## 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):

Cookie Cut Floor Objects at Selected Frame Object     Extend Frames to Shell Edges     Cookie Cut Floor Objects at Selected Joints at     Divide Quadrilaterals/Triangles into     Divide Quadrilaterals/Triangles at	ts Degrees Area	as	nual Mesh
Intersections with Visible Grids	🕑 ETABS		- 🗆 X
Selected Joint Objects on Edges     Intersections with Selected Frame Objects	File		
	Contents Index Search	Divide Challe	
OK Clos	Main Main Main	Form: Divide Selected Shells	
	Keyboard/Special Features     Menus     General	The Edit menu > Edit Shells > slab type shell objects that lie in objects. Select the shell objects menu > Edit Shells > Divide Sl Divide Selected Shells form. • Cookie Cut Floor Object Extend Formes to Shell	Divide Shells command divides the model datum plane into smaller to be divided, and use the Edit hells command to access the ts at Selected Frame Objects and
		EVIODA FRANCE TO SDOU	
Assignment - Floor Auto Mesh Options	ETABS Help		Construct box. Divides the
Assignment - Floor Auto Mesh Options  Floor Meshing Options  Default  For Defining Rigid Diaphragm and Mass Only (No S  No Auto Meshing (Use Object as Structural Element) Mesh Object Into by  Auto Cookie Cut Object into Structural Elements Mesh at Beams and Other Meshing Lines (Applie) Mesh at Vertical/Inclined Wall Edges (Applies to Mesh at Visible Grids (Applies to Horizontal Floor) Further Mesh Where Needed to Maximum Element	ETABS Help Stiffness - No Vertical Load Transfer - Applies to nt) Elements (Applies for 3 or 4 noded object les to Horizontal Floors Only) o Horizontal Floors Only) rs Only) ent Size of mm	b Horizontal Floors Only) s only with no curved edges)	Auto Mesh
Assignment - Floor Auto Mesh Options	ETABS Help  Stiffness - No Vertical Load Transfer - Applies to tt) Elements (Applies for 3 or 4 noded object es to Horizontal Roors Only) o Horizontal Roors Only) rs Only) ent Size of mm  W ETABS	b Horizontal Floors Only) s only with no curved edges)	Auto Mesh
Assignment - Floor Auto Mesh Options	ETABS Help Stiffness - No Vertical Load Transfer - Applies to tt) Elements (Applies for 3 or 4 noded object es to Horizontal Roors Only) o Horizontal Roors Only) rs Only) ent Size of mm	b Horizontal Floors Only) s only with no curved edges)	Auto Mesh
Assignment - Floor Auto Mesh Options	ETABS Help Stiffness - No Vertical Load Transfer - Applies to att) Elements (Applies for 3 or 4 noded object es to Horizontal Roors Only) to Horizontal Roors Only) to Horizontal Roors Only) a Horizontal Roors Only) b Contents Index Search Contents Index Search Main Main Main Main Menus Menus Menus General	b Horizontal Floors Only) s only with no curved edges)           Floor Auto Mesh Op           Form: Shell Assignment - Fi           During analysis, ETABS autor           are used to model floors. Mesl           realistically. In some cases, au           the analysis model may not be	Contract box. Divides the  Contract box. Divide
Assignment - Floor Auto Mesh Options	ETABS Help Stiffness - No Vertical Load Transfer - Applies to att) Elements (Applies for 3 or 4 noded object es to Horizontal Roors Only) o Horizontal Roors Only) rs Only) ent Size of mm @ ETABS File @ @ @ @ @ @ @ @ Contents Index Search @ Eting Started @ Keyboard/Special Features @ Menus @ General	Extend Prantes to shell a Horizontal Floors Only) s only with no curved edges) Floor Auto Mesh Op Form: Shell Assignment - Fi During analysis, ETABS autor are used to model floors. Mesh realistically. In some cases, au the analysis model may not be Use the Shell Assignment - I	Contract and the second secon

Please also watch this YouTube webinar:

WEBINAR: How to use Meshing Tools Effectively - YouTube

Users should experiment with the default meshing options **including the mesh size** 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:

## E Automatic Mesh Options (for Floors) Mesh Options for Slabs General Mesh O Rectangular Mesh Use Localized Meshing Merge Joints Where Possible Approximate Maximum Mesh Size 1.25 m mportant Note These settings apply to all slab-type shell objects in the model that use auto meshing Set View Options General Object Assignments Other Assignments View by Colors of Section Properties $\sim$ Objects Present in View Special Effects Joint Objects Object Shrink ✓ Invisible Object Fill Columns Object Edge Beams Extrude Frames Braces Extrude Shells All Null Frames Other Special Items Floors Joint Restraints and Springs ✓ Walls Diaphragm Extent Openings Connections All Null Shells Story Labels ✓ Wall Stacks Dimension Lines ✓ Links Architectural Plan Layers Tendon Horizon Design Strips Shell Analysis Mesh Line Gauge Slab Internal Ribs Area Gauge Isolated Column Footings Slab Rebar Soil Profile for Joints Soil Profile for Areas Shell User Mesh Lavere to Show for Tendone /Strine /Slah Rehan

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:

## 🗧 Slab Property Data

Property Name	Slab2
Slab Material	4000Psi 🗸
Notional Size Data	Modify/Show Notional Size
Modeling Type	Membrane $\checkmark$
Modifiers (Currently Default)	Modify/Show
Display Color	Change
Property Notes	Modify/Show

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



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

Sho	w Frame Loads
	Load Pattern V
	Load Type
	O Span Loading Applied Directly to the Frame Object (Forces)
	Span Loading Applied Directly to the Frame Object (Moments)
	All Loading that is Tributary to the Frame Object (Forces)
	All Loading that is Tributary to the Frame Object (Moments)
	O Temperature Values
	Open Structure Wind Loads
	Show Loading Values
	OK Class Apply
	UN Close Apply
0 000	9.054
003 003 003	0
USA LA STATEMENT	2 <u>0.018</u> <u>0.019</u>
3	
003 003 090	0000
	2-Way 1-Way
4	

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.