.2 Installation

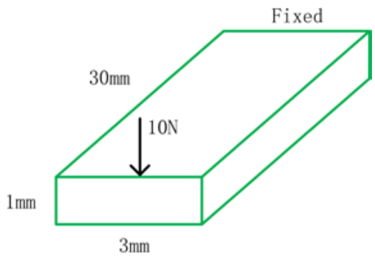
UnZip the file and find ‘InstallationGuide.pdf’, which will tell you how to install it. Remember to read it first before you install it.



2 Simulation process

The following is an example, but in your lab report you need to use different force and global seeds. You can not submit this example as your lab report.

2.1 Problem



The dimension of the part is shown above and we apply a force, 10N, on one end, while the other end is fixed. Now, we will use Abaqus to solve this problem. We assume this part is made by a kind of steel with Young's Modulus, $209000MPa$, and Poisson ratio 0.269.

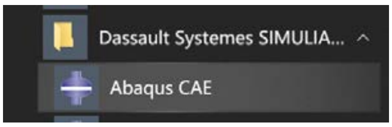
2.2 Unit

As Abaqus do not contain unit, you need to remember the unit you use and the unit of the results. The table below shows two common units in Abaqus. Here I choose the first one for this guidance, but in your report, you should choose the first one.

Length	Force	Mass	Time	Stress	Energy	Density	Ace
m	N	kg	s	Pa	J	kg/m^3	m/s^2
mm	N	kg	s	MPa	mJ	$tonne/mm^3$	mm/s^2

2.3 Launch the software

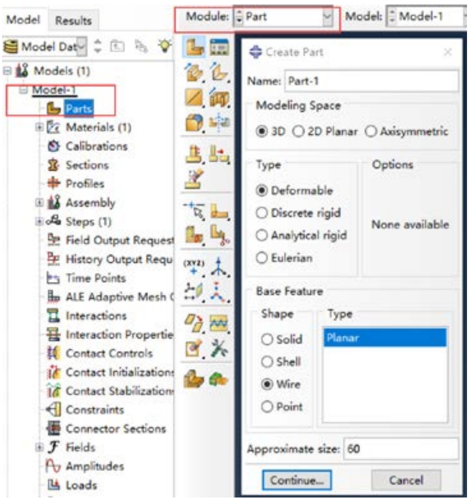
Press 'Startup' bottom to launch the software.



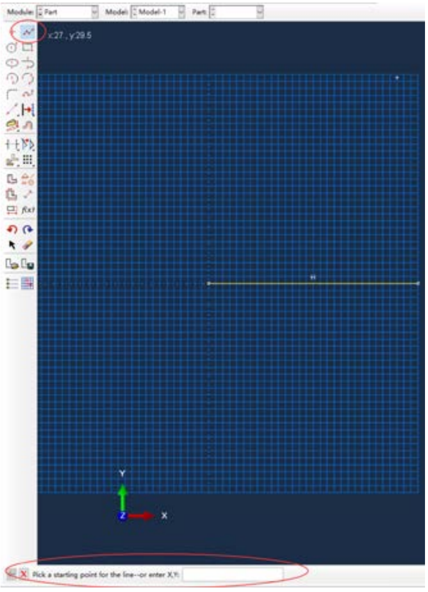
2.4 Create the part

As the length is much larger than the width and height, we model this part as a line.

Create the part with 'Standard/Explicit Model'. In the model database tree, under 'Model-Part', right click and the select 'create', then define the part like the following picture. Another method to create part is select 'Part' in 'Module' (combobox). Then click 'Continue'.

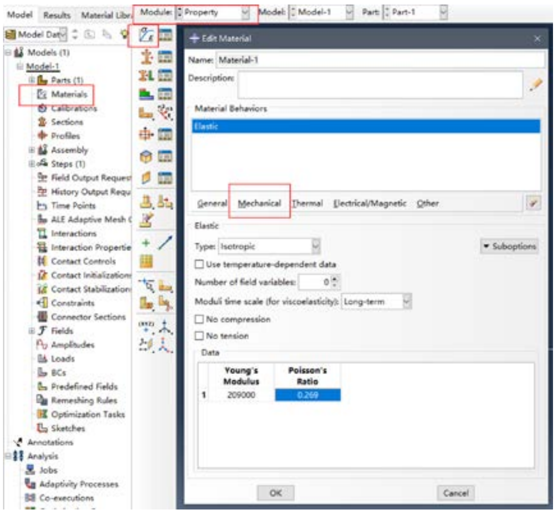


Next, choose command 'Line', and then input the two coordinate of the line, (0,0) and (30,0) in the bottom, prompt box. And follow the instruction in the bottom to finish the part.

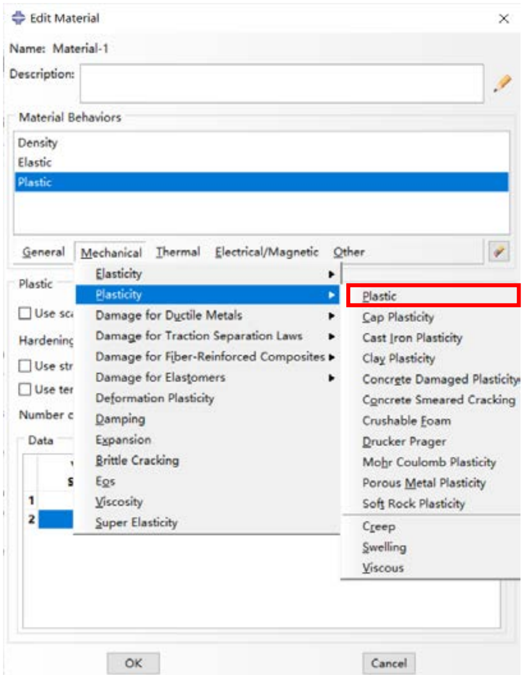


2.5 Define the material property

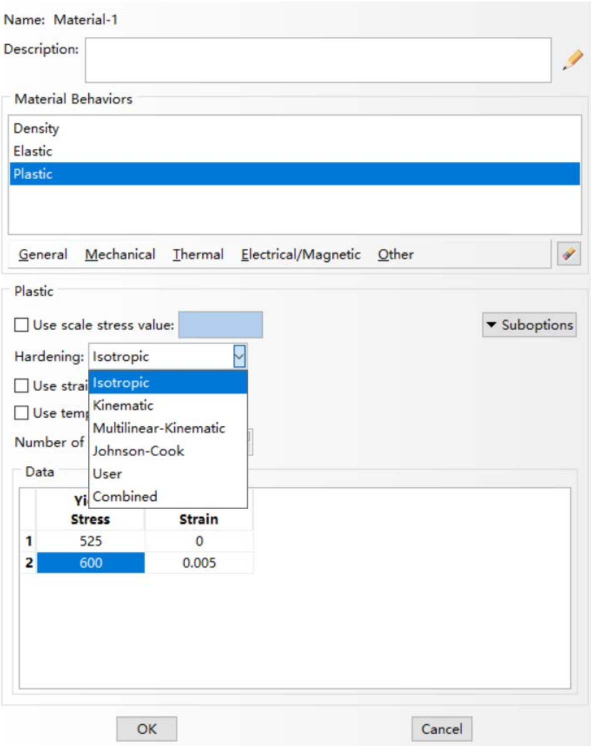
In the model database tree, double click on Materials, and then edit the materials for this part. Define the Young's modulus in Mechanical-Elastic.



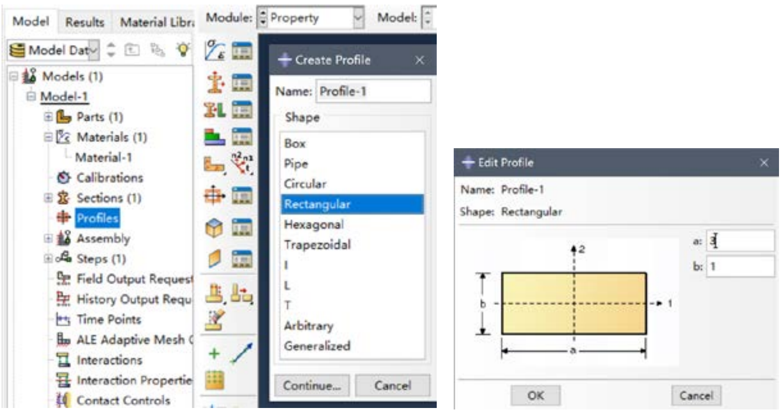
In this project, different hardening laws can be added by Mechanical->Plasticity->Plastic.



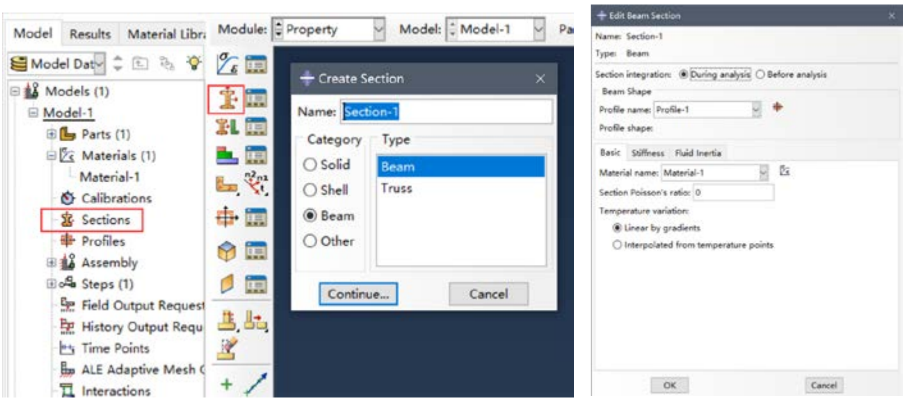
Then click the drop-down icon besides ***Hardening*** option. For linear isotropic plasticity, choose Isotropic in the list; while pick hardening for linear kinematic plasticity.



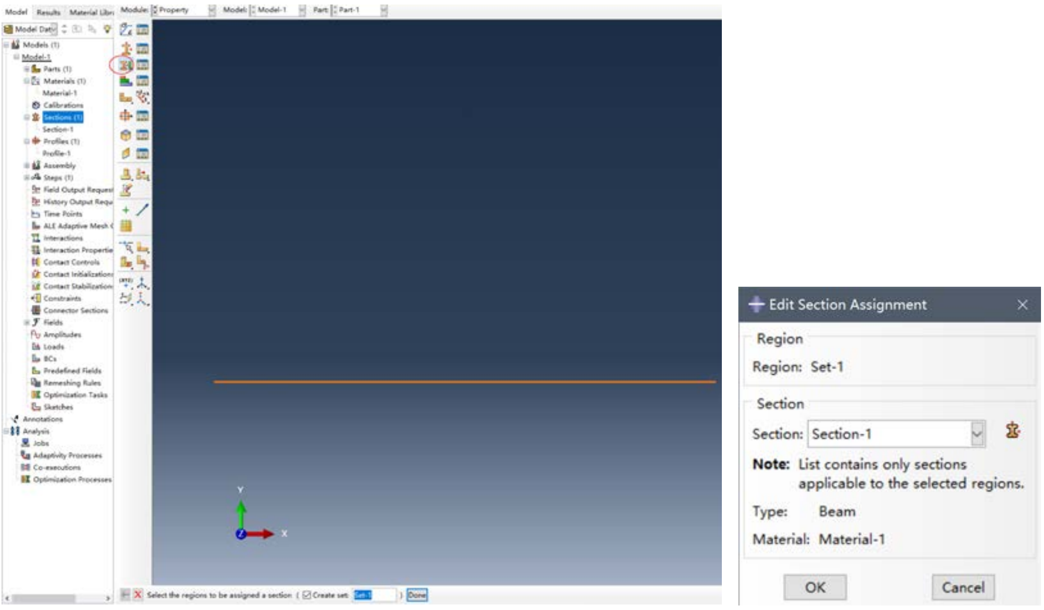
2.6 Define the profile of the part.



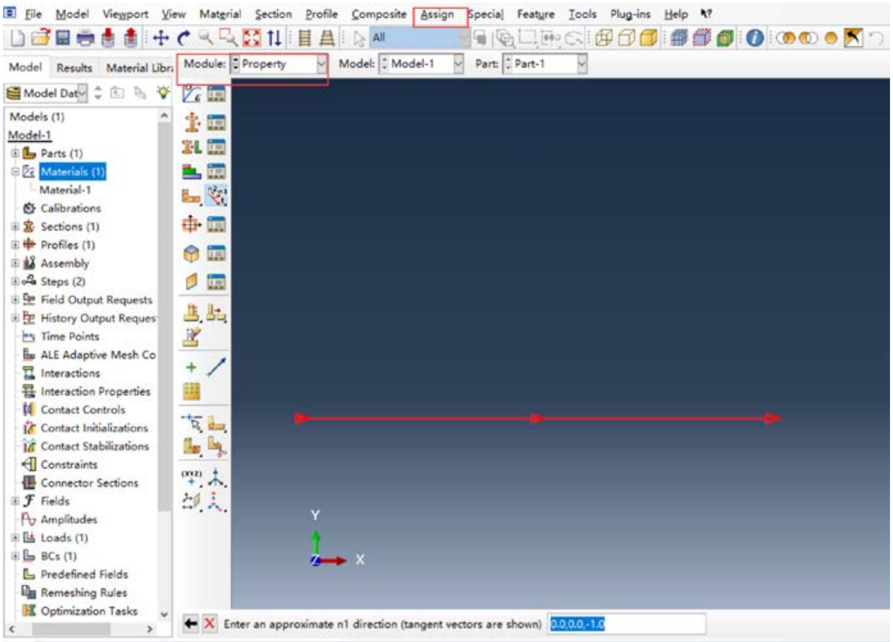
Define a section as the picture below and keep the material as ‘Material-1’ if you do not change the name of the material you defined.



Assign the section. Click the ‘Assign Section’ button and choose the line we created.

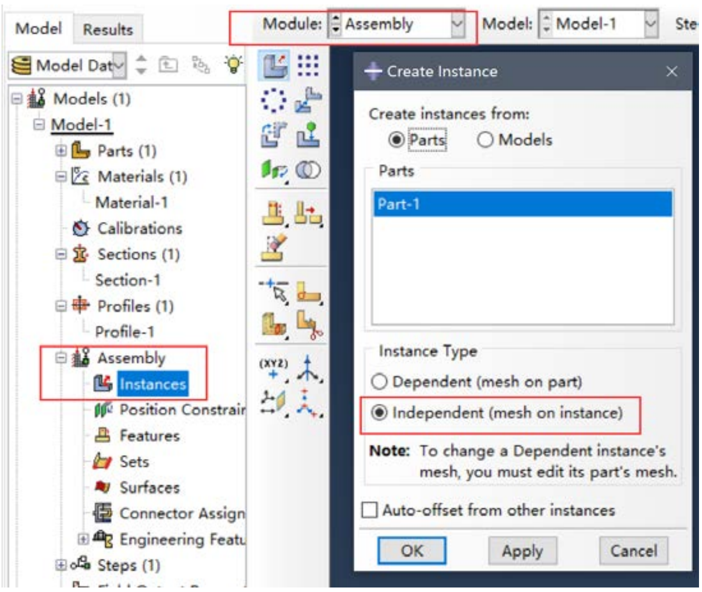


Assign the beam section oritation in the assign menu. Then select the line and click done. Define the direction as default.



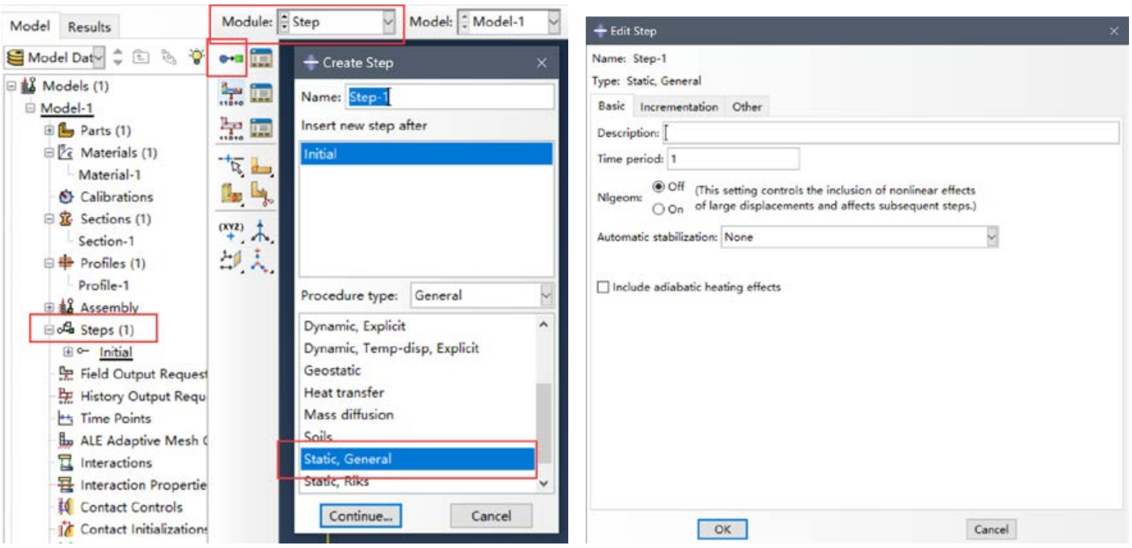
2.7 Assembly

Click the ‘Create Instance’ button, create instances from the part we created as Independent.



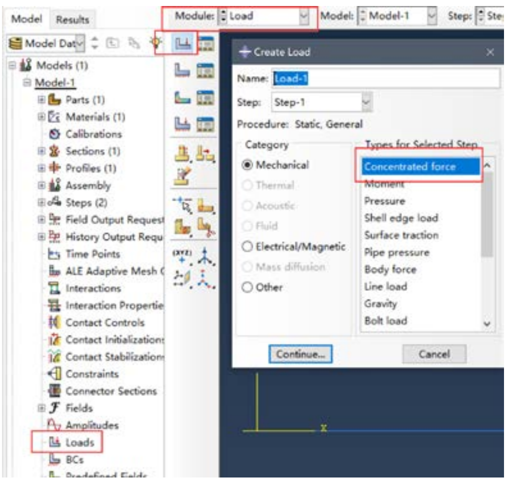
2.8 Step

Change the Module to Step, and Click ‘Create Step’ button. Keep the Edit Step windows default.

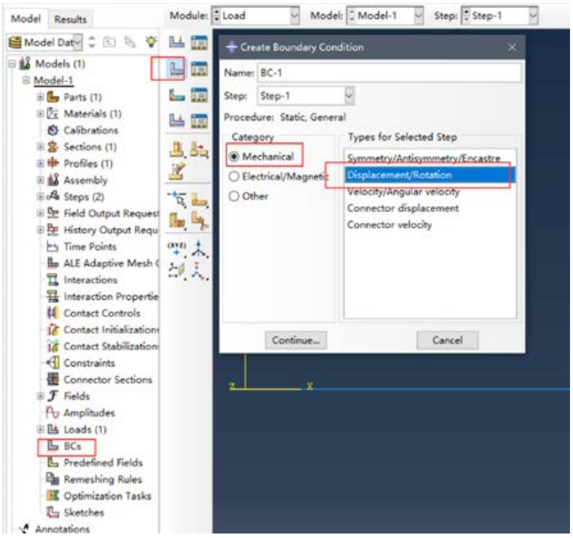


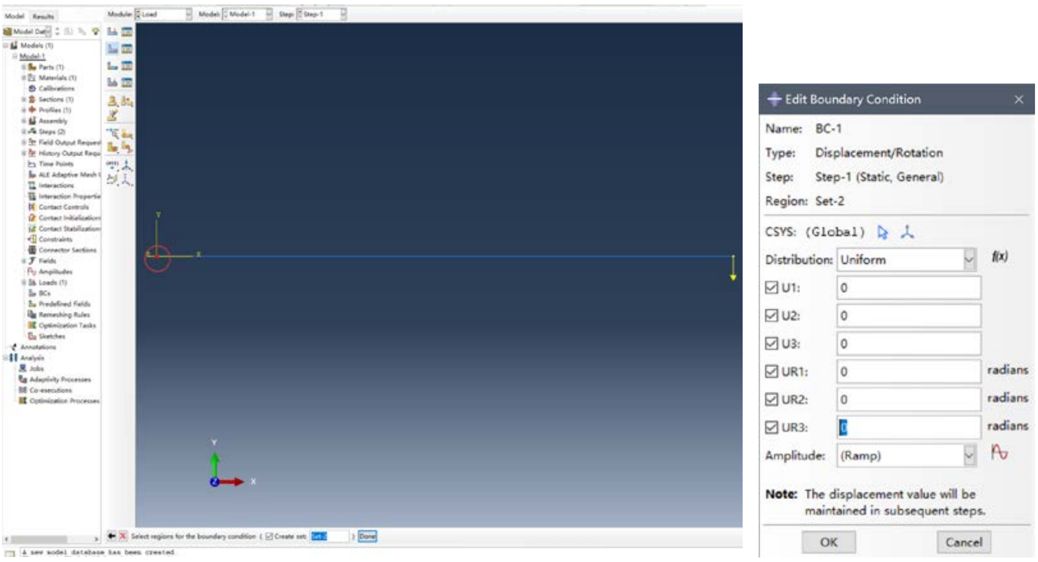
2.9 Load

Change the Module to Load, and Click 'Create Load' button to create a Concentrated force on the end of the line (Select the point). Then, define the force on -y direction.



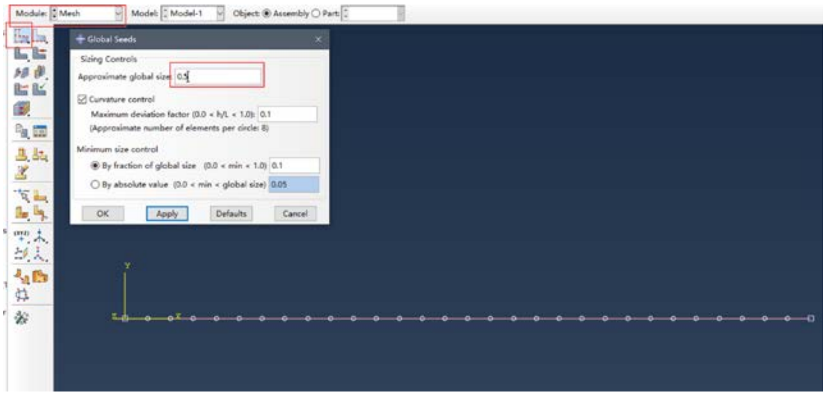
Next, we will fix the other end by adding boundary condition. Define the displace and rotation on the other point is 0.

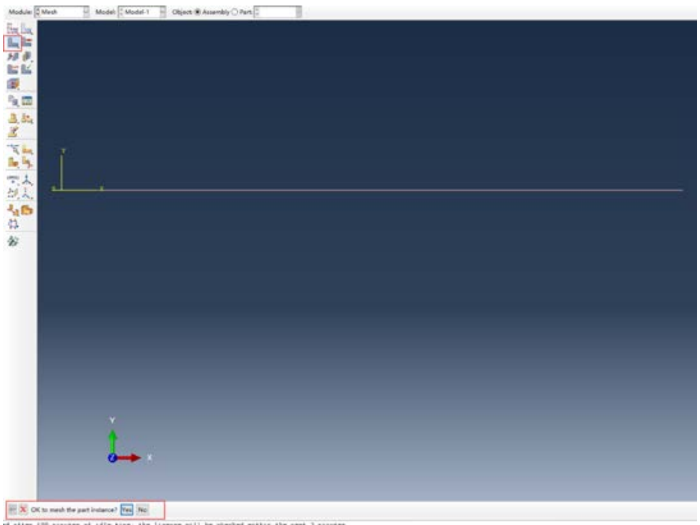




2.10 Mesh the assembly

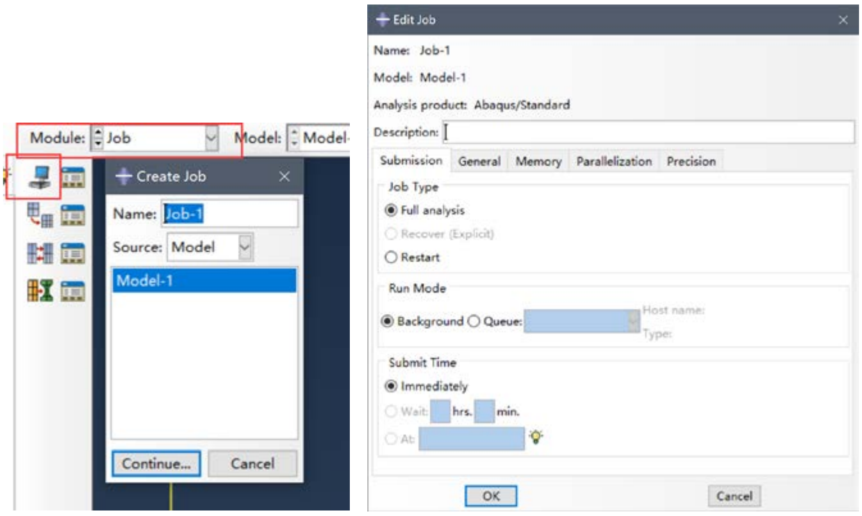
Change the Module to Load, and Click ‘Seed Part Instance’ button to define the global size as 0.5, and then click ‘done’ on the bottom to finish. In your report, define a different global size. Next, click ‘Mesh part instance’ to mesh the part.



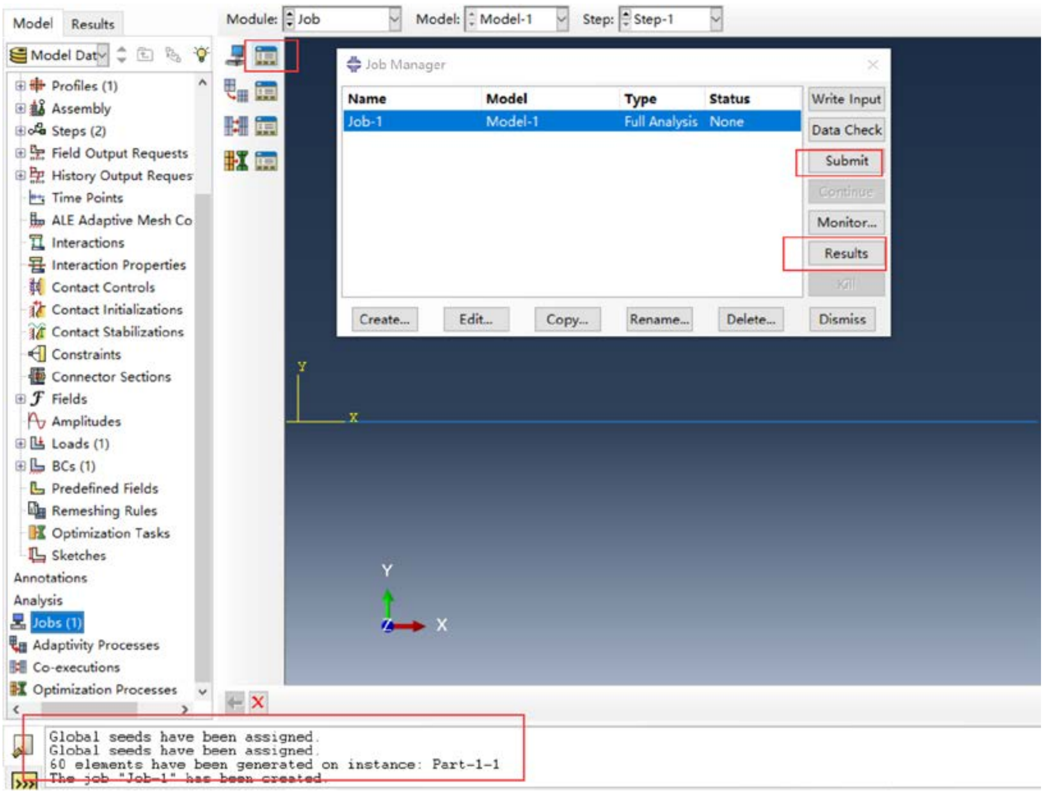


2.11 Job

Now, we come to analysis by creating the Job of the analysis. Change the Module to Load, and Click ‘Create Job’ button to create a job and remain the job parameter default.



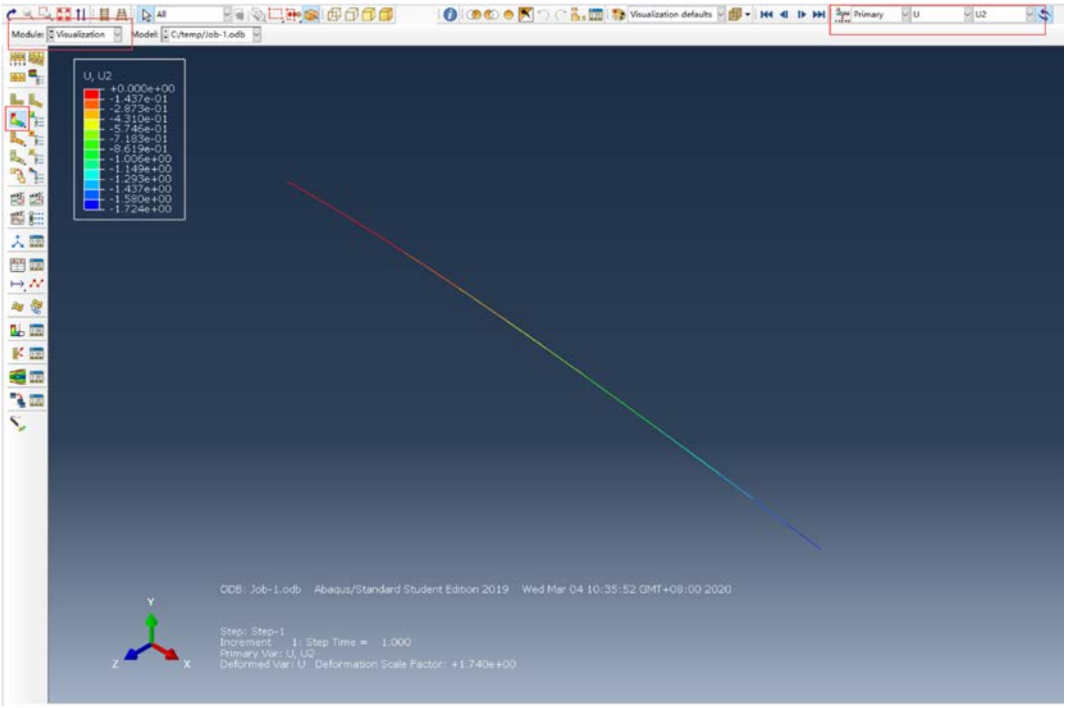
Then, open the ‘Job Manager’, and submit the job to solver. The processing information is shown in the bottom.



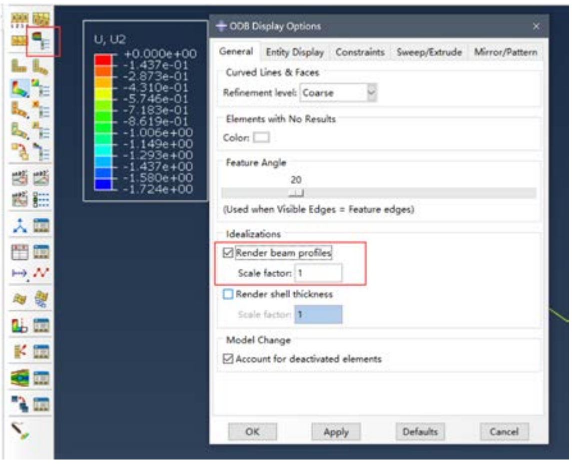
After the analysis is finished, click ‘Results’ to check the results.

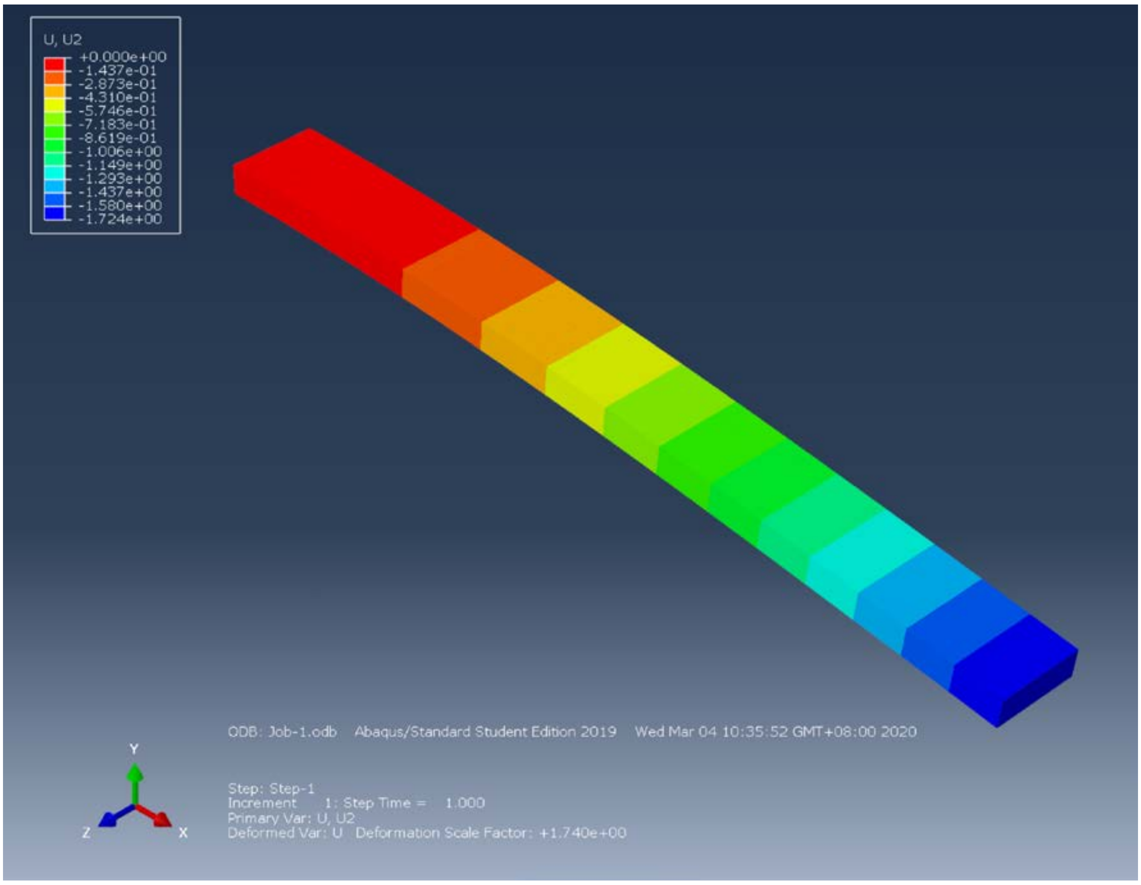
2.12 Visualization

After click ‘Results’ button, the modulus will change to ‘Visualization’. Find ‘Plot Contours on Deformed Shape’ command and go to the upper-right corner, change the combobox to U (displacement) and U2 (Y direction). Then, we can see the line with different color, and the upper-left corner shows the value related to the color. Here, we know the maximum displacement is on the tip of the line, where we applied the force on. The maximum displacement is 1.724 and the unit we use in this analysis is mm.



Since we have defined the profile of this part, now we are going to show the actual 3D shape by clicking the ‘ODB Display Options’ to render the beam profiles.





3 Calculate

Here, we will calculate the displacement manually using beam theory.

For a cantilever beam,

The deflection at the free end is given by:

$$f_m = \frac{F \cdot L^3}{3 \cdot E \cdot I}$$

Where

L: Distance between the supporting points

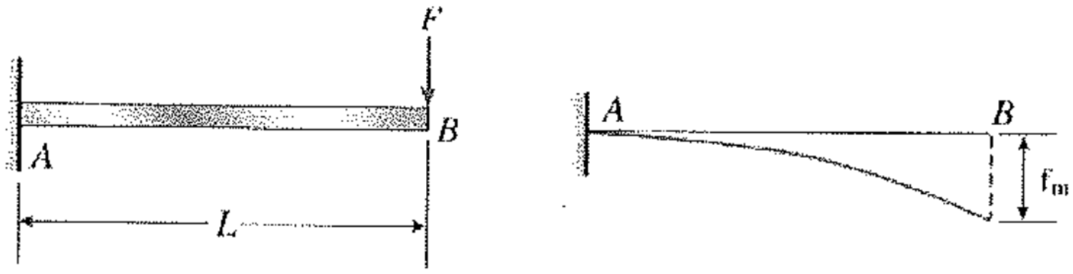
F: Deflection of beam

f_m Maximum deflection

F Applied force

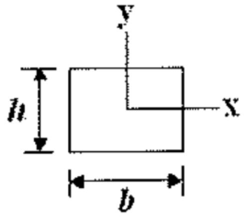
E Modulus of elasticity

I Moment of inertia of the cross-section



The moment of inertia, I, with respect to the x axis, for a rectangular section is given by:

$$I = \frac{b \cdot h^3}{12}$$



h: the dimension in the plane of bending (section height)
b: section width of the beam

verification

$$f_m = \frac{10 \times 0.03^3}{3 \times 209 \times 10^9 \times \frac{1}{12} \times 0.003 \times 0.001^3} m = 1.7225 \times 10^{-3} m$$

Thus, the simulation result is nearly the same.

4 Report requirements

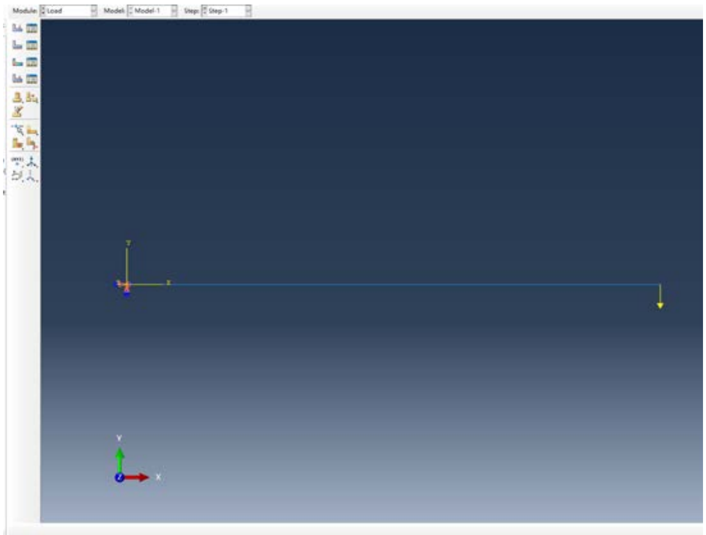
1. Redo this simulation with unit SI (m);

Length	Force	Mass	Time	Stress	Energy	Density	Ace
m	N	kg	s	Pa	J	kg/m^3	m/s^2

2. Define your force different from 10N, from 5 to 20;
3. When mesh your part, use a different global seed;
4. Render the beam profiles;
5. Calculate the displacement manually by beam theory.

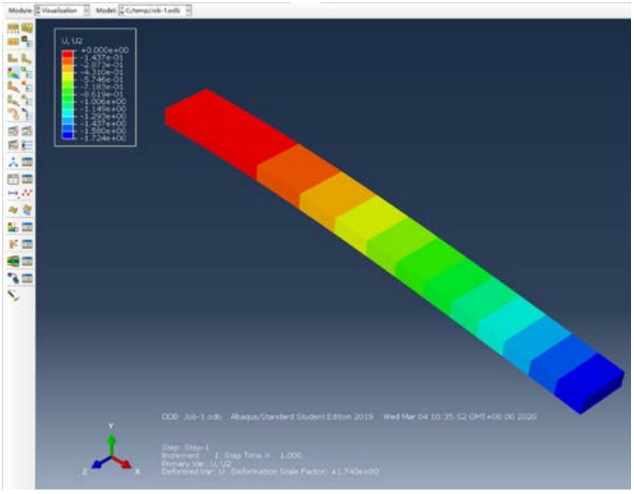
Your report must contain the following content.

a) Capture a picture of the load and constrain;



b) Provide the global seed you used in your simulation;

c) Capture a picture of the simulation results with the beam profiles rendered;



d) Provide the theoretical analysis by beam model;

e) Discuss the simulation results and theoretical analysis;

f) This is not a team work. Every student should finish it by themselves and submit a report.