**53\_Design of slabs in SAFE**

After successfully importing slab model from ETABS to SAFE 12. You need to make sure if the import was ok or not. Check slab loads by right clicking on it on different load cases. You can also show loads from Display menu. Also check load cases and load combinations. Also remember to change the slab type to "Stiff" for stiff elements as this is a bug in some versions of SAFE 12. Stiff elements are very stiff elements as the name suggests under the column, usually 1m thick or half of story height or full height depending on case to case.

After having verified the imported model, you can apply additional loads if you want in SAFE to areas as well as to beams. Generally stiffness modifiers are not applied to elements in SAFE but you have to option to apply these factors if you want. We will keep them 1.0 as default values. Remember that we apply modifiers in ETABS to account for redistribution of moments between slabs, beams and columns due to different rate of and extent of cracking between these elements. Stiffness modifiers are a simple way to account for reduced inertia. In SAFE while designing slabs we are not concerned with load paths between vertical and horizontal elements as we are just designing a plate. We will cover stiffness modifiers in much more detail in A3 course.

In this lecture, we are covering a very basic design of slabs to introduce you to the concept. Detailed analysis and design of slabs and foundations with cracking and long-term deflection is covered in next advance courses.

Specify the cover and bar diameter options from "Design" menu and select the location of columns for punching calculations from design overwrites if the slab is a flat plate or a flat slab.

Save and run the model. After analysis, you can show shell contours on different load cases just like in ETABS from this option. Here you have options to either show resultant, top face, mid surface or bottom surface stresses. Usually resultant forces are checked. Then you can show contours on deformed or un deformed shapes and even on extruded form.

To check deflection, click on this triangle button then select a load case or a combination to view deflection contours. Punching failure can be checked by this button. N/C means punching is not calculated at this location because we have a beam here. Punching is not calculated when a beam is present. You design for the punching reinforcement in beam for these cases. You can right click on punching ratios to see more detailed calculations. If punching ratio is more than 1.0 it indicates a failure in punching.

You can see these ratios easily in tables and then exporting the tables to Excel.

For reinforcement design you have 3 methods.

First one is to read shell contours on the worst load combination and calculate the capacity of the section outside the program for a singly or doubly reinforced RC section. For example, we need this is the capacity of this section with this reinforcement. We can then check M11 and M22 contour range to see if the moments are within this range. M12 is can be conservatively added to both M11 and M22. We will discuss these things in more detail in A3 course of this series.

You can download this tool to calculate moment capacity of a singly reinforced concrete section for free in this lecture.

Other way to design reinforcement in SAFE is by clicking on "Show slab design" shortcut button from here. In this tool, you have further two options; either to use strip based method or FE method. Strip method is only available if you have designed and modeled design strips. We will come to the topic of design strips shortly.

Let's design this slab from FE contours. From here select the design type either as flexural or shear. Here you have four options to for each direction of reinforcement. Then you have option to impose minimum reinforcement requirement. Here is the option to show rebars above specified value. What this means is if you have a typical mesh in the slab, you can see what additional reinforcement you require above that mesh. Let's start by keeping main mesh as zero by selecting none option.

In contour range you can specify the value of reinforcement in mm² per meter. For example let's put dia 12 bars spaced every 150mm that gives 753 mm² per meter. Now show top rebar in direction 1 and then in direction 2. Also see bottom rebar. All the reinforcement contours are within dia 12 at 150mm spacing so the provided reinforcement is sufficient for this slab.

Now let's apply a uniform top and bottom mesh of dia 12 at 150mm spacing by using this option. Now the contours will show you additional reinforcement in excess of dia 12 at 150. So the additional reinforcement is none, our design of slab is ok. You can also draw rebar objects in SAFE models and choose this option.