Differentiating 2D Elements-Tips and Tricks

There are several different 2D elements available in S-Frame Analysis which, when used properly, can greatly improve the efficiency and accuracy of the modeling process. S-Frame provides support for diaphragms, panels, and both triangular and quadrilateral finite elements. The different characteristics and common uses of each 2D element are highlighted below. Note that XY refers to the Global Coordinate System, and xy refers to the Local Coordinate System.

Diaphragm 2D Elements

A diaphragm is defined by a set of joints that are constrained to translate together in the XY plane and rotate about the Z axis.   Diaphragms are most frequently used as a simplified way to model the rigid-body effects of a floor system. One advantage of diaphragms is their relative ease of application and loading.   Diaphragms can also simplify the model and allow for faster analysis times (when compared to using 2D Finite Elements). S-FRAME Analysis even allows users to define inclined diaphragms, in which case they are constrained about the local axis instead.   There are three diaphragm types available in S-Frame Analysis, allowing users to model in a number of different ways:

- **General Diaphragm** – defined by _all_ joints within an XY plane (at a single Z-Coordinate, unless the diaphragm is inclined) belong to a constrained set. General Diaphragms do not have a center of mass, material, or thickness. The simplest way to add a General Diaphragm is to define a plane with three points.
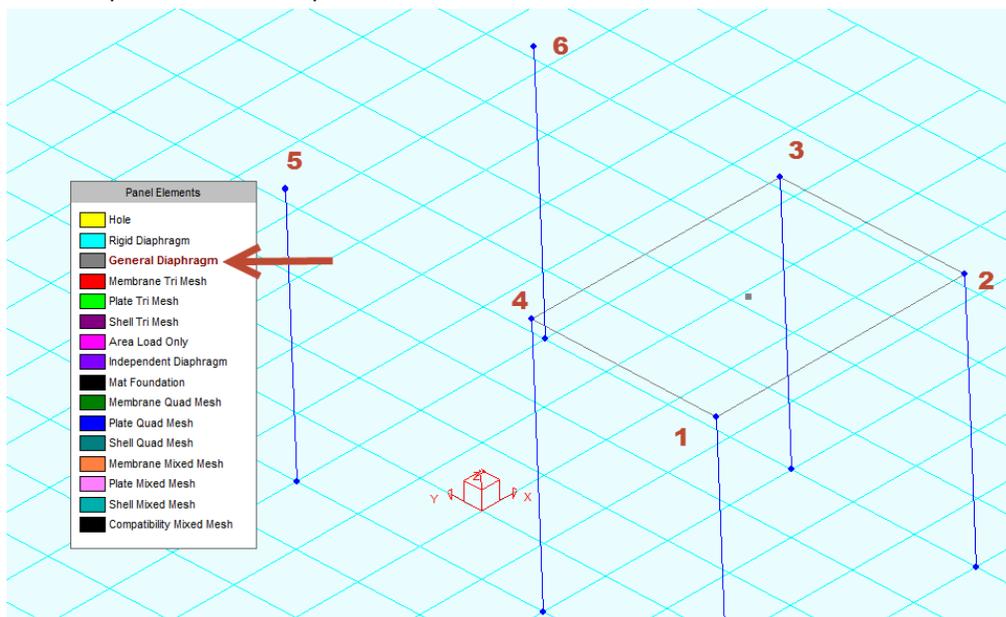
As shown above, a General Diaphragm was drawn between columns 1, 2, 3, and 4. The nature of a General Diaphragm means that all joints within the XY plane belong to that constrained set. Therefore, joints 5, and 6 are also part of the General Diaphragm. If the user does not want all

joints that are in the plane of the General Diaphragm to be part of the diaphragm constraint, there are ways to remove select joints.

- **Rigid Diaphragm** – constrains all joints within and along the borders of the object. If the above diaphragm type was changed to Rigid, only joints 1-4 would be included. A center of mass is supplied, which means that thickness and material density can be defined. These properties do not alter the stiffness, but do add mass to the diaphragm in the X, Y directions and rotational mass about the Z axis only. Rigid Diaphragms can be defined as triangular, quadrilateral, or polygon (panel) diaphragms using tools in the Geometry Window of S-Frame.

- **Independent Diaphragm** – If two or more Rigid Diaphragms are added in the same plane, a single set of joints with a common center of mass is created. If one of the diaphragms is made independent, S-Frame will maintain two independent sets of constrained joints with two independent centers of mass. If General and Independent Diaphragms are used on the same plane, all joints in the plane not belonging to the Independent Diaphragm will be slaved to the General Diaphragm. A potential use of Independent Panel Diaphragms would be the modeling of diaphragms in two side-by-side towers which may move independently of one another.

Panel 2D Elements

A panel element is polygon-shaped element that can be used to define area loads, diaphragm elements, or the extents of 3- and 4-joint finite elements. Finite element mesh panels can even be used to automatically generate meshes of triangular shell elements with springs assigned to the joints. This feature is useful to model mat foundations.

A Panel Object is defined by the following:

- An ordered list of joints (up to a maximum of 30)
- Material
- Thickness (applies to rigid diaphragms and meshable panels)
- Excluded Joint List (for diaphragms and meshes with irregular gaps inside the polygon)
- Prestress in x and y (applies to meshable panels)
- Mesh density (applies to meshable panels)
- Seed density (applies to meshable panels)
- Quad weight in percent (applies to meshable panels)

Example: Creating a Two-Way Area Load Panel

An Area Load Panel can be used to distribute area loads into member loads for the members in the same plane as the panel. The member load's magnitude is automatically calculated. The type of span (one-way, two-way) determines the magnitude of forces on each member.

1.) Select all of the members you would like to transfer the area load to, and then add them to the Area Load Members folder in the Groups window. You can, for example, choose to exclude bracing that will not be loaded by the area loads.
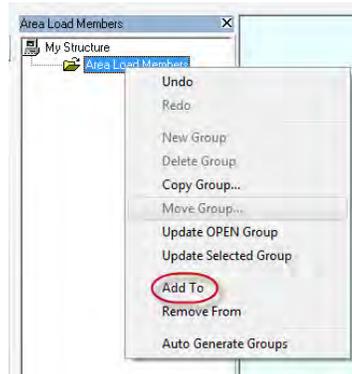

2.) Next, create the panel using the Panel Element Tool (ensure that the Area Load Only Panel type is selected). Select between 3 and 30 joints to define the panel extents.
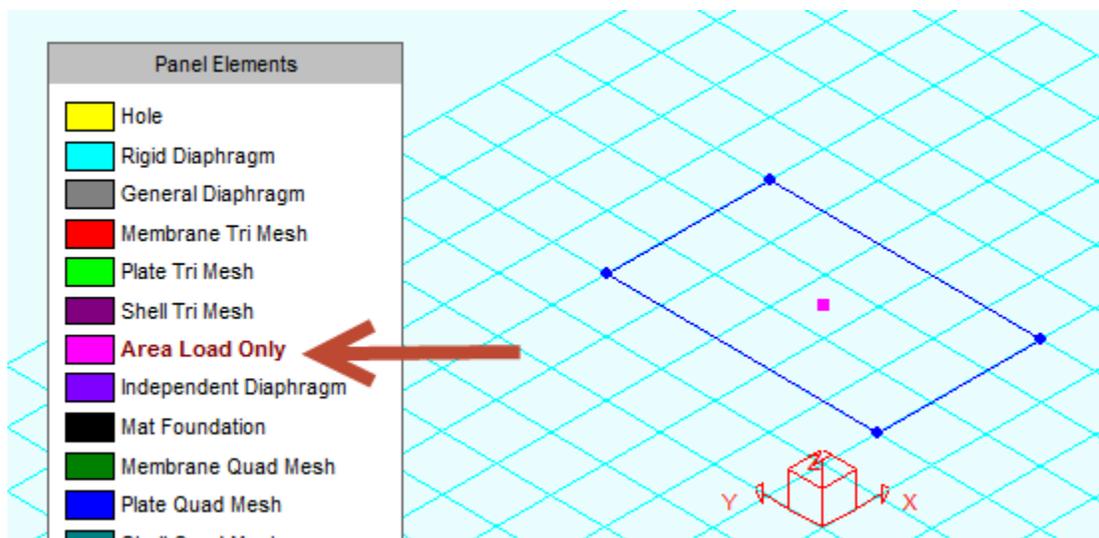
3.) Using the Span Direction Tool, select the Two-Way Span option in the data bar, then select the panel handle (the small square in the center of the panel). At this point you could

alternatively select a One-Way Span which would distribute the loads along either the short or long panel sides.
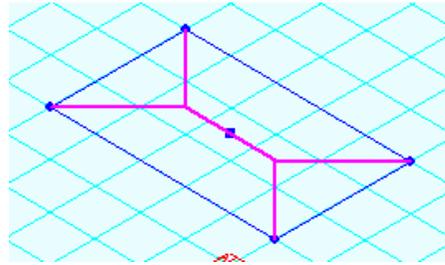
4.) To add an area load to this panel, open the Loads Window and select the Area Load Tool. Using the data bar immediately above the visual editor, enter the magnitude of your area load, as well as an orientation. Next, select the panel handle again to place the load.

Finite Element Types

Unlike diaphragms and panels, finite elements have a stiffness and mass dependent upon their material and thickness. These elements are used to more-realistically model physical components of structures.

Membrane Element

A membrane element has in-plane stiffness only, in both tension and compression. A membrane element can be thought of as the 2D equivalent of a truss element. Membrane structures are used to model very thin floor systems with minimal vertical loading, or fabric structures (remember that finite elements can be prestressed).

Plate Element (3D)

A plate element has out-of-plane stiffness only and is often used to model floor systems not designed to transfer in-plane loads.

Shell Element (3D)

Shell elements are the most frequently used finite elements and support both in-plane and out-of-plane loading. A shell element can be thought of as the 2D equivalent of a beam element. Shell elements are well suited to model walls, floors, foundations and roofs. Shells can even be used to model member geometry for an in-depth force and stress analysis of a member under specified loading conditions.

Further description of the 2D Elements can be found in S-FRAME's built-in Help System. A thorough explanation of the finite element method can be found in the following references:

Bathe, Klaus-Jurgen (2007). *Finite Element Procedures.*
Cook et. al. (2001). *Concepts and Applications of Element Analysis* (4th ed.)