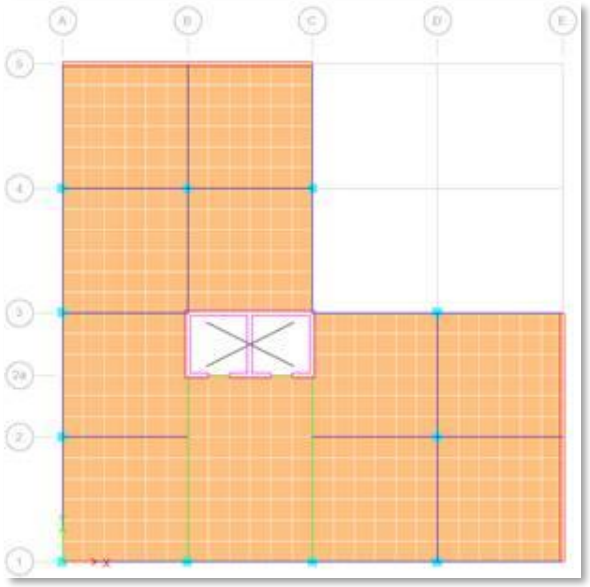
Technical Note: Gravity Load Collection Using Floor Object

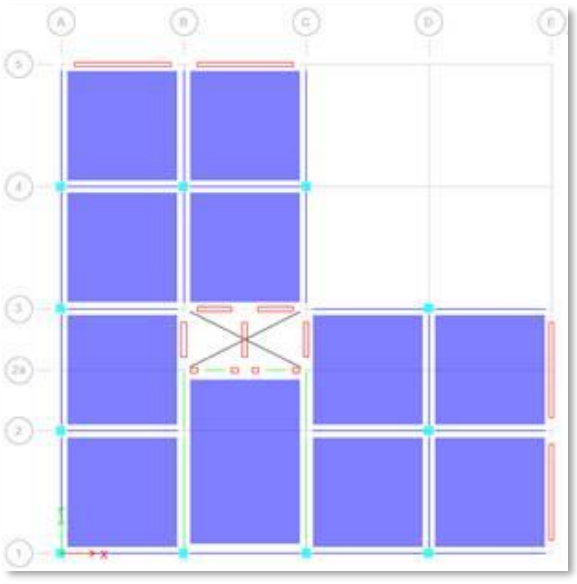
In ETABS, there are two approaches for floor gravity load collection:

1. Finite element method using plate or shell floor objects.
2. Conventional trapezoidal/ triangular or 1-way load tributary collection with 4-sided or 3-sided membrane element.

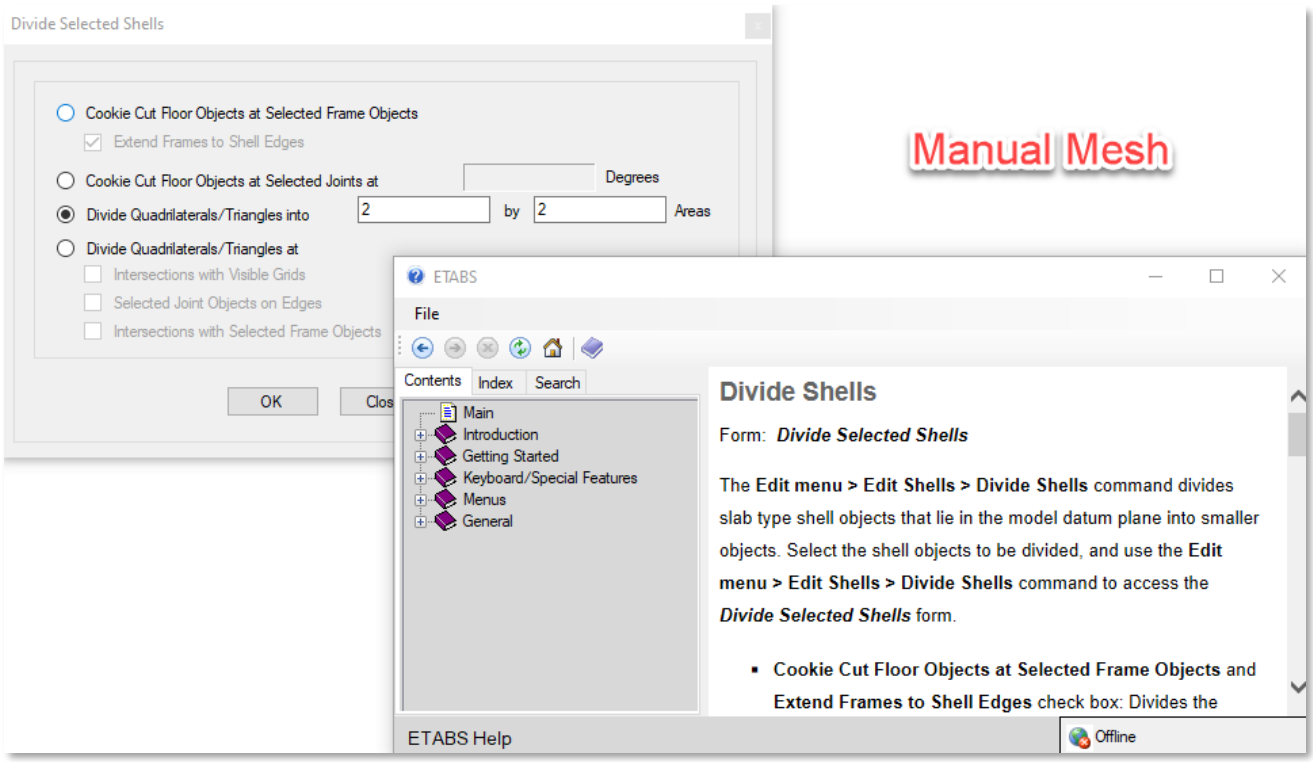
The logic is simple. If the area object has out-of-plane stiffness, namely the Shell or Plate object, ETABS would use the FEM approach. It is, therefore, necessary to refine the mesh of floor object into at least a 4x4 mesh for proper FEM analysis:

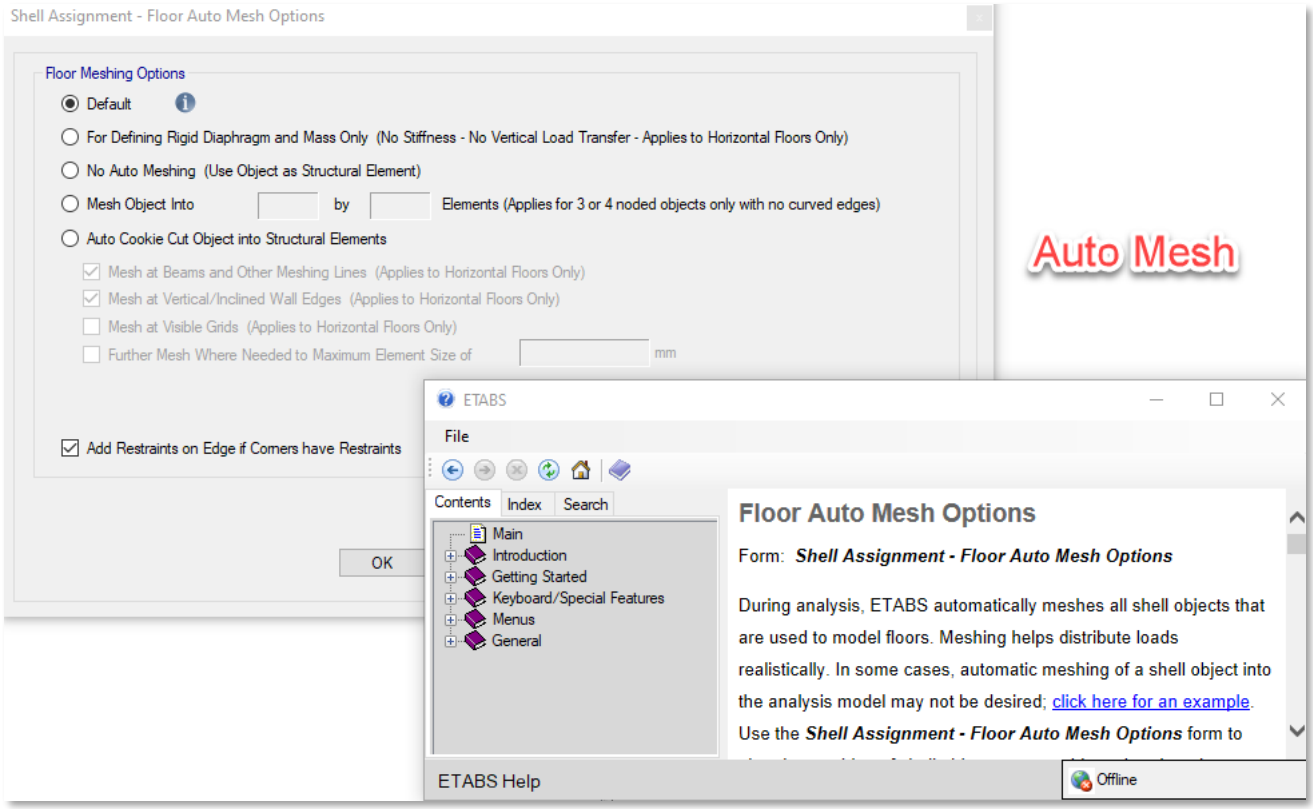


If the area object is the Membrane type and therefore has no out-of-plane stiffness, ETABS would automatically switch to the conventional trapezoidal/ triangular tributary collection method. The floor object needs to be meshed or input according to the conventional by-by-bay approach similar to the hand-calculation procedure (meshed floor objects shrunk to for clarity). **The corner joints of membrane and deck elements must be supported by columns, beams, walls, braces, etc, otherwise unrealistic gravity deflection may occur**:



Users must learn in detail how to use the manual and auto mesh options (detailed information on these options are in the Help system):



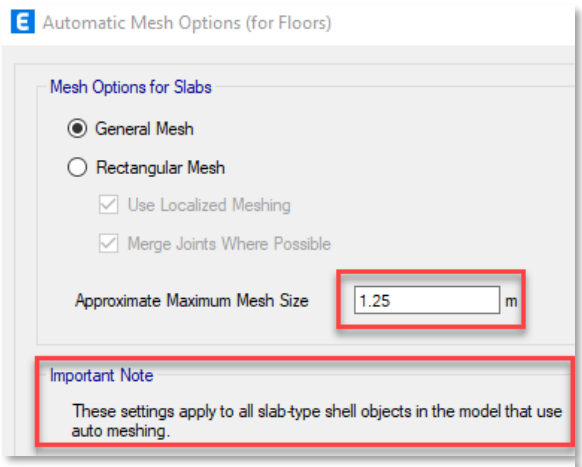


Please also watch this YouTube webinar:

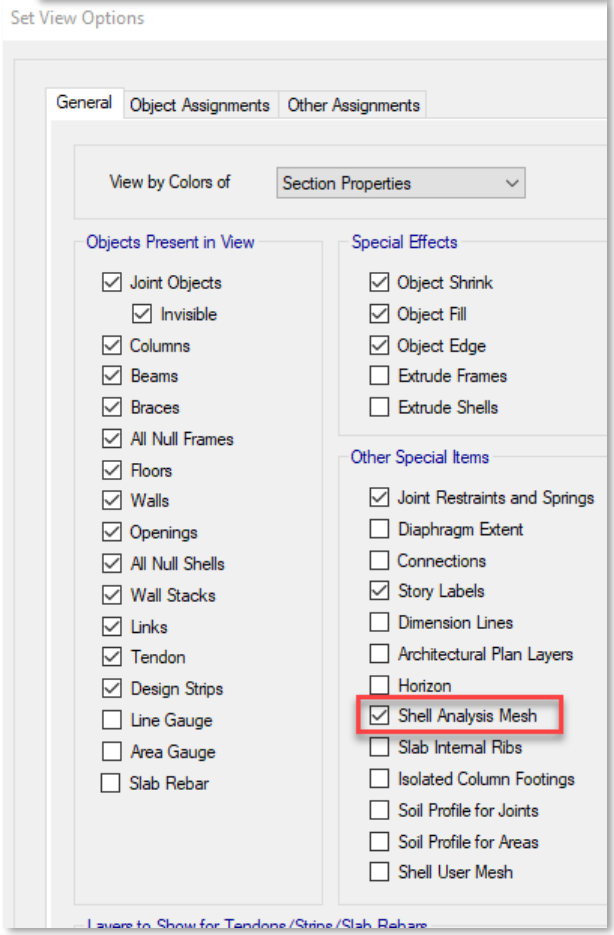
[WEBINAR: How to use Meshing Tools Effectively - YouTube]

Users should experiment with the default meshing options **<u>including the mesh size</u>** to see which option is more suitable for the floor shape and type:

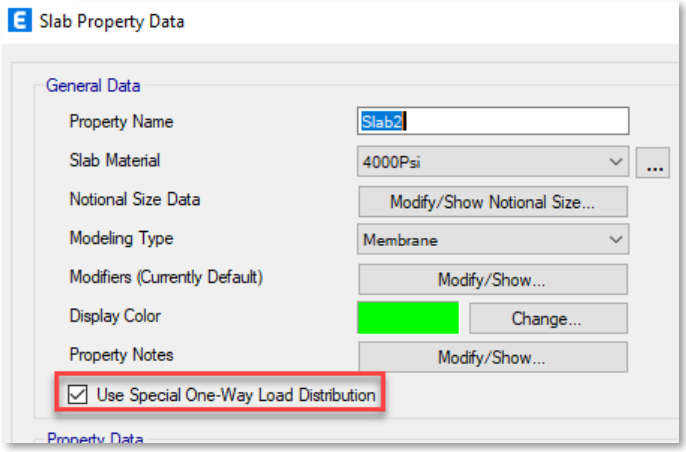
For more regular floor shapes, I recommend the Rectangular Mesh. For irregular floor shapes, the default General Mesh may be more suitable. This automatic meshing of floor objects especially for transfer plates, foundation caps, … etc, is a powerful tool. If the resulting mesh is good, there is no need to use the manual and auto mesh assignments.



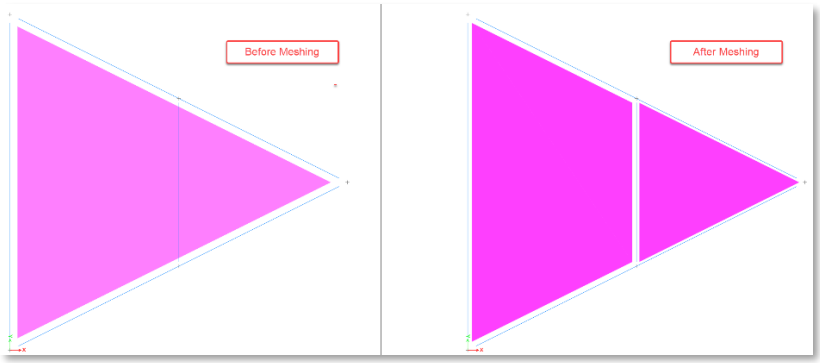
If the automesh algorithm does not produce the desired mesh, you can draw null lines to facilitate floor meshing. The membrane type area is automeshed at beams and null lines. Another alternative is to draw the floor objects manually. Users should always turn on the “Shell Analysis Mesh” in the view options to check if the analysis mesh is appropriate:



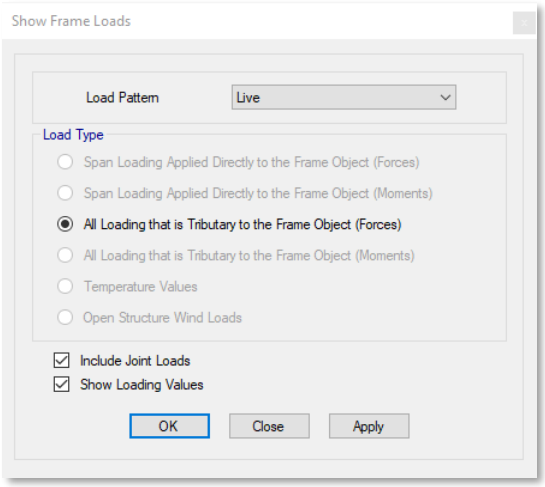
For 1-way load collection using membrane floor object in the <u>local 1-direction</u>, the mesh is the same as the conventional by-by-bay approach shown above. Furthermore, the users need to check the “Use Special One-Way Load Distribution” checkbox:

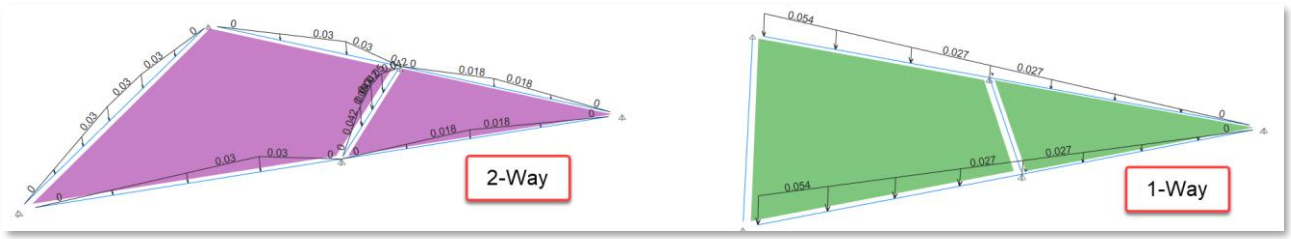


When using membrane floor for gravity load collection, it must be either **quadrilateral or triangular shape AFTER auto or manual meshing**:



After analyzing, users should review the tributary loads transferred to the supporting elements:





If the mesh is not good resulting in improper tributary loads distribution, users should assign such loads manually to the supporting objects.

When using conventional bay-by-bay mesh, the membrane object can only have uniform loads. For point loads and partial distributed loads, users may consider applying such loads as:

1. Averaged uniform load.
2. Loads directly on the supporting beams walls (corner nodes).
3. Switch to the FEM method using shell or plate objects.