Floor Auto Mesh Options

Form: Shell Assignment - Floor Auto Mesh Options

During analysis, ETABS automatically meshes all shell objects that are used to model floors. Meshing helps distribute loads realistically. In some cases, automatic meshing of a shell object into the analysis model may not be desired; click here for an example. Use the Shell Assignment - Floor Auto Mesh Options form to alter the meshing of shell objects to something other than the default settings. The use of Rectangular meshing for the default settings is specified using the Analyze menu > Automatic Mesh Settings for Floors command.

1. Select the shell object(s) for which the mesh is to be revised or marked so as not to be meshed at all.
2. Click the Assign menu > Shell > Floor Auto Mesh Options command to access the Shell Assignment - Floor Auto Mesh Options form.
3. Select the appropriate option(s):

- Default option. This is the setting that will be used on all shell floor object meshing unless another choice is specified on this form. Following are the three categories of shell objects used to determine the default settings – which one to use depends upon the properties of the shell object and its orientation.
    - Horizontal decks and planks use the auto cookie cut at beams and walls meshing option. Because these types of floor objects do not have out-of-plane stiffness, vertical loads are transformed from the floors onto the beams and walls using tributary area algorithms.
    - Horizontal slabs and layered slabs use the automatic rectangular meshing as specified on the Automatic Mesh Options (for Floors) form. For slabs with membrane behavior, vertical loads are transformed to vertical supports using tributary area algorithms.
    - Inclined floors use the no auto meshing option, i.e., they are not internally meshed and remain as a single element equivalent in size to the floor object.
- For Defining Rigid Diaphragm and Mass Only (No Stiffness - No Vertical Load Transfer-Applies to Horizontal Floors Only) option. No stiffness or vertical load transfer associated with these objects will be considered during the analysis.
- No Auto Meshing (Use Objects as Structural Elements) option. Tags the shell object to not be automatically meshed by ETABS into the analysis model, thereby treating the object as a Structural Element.

- Mesh Object into ____ by ____ Elements option. If the selected object has no curved edges and is defined by 3 or 4 nodes, this option can be used to mesh the selected shell object into a specified number of elements in the local 1 direction and a specified number of elements in the local 2 direction.
- Auto Cookie Cut Object into Structural Elements option. Allows control of meshing according to the following:
    - Mesh at Beams and Other Meshing Lines (Applies to Horizontal Floors Only) option. Similar to the default option for decks, planks and slabs with membrane behavior.
    - Mesh at Vertical/Inclined Wall Edges (Applies to Horizontal Floors Only) option. Meshes the selected object at walls, thereby transferring loads, as appropriate, to and from walls. Similar to the default option for decks, planks and slabs with membrane behavior.
    - Mesh at Visible Grids (Applies to Horizontal Floors Only) option. Meshes the selected objects at grid lines used in the model.
    - Further Mesh Where Needed to Maximum Element Size of {specify value} option. Meshes the selected object(s) using the specified maximum element size.
- Add Restraints on Edge if Corners have Restraints checkbox. Check this box to restrain the edges of the mesh elements when the corners of the shell object are restrained.
- Advanced - Modify/Show Auto Rectangular Mesh Settings button. Click the button to display the Automatic Mesh Options (for Floors) form. Use the form to specify how and where the object will be meshed and the approximate maximum size of the mesh when floor-type shell options use auto meshing.

4. Click the Apply button to complete the assignment.

When the Apply button is used, the Shell Assignment - Floor Auto Mesh Options form will remain open until it is closed by clicking the Close button. This allows selection of another shell object(s), to which different assignments can be made.

If only one assignment is being made to only one set of selected objects, the OK button can be used to both apply the assignment and close the form.