**32\_Assigning properties and loads to joints**

Let's start with the points.

We will cover two options for assigning properties to points

First is diaphragms and the second one is restraints. You can also access these tools from this shortcut for diaphragm or from this shortcut for shortcuts.

We already have define diaphragms in define menu but we did not define restraints. Because they are already defined in ETABS.

Lets first assign diaphragms to these points. Select all these points and click on assign menu then points and then diaphragm. A list will appear containing all the diaphragms we have defined under define menu.

Select d1 and apply. You will notice that all these points are connected through white lines and at center of mass of boundary of these points you will see the name of this diaphragm.

Remember that diaphragms are applied only to horizontal plane. So diaphragms in simple words can be applied at plan view. Two different stories can have the same diaphragm name. For example we have applied at roof this diaphragm we can also use the same diaphragm at second floor. They will act separately. But we cannot assign diaphragm at different levels inside a single story for example a stair.

You can draw slabs and after that select all the points and assign the diaphragm as well. But there is a difference between assigning the diagram to points and to slabs; we will cover this in our advance course on analysis to find out the differences. For the time being you can consider applying diagram to points after drawing the slab.

We will be now assigning restraints to points. Restraints can only be applied to points in ETABS. In some other softwares like in LUSAS you can also apply restraints directly to lines. You will notice there is no restraint option in line or area menus in ETABS.

We can assign restraints from these predefined buttons or by choosing as many restraints as we want by checking these boxes.

For example let's apply a fixed support to this point. Now let's apply a pin support. You will notice that appropriate boxes are being checked here automatically. Similarly you can apply free supports. This support is a roller support.

All of the superstructure in this example will have no supports. Only base which is the level of foundation will be restrained. Let's apply pin supports at the base. We will also discuss choosing between different types of supports in advance analysis course.

Now let's apply some point loads let's say to this point of this beam. You can apply loads to points by going to assign and then assign point loads and then click on force or by directly clicking on this shortcut.

From the load case list you can select appropriate load case in which you want to apply the loads and the units from here.

There are 6 boxes here, 3 for force and 3 for moments.

These forces will be applied in global x y or z directions. You can also apply moments here.

You can also have multiple directions loading in one case as well.

For example let's apply gravity load of 100kn here with minus sign. Because global z is positive upwards. You can review basics of sign convention in sign convention lecture.

Here you have some options to either replace the already existing load at this point if any or to add it to previous one or delete the already applied loads in this specific load case.

Be default replace option is selected which will replace the previous loads with new ones.

To delete previously applied loads you don't need to enter any value here. It will delete all the previous loads in this load case at this particular point.

These three options are common when applying loads to lines as well as area elements.