ANALYSIS OF BUILDINGS WITH RIGID, SEMIRIGID AND PSEUDO-FLEXIBLE DIAPHRAGMS

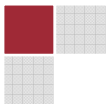
Bulent N. Alemdar, Ph.D., P.E.

Rakesh Pathak, Ph.D.

1. INTRODUCTION

A diaphragm is horizontal structural component and it functions as transferring story shears and torsional moments to lateral force-resisting members as well as distributing gravity loads to vertical members. Relative stiffness of the diaphragm with respect to stiffness of lateral members at the diaphragm level determines how it transfers shears and torsional moments. In addition, it distributes gravity loads based on either one-way or two-way load distribution. To this end, diaphragm action for lateral analysis can be defined as Rigid, Semirigid or Pseudo-flexible. For gravity analysis, type of deck placed inside a diaphragm determines how gravity loads on the deck are distributed.

RAM Frame – Analysis module accommodates several features providing great flexibility for analysis of buildings with diaphragms. The engineer can easily define diaphragm behavior by setting diaphragm type for lateral analysis. **Story Diaphragm** dialog (Figure 1) includes three sections: In-Plane stiffness, Out-of-Plane stiffness (bending) and Diaphragm Mesh controls. The In-Plane section shows a list of all diaphragms in the model. Note that more than one diaphragm per story is supported in the program. Regarding lateral analysis, the In-plane stiffness section provides 4 different diaphragm types: **Rigid**, **Semirigid**, **Pseudo-Flexible** and **Flexible\None**. Sections 2.1 and 2.2 provide more information about Rigid and Pseudo-Flexible diaphragms, respectively. If a diaphragm is set to Semirigid, then the program meshes the diaphragm by taking care of all meshing compatibility issues as well as loads on the diaphragm (including spatial mass distribution). See Section 2.3 for more information regarding Semirigid diaphragms. Section 4 provides in-depth information about all meshing procedures. Finally, Section 5.2 shows how P-Delta analysis is carried out for a semirigid diaphragm.



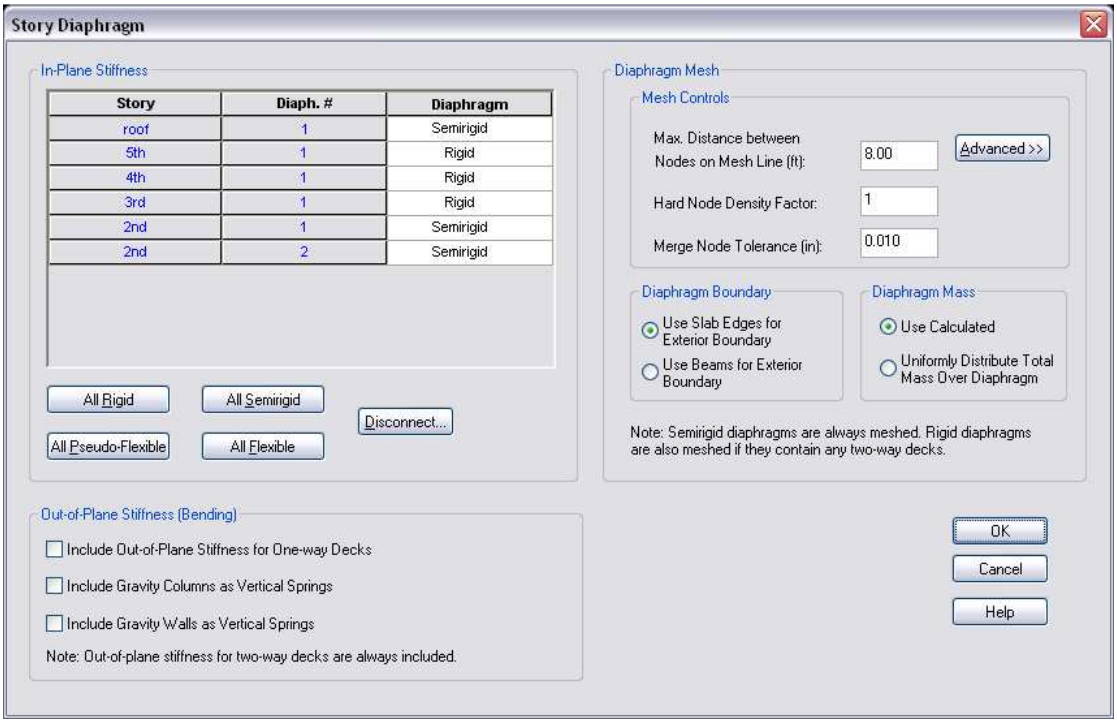
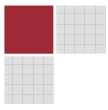


Figure 1 Story Diaphragm Dialog

A diaphragm is always meshed if it contains any two-way deck whether or not it is defined as semirigid for lateral analysis. The program always includes out-of-plane stiffness of two-way decks and this is needed for two-way load distribution for gravity loads (since two-way load distribution is carried out through the diaphragm's out-of-plane bending stiffness). Moreover, gravity loads (dead and live loads) as well as mass loads are automatically calculated and distributed spatially over the diaphragm. Section 4.3 explains how loads on meshed diaphragms are processed.

If a diaphragm includes one-way deck but it is defined as semirigid, or if a diaphragm contains both one-way and two-way decks, the diaphragm is always meshed. Then, the engineer is provided the option, "*Include Out-of-Plane Stiffness for One-Way Decks*" in the Out-of-Plane section of the Story Diaphragm dialog. Once it is invoked, the program includes out-of-plane stiffness for one-way deck as well. In addition, two more options are provided in the same section: Including gravity columns and walls as vertical springs. When they are invoked, the program inserts vertical springs at gravity wall and\or column locations and these springs has a stiffness equivalent to wall's or column's axial stiffness.

Section 3 provides more information regarding how gravity loads are handled for one-way and two-way decks. Finally, the Story Diaphragm dialog includes several mesh control parameters, which are also detailed in Section 4.



2. ANALYSIS OF BUILDINGS FOR LATERAL LOADS

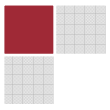
Diaphragm's in-plane stiffness plays a major role in transferring shears and torsional moments generated by applied lateral loads. A diaphragm is assumed rigid if it is relatively stiffer than lateral force-resisting members. Or, it can be assumed to be flexible if it is relatively less stiffer than the lateral members. A flexible diaphragm is defined as a diaphragm that has a limited capacity (semirigid diaphragm) or no capacity (pseudo-flexible diaphragm) to transmit torsional moments and it is relatively more flexible than the lateral members. One can further assume that story shears are transmitted to the lateral resisting members based on tributary areas of the frames. To this end, a flexible diaphragm can be viewed as a deep beam where lateral forces are applied through its web. Thus, the diaphragm experiences flexural deformations in its own plane. Collectors and chords are used to transmit shear and flexural forces to the lateral members.

In RAM Frame, a diaphragm can be modeled as rigid, semirigid or pseudo-flexible. This can be set for each diaphragm in ***Story Diaphragm*** dialog (Figure 1).

2.1. RIGID FLOOR DIAPHRAGMS

Rigid diaphragm behavior is well known and it is extensively used in analysis of buildings. Briefly, its salient characteristics are summarized as follows:

- It is assumed that the diaphragm is rigid enough so that no axial, bending and shear deformations are expected in the plane of the diaphragm. Thus, the diaphragm is displaced as one rigid component under applied lateral loads.
- Story shears are distributed to supporting lateral members based on their relative rigidities. Thus, stiffer members attract more story shears.
- Diaphragm deformation characteristics can be uniquely defined with three degrees of freedom: two lateral displacements in the plane of the diaphragm (i.e., one along global X-direction, and one along global Y-direction) and one rotation perpendicular to the diaphragm (i.e., rotation around global Z-axis).
- Diaphragm mass is lumped at diaphragm's mass center and it has three components: mass in X and Y direction and around Z-axis (i.e., rotational inertia of diaphragm mass).
- An analysis is carried out based on the assumption that applied loads are imposed through the center of mass of the diaphragm but they are resisted through diaphragm' center of rigidity. Thus, if diaphragm's center of rigidity and center of mass are not coincident, an inherit torsional moment is imposed on the diaphragm and it is further transmitted to the lateral members through the infinitely stiff in-plane action of the diaphragm.



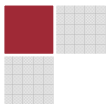
- All members connected to the diaphragm are displaced with the same amount of diaphragm displacements (compatibility requirement).

The in-plane stiffness of the floor systems of most building structures are very high compared to the stiffness of framing members. As a result, the in-plane deformations can be neglected, assuming that it is infinitely stiff for axial deformations. RAM Frame uses this assumption to reduce the size of the system equations of buildings with such floor types. If the "Rigid Diaphragm" option is selected, the program automatically carries out the necessary transformation for all the elements to arrive at a system equation that allocates only three in-plane degrees of freedom for each diaphragm. For dynamic analysis (Response Spectra Analysis), the predominant masses in such buildings are the ones that correspond to the three degrees of freedom per story (translation in X and Y direction and rotation around Z-axis). In the dynamic analysis of buildings with rigid diaphragms, RAM Frame evaluates corresponding stiffness and mass coefficients in these three degrees of freedom.

In the context of sloped framing, it is important to note that the diaphragm is always assumed to be rigid on a horizontal plane, even though the members in the floor may be sloped.

The program generates wind, seismic and dynamic loads for rigid diaphragms based on chosen building codes. It automatically calculates base shears, story loads and applies them on rigid diaphragms by taking care of any eccentric loadings. This process is fully automated. In addition, it is possible to define user-specified seismic and wind cases for rigid diaphragms. Regarding Seismic Story Forces for buildings with rigid diaphragms, force, direction angle, and the X and Y coordinates are needed to be defined for each diaphragm. By default, X and Y coordinates are at diaphragms' mass centers. If any eccentric loading is needed, then the user needs to adjust X and Y locations so that any difference between X, Y coordinates and mass center coordinates is considered as diaphragm eccentricity. Wind Story Forces for buildings with rigid diaphragms, the user needs to enter force, direction angle and wind exposures, in X and Y directions for each diaphragm.

For eccentric wind, seismic and dynamic loads, it is referred to RAM Frame – Analysis manual. P-Delta analysis for buildings with rigid diaphragms is explained in Section 5.1.



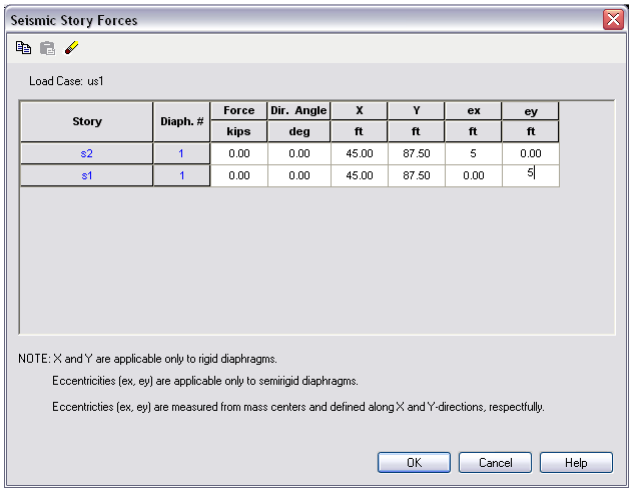


Figure 2 User Specified Seismic Story Forces

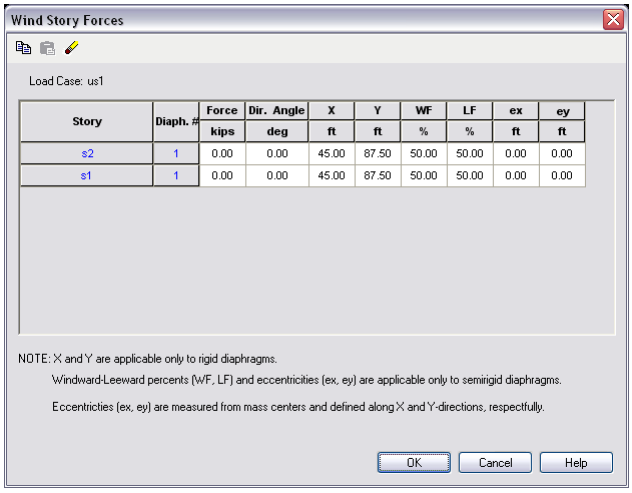


Figure 3 User Specified Wind Story Forces

2.2. PSEUDO-FLEXIBLE FLOOR DIAPHRAGMS

It can be assumed that a flexible diaphragm is not able to transfer torsional moments and that story shears are transferred to lateral members based on tributary areas of the frames. Under these observations, it is fair to conclude that each frame acts independently from the others. Therefore, behavior of a flexible diaphragm can be represented by the help of independently acting frames in the diaphragm followed by resolving story shears for each frame. Such a framing system is referred to as **Pseudo-flexible** diaphragm in RAM Frame. It should be noted that the diaphragm behavior is not directly captured by utilizing the

diaphragm stiffness, but by considering frame partitioning of the diaphragm (i.e., by use of independently acting frames with distributed story shears). To this end, frame *effective participation ratios* are required for each frame, which explicitly define what percentage of the story forces are resisted by each frame at the diaphragm. Note that the effective ratios can be approximated according to tributary area of each frame.

The procedure for pseudo-flexible diaphragm can be summarized as follows:

1. The user is expected to assign frame numbers to members
2. The user is expected to assign effective participation ratios to frames in the **Pseudo-Flexible Diaphragm Properties** dialog (Figure 4). These ratios are used to determine how much of the diaphragm load goes to which frame.
3. Based on chosen building code, the program calculates wind and seismic loads and resolves them at diaphragm levels. If the diaphragm is pseudo-flexible, the program further distributes these loads among the participating frames in the diaphragm based on their effective participation ratios.

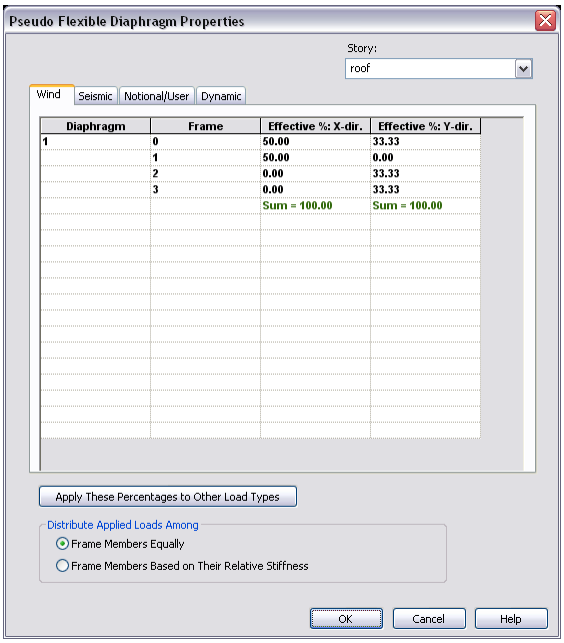


Figure 4 Pseudo-Flexible Diaphragm Properties

4. Once loads for each frame are known, they are distributed among members in the frames, based on either members' relative rigidities or on equal distribution among members.

5. Finally, frame joint forces are calculated and they are applied as *global* nodal loads at member joints.

In computation of interstory drifts and drift\height ratios for flexible diaphragms, the diaphragm member displacements are averaged and these averaged values are used as the pseudo-flexible diaphragm displacements. Thus, all drift related quantities are based on these averaged values.

2.3. SEMIRIGID DIAPHRAGMS

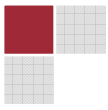
Distribution of horizontally applied loads through diaphragms to lateral load-resisting elements depends on relative rigidity of diaphragms and the resisting lateral elements. For the cases where diaphragm deflections and resisting members' deflections are in the same order of magnitude, then such diaphragms cannot be categorized as **Rigid** or **Flexible\None**. Instead, it is referred to as a **Semirigid** diaphragm, which basically represents a diaphragm condition between **Rigid** and **Flexible\None.** Analysis with semirigid diaphragms considers diaphragms stiffness and thus, it reflects real diaphragm deflections and provides a more involved load distribution among the resisting members. Note that diaphragm properties (such as thickness, E, etc…) and its dimensions do not alone reflect whether diaphragm is categorized as rigid, none-rigid or semirigid. Rather, its interaction with the resisting members and relative stiffness of diaphragms as well as of the resisting members plays a major role in this decision. Many building codes provide some guidelines for determination of diaphragm types.

If a diaphragm is defined as **Semirigid** in RAM Frame, then the program automatically meshes the diaphragm. Most of the meshing issues are taken care of by the program so that the user does not need to go over this tedious process manually. If applicable, crack factor is applied to in-plane thickness of the diaphragm. Hence, in-plane (membrane) stiffness of the semirigid diaphragm is reduced by crack applying crack factor.

2.3.1. ECCENTRIC LOADING

Several building codes require eccentric loading on diaphragms to account for accidental torsion effects. Referring to models with rigid diaphragms, such eccentric loading is accomplished by simply shifting loads to a new position that accounts for eccentric distances. For instance, concentrated story loads for seismic load cases are first calculated at diaphragm's mass center and then they are applied at their shifted locations. For wind load cases, a similar procedure is followed: story wind loads are calculated based on building profiles (one for X and one for Y-direction) and then they are lumped and applied at their shifted locations.

On the other hand, for models with semirigid diaphragms, the above procedure is not applicable since such diaphragms are meshed and loads on these diaphragms are not concentrated at a certain location but they are distributed over the diaphragm for



seismic\dynamic load cases and they are applied along building edges for wind load cases. To address these issues for semirigid diaphragms, different methods are implemented and they are explained in the following sections.

2.3.1.1. Eccentric Seismic and Response Spectra Load Cases on Semirigid Diaphragms

After a semirigid diaphragm is meshed, the program processes all mass loads over the diaphragm. At the end of this process, all the mass are converted into point (nodal) mass loads and they are directly applied at mesh nodes. In order to explain the current method used for seismic and response spectra load cases, an example is provided in this section (in the following example, a diaphragm loaded in (+) Y direction with (+) eccentricity). The following steps are pursued for this purpose:

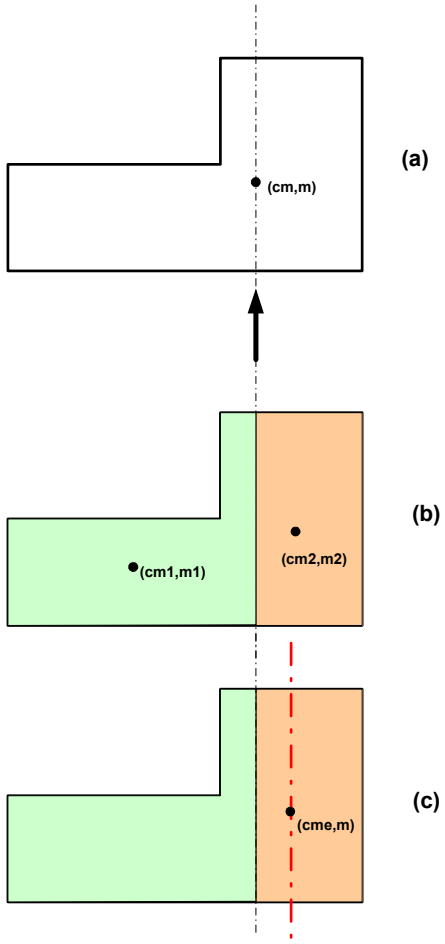


Figure 5 Procedure for Eccentric Seismic Loads for Semirigid Diaphragms

The program calculates mass center for the diaphragm. This is shown in Figure 5a. Mass center line is determined (i.e., a line passing through the mass center and parallel to global Y-axis).

For the load case in Y-direction, the diaphragm is divided into two zones: one zone at the left side and one zone of right side of the mass center line (Figure 5b). For each zone, mass centers are calculated according to the following equations:

$$cm_1 = \frac{\sum_{i=1}^{n} m_i\, c_i}{\sum_{i=1}^{n} m_i} = \frac{\sum_{i=1}^{n} m_i\, c_i}{m_1} \quad (1)$$

$$cm_2 = \frac{\sum_{j=1}^{m} m_j\, c_j}{\sum_{j=1}^{n} m_j} = \frac{\sum_{j=1}^{m} m_j\, c_j}{m_2} \quad (2)$$

Where cm_1 and cm_2 are the mass center coordinates for the zone 1 and 2, respectively (see Figure 5b). Note that the index "i" is reserved for nodes located inside Zone 1, and the index "j" is reserved for nodes in Zone 2.

If it is a rigid diaphragm, calculated seismic story loads are concentrated at a single point "cm_e", (i.e., $cm_e = cm + eccentricity$ for (+) eccentricity in Y-direction). In other words, the mass center is shifted to its new location, "cm_e" to account for eccentric loading. Once the concentrated load is applied at this location, additional moment (i.e., accidental torsion) is created at "cm".

For a semirigid diaphragm, two new terms are introduced, "$\bar{m}_1$" and "$\bar{m}_2$", and they are obtained from solving the following two equations:

$$cm_e = \frac{\bar{m}_1\, cm_1 + \bar{m}_2\, cm_2}{\bar{m}_1 + \bar{m}_2} \quad (3)$$

$$\bar{m}_1 + \bar{m}_2 = \sum_{p=1}^{k} m_p = m = \text{Total mass of the diaphragm} \quad (4)$$

Then,

$$\bar{m}_1 = -\left(\sum_{p=1}^{k} m_p\right)\left(\frac{cm_2 - cm_e}{cm_1 - cm_2}\right) \quad (5)$$

$$\bar{m}_2 = +\left(\sum_{p=1}^{k} m_p\right)\left(\frac{cm_1 - cm_e}{cm_1 - cm_2}\right) \quad (6)$$

The terms "$\bar{m}_1$" and "$\bar{m}_2$" can be interpreted as *the amount of lumped masses at each zone so that the final system has a combined mass center at "cm_e".* Then, the program calculates effective mass coefficients for each zone as follows:

$$\alpha_1 = \frac{\bar{m}_1}{\sum_{i=1}^{n} m_i} = \frac{\bar{m}_1}{m_1} \quad (7)$$

$$\alpha_2 = \frac{\bar{m}_2}{\sum_{j=1}^{n} m_j} = \frac{\bar{m}_2}{m_2} \quad (8)$$

Finally, each point mass (or calculated seismic point load at each node) in zones 1 and 2 are multiplied with α_1 and α_2, respectively. Once, each point mass (or point seismic load) is modified as explained above, the analysis is carried out with this modified configuration.

Note that it is for seismic load cases when each seismic point loads are modified with α_1 and α_2 . Similarly, it is response spectra load cases when each point mass is modified with α_1 and α_2.

The following is summarized based on the above procedure:

$$\alpha_1 = \frac{\bar{m}_1}{m_1} \ , \quad \alpha_2 = \frac{\bar{m}_2}{m_2} \tag{9}$$

And also note that diaphragm mass is always *conserved*:

$$m_1 + m_2 = \bar{m}_1 + \bar{m}_2 = m \tag{10}$$

$$\alpha_1 m_1 + \alpha_2 m_2 = m \tag{11}$$

Figure 6a shows an example of eccentrically loaded diaphragm (i.e., Y+E). Note that the arrows of the right side of the diagram are relatively longer than those on left side, which indicates that these seismic point loads are amplified with α_2 whereas those on left side are reduced with α_1. In addition, the deflected shape given in Figure 6b indicates presence of eccentric loading.

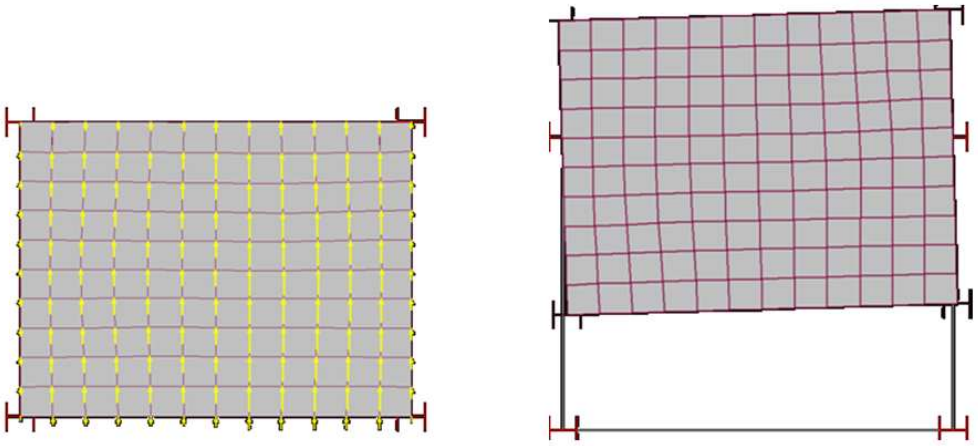
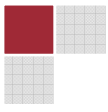


Figure 6 (a) Eccentric seismic loads, (b) deflected shape under eccentric seismic loads

If it is a seismic load case, the seismic point loads are modified as explained above. If it is a dynamic load case (Response Spectra Analysis), nodal masses are modified according to the same method explained.

2.3.1.2. ECCENTRIC WIND LOAD CASES ON SEMIRIGID DIAPHRAGMS

Wind pressures on semirigid diaphragms are calculated for each Windward and Leeward surfaces of the building. Then, these pressures are converted to uniform line loads and applied along semirigid diaphragm edges as shown in Figure 7.



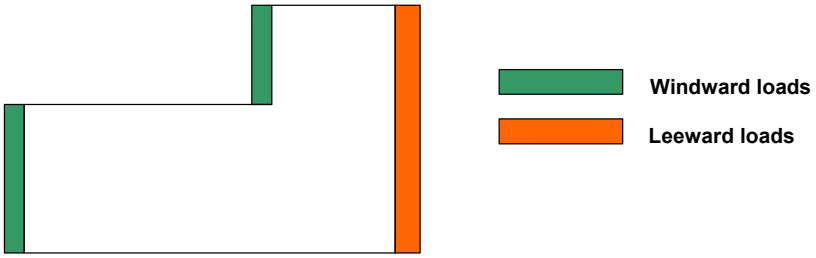


Figure 7 Windward and Leeward Wind Loads on Semirigid Diaphragm

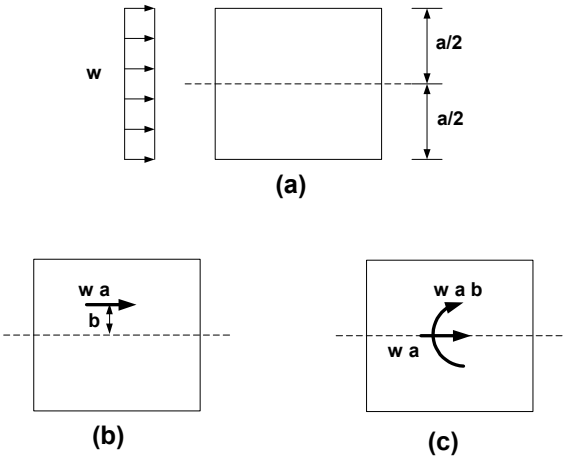


Figure 8 Procedure for Eccentric Wind Loads for Semirigid Diaphragms

Figure 8a shows a uniform load applied at Windward side of the building. If diaphragm type is rigid, then the program converts the uniform load into a concentrated load and applies it at mid-width location of story (Figure 8b). If there is any eccentricity defined, the concentrated load is shifted to its location (indicated in the figure with "b"), or associated moment (w * a * b) is applied at the mid-width location (Figure 8c).

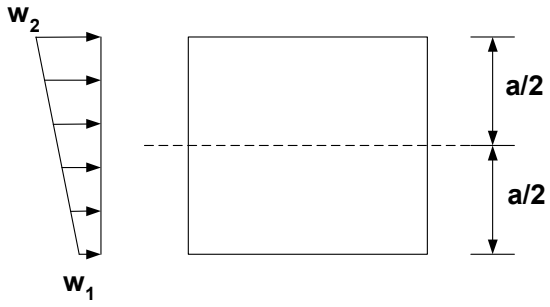


Figure 9 Trapezoidal Load Pattern for Eccentric Wind Loads

In order to account for eccentric loading on a semirigid diaphragm, uniform wind line load is represented with a trapezoidal load pattern (Figure 9). The following equations are solved for new load pattern:

Force Equilibrium:

$$(w_1 + w_2)\,\frac{a}{2} = w\,a \tag{12}$$

Moment Equilibrium:

$$w_1\,\frac{a^2}{8} - w_1\,\frac{a^2}{8} + \left(\frac{w_1+w_2}{2} - w_1\right)\frac{a^2}{24} + \left(\frac{w_1+w_2}{2} - w_1\right)\frac{a^2}{8} + \left(w_2 - \frac{w_1+w_2}{2}\right)\frac{a^2}{12} = w\,a\,b \tag{13}$$

And solving the above two equations, one can find

$$w_1 = w - \frac{6\,b\,w}{a} = w\,\left(1 - \frac{6\,b}{a}\right) \tag{14}$$

$$w_2 = w + \frac{6\,b\,w}{a} = w\,\left(1 + \frac{6\,b}{a}\right) \tag{15}$$

It should be noted that the trapezoidal load profile generates the same amount of total load as of the uniform load, plus it also generates the same moment (w*a*b). This is graphically portrayed in Figure 10.

Figure 11 shows an example of eccentric wind loads on a semirigid diaphragm. Note that if there is no eccentricity defined (Figure 11a), the loading profile is linear. Otherwise, it is converted to a trapezoidal profile (Figure 11b) to reflect the effect of eccentric loads.

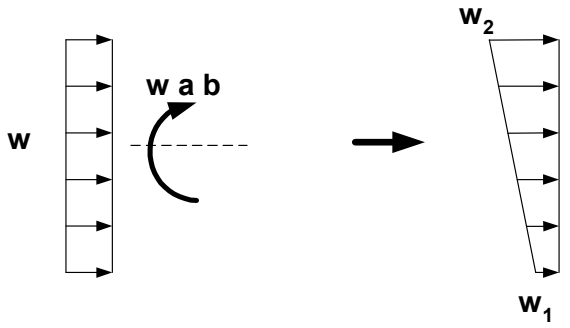
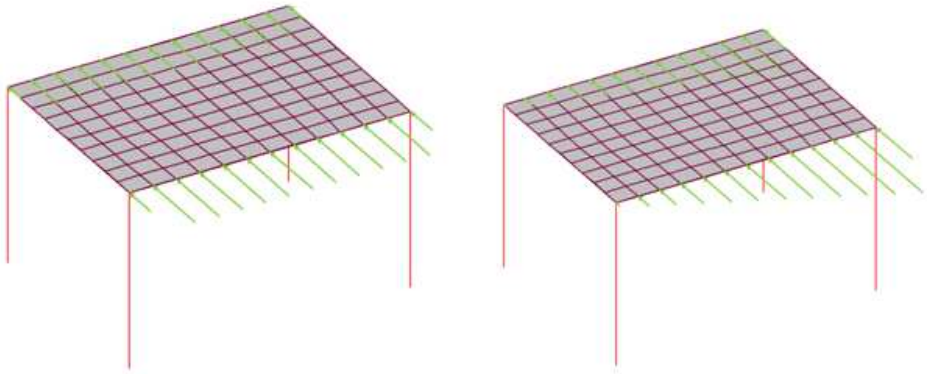


Figure 10 Equivalent Loading Systems



(a) (b)

Figure 11 Eccentrically Loaded Diaphragm for with Load Cases

2.3.2. Seismic Load Case

If a diaphragm defined as semirigid, mass associated with the diaphragm (including mass from surface, line and point loads, which are defined in the Modeler, and member self mass including decks) is distributed to diaphragm nodes. "Center of Mass" command on the toolbar menu shows lumped mass for rigid diaphragms as well as nodal mass assigned to nodes on semirigid diaphragms. Also, "**Criteria, Mass & Exposure**" report shows nodal mass quantities for semirigid diaphragms.

Based on chosen building code, the program calculates building base shear and then it distributes it over the height of the building as a lateral load to be applied to each diaphragm. Finally, the loads are distributed to nodes on the semirigid diaphragm, which are in proportion with nodal masses at each node. If t here exist any eccentricity, these loads are modified as explained in Section 2.3.1.1. "**Loads and Applied Forces**" report tabulates loads applied to each node for semirigid diaphragms.

2.3.3. Response Spectra Load Case

Analysis is conducted based on distributed nodal masses over semirigid diaphragms. Thus, analysis results reflect the effect of spatial distribution of mass related to semirigid diaphragms. If t here exist any eccentricity, nodal masses are modified in as explained in Section 2.3.1.1.

2.3.4. *Wind Load Case*

The wind loads on semirigid diaphragms are automatically computed for both the windward and leeward side. The wind loads are applied on the diaphragm edges as line loads. The edges on a diaphragm are internally marked as the windward edge or the leeward edge depending upon the direction of the wind i.e. +X, -Y etc. The computations of windward and

leeward forces are based on the respective pressures and the normal areas on which the loads are applied. When computing the line load on a diaphragm edge it is assumed that the edge is covered both above and below the diaphragm. The program applies forces to the diaphragm edges even if they are partially or fully covered by some other edges e.g. see orange edge in Figure 12**.** If t here exist any eccentricity, wind line loads are modified as explained in Section 2.3.1.2.

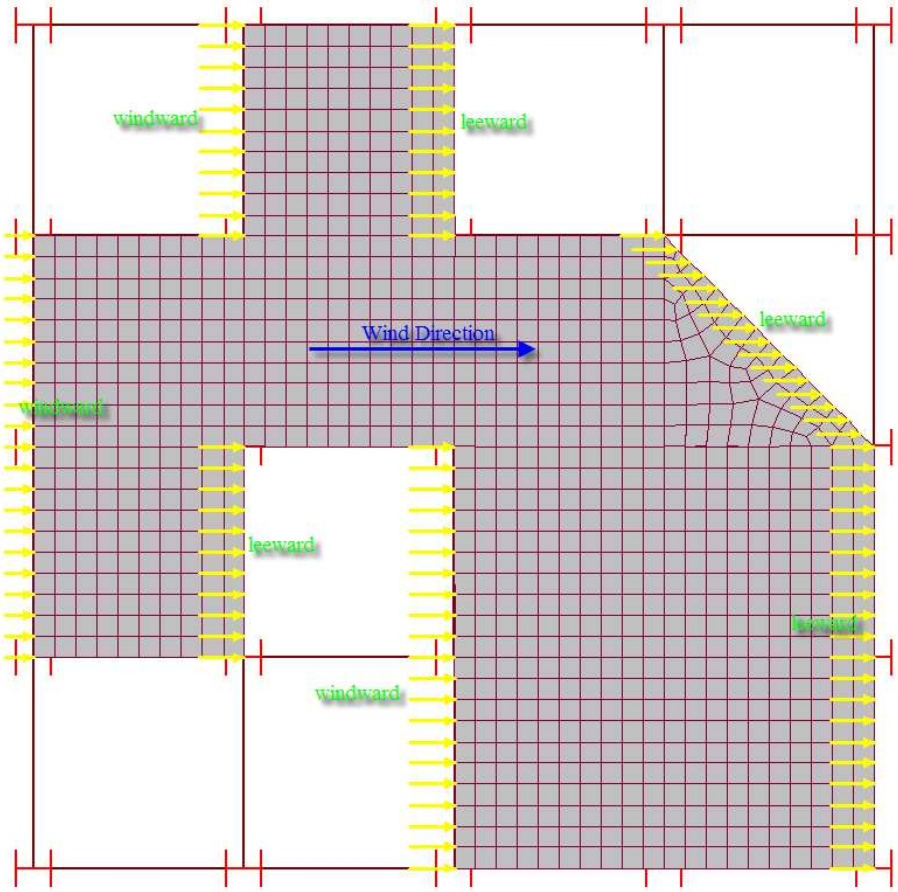
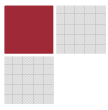


Figure 12 A semirigid diaphragm showing windward and leeward forces

2.3.5. User Defined Seismic Load Cases

Currently the program allows the engineer to specify a user-specified seismic load case as shown in Figure 13. Regarding semirigid diaphragms, force, a direction angle, and eccentricities are needed to be entered (coordinates X and Y are not considered for semirigid diaphragms. they are only applicable for rigid diaphragms.



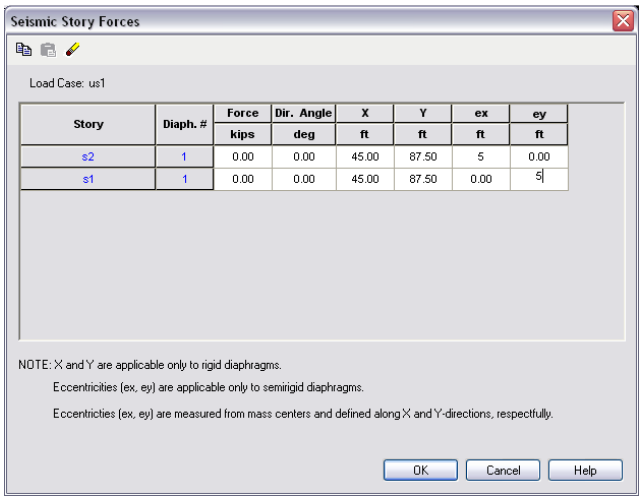


Figure 13 User defined Seismic Story Forces Dialog

Note that eccentricity definitions (**ex** , **ey**) are measured from diaphragm mass center and they are defined along X and Y directions, respectively:

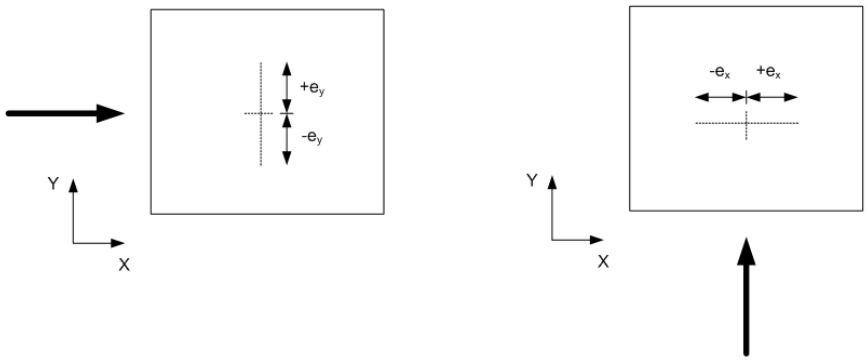
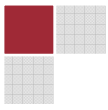


Figure 14 User Specified Seismic Loads for Semirigid Diaphragms

2.3.6. User Defined Wind Load Case

The program allows the engineer to specify a user-specified wind load case as shown in Figure 15. Regarding semirigid diaphragms, force, a direction angle, windward and leeward face percentages and eccentricities are needed to be entered (coordinates X and Y are not considered for semirigid diaphragms.



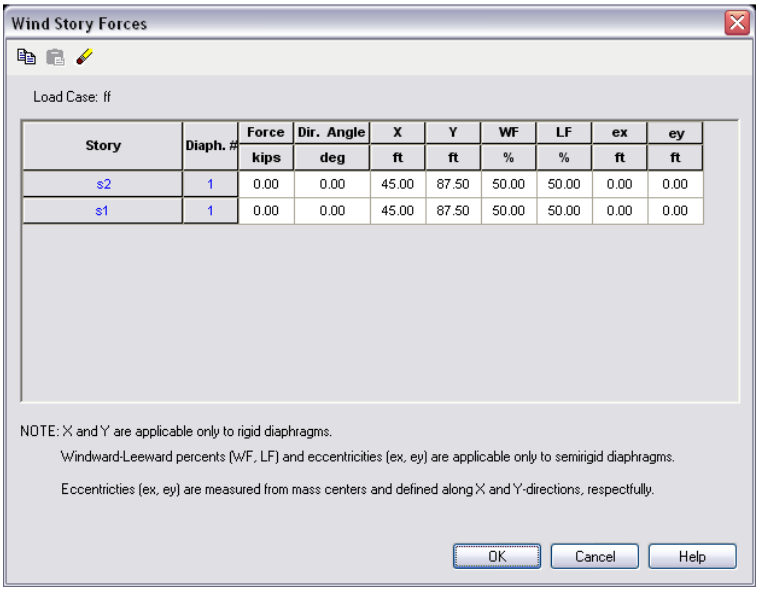


Figure 15 User defined Wind Story Forces Dialog

Note that (**ex** , **ey**) are eccentricities defined along X and Y directions, respectfully and they are measured from center of exposure.

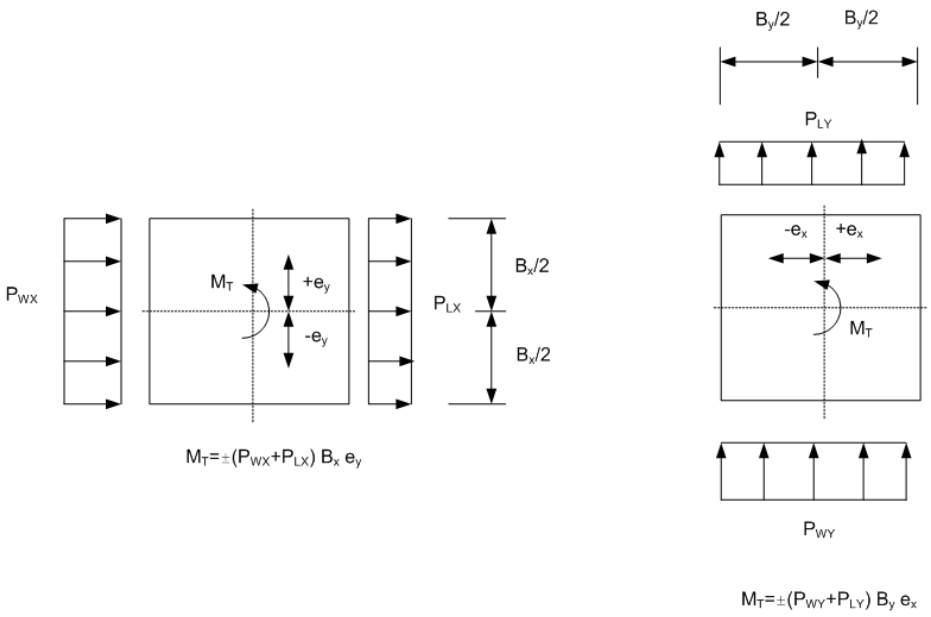


Figure 16 User Specified Wind Loads for Semirigid Diaphragms

2.3.7. Sloped Diaphragm Modeling

Sloping semirigid diaphragms are allowed in the program. Note that this requires that decks must be redefined in the modeler (each sloped plane must be modeled as a separate deck

polygon). In the following example, roof floor is defined as sloped as indicated in Figure 17 (note that columns are lowered at top and bottom perimeter lines and other columns at center are not). In order to correctly define sloped roof, two separate deck polygons are modeled (Figure 18) and each deck polygon represent a unique sloped plane. The program automatically calculates the plane of sloped diaphragm based on members located inside each deck polygon. Figure 19 shows sloped roof model. In addition, any loads (dead, live, snow or roof loads) on sloped diaphragms are also corrected: live loads computed on projected areas and dead loads are computed according to true sloped areas.

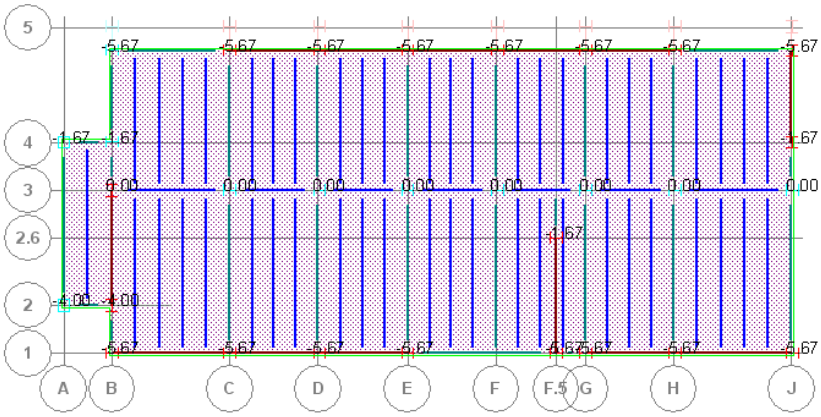


Figure 17 Plane view of sloped roof floor

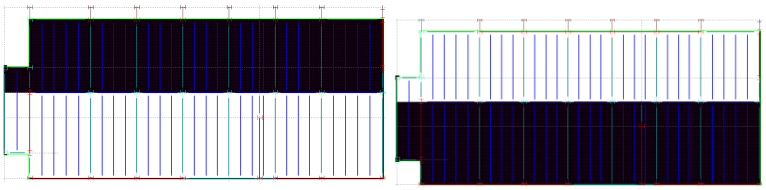


Figure 18 Each deck polygons uniquely define a sloped plane

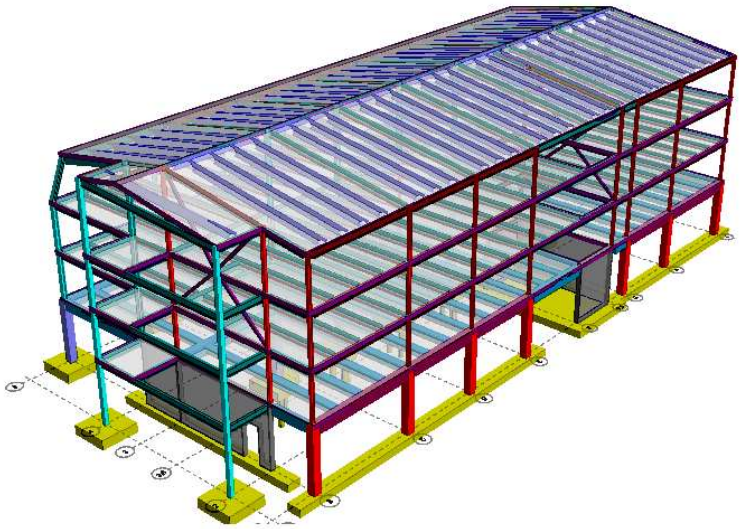


Figure 19 Model with sloped semirigid diaphragm

3. ANALYSIS OF BUILDINGS WITH GRAVITY LOADS

Gravity loads are distributed to vertical members through diaphragms by means of two methods: one-way load distribution and two-way load distribution. Each diaphragm can be composed of one or more decks and each deck type (one-way or two-way deck) is assigned in RAM Modeler. One-way load distribution is well known tributary area load distribution so that gravity loads are projected to beams, walls and columns based on one-way deck orientation and load tributary areas. Two-way load distribution is carried out through diaphragm's out-of-plane (bending) stiffness and hence, the diaphragm is meshed for this purpose.

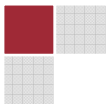
3.1. GRAVITY LOADS ON ONE-WAY DECK

One-way load distribution is well known tributary area load distribution so that gravity loads are projected to beams, walls and columns based on one-way deck orientation and load tributary areas. Further information is provided in RAM Frame manual.

3.2. GRAVITY LOADS ON TWO-WAY DECK

Two-way load distribution is carried out through diaphragm's out-of-plane (bending) stiffness. Diaphragm is meshed and loads are resolved and applied at meshed nodes. Note that the program always meshes a diaphragm if it contains a two-way deck.

An example is given in Figure 20: one way deck and two way deck are defined within the same diaphragm. As shown in Figure 21, only two-way deck is directly loaded with gravity loads and the loads on one-way deck are handled with tributary area approach.



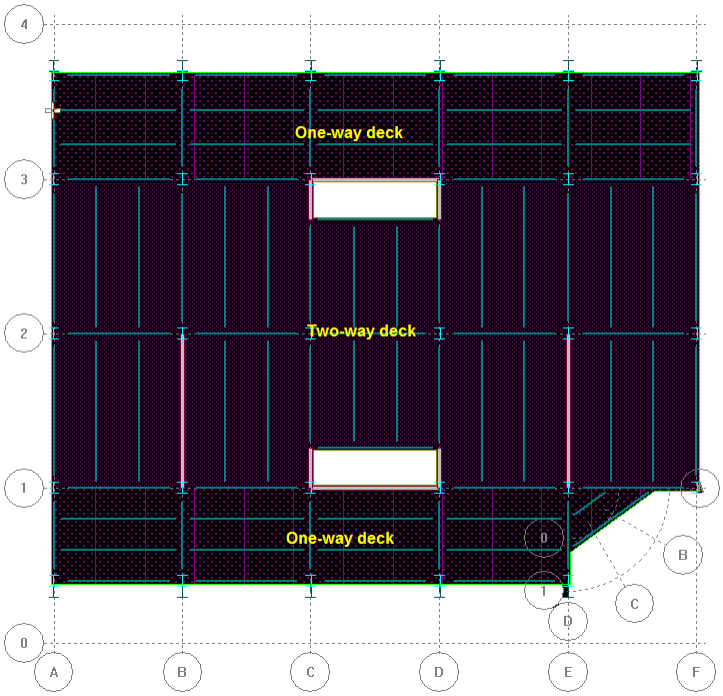


Figure 20 . One-way and two-way deck example

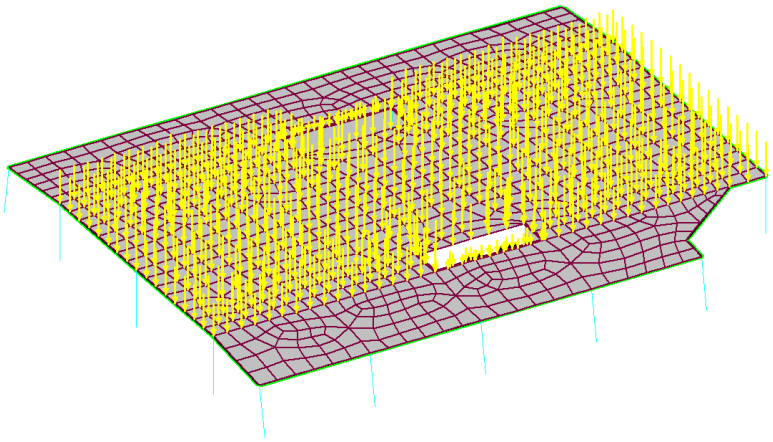


Figure 21 . Two-way deck area subjected to gravity loads

It is important to know that **out of plane stiffness for two-way deck is always included in analysis**. On the other hand, it is not included for one-way deck by default. In order to include out-of-plane stiffness for one way deck, the user option "*Include out of plane stiffness for one way decks*" in **Story Diaphragm** dialog must be check out.

The followings are important to note for gravity loads on two-way decks:

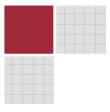
- Gravity loads (dead, live, roof and snow loads) are directly applied to mesh nodes of two-way decks
- User specified live load reductions for members are used for these members located inside two-way decks.
- Decks can be defined as a two-way system in the Modeler. Note that a bending thickness is needed for considering out-of-plane action (bending or plate action) in two-way load distribution.
- A diaphragm is directly meshed if it includes a two-way deck.
- Out of plane stiffness for shells in a two-way deck is always included in analysis.
- Applied story forces command can be used to visually see gravity loads on two-way decks (Figure 21).
- Gravity loads report shows calculated gravity loads on two-way decks.
- Gravity columns and walls located inside a meshed diaphragm can be also represented with vertical springs
- Referring to transfer columns,
    - If a transfer gravity column is sitting on a one-way deck, the load in the column is projected to nearest lateral beams or walls based on the deck orientation.
    - If a transfer gravity column is sitting a on two-way deck, the load in the column, is applied to the deck directly.

3.3. Diaphragm Out-of-Plane Stiffness

The Story Diaphragm dialog (Figure 22) includes several options related to out-of-plane stiffness (bending) for a meshed diaphragm. If a diaphragm includes a two-way deck, its out-of-plane stiffness is automatically included in the program. On the other hand, the same is optional for one-way decks. If the use wants to include the bending stiffness for one-way decks, Included **Out-of-Plane Stiffness for One-Way Decks** should be checked. In addition, the followings are also provided:

- **Include Gravity Columns as Vertical Springs**: if it is invoked, the program inserts vertical springs at the locations of all gravity columns inside the meshed diaphragm. A stiffness value of AE/L is assigned to these springs.
- **Include Gravity Walls as Vertical Springs**: if it is invoked, the program inserts vertical springs at the locations of all gravity walls inside a meshed diaphragm.

Model Data report tabulates calculated spring stiffness values for these gravity columns and walls. In addition, **Spring Forces** report (available after the analysis run) shows how much loads are carried by these springs.



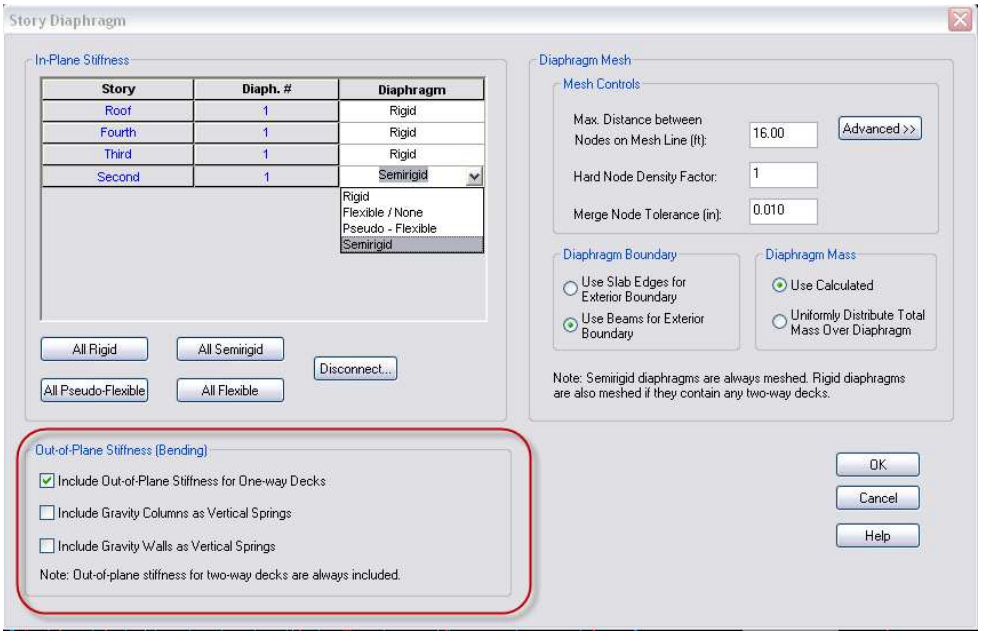


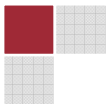
Figure 22 Out-of-Plane Stiffness Controls

4. DIAPHRAGM MESHING AND LOADS ON MESHED DIAPHRAGMS

A diaphragm is meshed if and only if the diaphragm is designated as semirigid or there exists a two-way deck inside the diaphragm. This is still correct for a rigid diaphragm with a two-way deck. The meshing process is fully automated. It detects all wall intersections with the diaphragm (hence, the diaphragm and walls are attached at the same mesh points to provide proper load transverse from the diaphragm to walls) as well as deck polygons, openings, slab edge polygons. The **Story Diaphragm** dialog (**Error! Reference source not found.**) provides several options used for controlling meshing details.

4.1. DIAPHRAGM BOUNDARY

It determines the extension of meshed area of the diaphragm. If "Use Slab Edges for Exterior Boundary" is chosen, the program uses slab edges as an exterior boundary of the meshed diaphragm (Figure 23). It is important to note that the area between the most exterior beams and slab edges are also meshed. If "Use Beams for Exterior Boundary" is chosen, the program uses the most exterior beams as exterior boundary of the meshed diaphragm (Figure 24). Note that there should be a closed beam loop constructed with exterior beams. Otherwise, the program fails to mesh in the absence of a closed boundary if this option is chosen, and then it automatically switches to the other option.



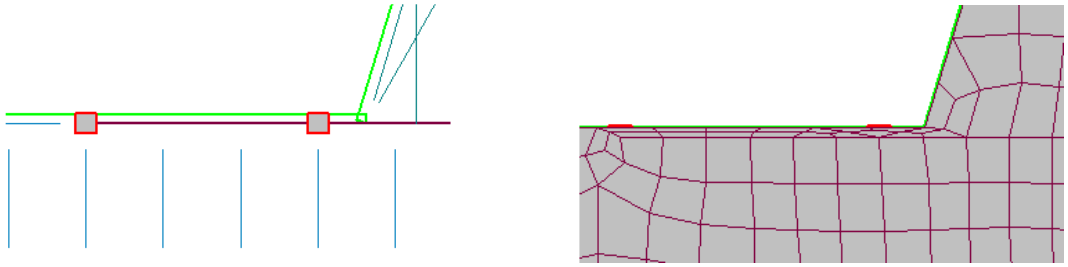


Figure 23 . Generating diaphragm mesh (*Use Slab Edges for Exterior Boundary*)

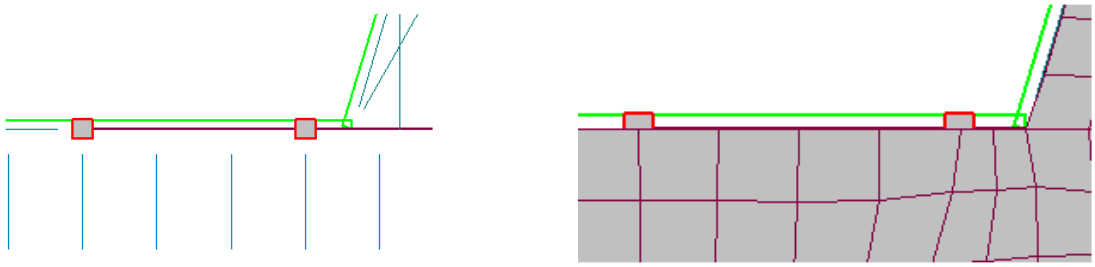


Figure 24 Generating diaphragm mesh (*Use Beams for Exterior Boundary*)

For some configurations, slab edge offset distance may be too small (only a few inches) and there are many beams running parallel to slab edges. If the first option is chosen, then it is very likely that very small distorted elements are created between exterior beams and slab edges. Thus, the user is advised to use the second option, which leads to a more balanced meshed configuration.

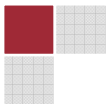
For slabs with no beams (like concrete flat slabs) or no closed beam loop exists, then user needs to use the first option.

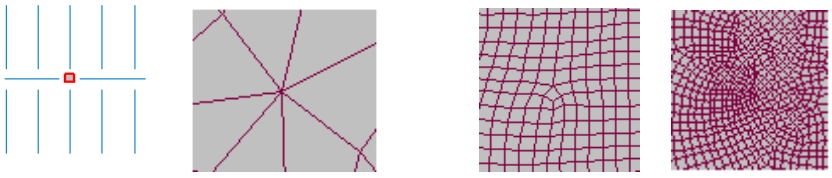
4.2. Hard Node Density Factor

This factor determines how dense a mesh is generated around a hard node located inside the diaphragm. A hard node is simply a column or wall end points. For most cases, a value of 1.0 generates a good mesh around a hard node. An example is defined in Figure 25.

4.3. Loads on Diaphragms (Physical and Mass Loads)

Once a diaphragm is meshed and represented with a collection of shells, all loads on the diaphragm are converted to point loads which are directly applied to the shells. The loads on diaphragms are simply surface, line or point loads as defined in the Modeler. Any overlapped surface loads are automatically resolved. Both mass loads and physical loads are processed the same way. The program can also include self weights and mass from elements including decks.





(a) (b) (c) (d)

Figure 25 Hard node density factor, (a) a column and mesh generated around the column based on the factor (b) 0.1, (c) 1.0, and (d) 5.0

In the following figures, mass loads are shown as an example but other types of loads are also processed the same way.

LINE LOADS

They are directly applied to the meshed surface. The program converts line loads into equivalent point loads which are applied to shells' nodes (note that if a line load is crossing any shell element, then the portion of the line load located inside the shell is extrapolated to the shell's nodes as equivalent nodal loads). An example is given in Figure 26.

POINT LOADS

They are directly applied to the meshed surface. If point load is located directly on a shell node, it is applied to the node directly. If the point load is located inside a shell element, it is extrapolated to the shell's nodes as equivalent nodal loads. An example is given in Figure 27.

SURFACE LOADS

They are also applied to mesh nodes. The program first detects overlapping surface load areas (note that the surface load placed last or at very top is considered active load). Each surface load on each shell is calculated and then, it is converted to equivalent point loads to be applied at the shell's nodes. An example is given in Figure 28.

MEMBERS' SELF WEIGHT AND MASS

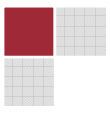
They are directly applied to the members as point loads. This includes all gravity and lateral beams, columns and walls. An example is given in Figure 29.

SELF WEIGHT AND MASS FOR DECKS

Similarly, if deck mass is included in analysis, it is again treated as a surface load and applied to the meshed surface.

DIAPHRAGM MASSES

Two additional options are provided to the user:

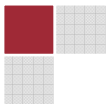


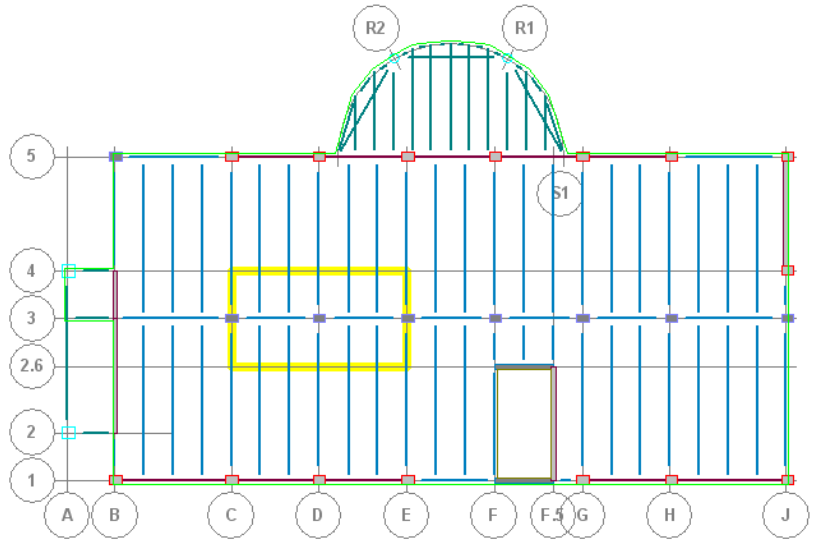
- Use Calculated: the program processes all masses related loads according to the procedure explained above. In contrast to rigid diaphragm approach where diaphragm mass is lumped at diaphragm mass center, this procedure considers a spatial mass distribution over semirigid diaphragms.
- Uniformly Distribute Total Mass Over Diaphragm: The total mass calculated for the diaphragm is distributed to shell elements based on each shell's effective area. This method can be used if a truly uniform mass distribution over the diaphragm exists. Also, it is provided as a alternative method if "Use Calculated" method fails to resolve mass polygons over the diaphragm due to some numerical difficulties.

MESH CONTROL PARAMETERS

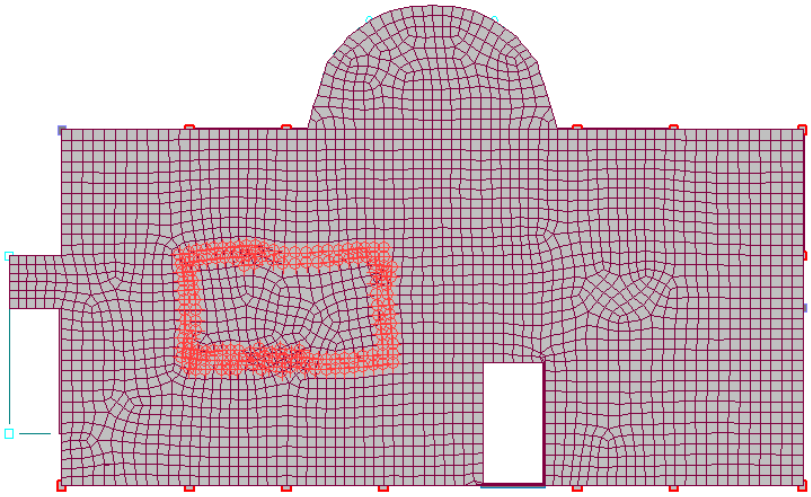
These control parameters do not only affect the mesh generated for semirigid diaphragms but also the mesh generated for walls. See the RAM Frame Manual for a complete list of these parameters. Only the most important parameters are explained below.

- **Max. Distance between Nodes on Mesh Line:** it allows the user define the maximum distance between nodes on wall edges and nodes on semirigid diaphragm edges. The program may generate additional nodes closer than user entered value, but it is never allowed to be larger than that value. The user can generate coarse or fine mesh by setting **Max. Distance between Nodes on Mesh Line**. Usually, a coarse mesh gives conservative results since the walls and diaphragms are stiffer. A finer mesh leads to more flexible walls and diaphragms, which leads better results but it may significantly increase analysis time. Figure 30 demonstrates an example, which shows how this parameter affects generated mesh density: Figure 30a shows a mesh generated with 16 feet whereas it is 2 feet in Figure 30b.
- **Merge Node Tolerance:** Once meshing process for walls and semirigid diaphragms is completed, the program generates a finite element model for analysis, which is basically composed of nodes and elements (columns, beams, braces and shells in walls and meshed diaphragms). During this process, it is possible that several nodes are very close. To have a good (and valid) analytical model, these nodes are merged to a single node and elements connected to this node are adjusted accordingly. The **Merge Node Tolerance** is used in such a way that if distance between any 2 (or more) nodes are found to be smaller than the **Merge Node Tolerance**, then they are treated as a single node (i.e., these close nodes are merged together).
- **Advanced Mesh Parameters**:
    - *Geometry Tolerance* is mostly used in geometry calculation as a threshold tolerance value. This should not be confused with a merge (close) node tolerance. Examples are as follows: it is used as a tolerance to check a node is on a line or to check a point is inside a polygon, etc...



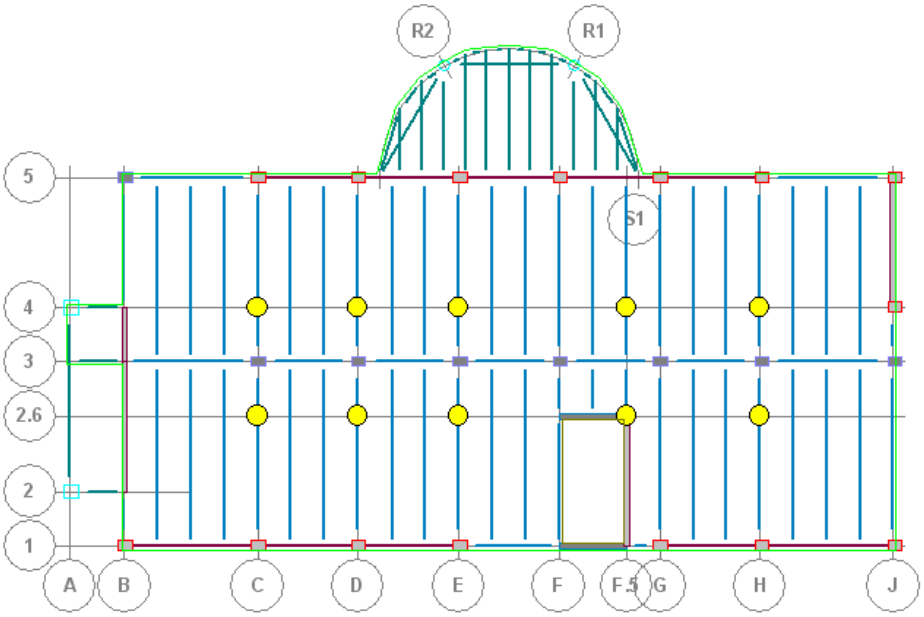


(a)

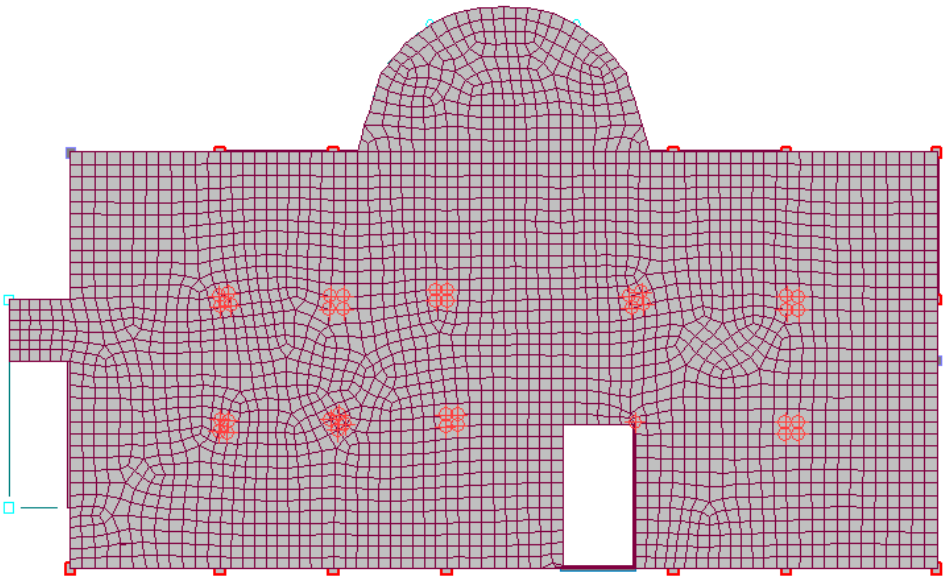


(b)

Figure 26 Line load example, (a) Applied line loads (indicated with yellow lines), (b) converted to point loads applied to mesh nodes directly

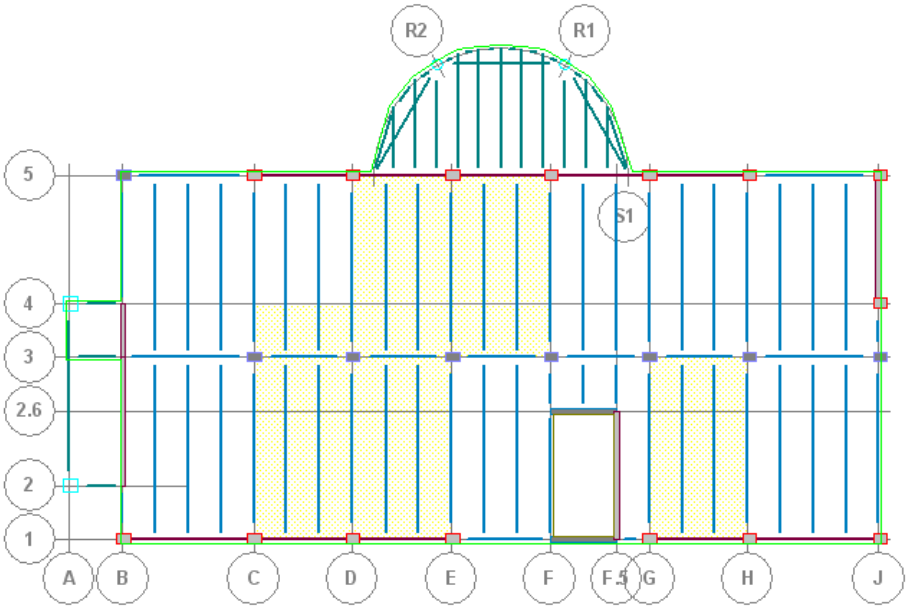


(a)

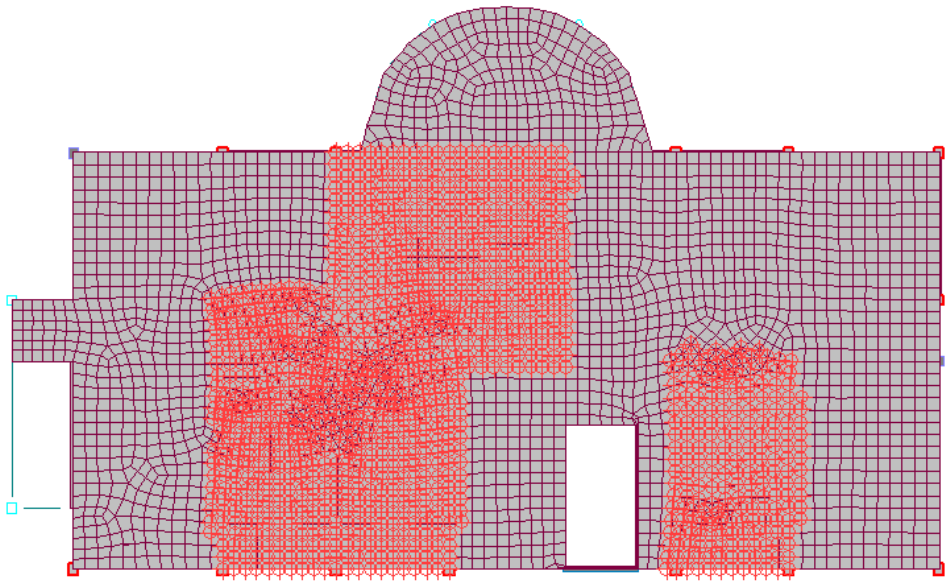


(b)

Figure 27 Point load example, (a) Applied point loads (indicated with yellow circles), (b) converted to point loads applied to mesh nodes directly

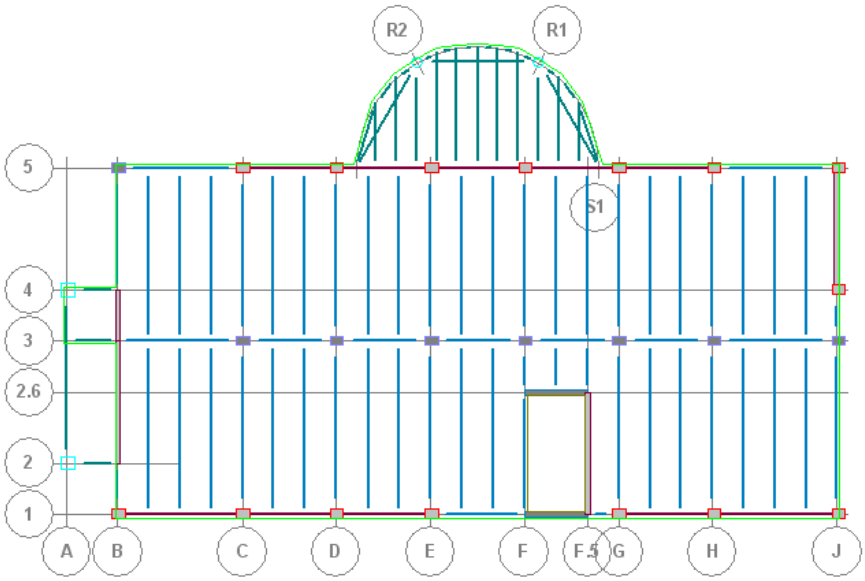


(a)

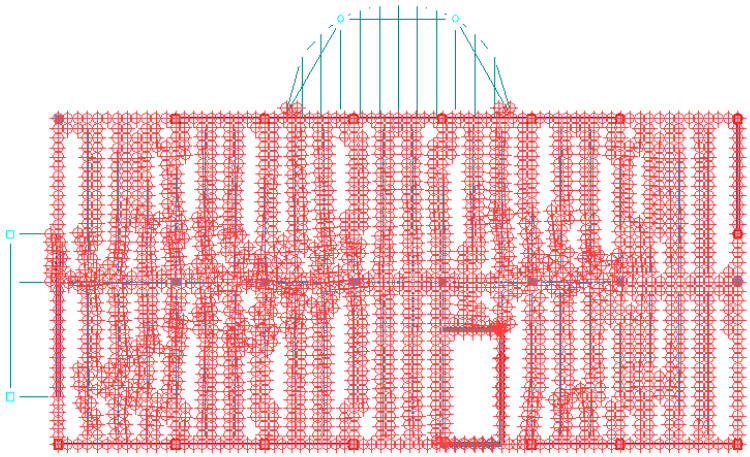


(b)

Figure 28 Surface load example, (a) Surface loads (indicated with yellow polygons), (b) converted to point loads applied to mesh nodes directly

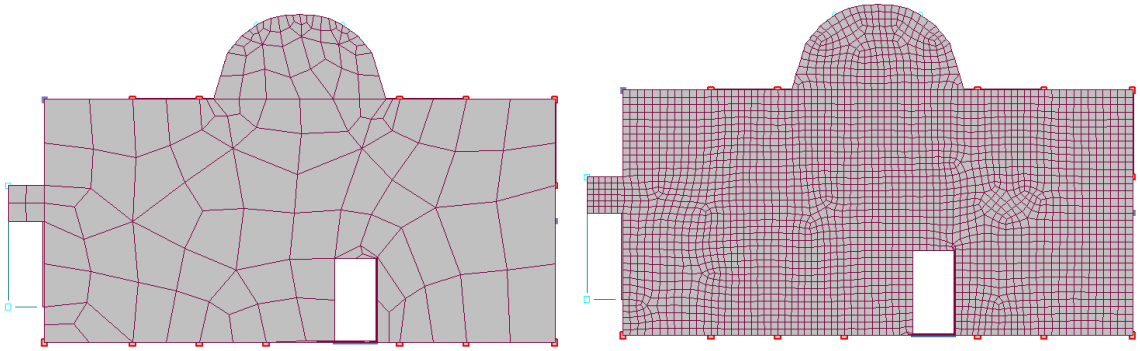


(a)



(b)

Figure 29 Members' self weight or mass loads, (a) All gravity and lateral members shown, (b) Calculated loads directly applied to members



(a) (b)

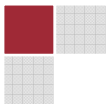
Figure 30 . (a) the mesh is generated with 16 foot max. allowed distance,

(b) the mesh is generated with 2 foot max. allowed distance

4.4. CLOSING REMARKS ON DIAPHRAGM MESHING

The following points are important to note:

- Only lateral members are included during the meshing process. Gravity members are only considered in finding closed beam loop if "Use Beams for Exterior Boundary" option is chosen or for inserting vertical springs as gravity columns and walls.
- All openings are accounted for but penetrations are not. After the meshing is completed, the diaphragm is represented with a collection of finite shell elements. For each shell, the program calculates its properties (such as thickness, E, etc…) based on the information assigned to decks in the Modeler. This process is internally taken care of by the program so that the user does not need to go through this tedious task manually. The "Model Data" report provides more information regarding calculated shell's properties.
- During the meshing process, the following details are taken care of by the program automatically:
    - *Walls and Semirigid Diaphragms*: when a wall is found to be a part of the diaphragm, the mesh around the wall is generated in such a way that both the diaphragm and the wall are attached to the same mesh nodes. This provides compatibility between the diaphragm and the walls (hence, the load transfer from the diaphragm to the wall is ensured in analysis).
    - *Columns and Semirigid Diaphragms*: Similarly, when a column is found to be attached to a semirigid diaphragm, the interaction between them is taken into account in such a way that both the column and the diaphragm are attached to the same column point (hence, both load transfer and compatibility fulfilled).



- *Beams and Semirigid Diaphragms*: Beams located inside the diaphragm are also meshed along their lengths. Note that the meshed beams are attached to the diaphragm at the nodes created along their lengths. It is always guaranteed that no individual shell element crosses the meshed beams.
- *Deck Geometry*: Any changes in deck geometry are considered during the meshing process. Thus, no finite element is created that crosses decks with different properties (i.e., E or thickness).
- *Sloped Diaphragms*: A sloped diaphragm can be defined by setting correct orientation of decks located inside the diaphragm. Each deck represents a unique plane and by adjusting column and wall elevations inside the deck, one can create a sloped plane for the deck. It is assumed that each sloped deck constitute a flat-plane (no warped surface allowed).
- *Column Drop Zones*: The program considers column drop areas during meshing process. Generated final mesh visually shows column drop zones.

5. P-DELTA EFFECTS

Additional moments are induced on a structure due to the movement of its mass\weight under lateral loads. This structural phenomenon is called the (large) P-Delta effect. Various codes of practice require that these effects be considered in the analysis and design of building structures.

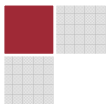
In RAM Frame, two different methods are implemented to consider P-Delta effects in analysis. Either mass or gravity loads (DL+LL) can be used in each one of the methods.

5.1. NON-ITERATIVE P-DELTA METHOD

This method is suited for models with rigid diaphragms (in-plane stiffness action considered). It is an approximate method developed by Wilson et al (1987), and it is referred to as the Geometric Stiffness Method. Small deflections are assumed and it involves modifying the stiffness matrix by subtracting what is called the "geometric stiffness matrix".

This method forces P-Delta effects on both static and dynamic (Eigen value extraction) analyses. The softening effect of P-Delta analysis results in additional story displacements, horizontal shears and overturning moments. The same softening effect also results in lengthening of the periods of the structure. Note that the reactions components obtained from analysis where P-Delta effects are considered contain additional horizontal shears and overturning moments.

For buildings with columns disconnected from a rigid diaphragm, the P-Delta modifications are made only to the rigid diaphragm stiffness. In some building floor type configurations, users may disconnect a significant number of columns from the diaphragm resulting in small overall diaphragm stiffness. P-Delta corrections may introduce singularity in the lateral stiffness corresponding to such diaphragms. RAM Frame detects such singularities and



aborts the program. Users may then wish to combine the mass of such a diaphragm to the ones above or below it. Singularities could also occur in other types of buildings if P-Delta effects are high.

5.2. PRELIMINARY P-DELTA ANALYSIS METHOD

RAM Frame utilizes a non-iterative method to address P-Delta effects for rigid diaphragms in which global building stiffness are modified due to diaphragm mass\weights. Unfortunately, this method is strictly suited for only rigid diaphragm idealization where diaphragm response is represented with three degrees of freedom (two for lateral translation and one for rotation around global Z-axis).

The Preliminary P-Delta Analysis method is developed to be used for models with semirigid diaphragms. It involves a preliminary assessment of column\brace axial loads under only gravity or mass loads. Briefly,

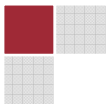
- The model is run against only gravity loads (dead and live) or mass loads. Axial forces for columns and braces are stored (only columns and braces located inside a semirigid diaphragm are considered). These axial forces are also referred to as P-Delta Axial Forces.
- Element geometric stiffness matrix for each column and brace is calculated based on the P-Delta axial force of each element. Then, element stiffness matrix is modified with the geometric stiffness matrix. Note that this modification is applied only once.
- The program runs all load cases while modified element stiffnesses are used to assemble building stiffness matrix.

This process is automated and transparent to the user. It can be regarded as a 2 iteration analysis: in the 1^{st} iteration, P-Delta axial forces in columns\braces due to gravity\mass loads are detected and then, in the 2^{nd} iteration, these axial forces are used to predict P-Delta effects for all remaining load cases.

It should be noted that the current implementation fully satisfies equilibrium equations. The main assumption imposed with this approach is that axial forces due to gravity\mass loads do not significantly change under lateral loads (in fact, this is the same assumed for the non-iterative P-Delta method implemented for rigid diaphragms).

Note that P-Delta effects are reflected in each individual element stiffness matrix instead of in assembled building global stiffness matrix as observed in non-iterative P-Delta method. Thus, one may conclude that the non-iterative P-Delta method modifies global stiffness matrix whereas the Preliminary P-Delta Analysis method modifies each individual element stiffness matrix. In either case, geometric stiffness matrices are calculated.

In calculating P-Delta axial forces, it is important to note that primarily diaphragm's out-of-plane (bending) stiffness play an important role in distributing vertical loads (gravity or mass) loads to columns and braces. The diaphragm's in-plane stiffness (membrane stiffness) has a secondary impact. Thus, if a semirigid diaphragm is composed of one-way



deck and if its out-of-plane stiffness is not included, then the vertical load distribution is carried our according to the diaphragm's in-plane (membrane stiffness) only. Live load reductions are also applied to P-Delta axial forces for each column\brace. Furthermore,

5.3. P-DELTA LOADS

Two sources of loads that are used in P-Delta effects are considered in the program:

- **Use Mass Loads:** This option uses building mass to calculate P-Delta effects (the building mass is converted to building weight to be used in P-Delta calculations). It is briefly related to mass associated with diaphragms (including member mass as well as any mass load defined with surface\line and point loads). This is the default method used in the program. RAM Frame provides users with an option for specifying factors for magnifying the P-Delta effects. To account for P-Delta effects due to live loads and/or factored loads, users may want to specify values larger than unity for the P-Delta factor.
- **Use Gravity Loads:** This option uses applied dead and live loads to calculate P-Delta effects. Different scale factors can be used for dead and live load components. Live load reduction factors are also applied for live loads. The program uses the gravity loads as summarized in **Diaphragm Gravity Loads** dialog (user defined gravity loads can be also used for P-Delta calculates for diaphragms with one-way decks).

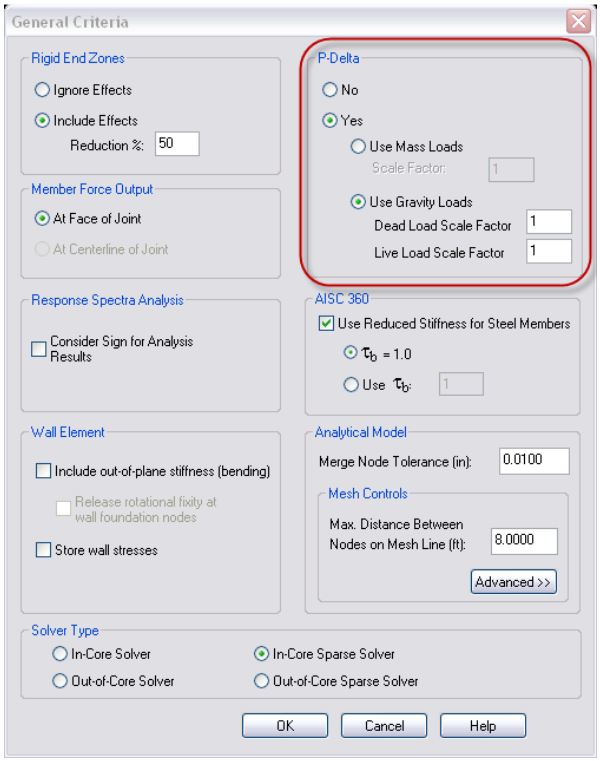
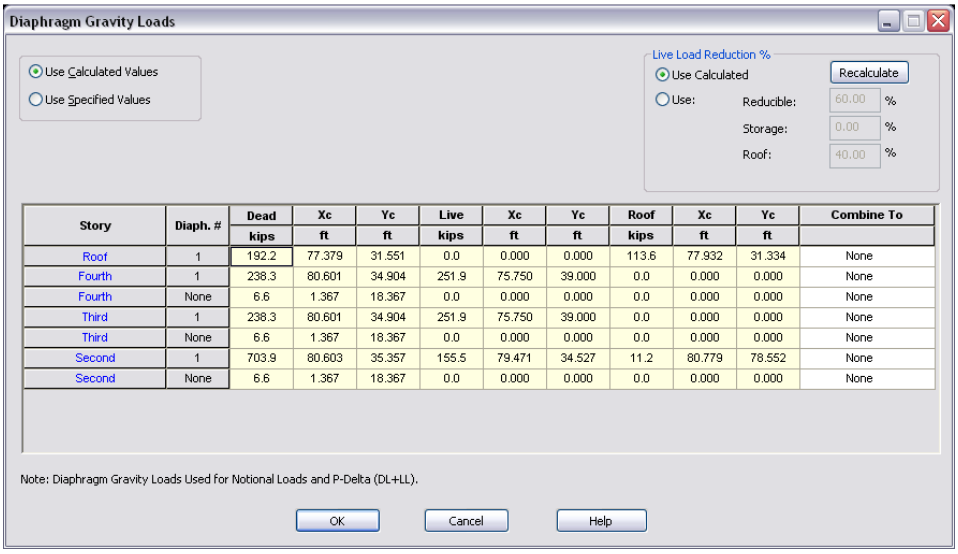


Figure 31 P-Delta Options



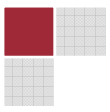
Story	Diaph. #	Dead	Xc	Yc	Live	Xc	Yc	Roof	Xc	Yc	Combine To
		kips	ft	ft	kips	ft	ft	kips	ft	ft	
Roof	1	192.2	77.379	31.551	0.0	0.000	0.000	113.6	77.932	31.334	None
Fourth	1	238.3	80.601	34.904	251.9	75.750	39.000	0.0	0.000	0.000	None
Fourth	None	6.6	1.367	18.367	0.0	0.000	0.000	0.0	0.000	0.000	None
Third	1	238.3	80.601	34.904	251.9	75.750	39.000	0.0	0.000	0.000	None
Third	None	6.6	1.367	18.367	0.0	0.000	0.000	0.0	0.000	0.000	None
Second	1	703.9	80.603	35.357	155.5	79.471	34.527	11.2	80.779	78.552	None
Second	None	6.6	1.367	18.367	0.0	0.000	0.000	0.0	0.000	0.000	None

Figure 32 Gravity loads for P-Delta Effects

5.4. Closing Remarks for P-Delta Analysis

In RAM Frame, in-plane diaphragm action can be defined with one of four types: **Flexible/None, Pseudo-Flexible, Semirigid** and **Rigid**. Regarding P-Delta effects, only **Rigid** and **Semirigid** diaphragm types are considered. In addition, the program allows one-way or two-way load distribution for gravity loads and this can be defined with one-way or two-way deck definition in the Modeler.

Briefly, whole diaphragm behavior is defined with a combination of its in-plane and out-of-plane behavior. The in-plane behavior (transferring wind, seismic or dynamic loads to lateral load resisting members) can be defined with one of the four types. The out-of-plane behavior (distributing gravity loads to beams and columns through either one-way or two-way load distribution) can be determined with one-way or two-way deck assignment. Mass or gravity loads are used for P-Delta effects. Considering all these choices, there exists different possible ways of handling P-Delta effects in the program. The following figure shows all possible choices and briefly explains how the program proceeds:



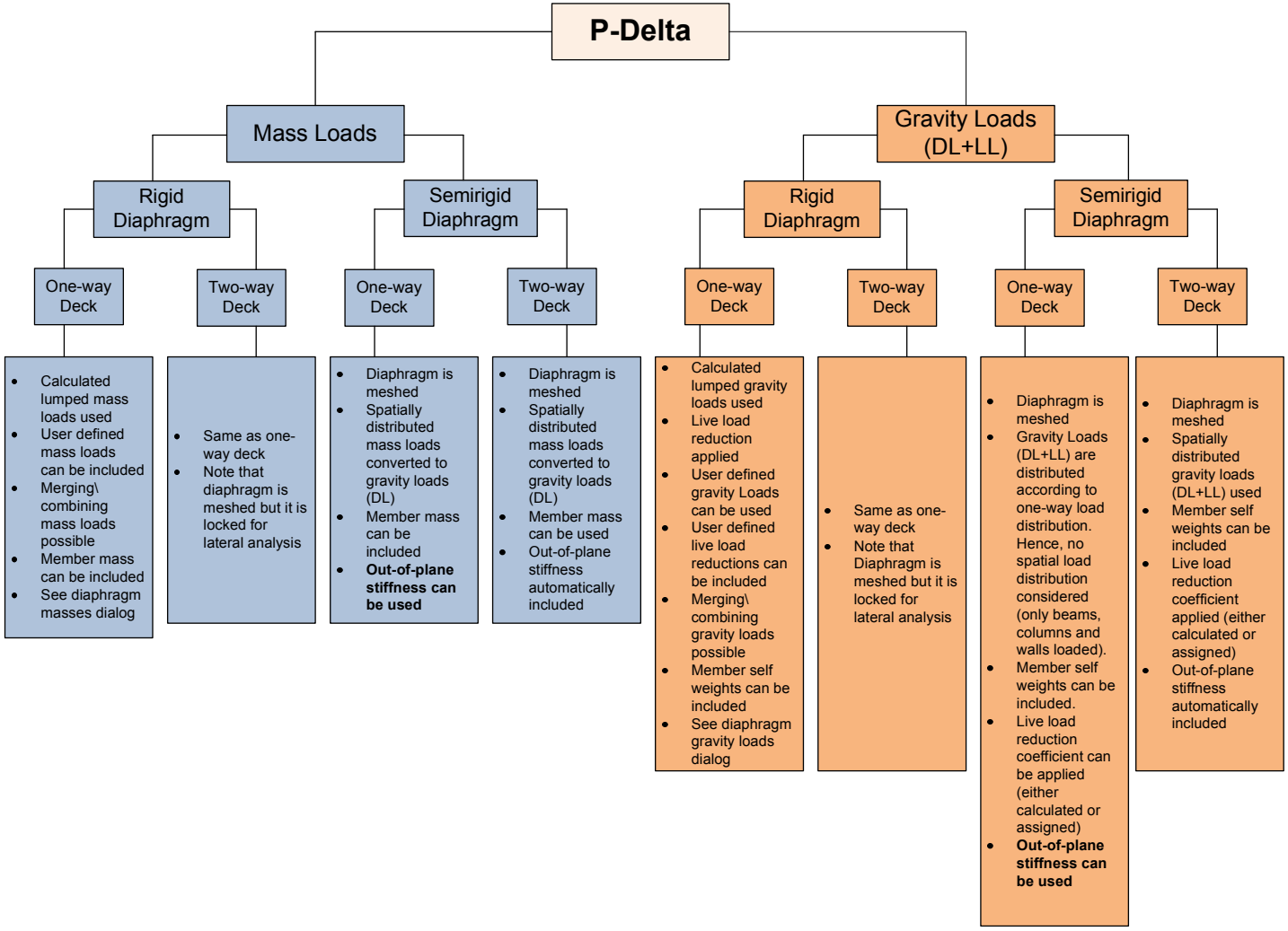


Figure 33 P-Delta Implementation in RAM Frame