**35\_Assigning properties to areas**

Let's learn how to assign properties and loads to area objects. You will click on assign menu and then assign shell area to assign different properties to them. Areas in ETABS can be slabs, ramps and walls. You can assign them a slab or wall section from this menu or make them an opening by click here. You can do these things also by directly click on this shortcut for slab section assign or on this shortcut to assign them as opening.

You can also them diaphragms or rotate their axis from here just like we did in points and lines. You can also click on these shortcuts.

Stiffness modifiers are applied from this tool to areas including slabs and walls. Stiffness modifiers concept in area is also the same what we have discussed already in lines in previous lecture.

And finally there are slab and wall automatic meshing options. We did manual meshing from edit menu in basic edit menu lecture. We can create invisible mesh for analysis automatically by this option.

A point to remember here is to apply auto meshing to only shell objects. There should be no meshing for membrane objects.

For example this is a solid slab resting on columns and is defined as shell object. We need to do meshing for object either manually or automatically. Usually both options are combined. Slabs or walls are drawn in small or finite size or manually meshed to make them smaller and then meshed automatically using this tool.

But why we do meshing? Meshing is very important to properly transfer load and to read correct stresses. Because the analysis is finite element analysis for stress we need to make the elements as small as possible in order to read correct results.

This is an advance topic and we will cover them in advance analysis course. We will learn concepts about patch test, proper mesh size, mesh convergence and stress concentrations from finite element analysis point of view.

For the time being we will be doing automatic meshing for shell objects from this tool by click on all these boxes and then further dividing it in every unit meter to make it finite. These options are for floors or slabs and these for walls and ramps.

These divisions will be invisible on screen unless you click on building view options and turn on analysis mesh.

This mesh is internal to ETABS.

This was for slab meshing.

To do wall meshing select this wall and go back to the same tool and this time select to subdivide objects in max size of 1m and check on add restraints on edges.