SAP2000 Tutorials
HW 7 problem 1

By Andrew Chan

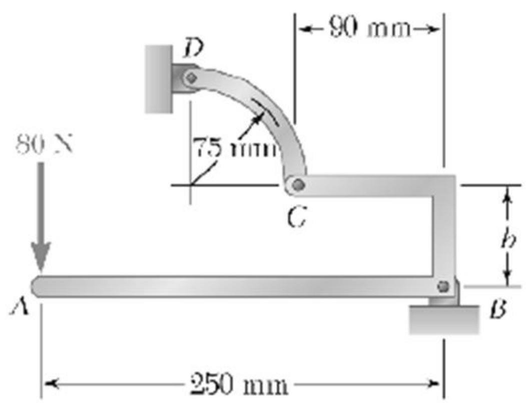
Brief:

This tutorial is intended to give you an understanding of how to analyze a problem in statics with SAP2000. We will determine the reactions at point B and D in terms of the X and Y components. This tutorial will also introduce you to basic concepts in designing, implementing, and analyzing an object in the SAP2000 environment. With an emphasis on showing you how to implement a nonlinear shape, in our case a curved object.

Example:

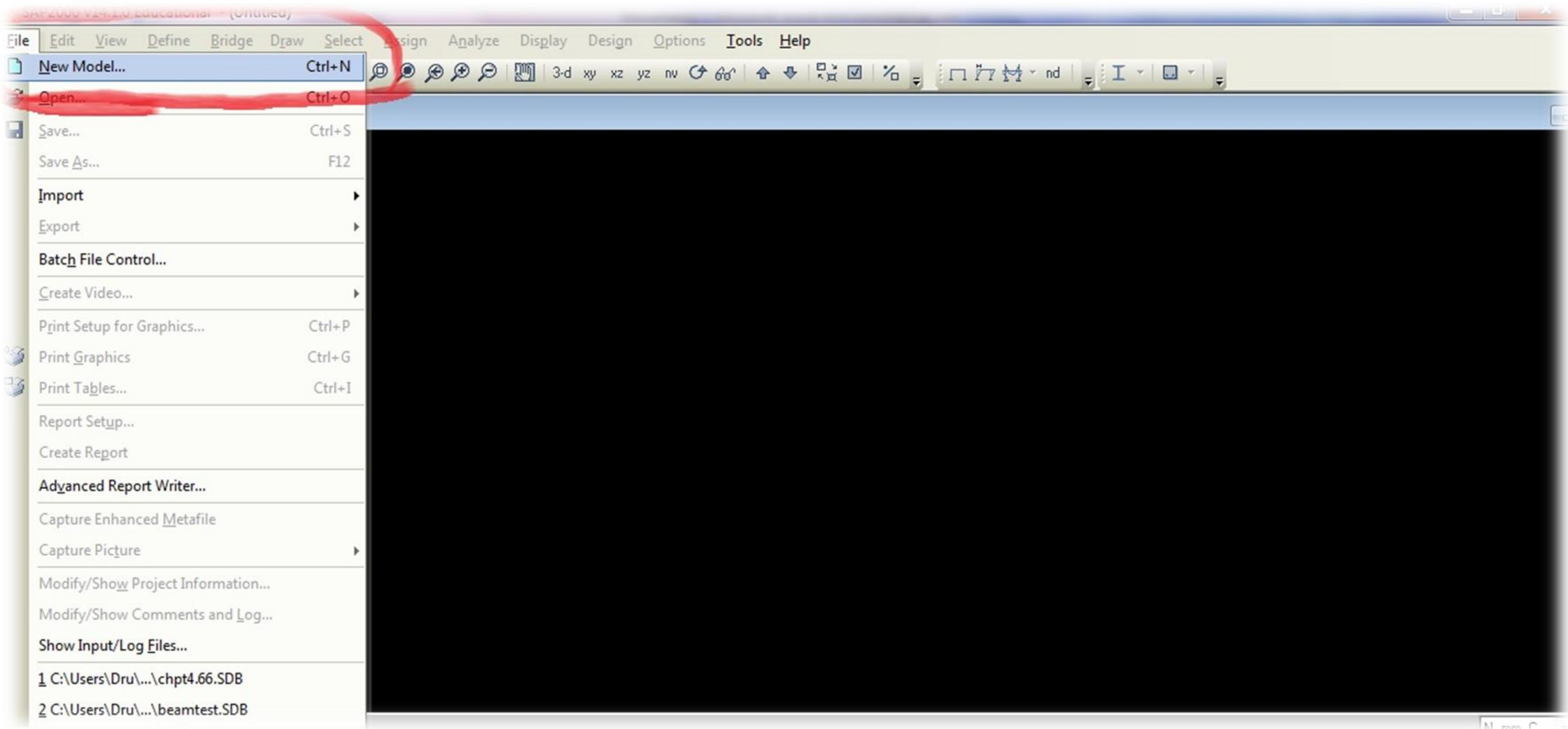
The problem we have selected is shown below.

Chapter 4, Problem 66



Determine the reactions at *B* and *D* when $b = 60$ mm.

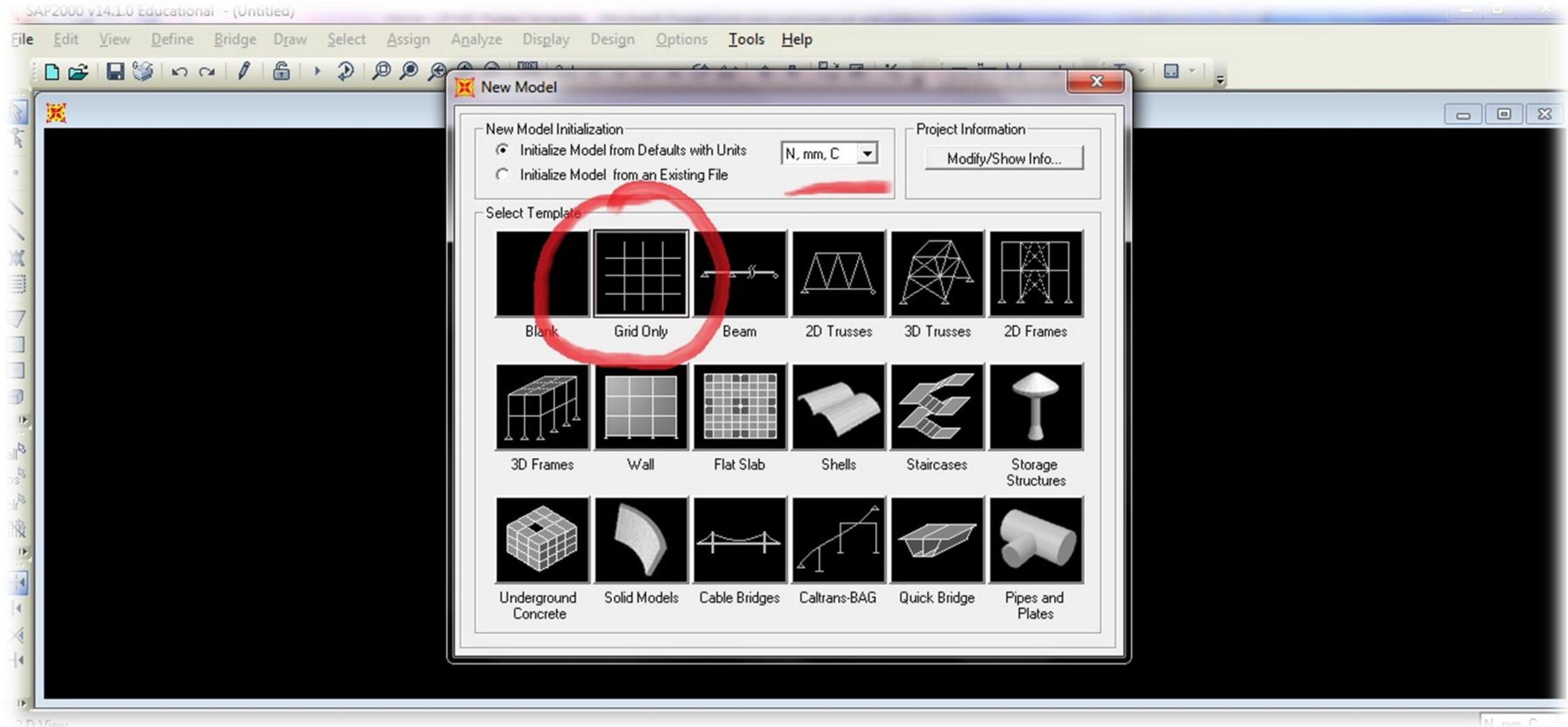
Step 1:



Directions:

After you have installed and start SAP2000. We want to click on the **File menu.** And then select a **New Model** to create a new model in our working environment.

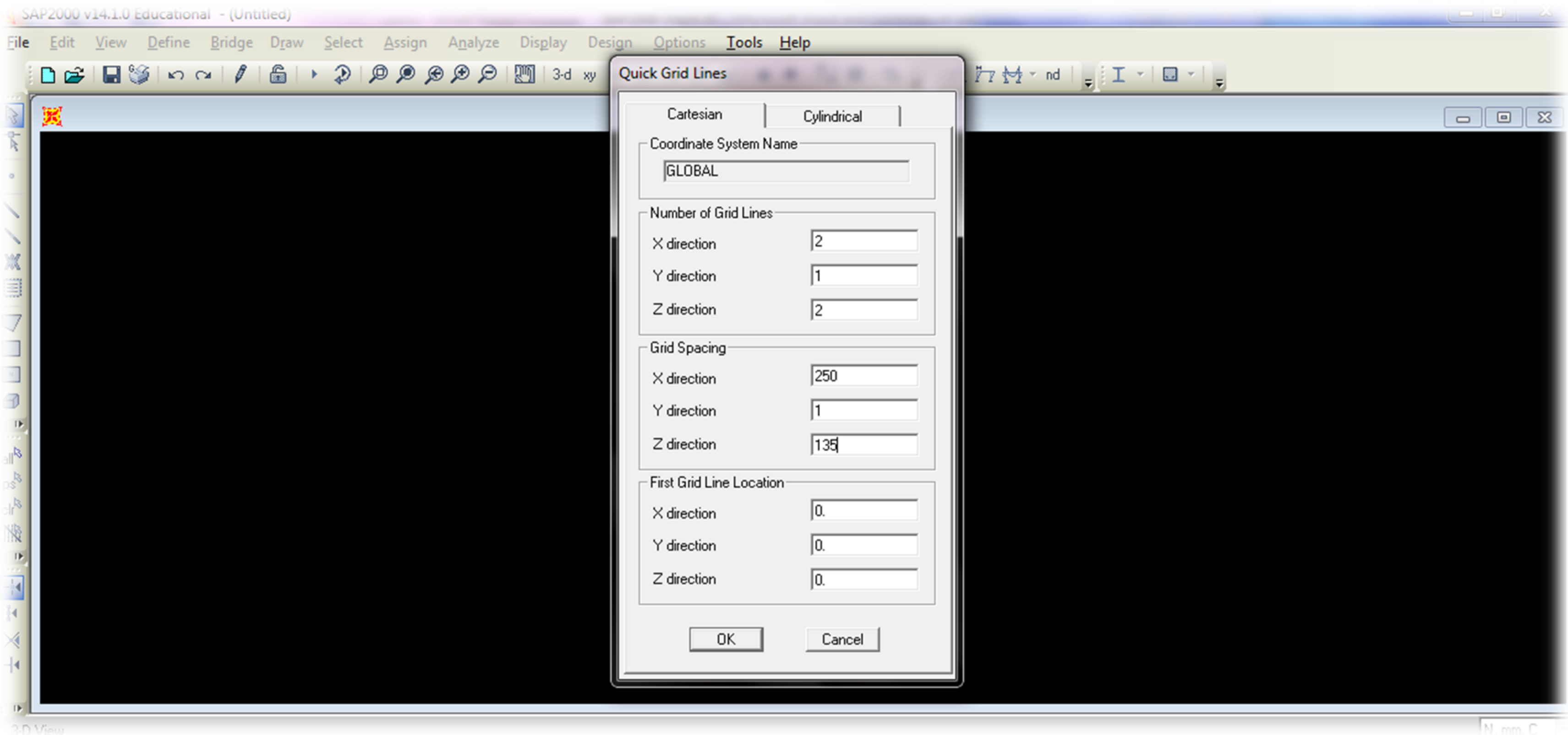
Step 2:



Directions:

Define your units by clicking on the **define units drop down menu** and select **N, mm, C.** Now click on **Grid Only**. This will assign our units to Newton, millimeters, and degrees in Celsius.

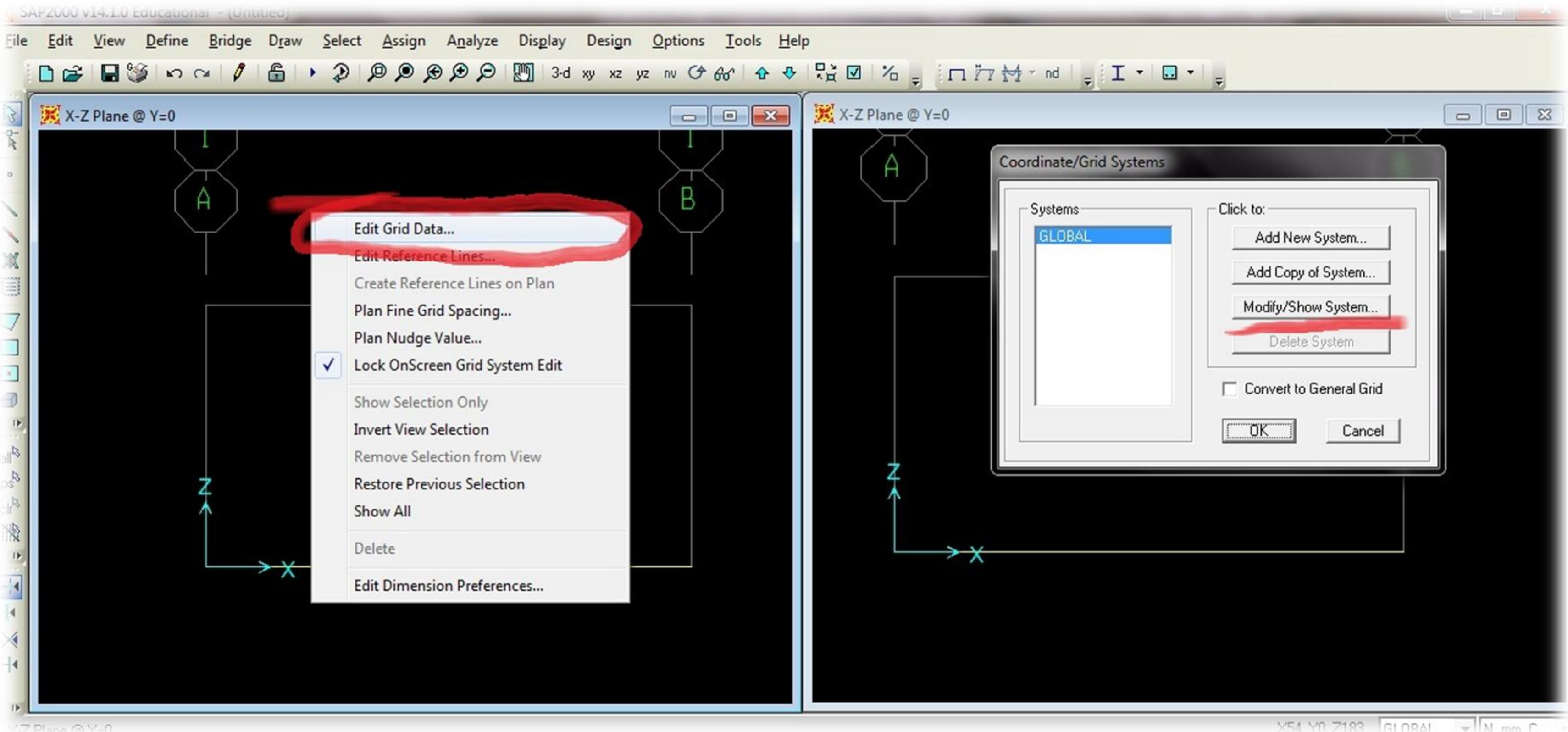
Step 3:



Directions:

The **Quick Grid Lines** menu should display. Under **Number of Grid Lines** enter 2, 1, 2 for the x, y, and z directions. Next under **Grid Spacing**, enter in 250, 1, 135 for the x, y, and z directions. Then click **Ok.** This will specify our grid to encompass the layout of the entire problem. We will edit it further below.

Step 4:

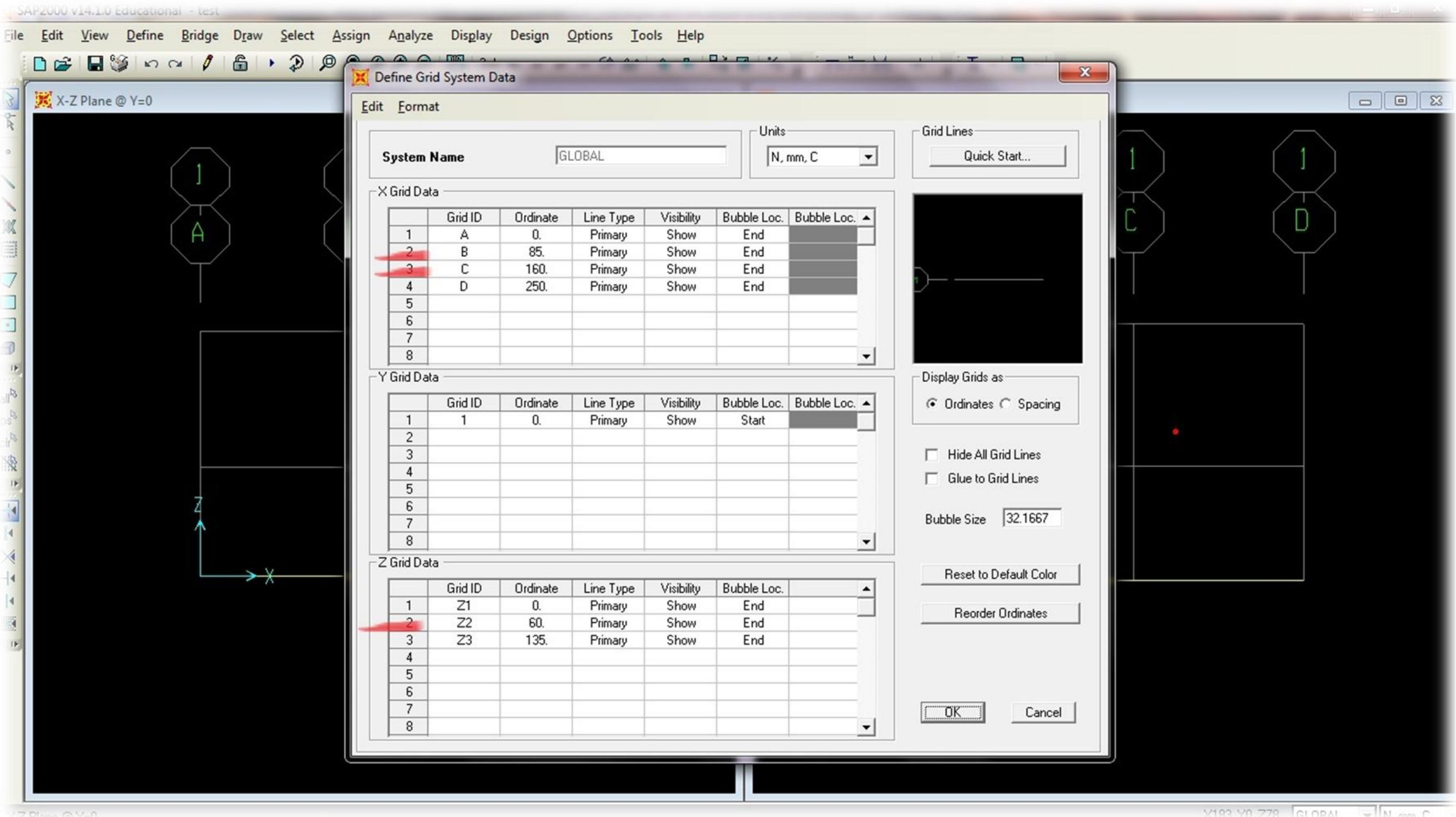


Directions:

Now since the 250 mm beam is laid out, we want to go into the **Edit Grid Data** menu by right clicking on the **empty black screen.** Next we click on the **modify/show system**. We want to modify the grid so that we may easily lay down the other sections of the object.

Note: Beginning on this page, I will be starting to compile images together for breivty. The order of procedures run from left to right.

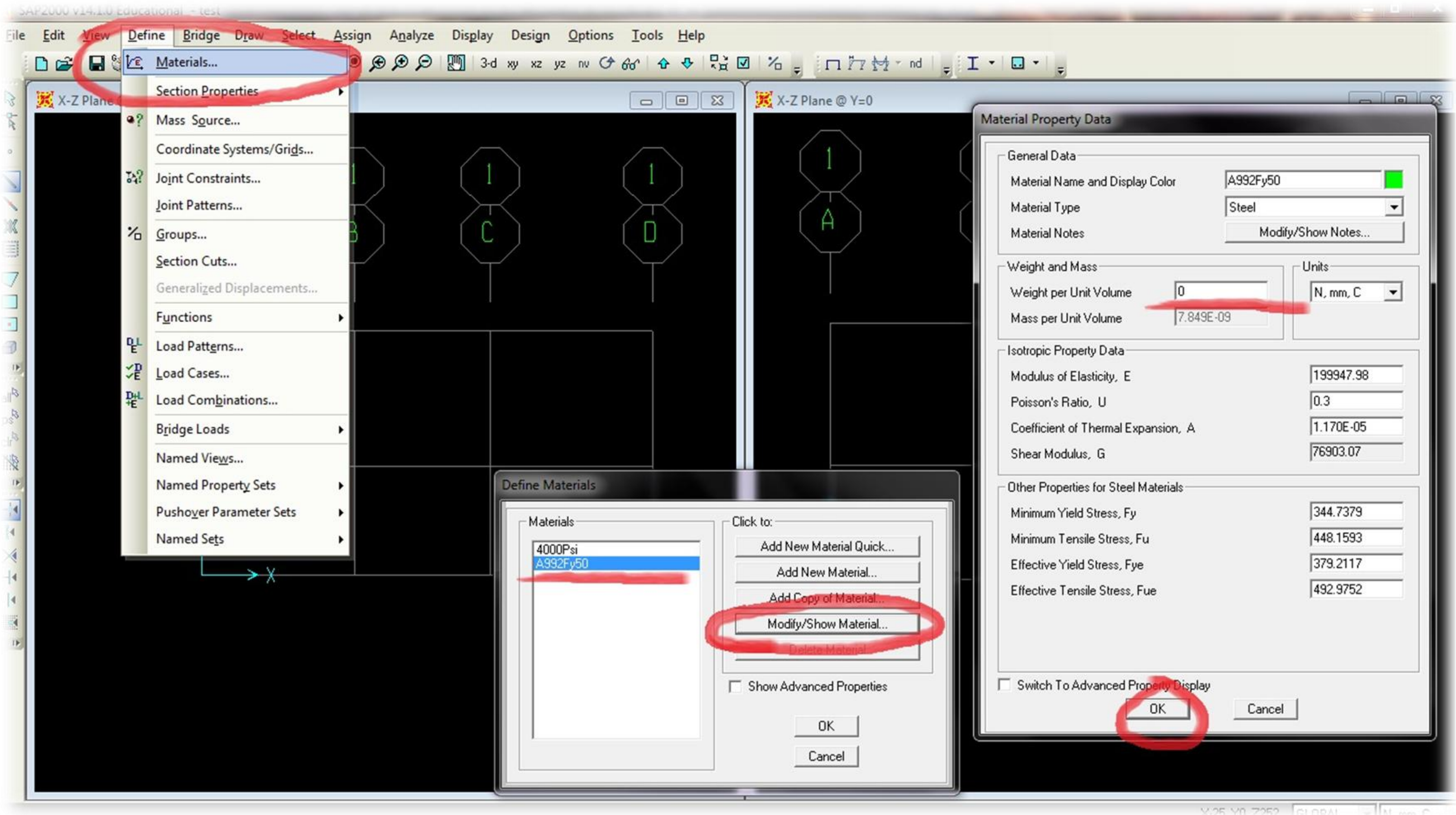
Step 5:



Directions:

In order for us to better lay out the grid, please modify the **Define Grid System Data** to reflect that which has been highlighted above. This preferred way of grid laying and allows for further customizations and precision.

Step 6:

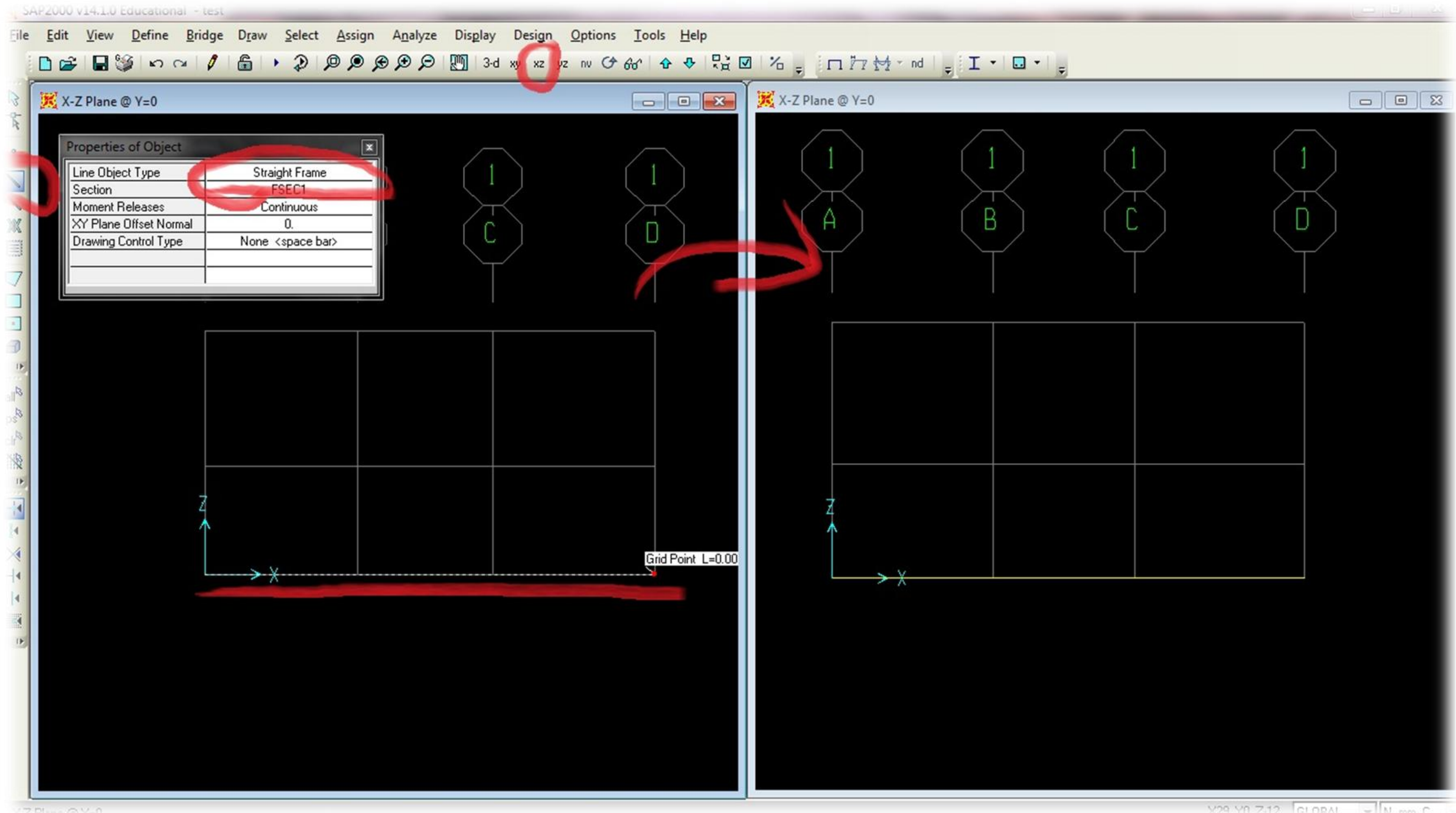


Directions:

Now let us define our materials. We will be using the default **FSEC1** and this is assigned by default to **A992Fy50** a type of steel. So first click on the **Define menu → Materials → Select A992Fy50 in the Define Materials → now select the Modify/Show Material.** Now we want to define the material as weightless, so we set the **Weight per Unit Volume to 0.** And now click **Ok.**

Note: We want to define our material's mass as 0 so as to not consider it in our calculations. This is not always the case.

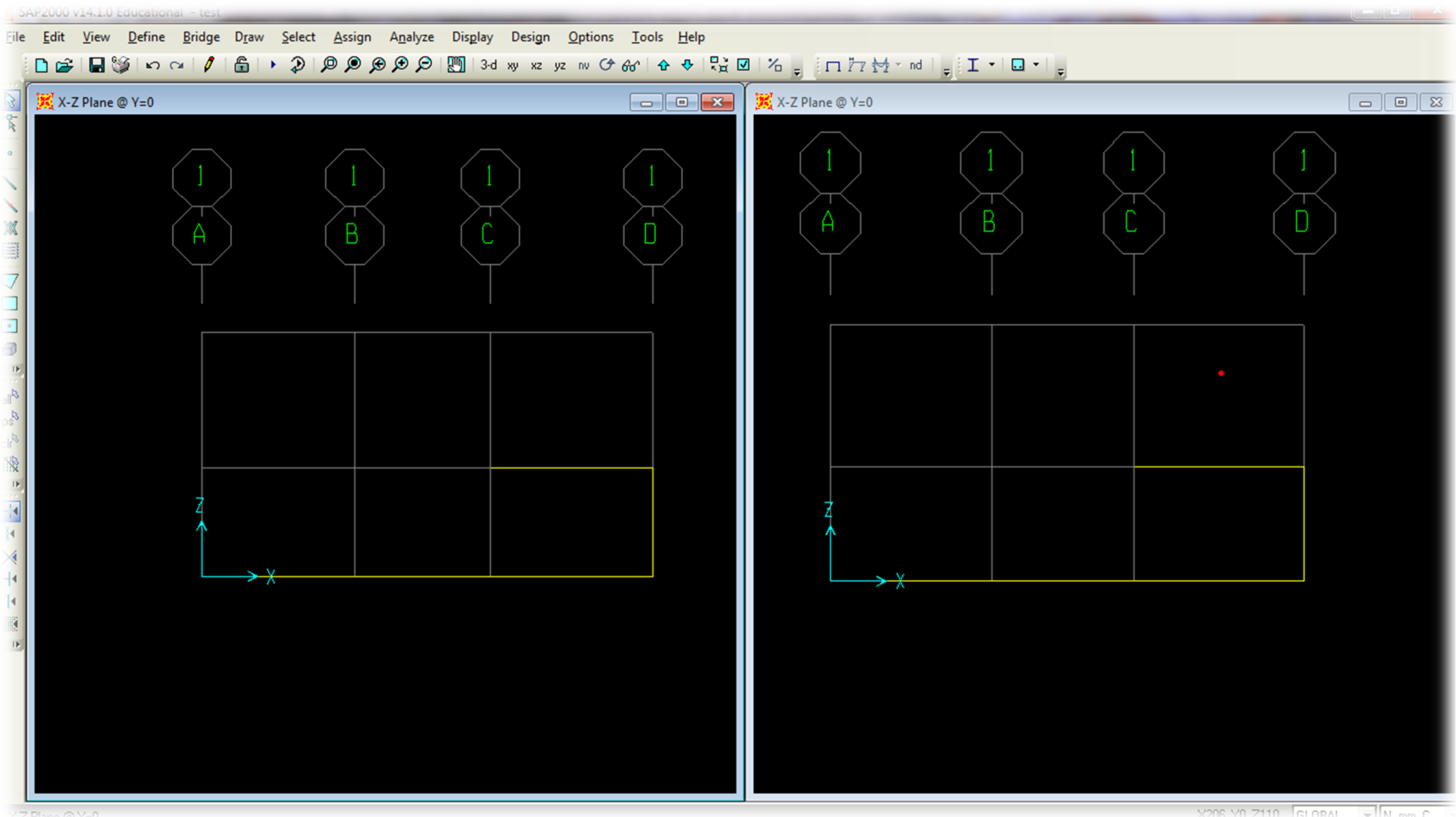
Step 7:



Directions:

Lets now begin drawing, so first we should ensure that we are optimized for viewing by clicking on the **X-Z plane** button**.** Next lets click on the **Draw Frame** command. Ensure that the object type is **Straight Frame** and lets lay out a line from the z-x axis to the end of our grid. After everything is done, your model should look like the **window above on the right** with a 250 mm long yellow beam laid out.

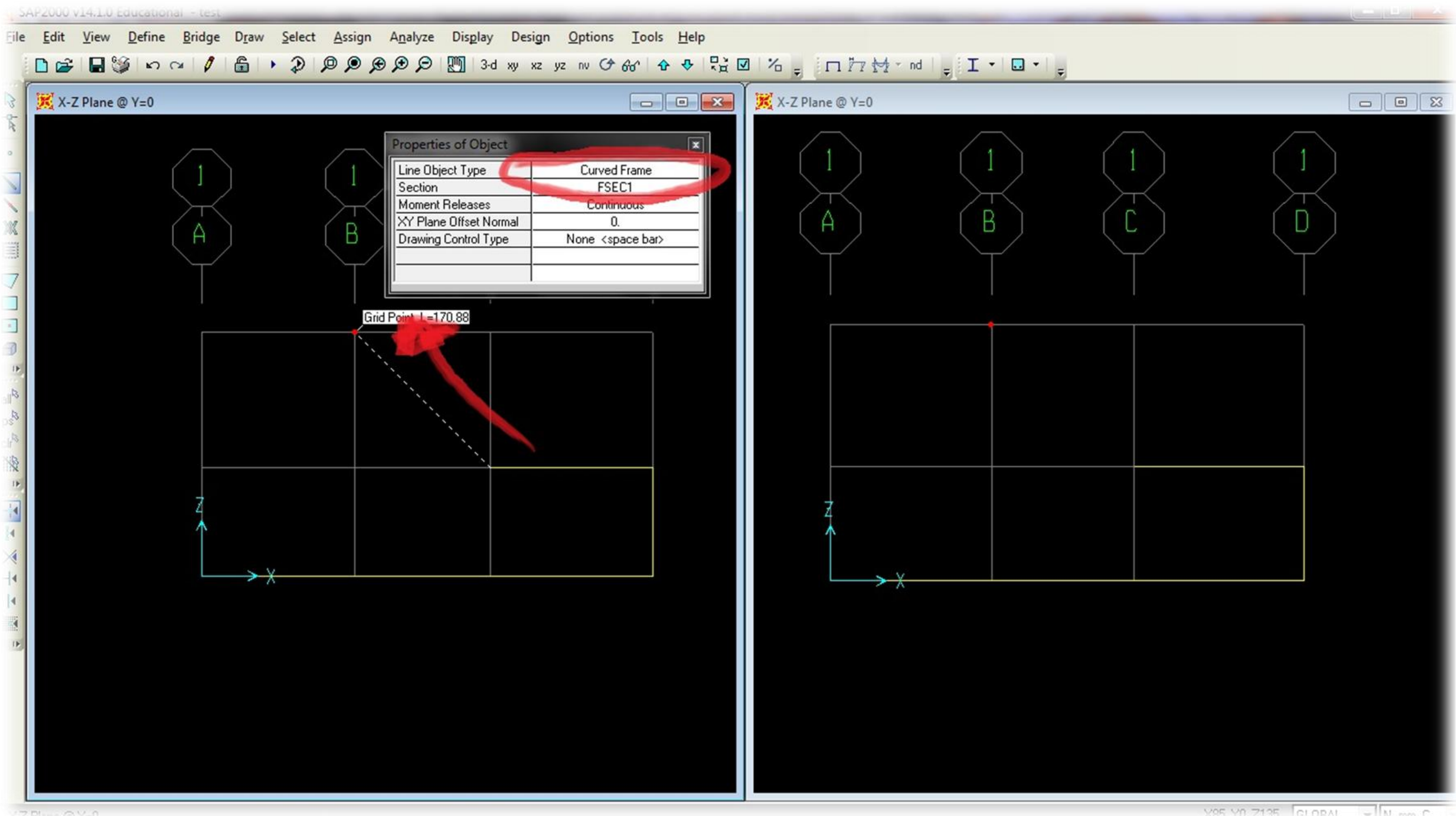
Step 8:



Directions:

And now we layout the rest of the members. Notice how the grid spacing we set up before hand aided us greatly in designing our model.

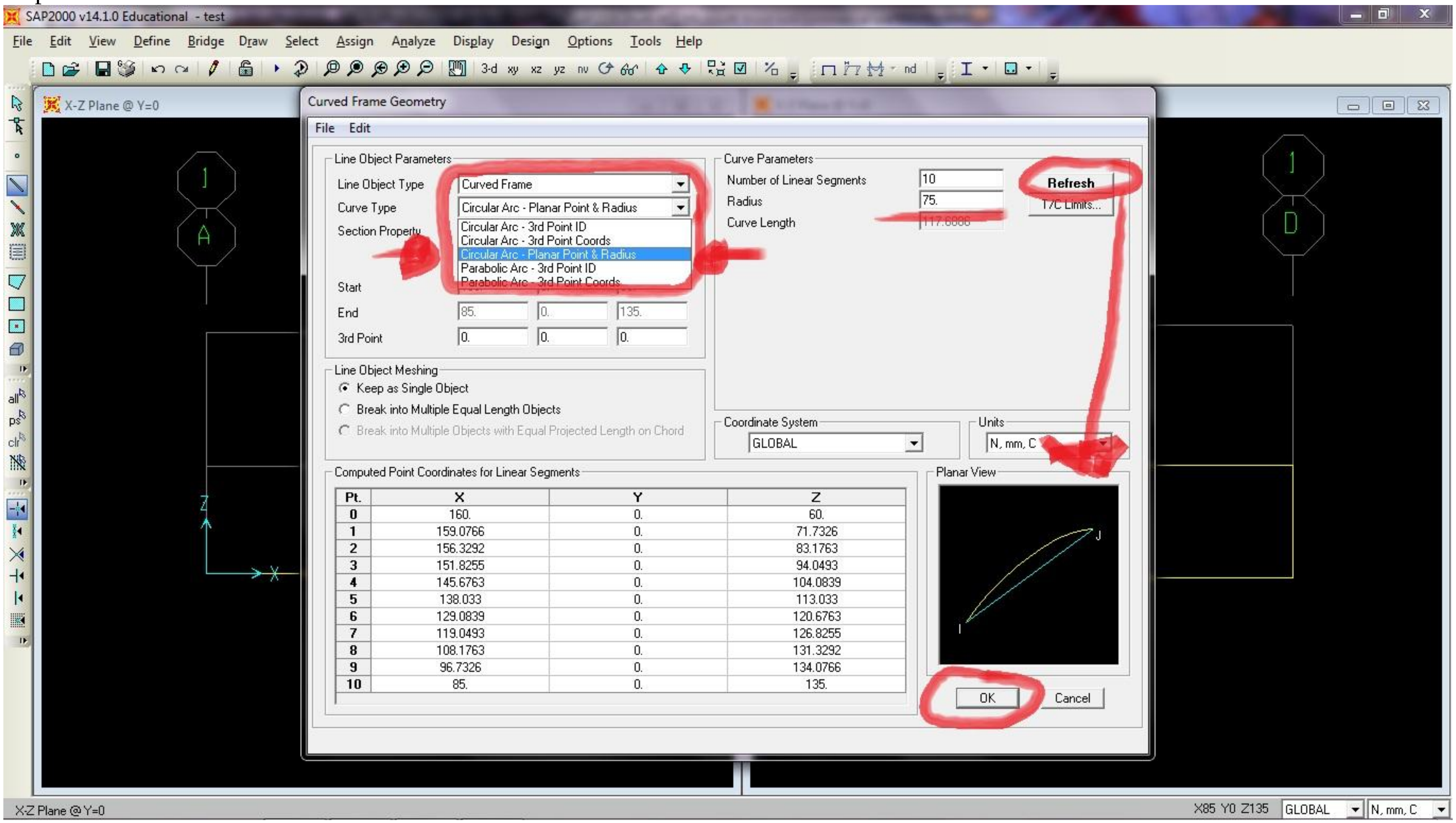
Step 9:



Directions:

We will now design the curved section of our problem. So begin by selecting the drawing tool but this time we will change the **Line Object Type** to a **Curved Frame.** After this is done layout the curved section as shown in the picture above.

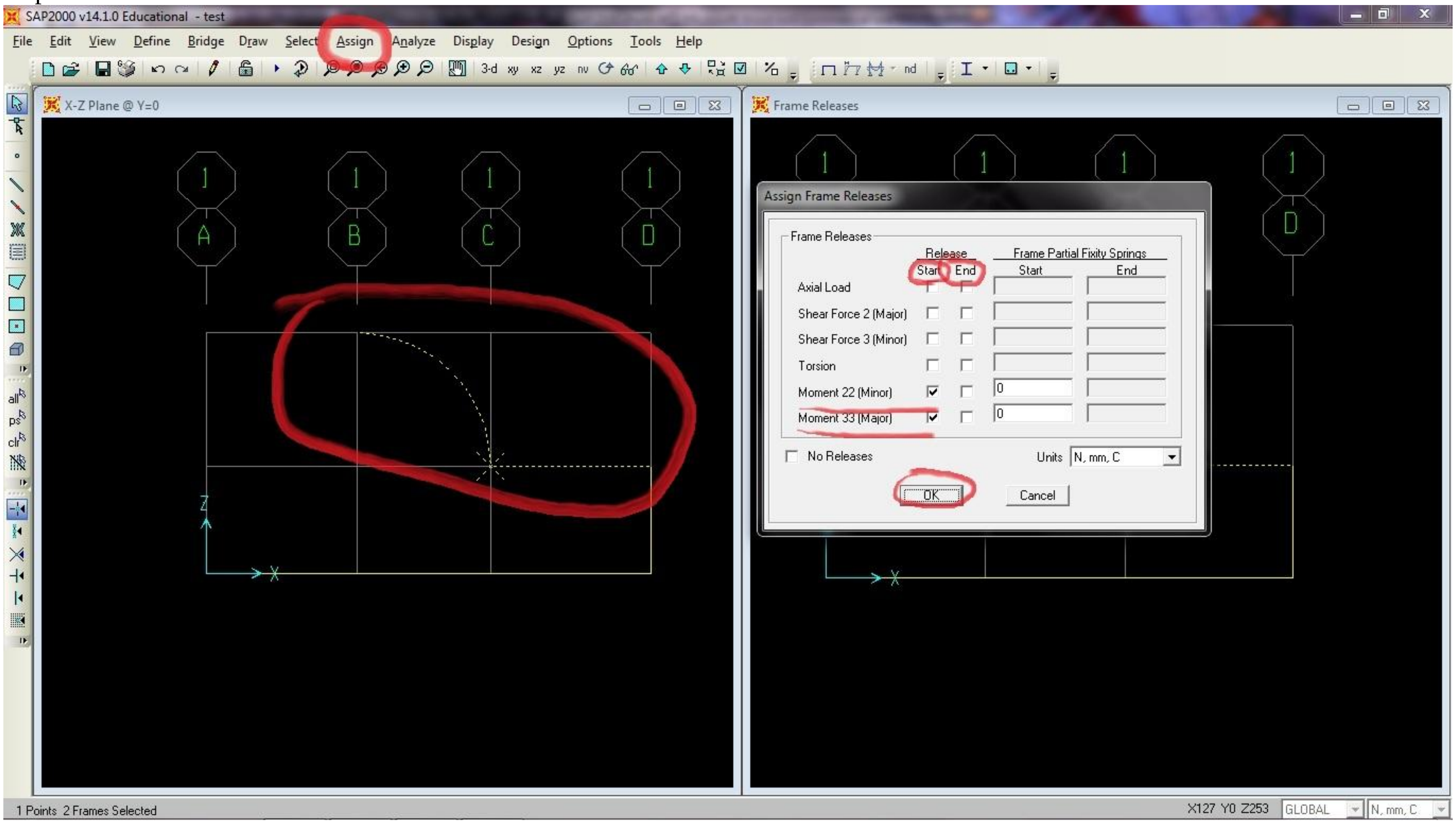
Step 10:



Directions:

The **Curved Frame Geometry** window will now display. And so we want to select under **Curve Type, Circular Arc – Planar Point & Radius.** After which the under the **Curve Parameters** we should now see **Radius,** so let's define the radius as **75 mm.** Hit **refresh** to see the model update and then click **Ok.**

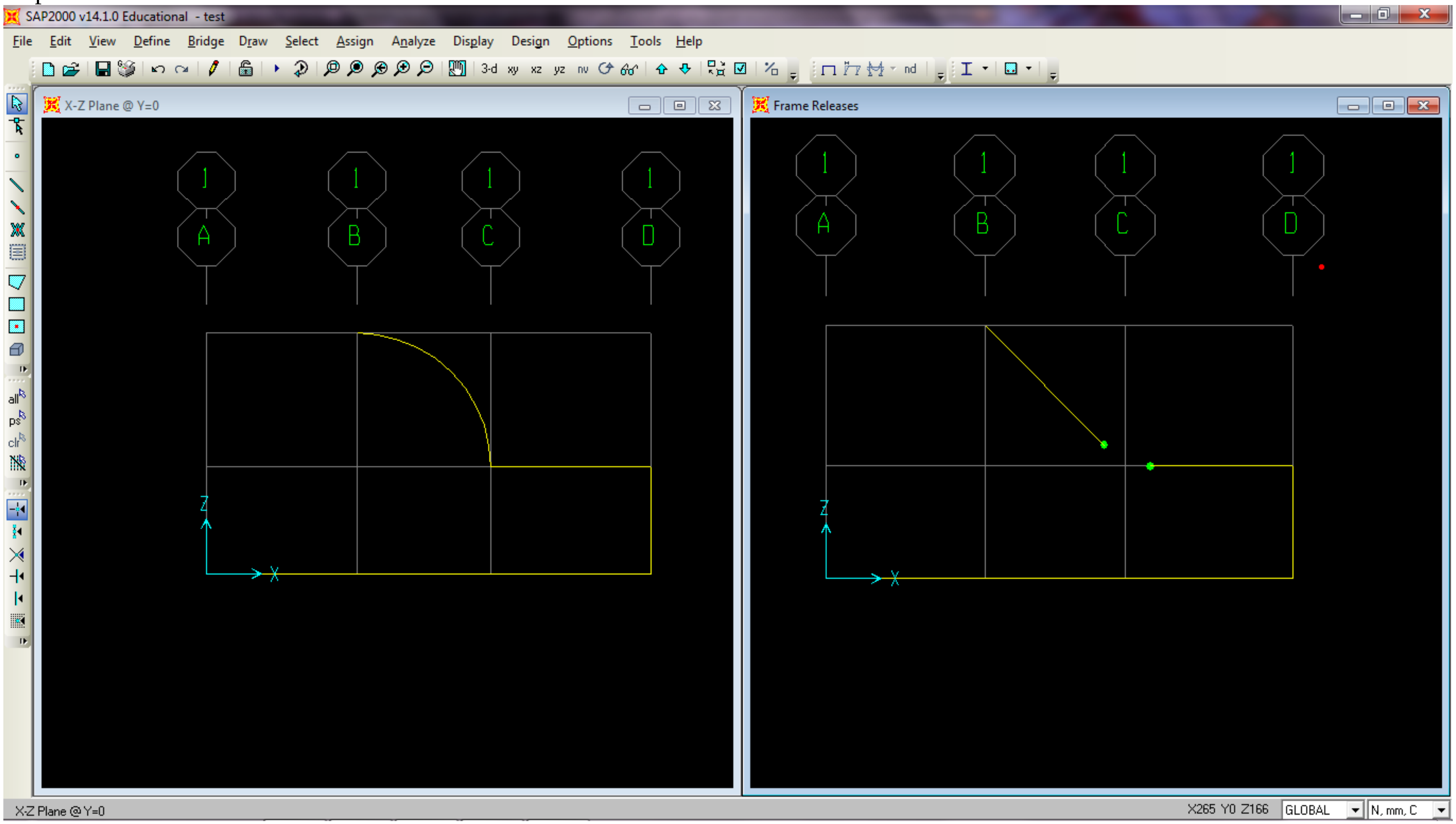
Step 11:



Directions:

We now need to define our joints. SAP2000 automatically defines the joints as a continuous section. But there is a pin at point C. To define that point we first highlight the two members involved and the joint, then we click the **Assign Menu → Frame → Releases/Partial Fixity…** The **Assign Frame Releases** should appear. Depending on your layout you may need to alternate between which end is the starting point and ending point that requires the release. Release both **Moment 22 and 33, minor and major.**

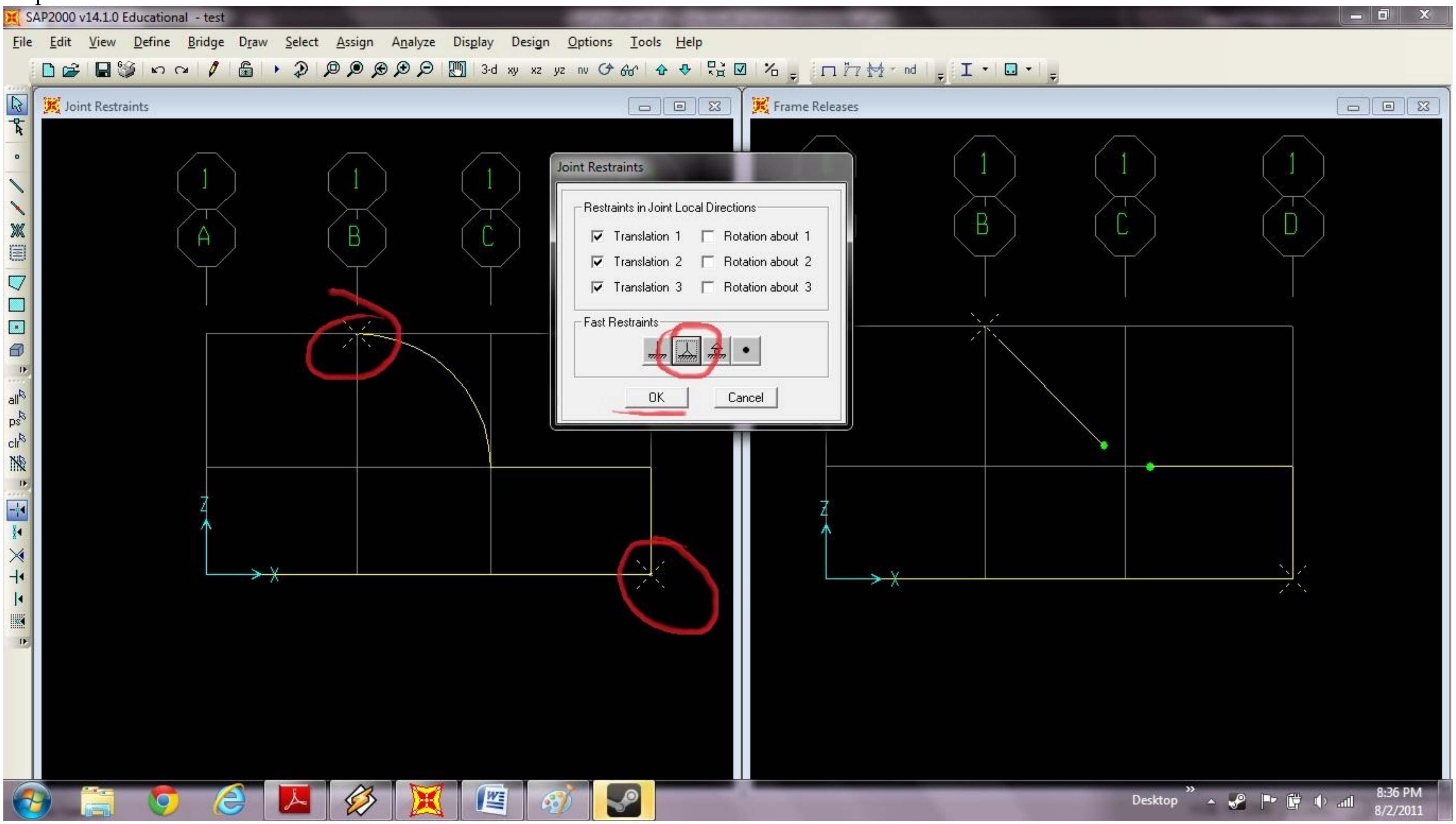
Step 12:



Directions:

If the steps above were done correctly, we should observe the slide on the right. Notice the opening, this indicates that between the two members there is a pinned connection.

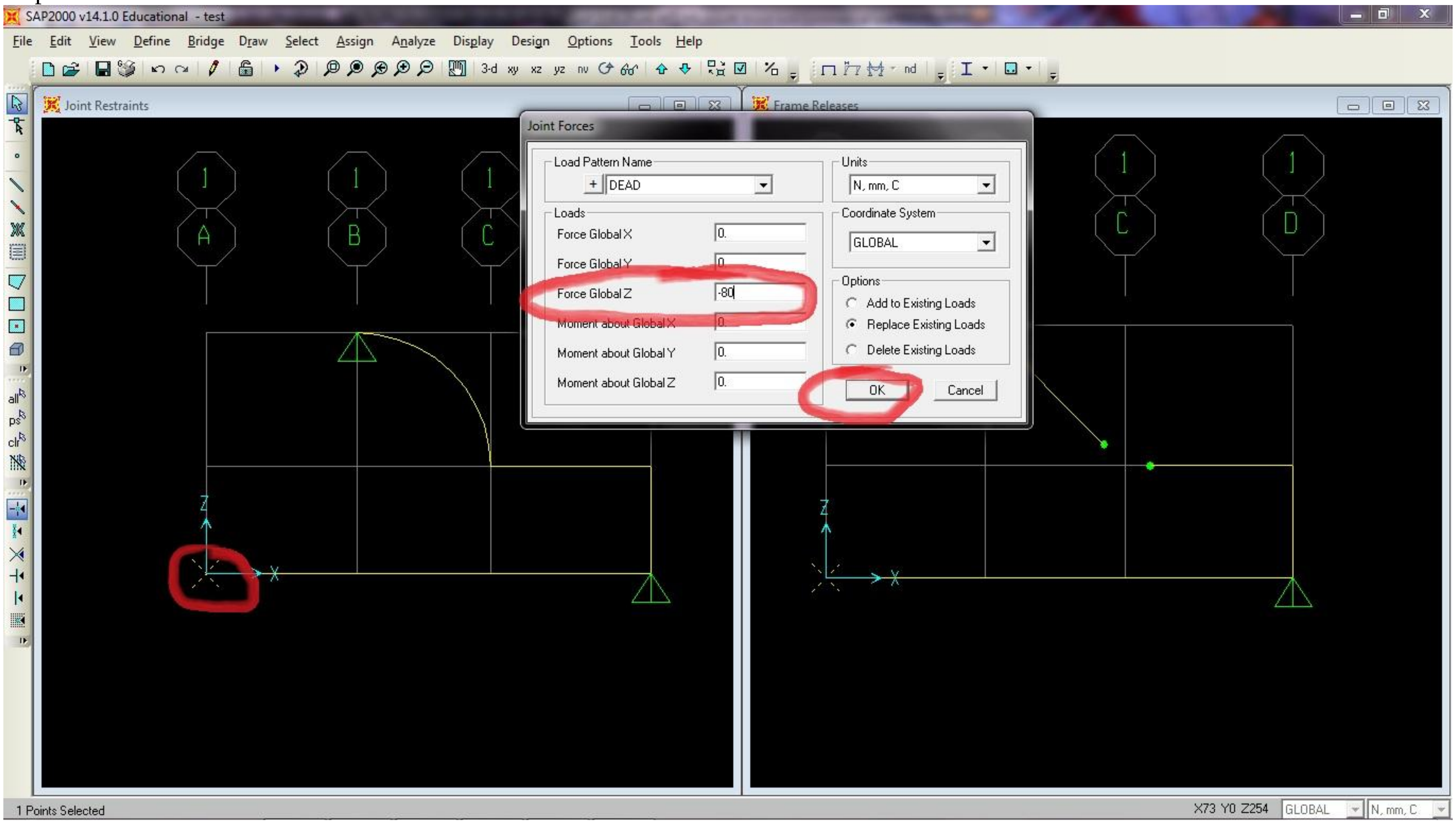
Step 13:



Directions:

We want to assign fixed pin supports to point D and B of our model. So we proceed by selecting the points B and D. Now we select **Assign → Joints → Restraints.** This leads to the **Joint Restraints** menu. Select the restraint that looks like a triangle. This represents a pinned fixed support. And click **Ok.**

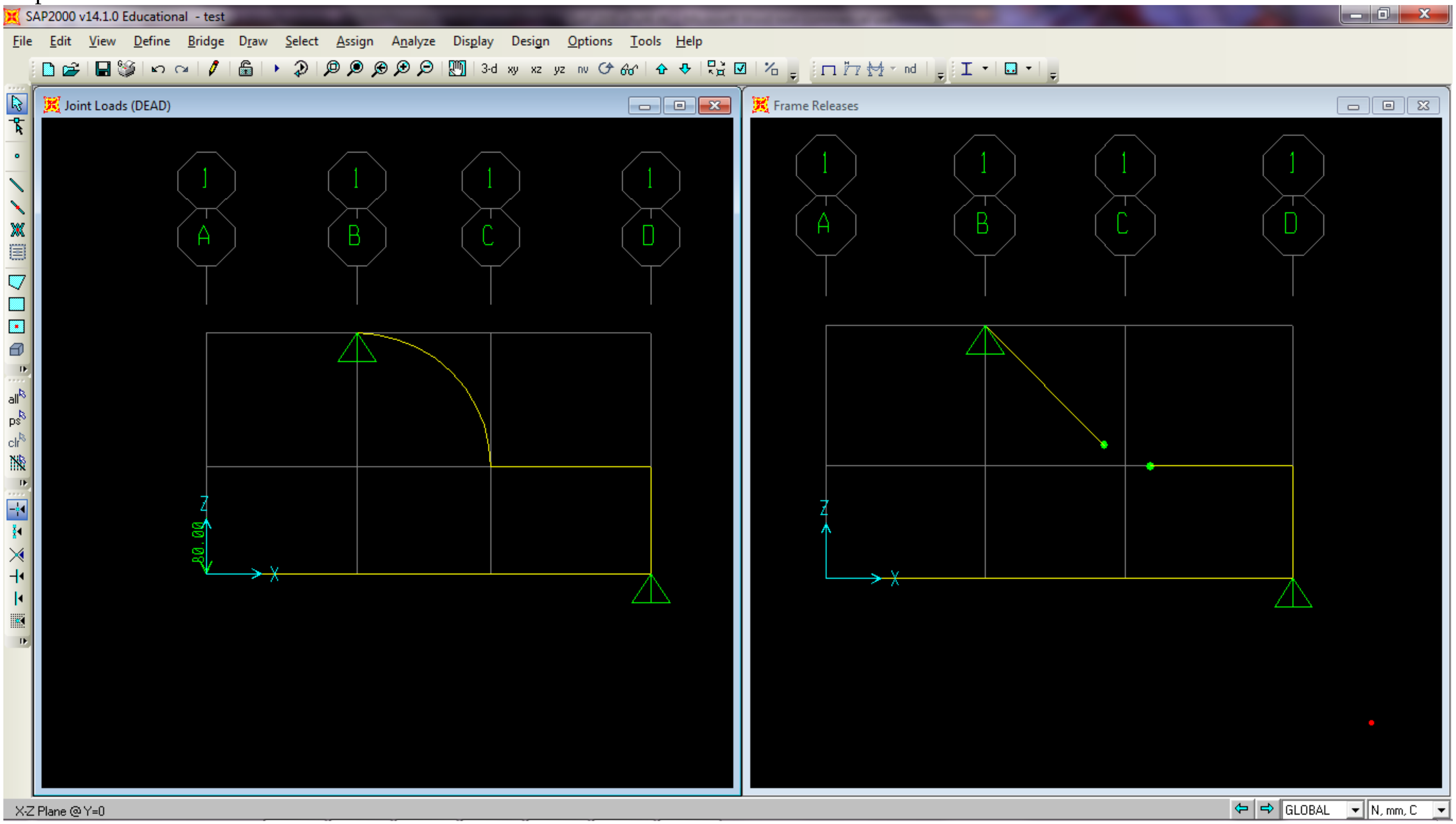
Step 14:



Directions:

Lastly we want to assign the **80 N force.** We proceed by selecting the point where we want to assign the load. In this case it is at the end near the **Z and X axis**. Then we select the **Assign →Joint Loads → Forces** and the **Joint Forces** menu should display. Now we assign a **-80 value for our Force Global Z** and click **Ok.**

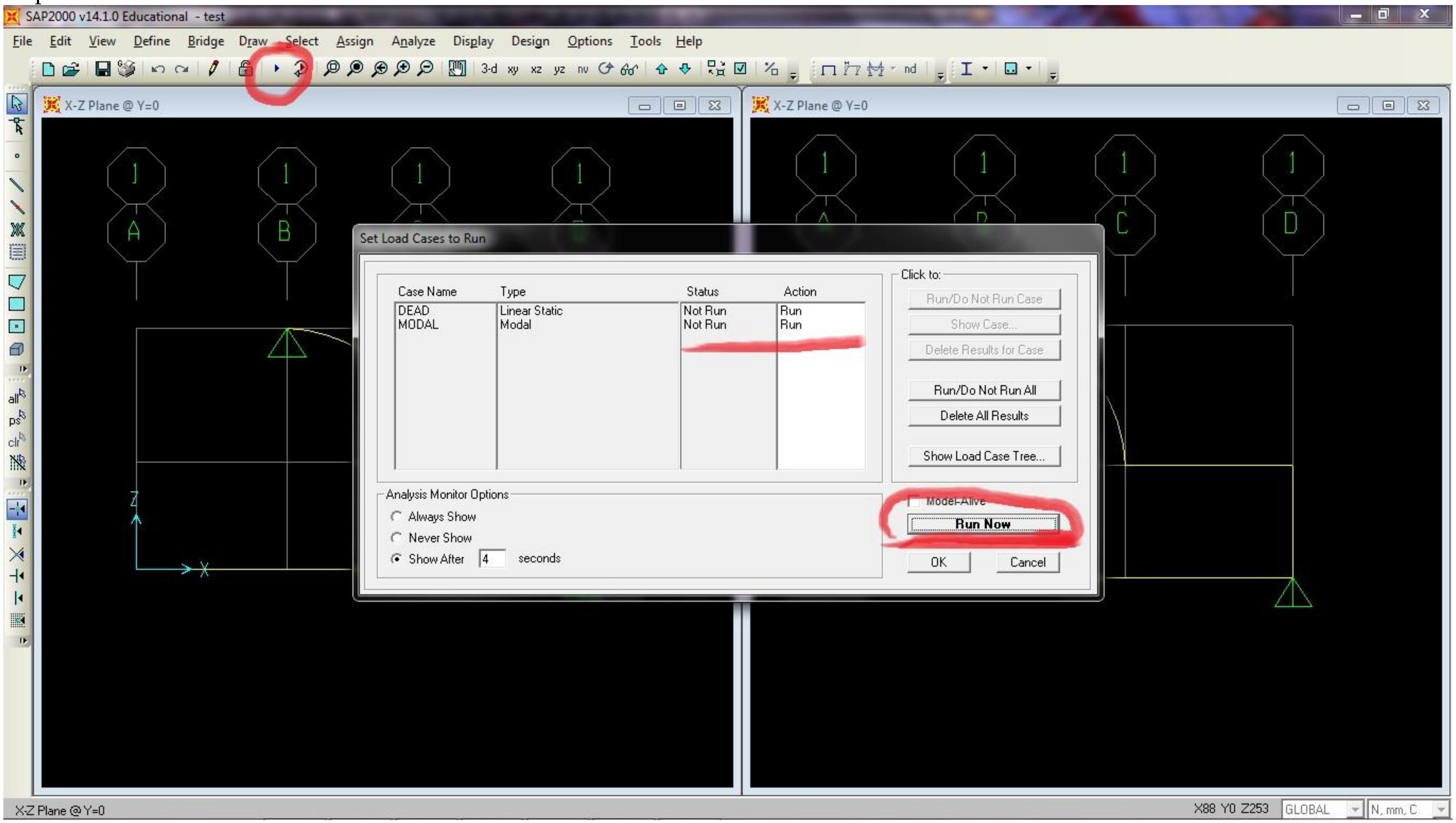
Step 15:



Directions:

If done correctly, you should see what is displayed above. We can see a force of **80 N** being exerted on the end of the 250 mm length beam, the pinned connections at point **C,** along with the fixed pinned supports at **B and D.** If the screen is not displaying the force, go to **Display → Show Load Assigns → Joints** and click okay on the popup window that shows up. If you have trouble viewing your pinned connection, go to **Display →Show Misc Assigns → Frame/Cable/Tendon** and finally select the **releases/partial fixity** and click the **ok** button.

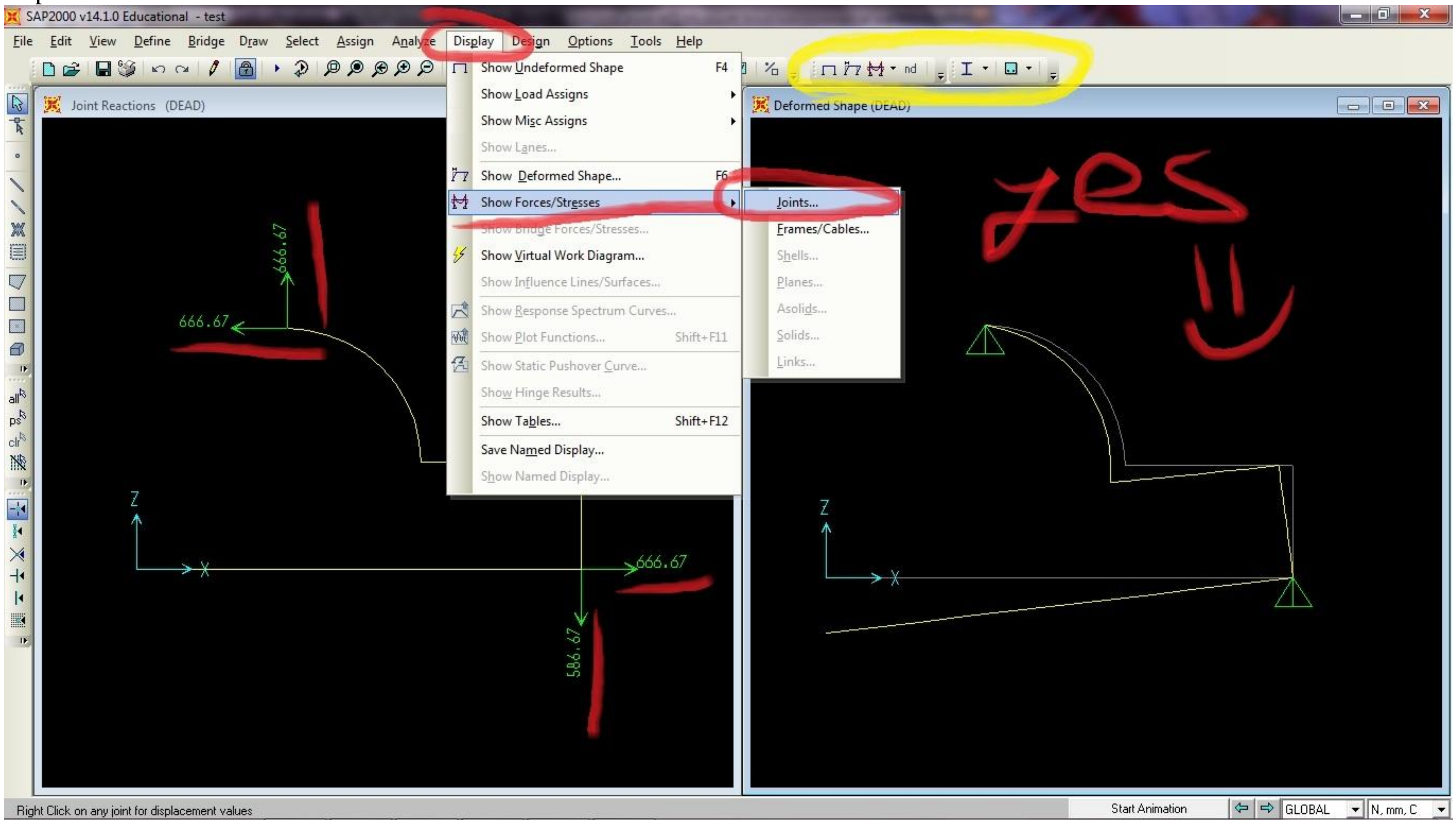
Step 16 ANALYSIS:



Directions:

Now we are finally ready to run the analysis! Proceed by clicking on the side ways triangle button ► or run button. The **Set Load Cases to Run** menu should appear. Ensure that it looks like the setup above and then click **Run Now.**

Step 17 SEE YOUR RESULTS:



Directions:

To display the results that we want to see, we first go to **Display → Show Forces/Stresses → Joints** then click ok. We should see the results that I have on the left in the image above. Notice that Point D displays has 666.67 in the Z and 666.67 in the X direction. If we apply the Pythagorean theory we should get **942.8 N** and similarly for B we get **888 N** which is what we want! If we play with the tool bar, highlighted above in yellow, we can even switch to a deformed shape to further analysis our model.

This concludes this tutorial.