**39\_Checking your models before analysis**

Just before running the model you should check your model if it is properly configured. Check if you have defined and assigned all the sections and their modifiers. For example, let's check the load cases, load combinations, property modifiers, and mass source options.

You should also check your model geometry by extruding it to see if you input dimensions of any member in meters instead of millimeters mistakenly. For example lets extrude this structure by clicking on building view options and then click on extrude.

The geometry looks okay. Now let's change any section depth intentionally to a very large value. Click on define sections and then add a new section with 0.2m width and 1000m depth. Click ok and go back to main model screen. Now extrude the geometry again.

By checking like this, you will come to know these problems before running analysis and can rectify them.

Similarly you can check by showing different tables from display menu. Click on display menu, then on table option. from here select any option you like to investigate. Let's say you want to check restraints applied to points. Click on point assignments option from here.

Also make sure each node is connected properly in continuation manner. For example let's draw two beams without use of snaps by clicking on this point beams seem connected but if we click at this area, we will notice 2 points in status bar. Zoom in this area and you will see two different points. You can merge these points from merge point option in edit menu.

Similarly this slab is not drawn properly because we zoom at this detail we will find out that slab point is not connected with column point so column will not carry any load from this slab. So make sure to connect all nodes properly.

Similarly there are also some limits on drawing number of sides of shell and membrane objects. For example shell can have maximum four edges. If you draw a shell having 5 edges or more make sure to divide it properly by doing meshing or redraw it using maximum of four sides.

Also if you apply loads to null areas or lines there will be a loading transfer error.

You can check all these errors before running analysis from check model tool in analyze menu.

Here you can check all boxes and keep the tolerance default. ETABS will report you a log text file containing information for all the errors. You can find them by label option and zoom in that specific area one by one to remove those errors.

Ideal a model should have zero geometry error but in complex buildings there will always be certain degree of error in geometry. We will discuss on how to handle them in our next advance course on modeling.