



PennState
College of Engineering

ME 461: Finite Elements in Engineering

Comparing the FE theory to Abaqus

Guide Packet



Table of Contents:

1. Introduction.....	3
2. Matlab Component.....	3
3. Geometry Creation	4
4. Assign Properties	7
5. Assembly	10
6. Create Step	11
7. Create Boundary Conditions	12
8. Create Mesh	17
9. Create a Job	19
10. Visualize the Results	20

1. Introduction

Let's compare our simple problem by hand (using matlab to run the numbers) and abaqus

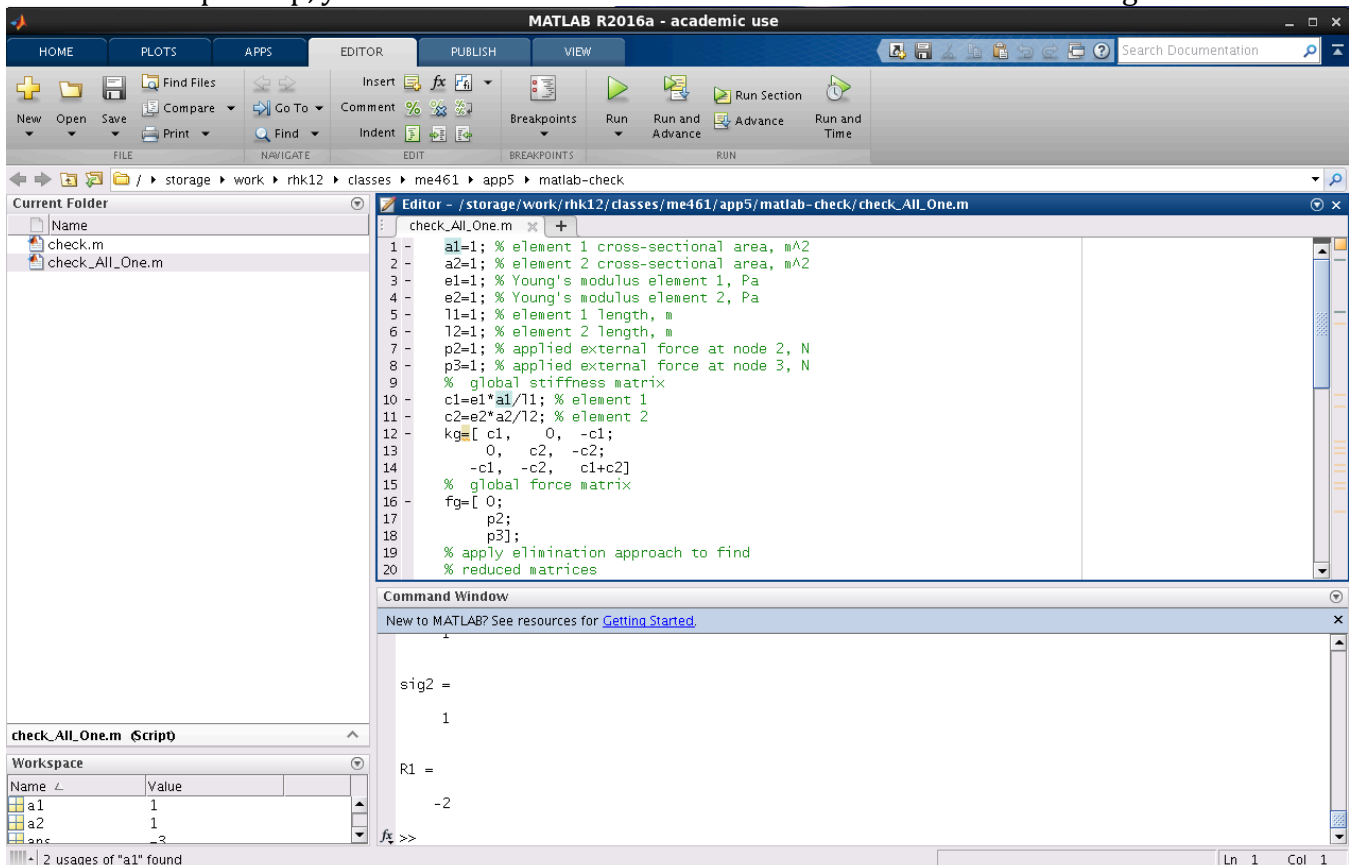
Once on ACI, open a browser and for the matlab part, download and run the check_All_One.m file from Canvas. This is the analytical solution we computed by hand, but I put it into matlab so you can change the numbers if you want to explore different cases in Abaqus.

2. Matlab Component

To run matlab do the following:

```
[rhk12@comp-int-10 matlab-check]$ module load matlab
[rhk12@comp-int-10 matlab-check]$ matlab &
2) 4790
[rhk12@comp-int-10 matlab-check]$
```

Once matlab opens up, you can load the m file and run it. You should see something like this:



The screenshot shows the MATLAB R2016a - academic use interface. The Editor window displays the script `check_All_One.m` with the following code:

```
1 - a1=1; % element 1 cross-sectional area, m^2
2 - a2=1; % element 2 cross-sectional area, m^2
3 - e1=1; % Young's modulus element 1, Pa
4 - e2=1; % Young's modulus element 2, Pa
5 - l1=1; % element 1 length, m
6 - l2=1; % element 2 length, m
7 - p2=1; % applied external force at node 2, N
8 - p3=1; % applied external force at node 3, N
9 - % global stiffness matrix
10 - c1=e1*a1/l1; % element 1
11 - c2=e2*a2/l2; % element 2
12 - kg=[ c1, 0, -c1;
13 -      0, c2, -c2;
14 -      -c1, -c2, c1+c2]
15 - % global force matrix
16 - fg=[ 0;
17 -      p2;
18 -      p3];
19 - % apply elimination approach to find
20 - % reduced matrices
```

The Command Window shows the output of the script:

```
1
sig2 =
    1
R1 =
    -2
```

The Workspace window shows the following variables:

Name	Value
a1	1
a2	1
ans	-3

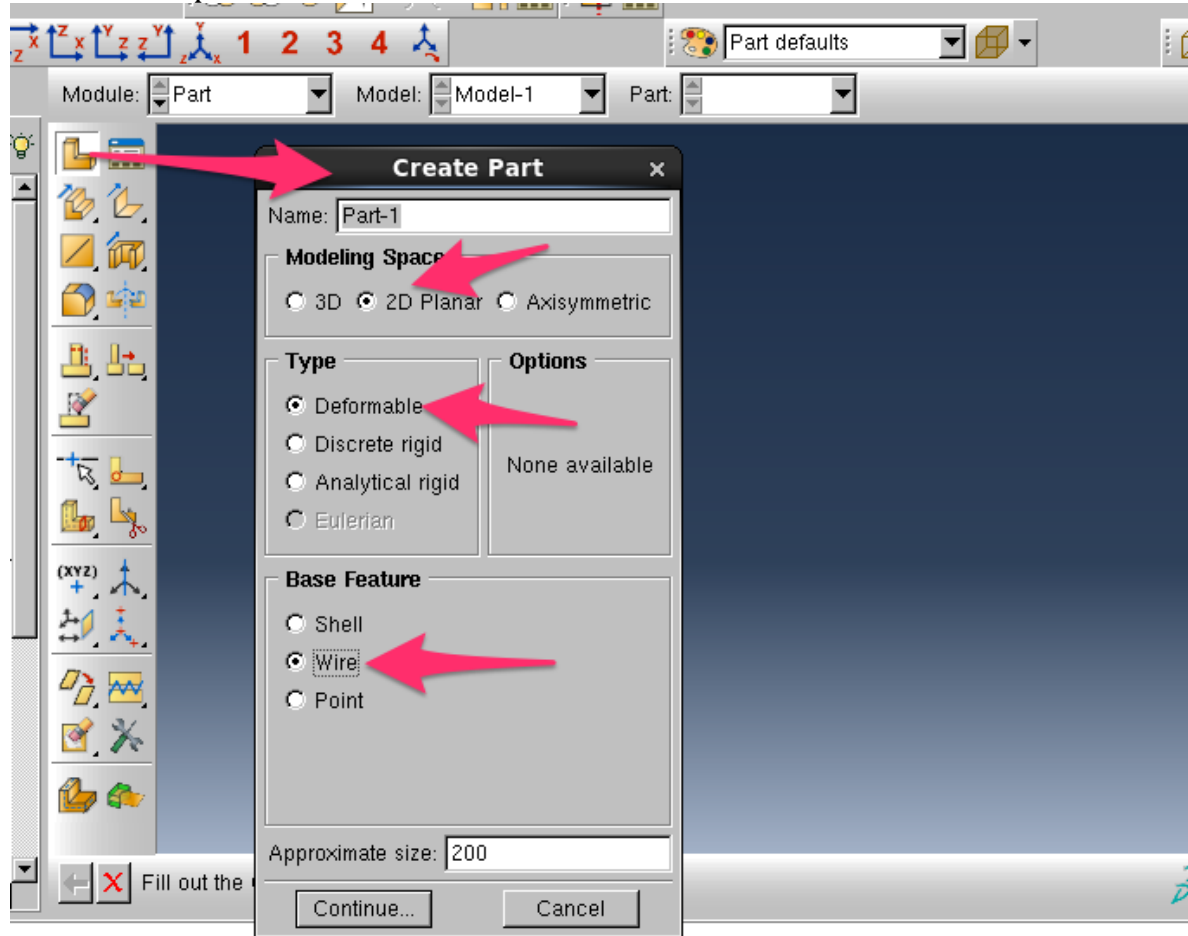
The Command Window also displays a message: "New to MATLAB? See resources for [Getting Started](#)."

You can compare the numbers to the values we computed in class.

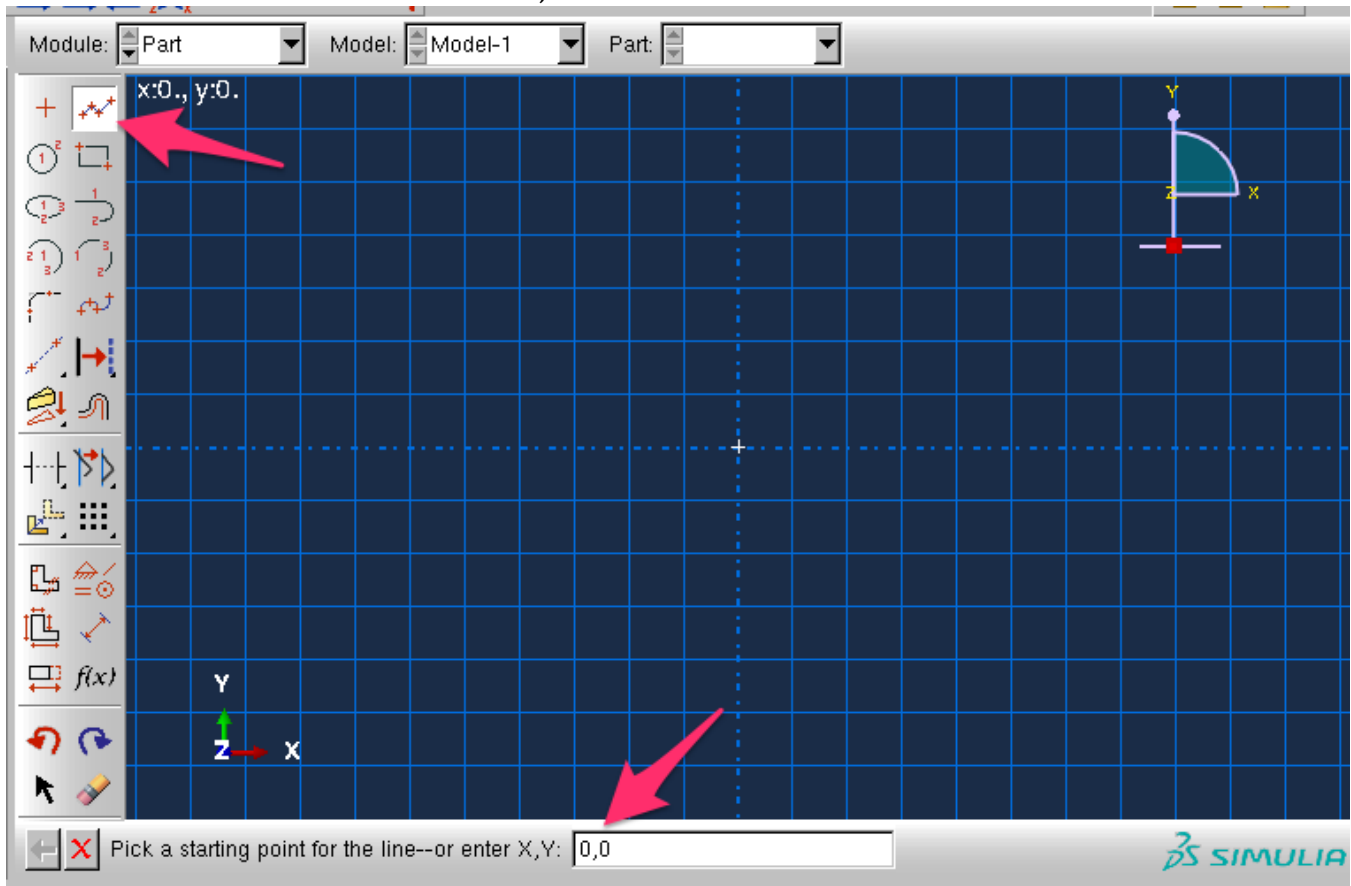
Next we will move on to the Abaqus part.

3. Geometry Creation

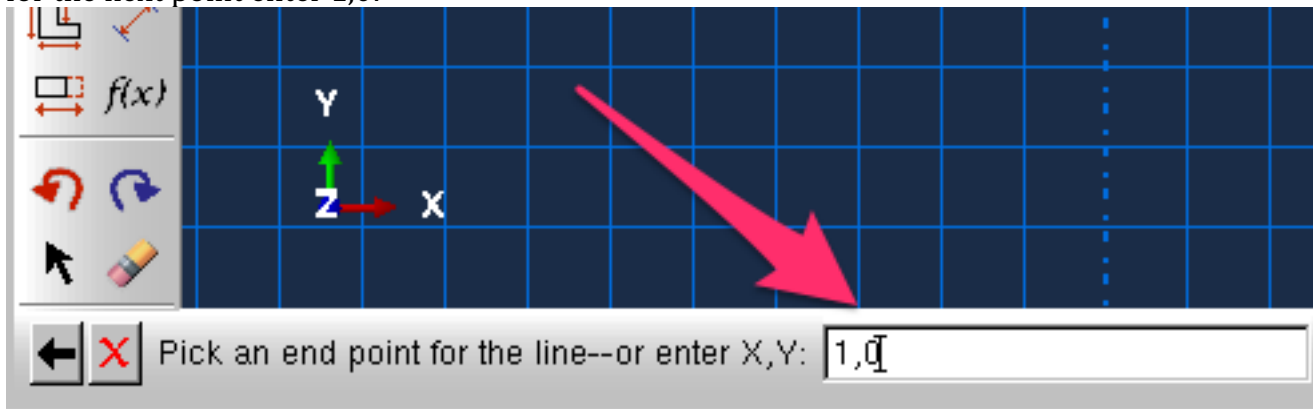
Create a new part:



Click the line tool and start the line at 0,0:

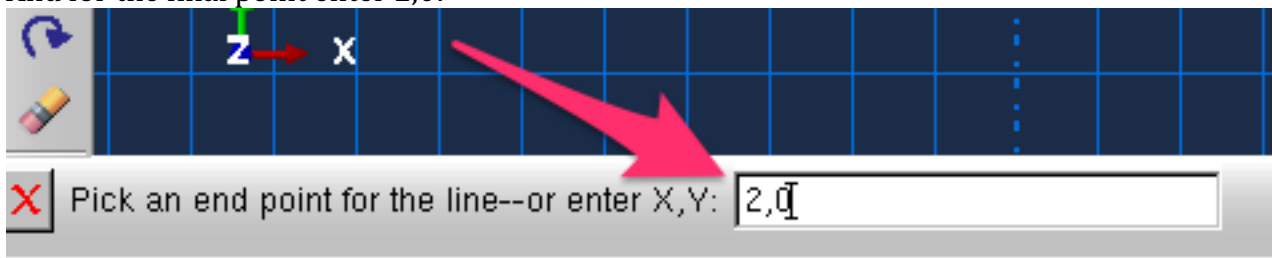


for the next point enter 1,0:

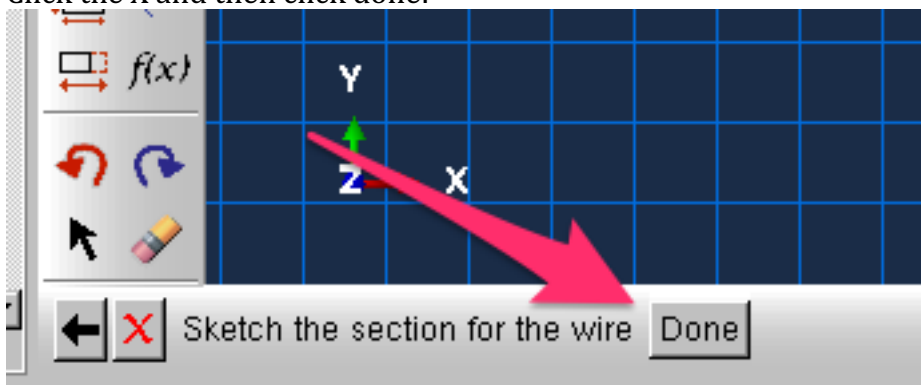




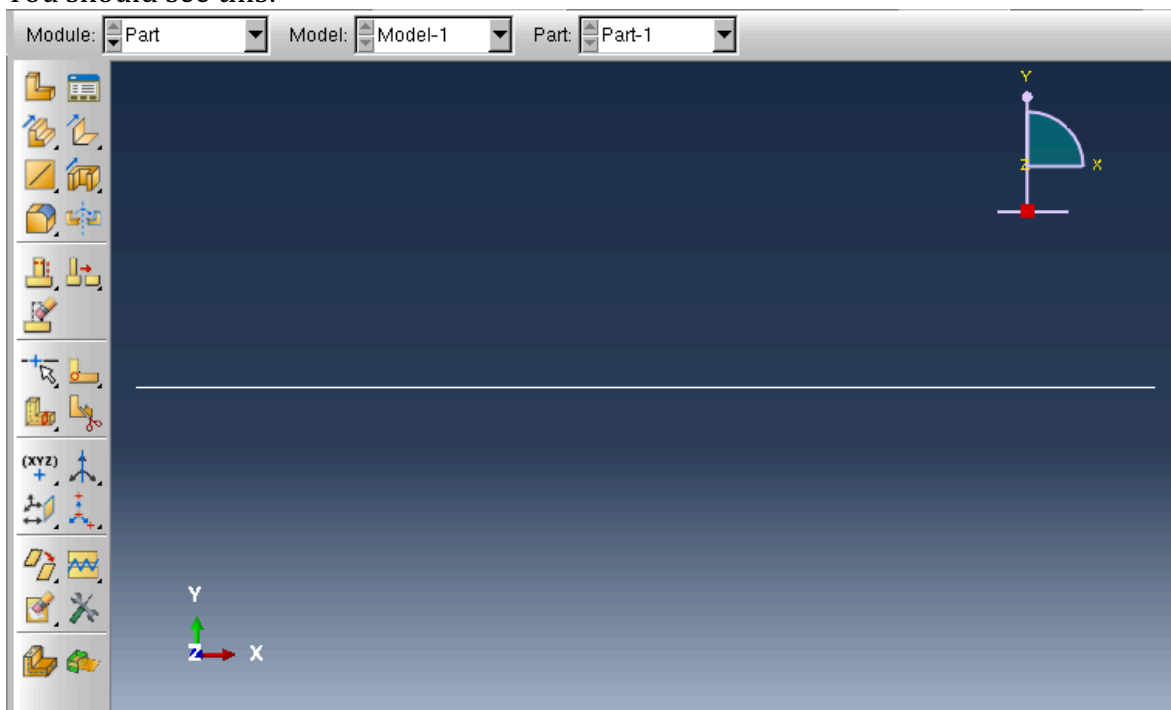
And for the final point enter 2,0:



Click the X and then click done:

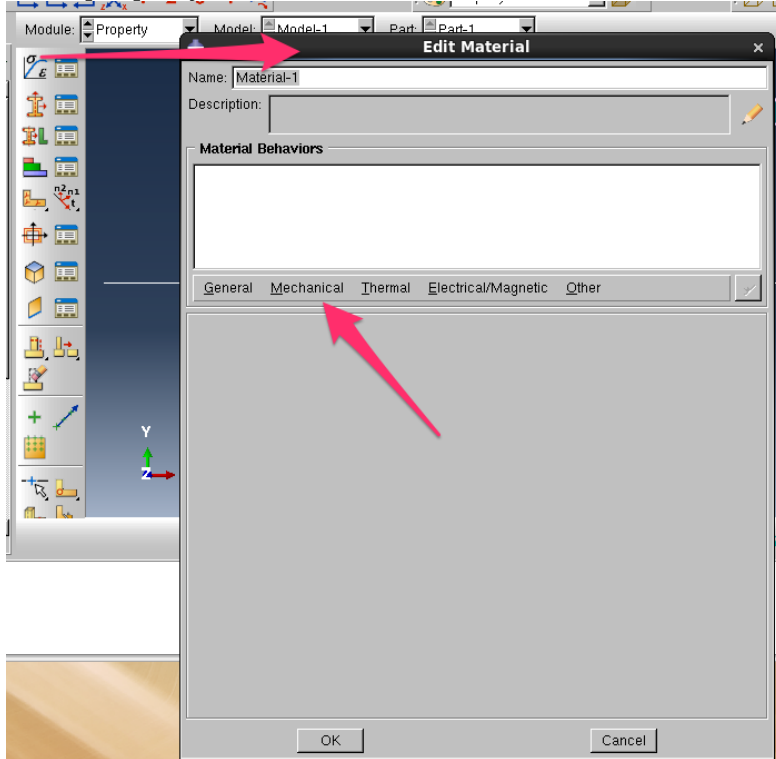


You should see this:



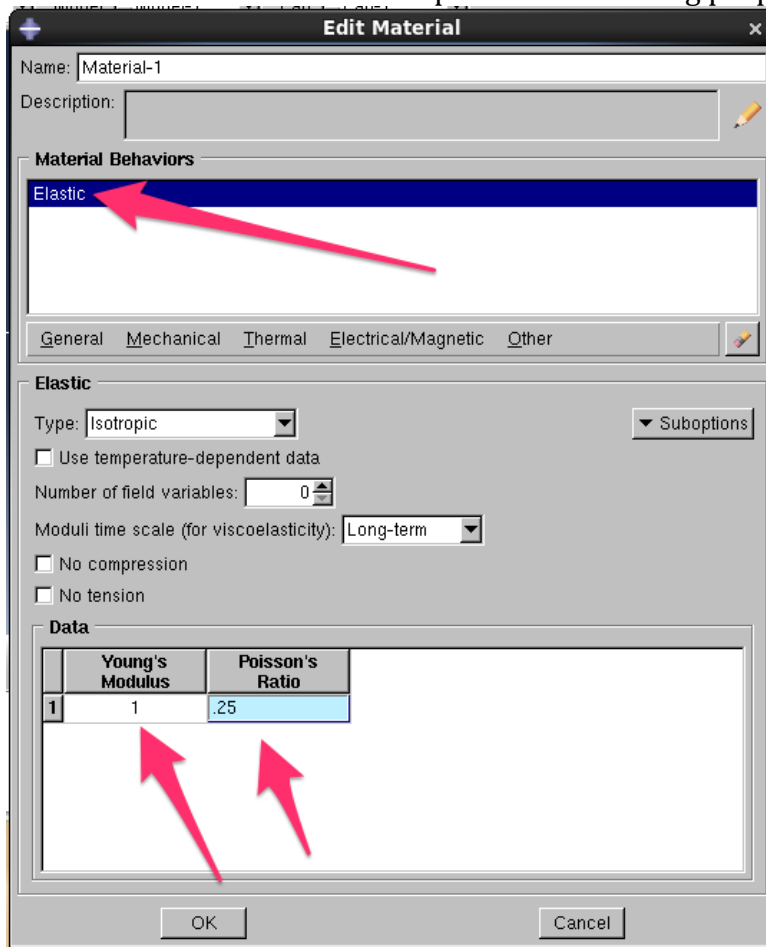
4. Assign Properties

Go to property module and add a new material:

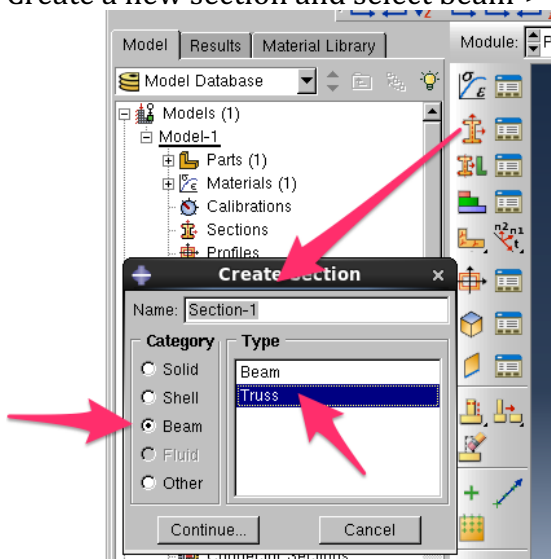




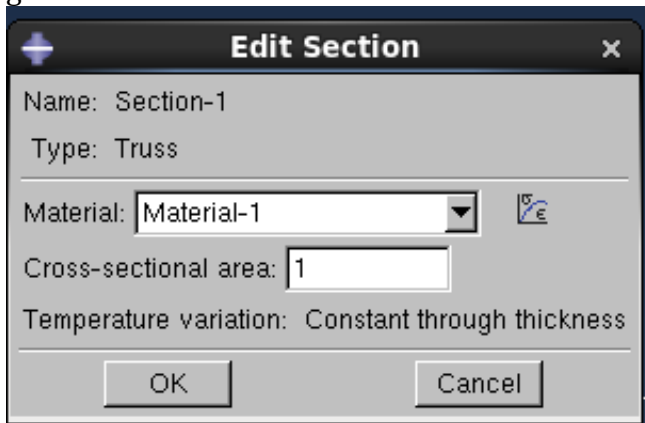
Go with an elastic material and put in the following properties:



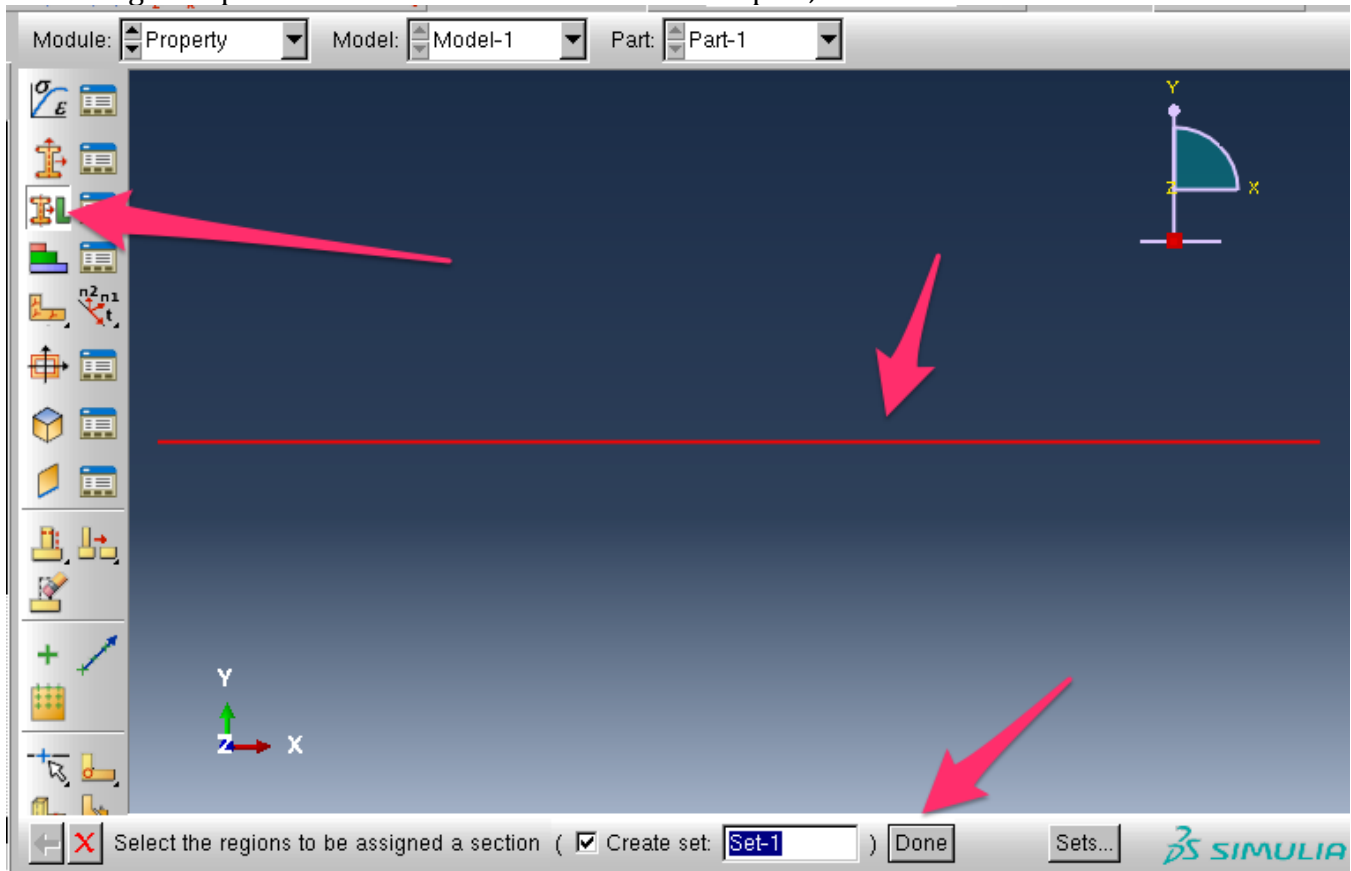
Create a new section and select beam-> truss:



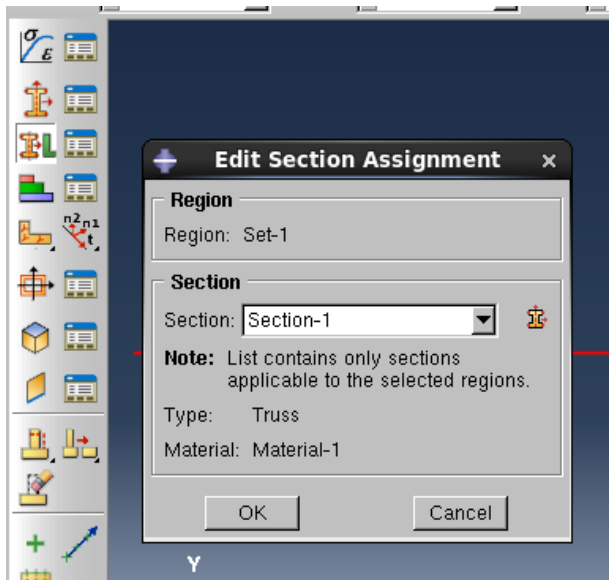
give the section a cross-sectional area of 1:



Now assign the part to the section. To select the entire part, use the box select:

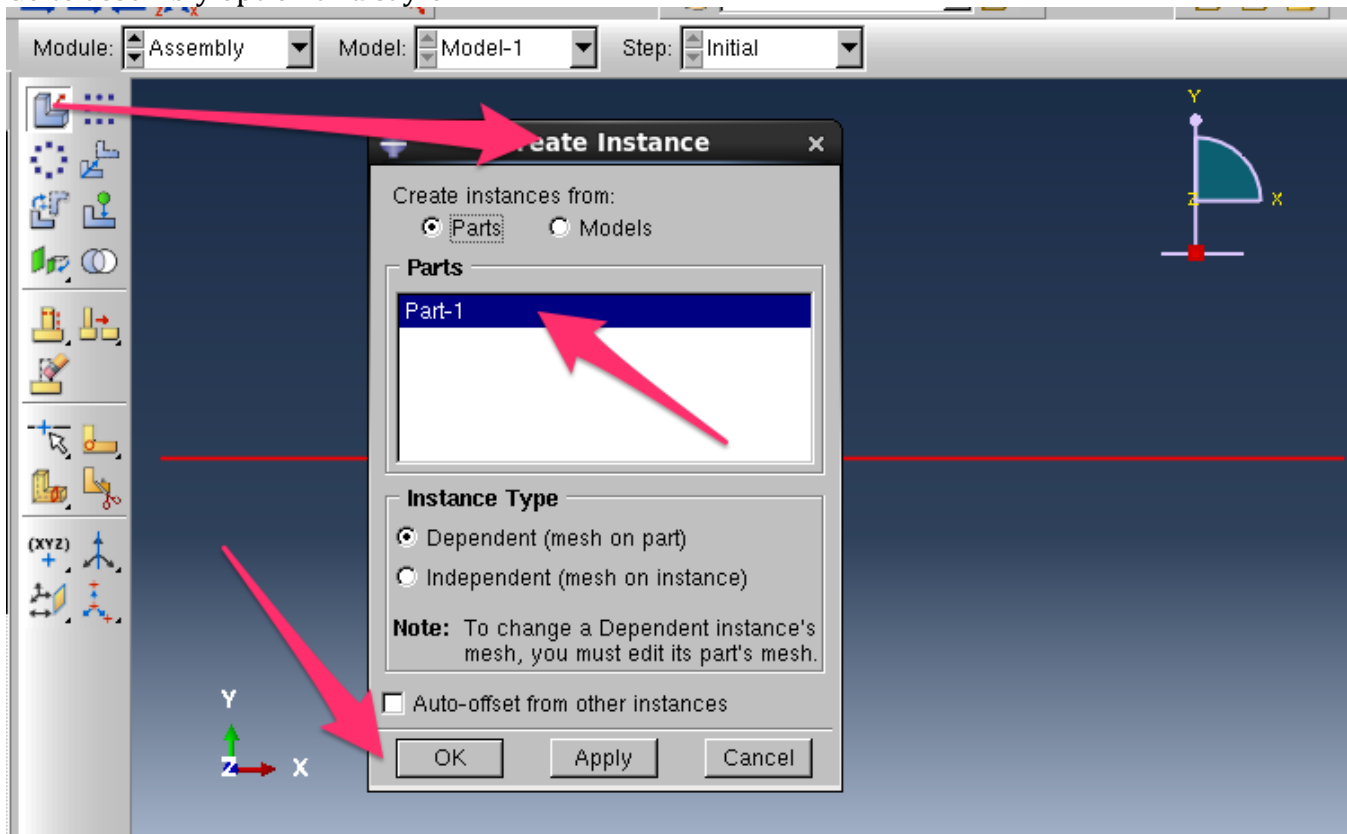


You should see this:



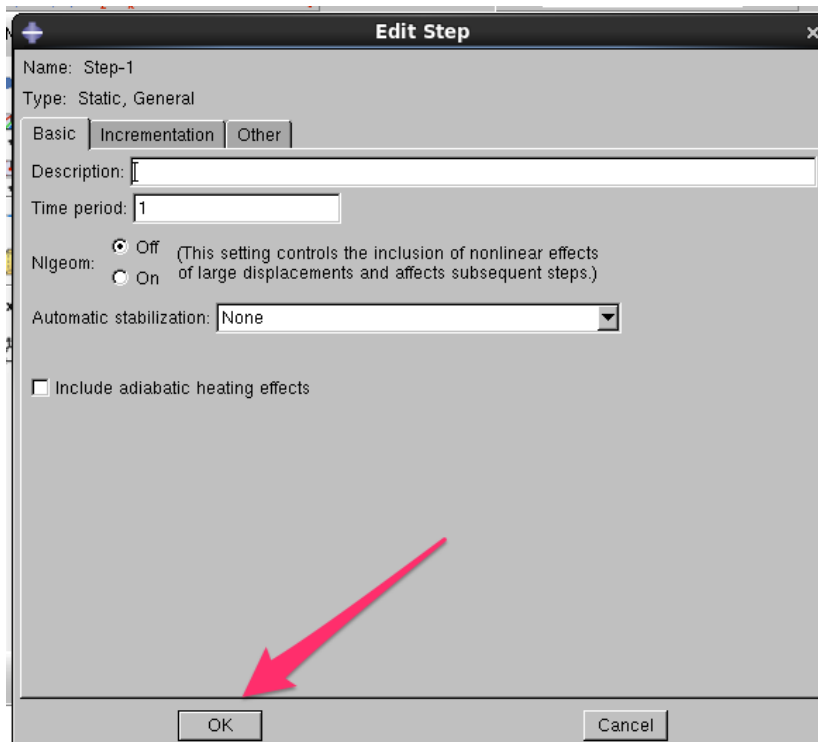
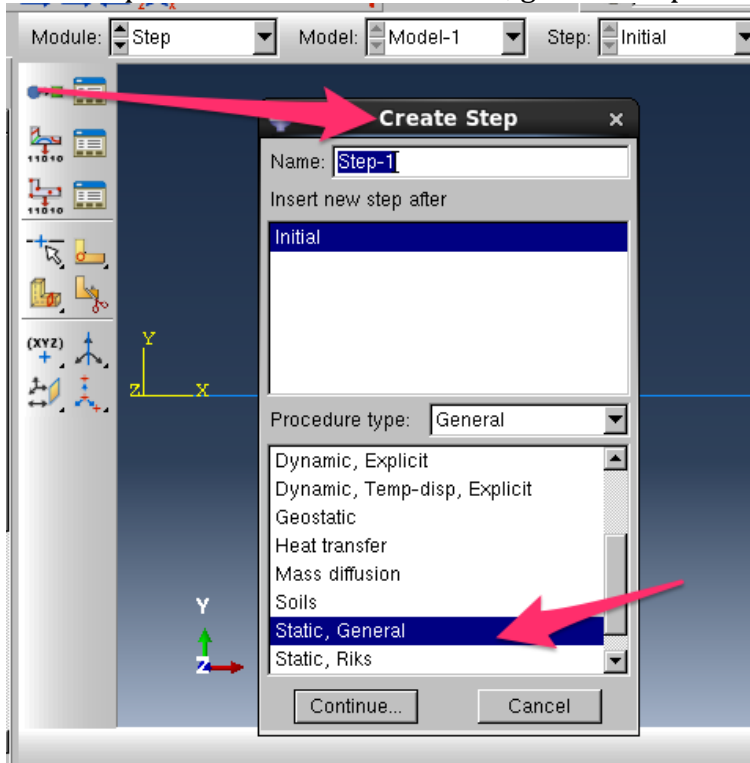
5. Assembly

Go to assembly option and say OK:



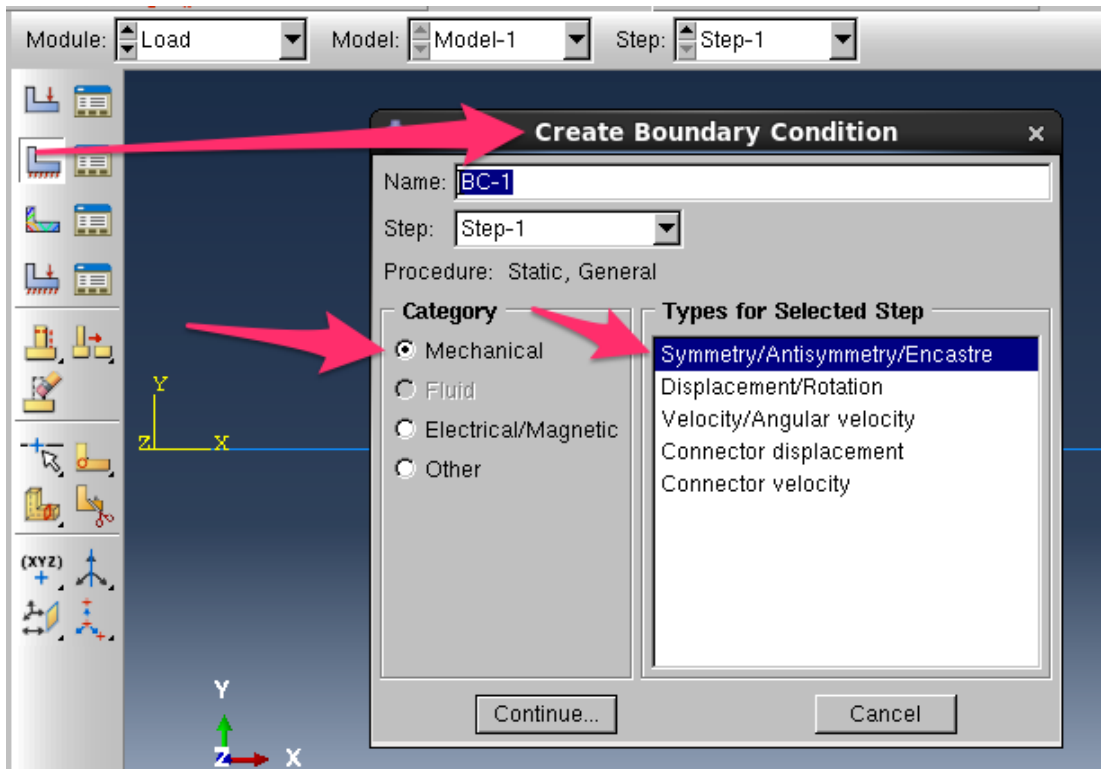
6. Create Step

Go to step module and create a static, general step. Accept all the defaults.

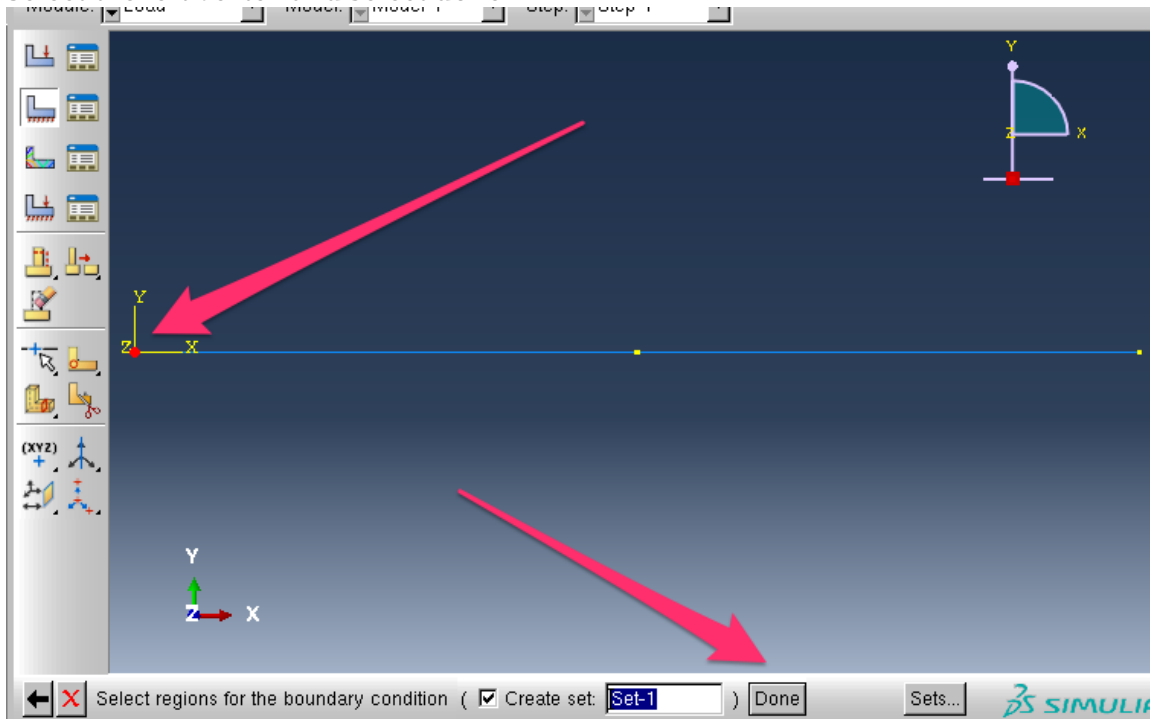


7. Create Boundary Conditions

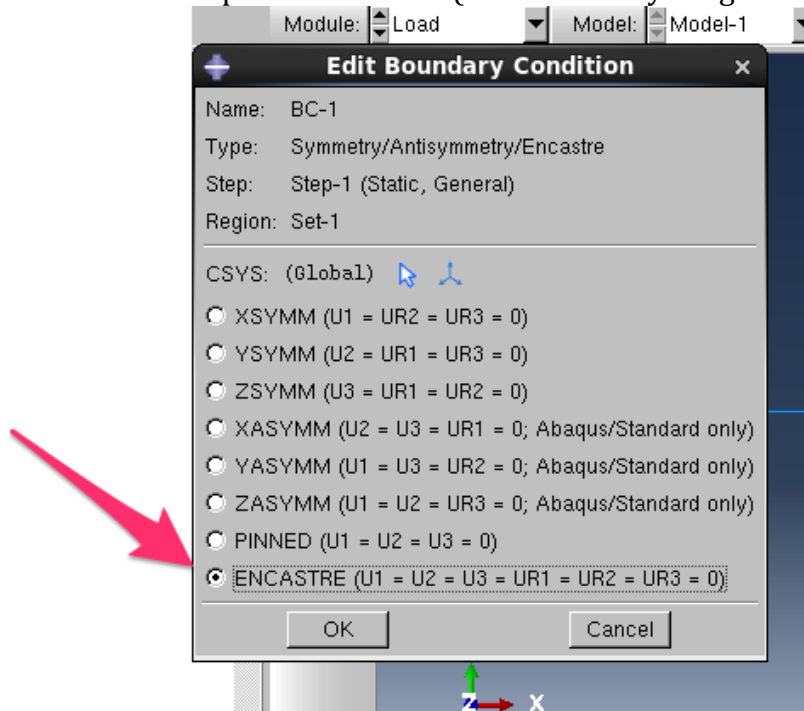
Go to load module and select the boundary condition option. We will go with an “encastre” option now:



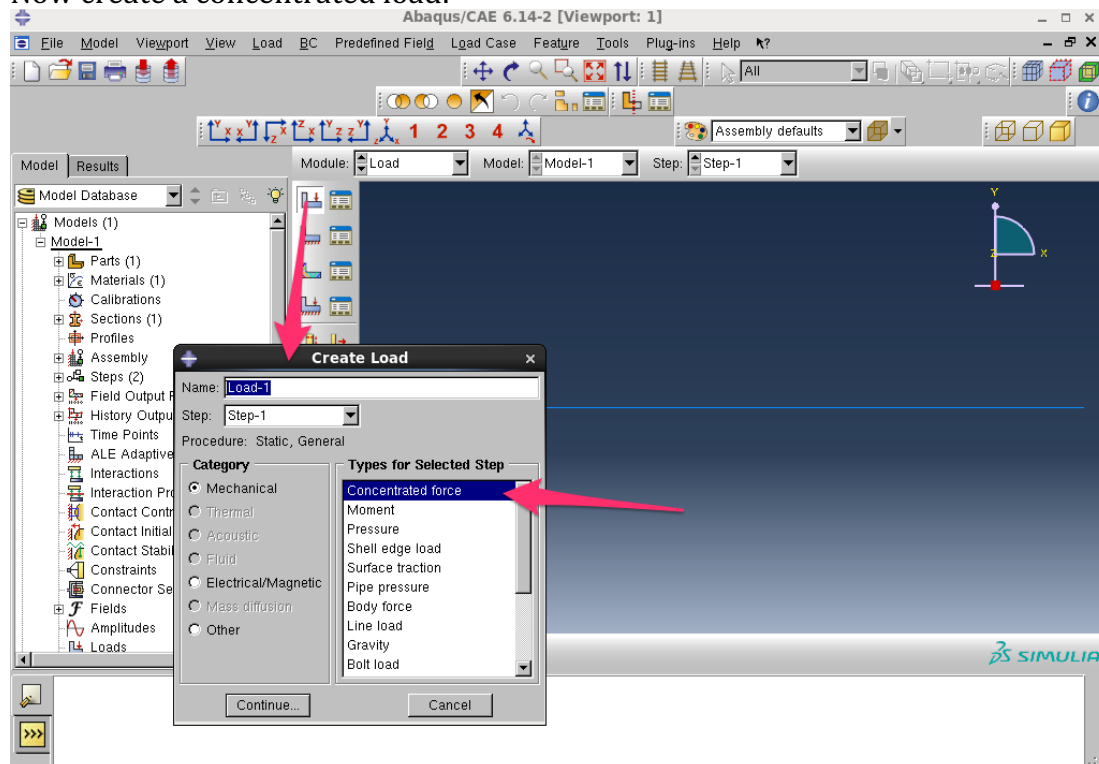
Select the left vertex and select done:



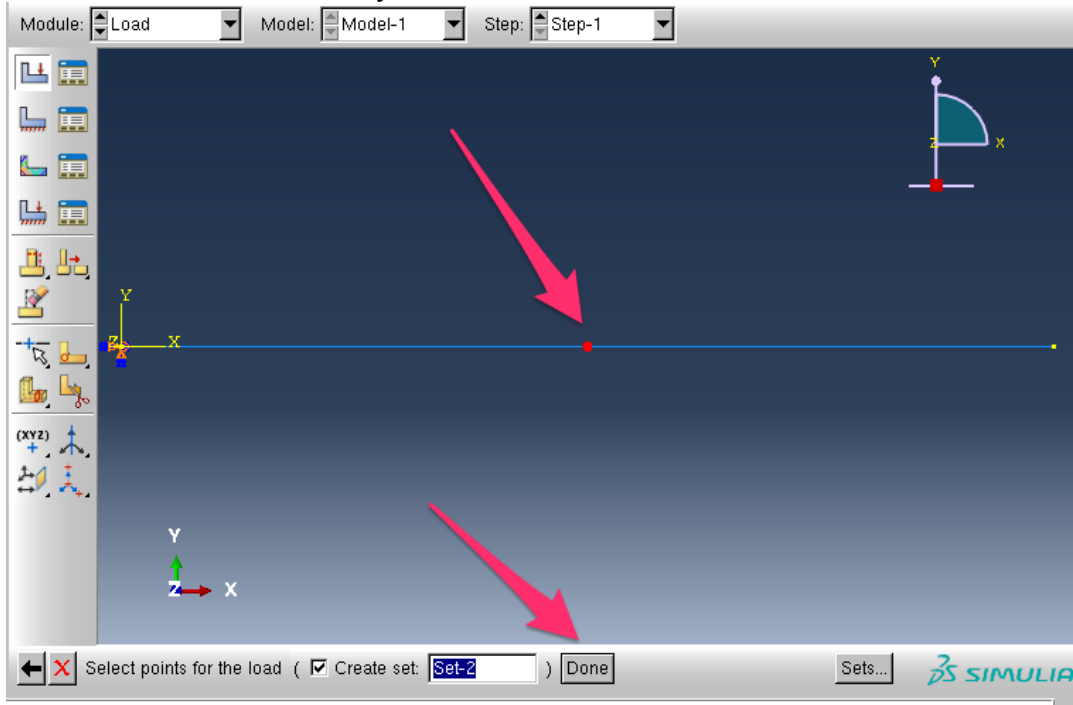
Select the last option “Encastre” (it means everything constrained):



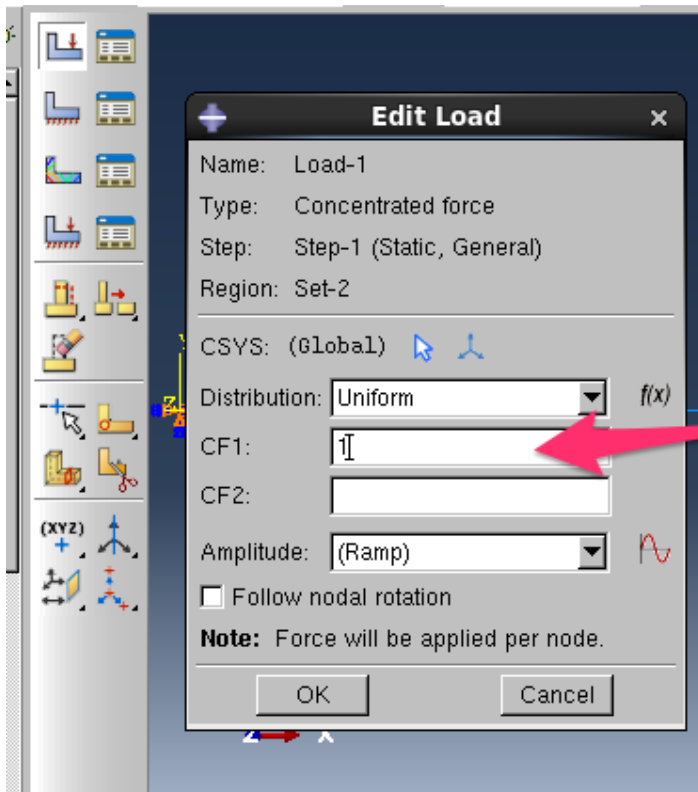
Now create a concentrated load:



Select the middle vertex, say done:



Enter 1 in the X direction:

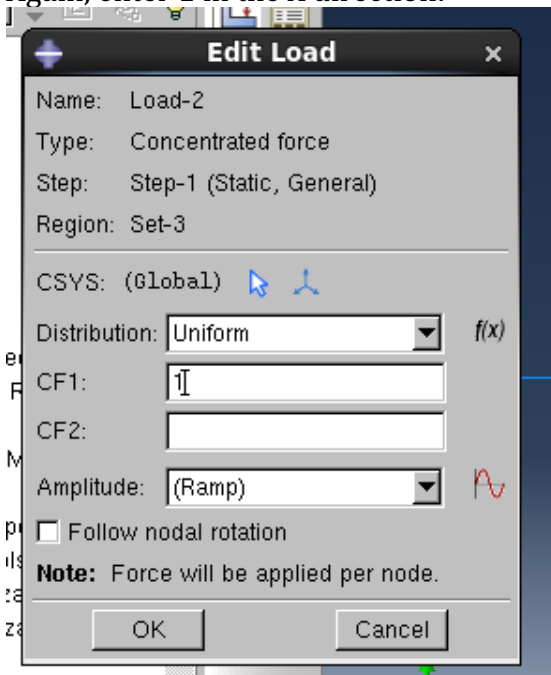




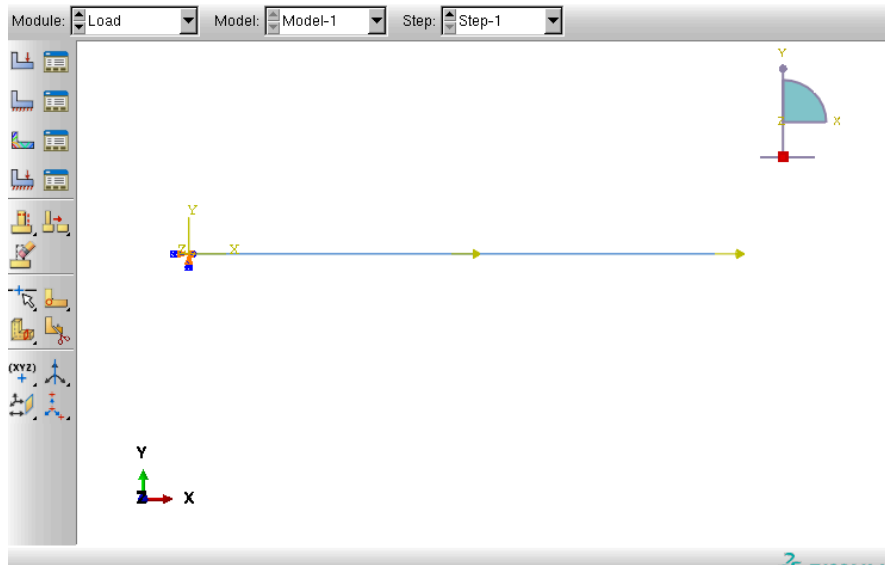
Do this procedure again to apply load at right end:



Again, enter 1 in the X direction:

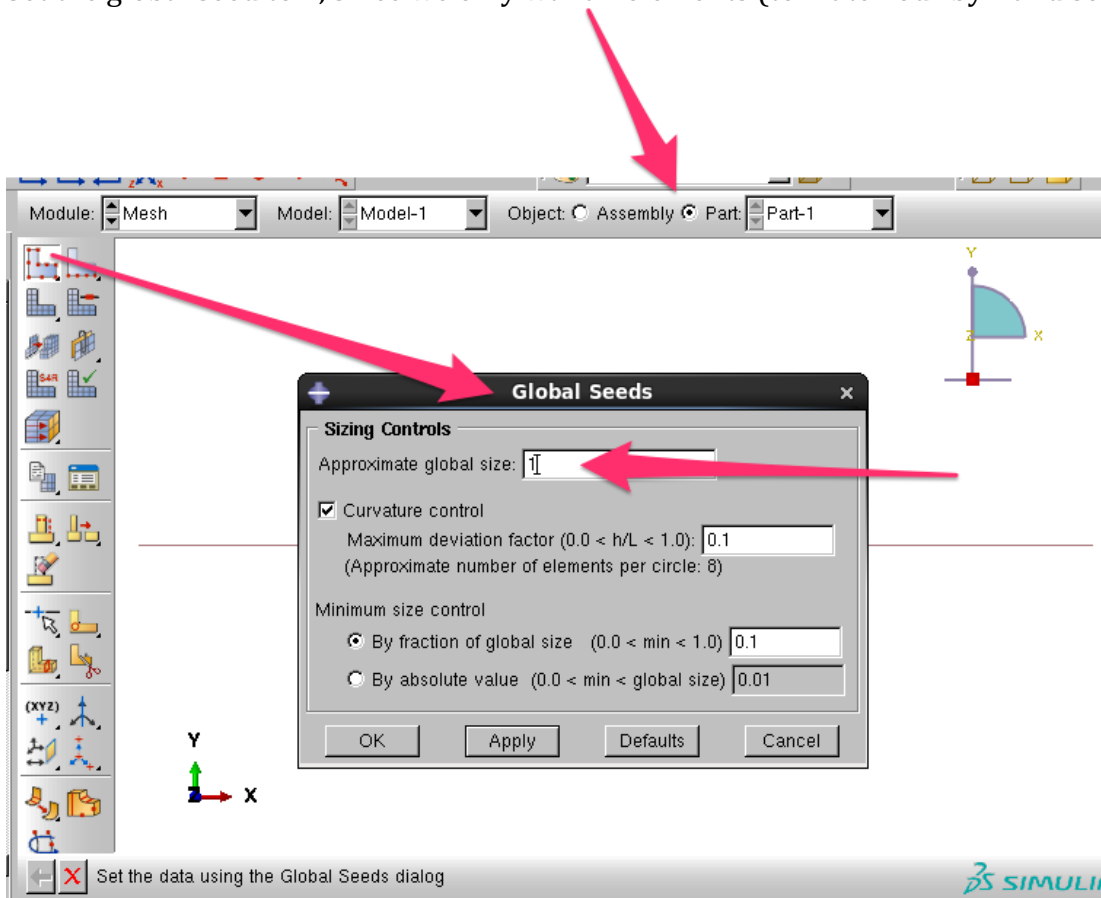


Should look like this:

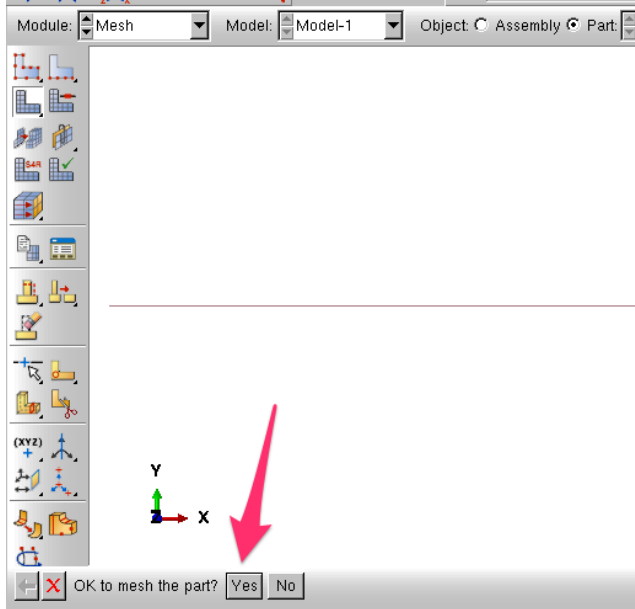


8. Create Mesh

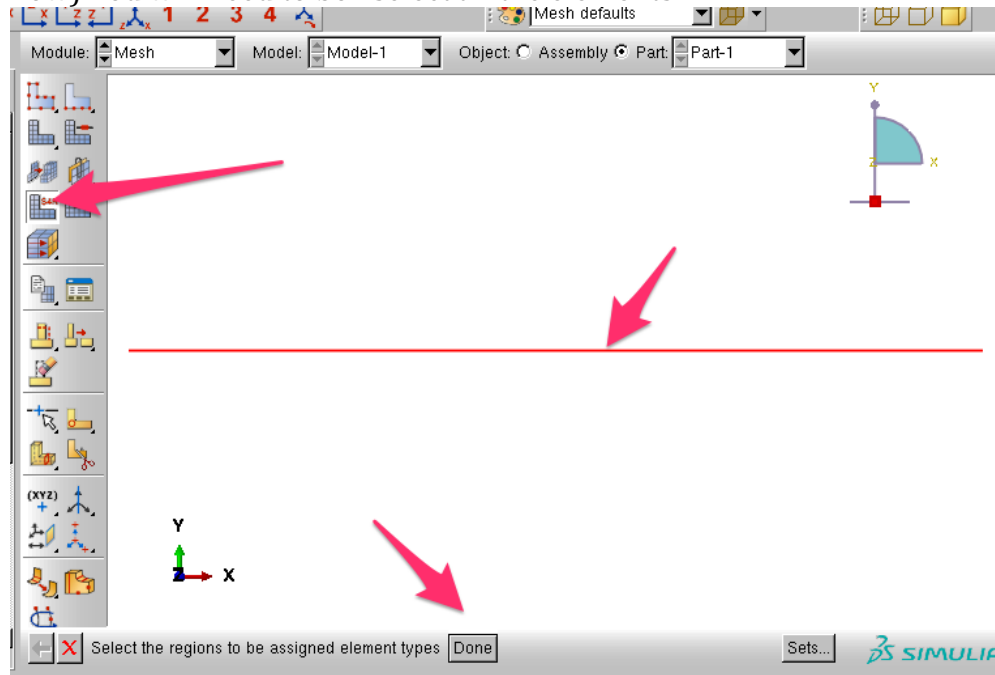
Set the global seed to 1, since we only want 2 elements (to match our by-hand solution):



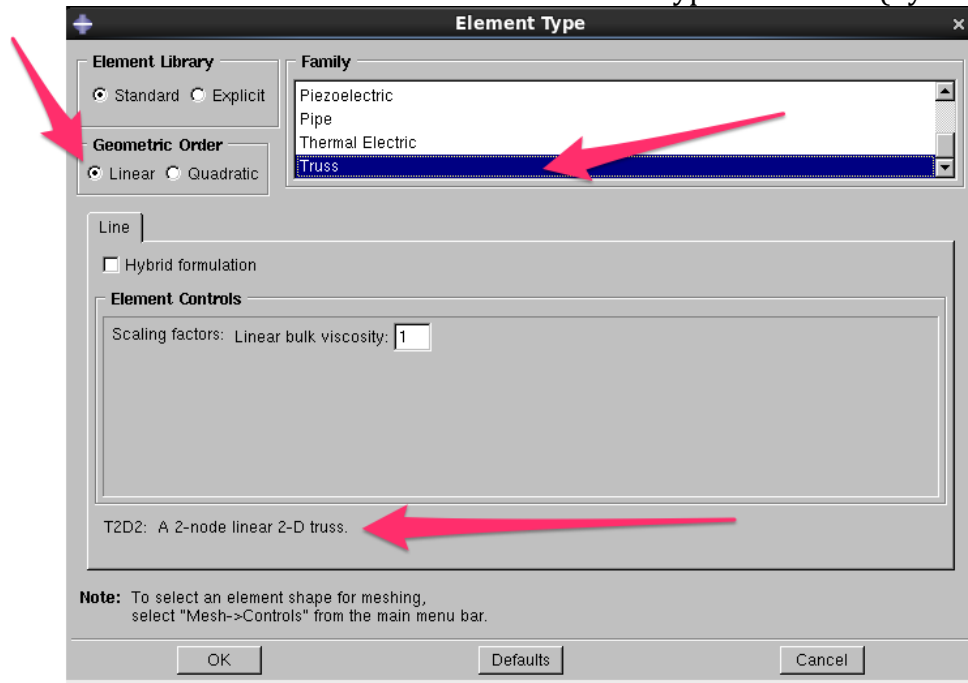
Mesh the part:



Let's check the element type to be sure they are linear trusses (since that is the theory we know now) You will need to box select all the elements:

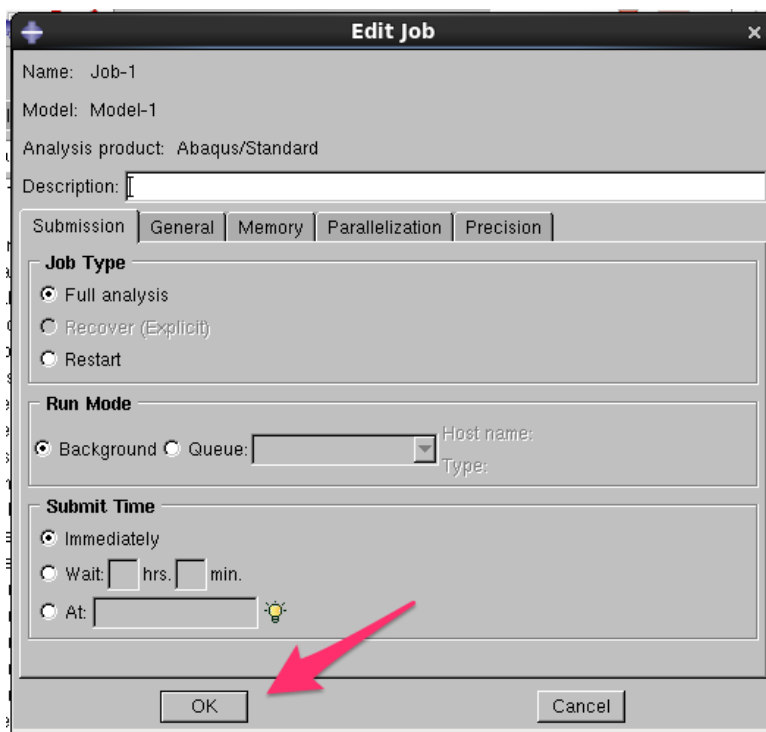
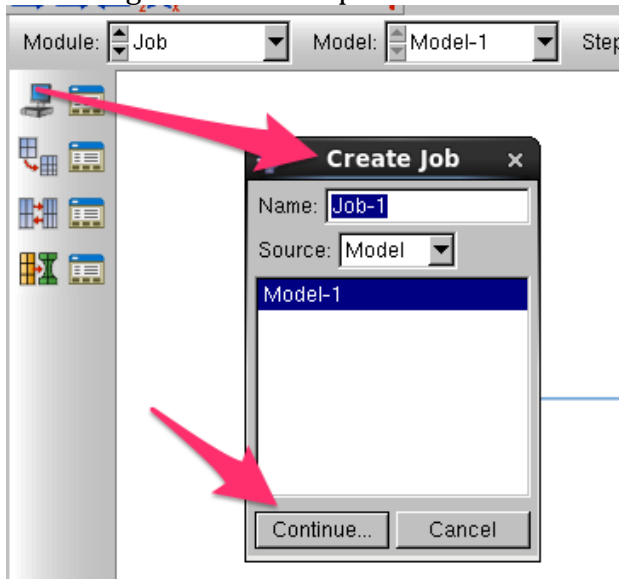


You want to make sure the linear truss element type is selected (by default it is beam):



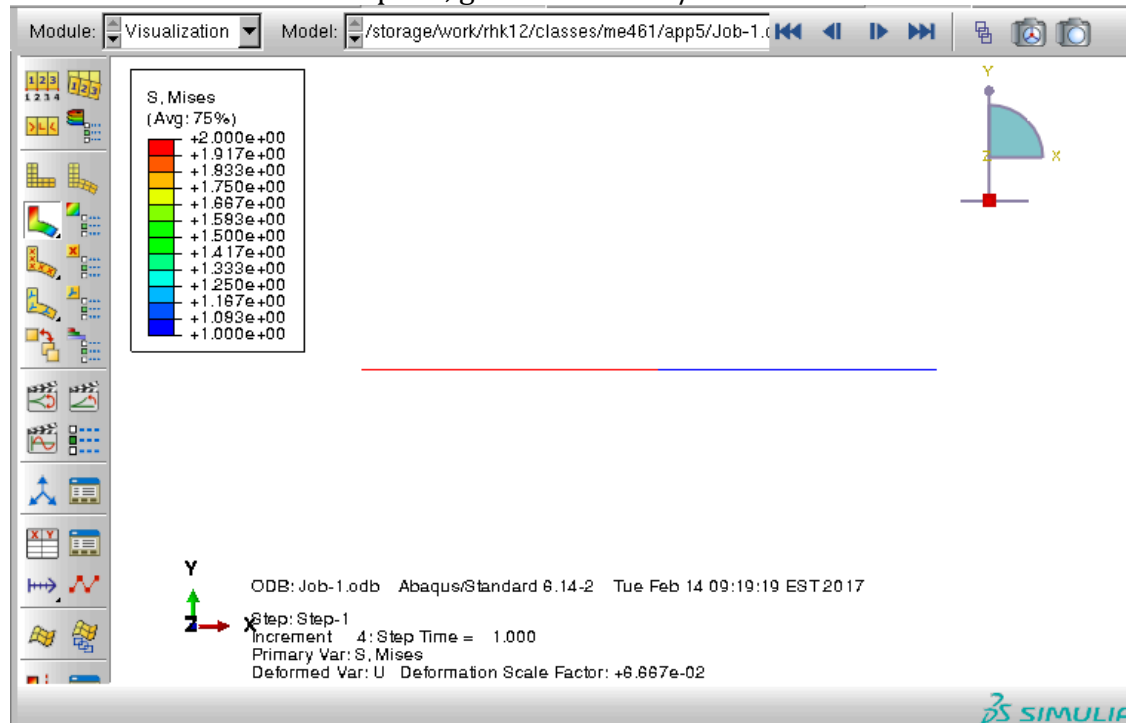
9. Create a Job

Go through the usual steps to create and run the job:

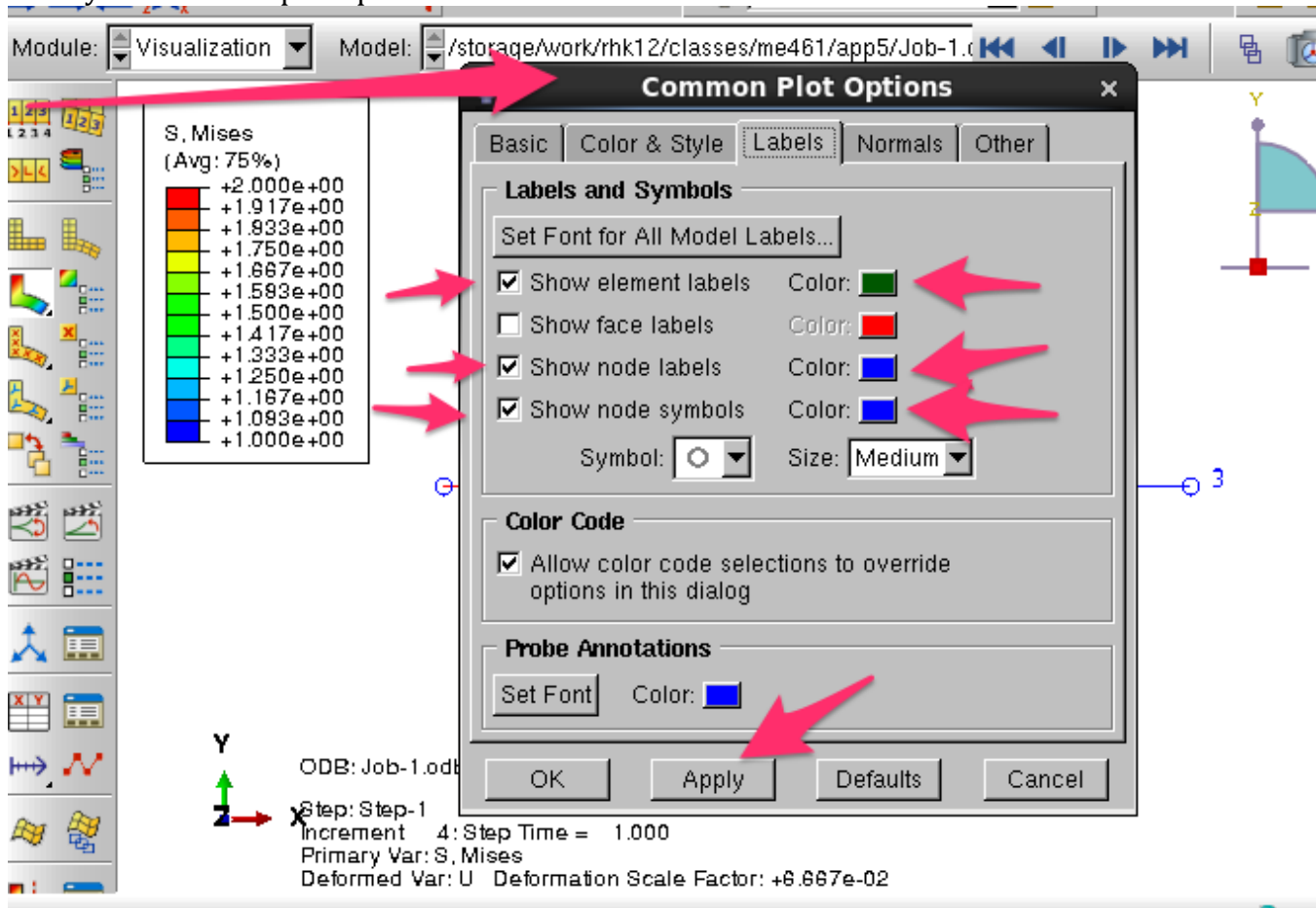


10. Visualize the Results

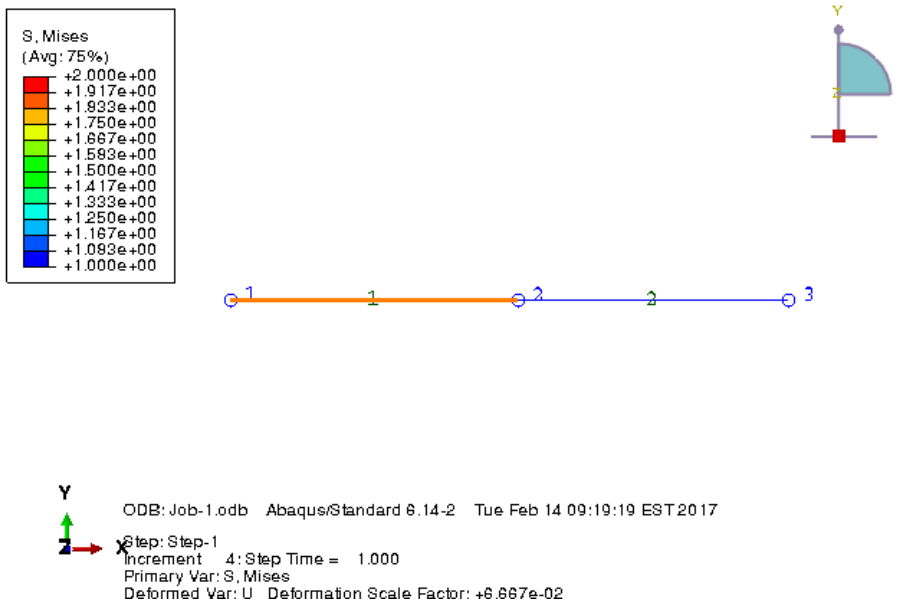
Once the simulation is complete, go to the results/visualization module and view the results:



Modify the common plot options to show the element and node labels:



You should see something like this:

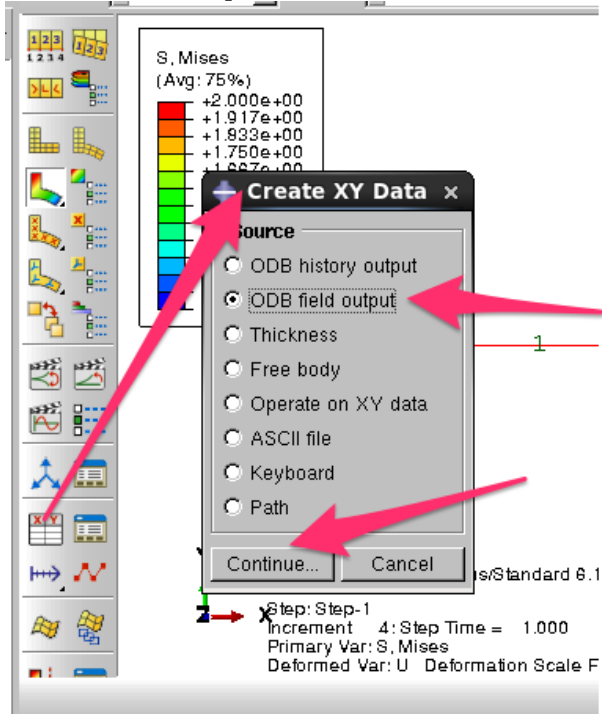


Note that the number here is different from the one we used in the by-hand calculation.

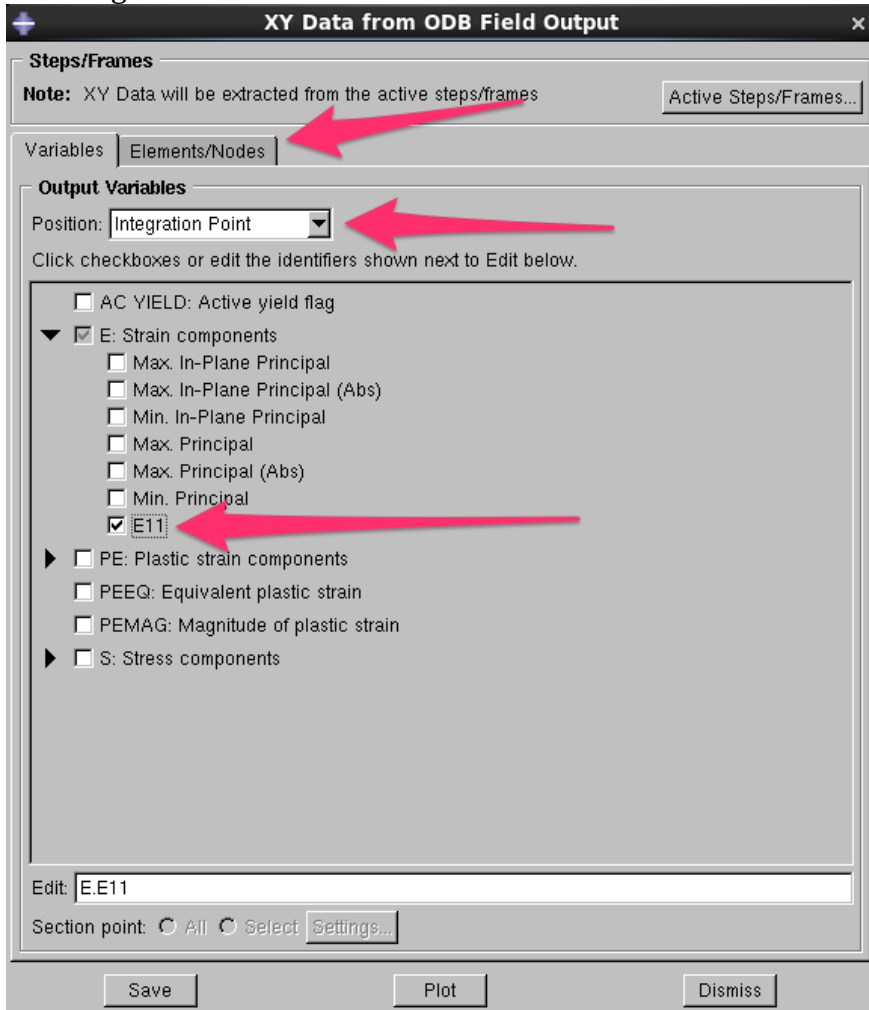
You can find XY plots for the displacements, strains, stresses and reaction forces in the lecture notes on Canvas.

Here I will give one example how to create a XY plot. You just follow the same procedure for the rest.

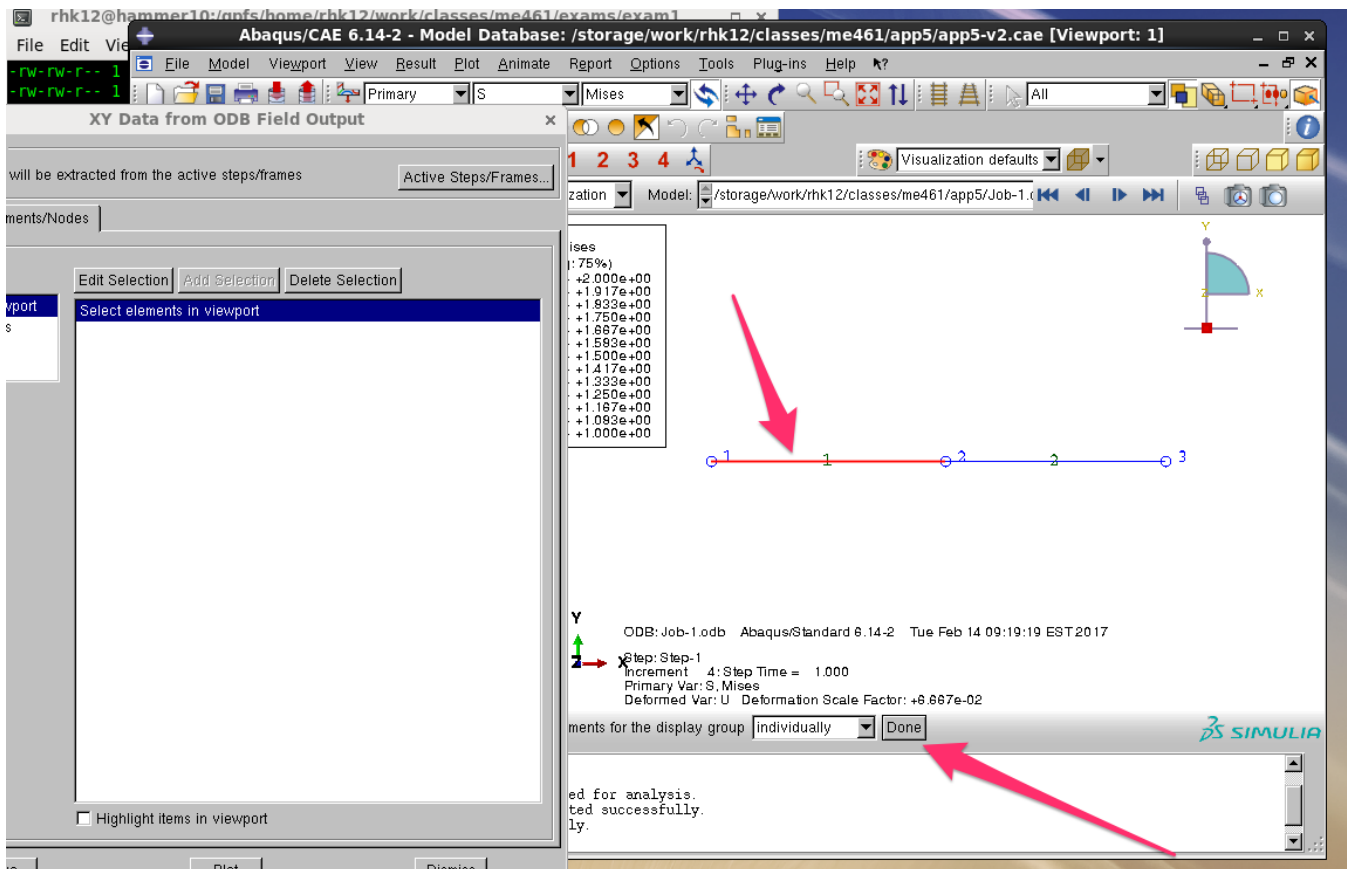
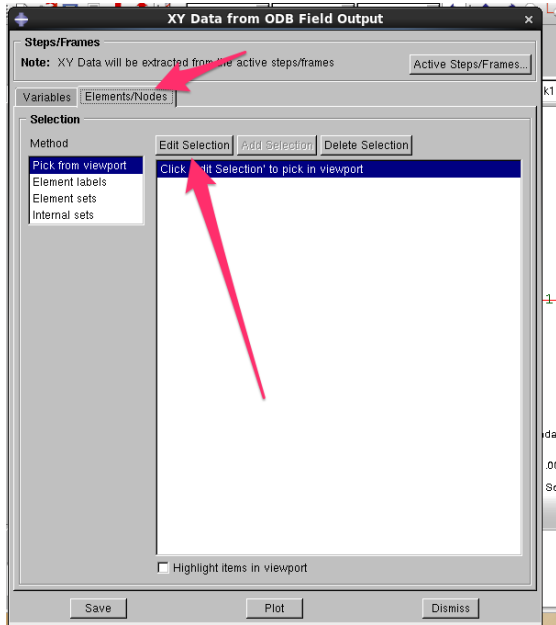
To create an XY plot data of an element or node:



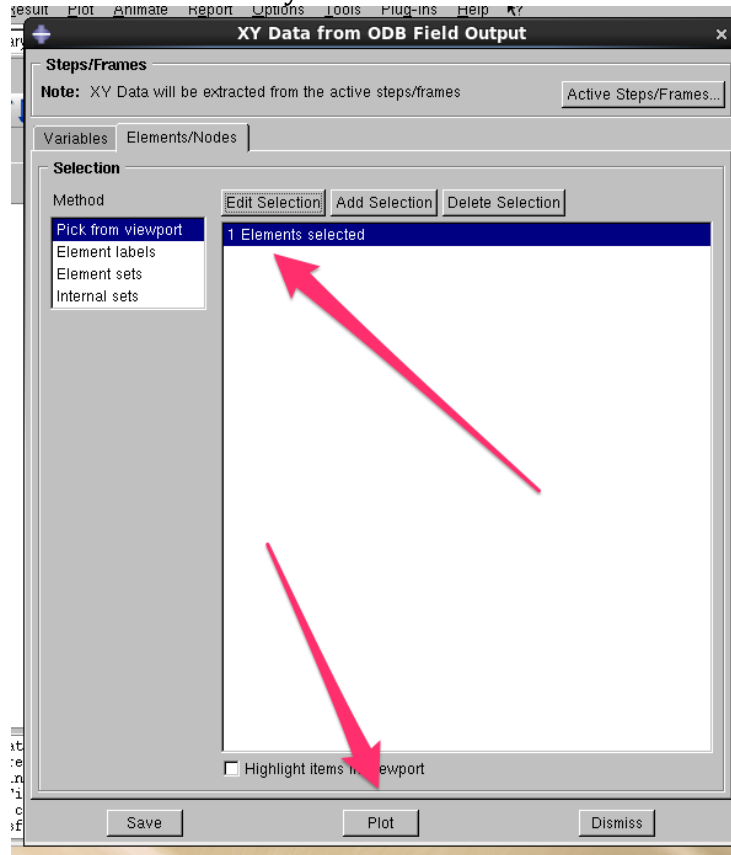
First select your output variable position, for elements you can select integration point. For our trusses, we only have one (we will learn about integration points later in the semester). Then you select which variable you want to plot. Things like stress and strain are element variables, while things like displacement, velocities, accelerations and reaction forces are nodal values. Here I'm selecting the Strain in the XX direction:



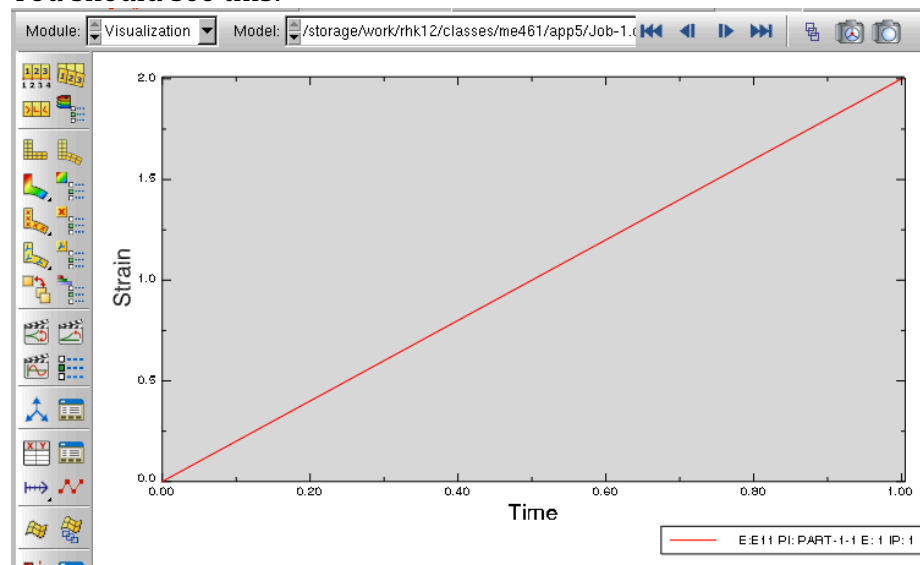
After we do this we need to select the element(s) that we want to plot for. Here I will select element 1:



You should see that you have 1 element selected. Now click plot:



You should see this:



You can find all the other plots in the lecture notes.