**43\_Reading shell results**

You can view shell forces by the same way you did for frame results. That is by click on this small arrow of this shortcut an then clicking on shell forces option.

Here you have a list of combination of load cases then you can view either forces or the stresses, then you can specify contour range in selected units. Shell forces are shown as surface contours.

Then there are options to average stress values at nodes when different elements are connected at common nodes. It is recommended to use none option. By choosing none option you will have better idea of the mesh quality and will know if you need to refine it or not by checking discontinuities and stress concentrations from results.

We will cover advance topics on meshing in advance analysis course.

Stress averaging at all joints might be misleading sometimes.

Here you can check if you want to display contour results on deformed shape under the same load case or to show contours on un deformed shape.

Now let's talk about what results you can display in ETABS for shells.

You can display forces or stresses in each local direction. Remember that like in frames, shells also have their own local axis and the results are always displayed in local axis of the elements.

For example f11 is the force acting on face 1 and directed towards 1 local direction.

Similarly you can display different stresses.