1.4.6 Calculations considering diaphragms

All of the available calculation in FEM-Design can be performed with diaphragms or without diaphragms if the diaphragms were defined in the model. By the different types of calculation the results will be analogous with the adjusted option. But there are several restrictions and exceptions by the diaphragm calculation. These issues will be detailed here.

If the "Apply diaphragms in analysis" option is adjusted then all of the desired calculations will consider the effect of diaphragms and the user will get some results.

All of the calculations will be perfomed considering diaphragms or without considering diaphragms depending on the adjusted option.

1.4.6.1 The mechanical behaviour of the diaphragms

The usage of the diaphragm tool is optimal by storeys of high-rise buildings but the diaphragm modelling tool is also useful by other engineering problems. Thus it means that the diaphragm is basically a modelling tool.

The diaphragms could be several regions, but **they must be horizontal**. The shapes of the diaphragms can be arbitrary and they can be separated also on the specific storey levels. There can be more diaphram regions on the same vertical level and they will work independently from each other.

The finite element nodes which are lying in the plane of the diaphragm will work together. It means that **the relative translations of those nodes in the horizontal direction will be zero** and **the rotation about the vertical axis will be the same**. See Fig. 1.4.6.1.1.

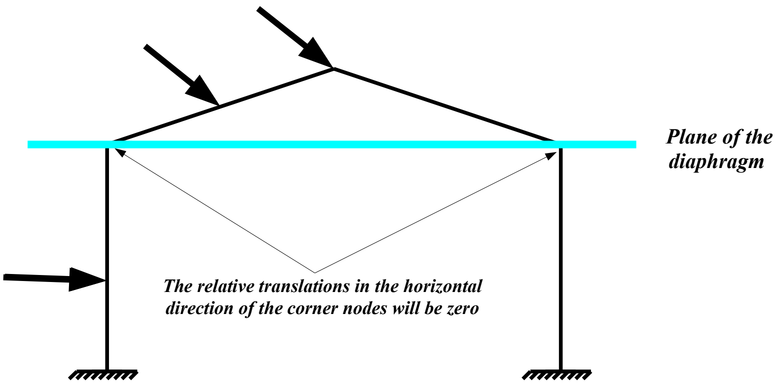

Figure 1.4.6.1.1 – The behaviour of the diaphragm

If there are plates, bars, trusses, etc. in the planes of the diaphragms (**without eccentricities**) the membrane (normal) stiffnesses of those elements are theoretically infinite, which can be imagined as an infinite rigid disc, therefore the **in-plane forces** (membrane forces) **equal to zero** in those elements which are lying in the region of the diaphragms (**without eccentricities**) because **there are no membrane strain in them** (e.g. for shell elements see Eq. 1.4.29). The same analogy is true by trusses, beams and edge connections which are lying in the diaphragms.

$$\begin{bmatrix} \varepsilon_{x'} \\ \varepsilon_{y'} \\ \gamma_{x'y'} \\ \kappa_{x'} \\ \kappa_{y'} \\ \kappa_{x'y'} \\ \gamma_{x'z} \\ \gamma_{y'z} \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ \kappa_{x'} \\ \kappa_{y'} \\ \kappa_{x'y'} \\ \gamma_{x'z} \\ \gamma_{y'z} \end{bmatrix} \quad \text{thus:} \quad \begin{bmatrix} n_{x'} \\ n_{y'} \\ n_{x'y'} \\ m_{x'} \\ m_{y'} \\ m_{x'y'} \\ q_{x'z} \\ q_{y'z} \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ m_{x'} \\ m_{y'} \\ m_{x'y'} \\ q_{x'z} \\ q_{y'z} \end{bmatrix} \qquad \text{(Eq. 1.4.29)}$$

However the out-of-plane stiffnesses of the diaphragms are considered, which means that under the vertical loads the out-of-plane internal forces (bending moments and shear forces) are not zero in the elements which are in the plane of diaphragms.

Due to the infinite membrane rigidity of the diaphragms what was mentioned before, and the lack of some internal forces in the elements which were placed in the plane of the diaphragms the **diaphragms calculation is not really appropriate by RC calculations** (e.g. crack section analysis or RC design).

If eccentricities are applied in plates, bars which are lying in the plane of the diaphragms, **the eccentricities** of those elements **will be ignored during the diaphragm calculation**!

During the mechanical calculation master nodes in the regions of each diaphragms are selected automatically. One master node belongs to one diaphragm. These **master nodes** must have **6 degrees of freedom**. It means that it could be such a structure where this master node does not exist (e.g. a 3D structure built with only truss elements, because one node of a truss element has only 3 degress of freedom). In this case the diaphragm calculation is impossible and an error message will be send to the users.

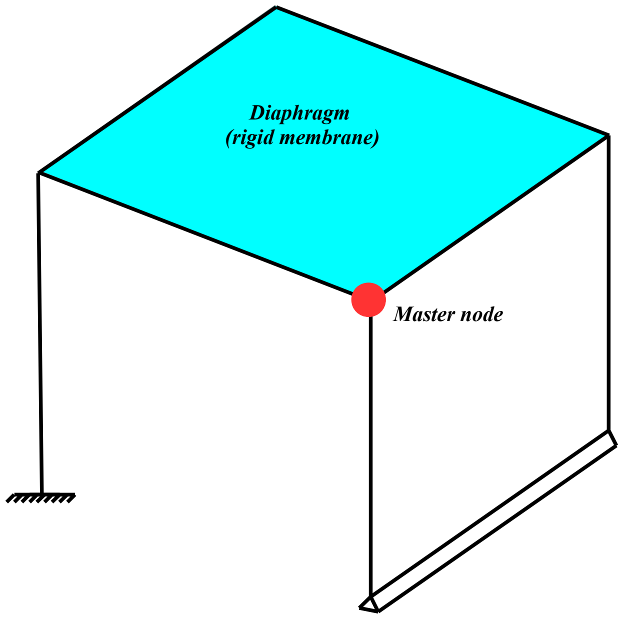

Figure 1.4.6.1.2 – The behaviour of the membrane rigid diaphragm

According to the **rigid membrane behaviour** of the **diaphragms** the horizontal translations of the rest of the nodes in the diaphragms will be calculated based on the horizontal translations and the rotation around the vertical axis of the master node. See Fig. 1.4.6.1.2.

WARNING: The diaphragms have to be horizontal. The support elements and 3D solid elements are not able to connect to a diaphragm beacause according to the mechanical behaviour it does not make any sense.

If there is a diaphragm which is connected to several structural elements but in the plane of the diaphragm zero number of finite element nodes are lying then the finite elements won't work as a diaphragm. It means that for the mechanical calculation the finite element nodes must be in the plane of the diaphragm if they ought to be work together as a rigid membrane.

If eccentricities are applied in plates, bars which are lying in the plane of the diaphragms, the eccentricities of those elements will be ignored during the diaphragm calculation!

Due to the infinite membrane rigidity of the diaphragms and the lack of some internal forces in the elements which were placed in the plane of the diaphragms the diaphragms calculation is not really appropriate by RC calculations (e.g. crack section analysis or RC design).

1.4.6.2 The calculation of the shear center of a storey with diaphragm

The coordinates of an arbitrary selected key node on the diaphragm in the global system (see Fig. 1.4.6.2.1) are:

$$x_m ; y_m \qquad \text{(Eq. 1.4.6.1)}$$

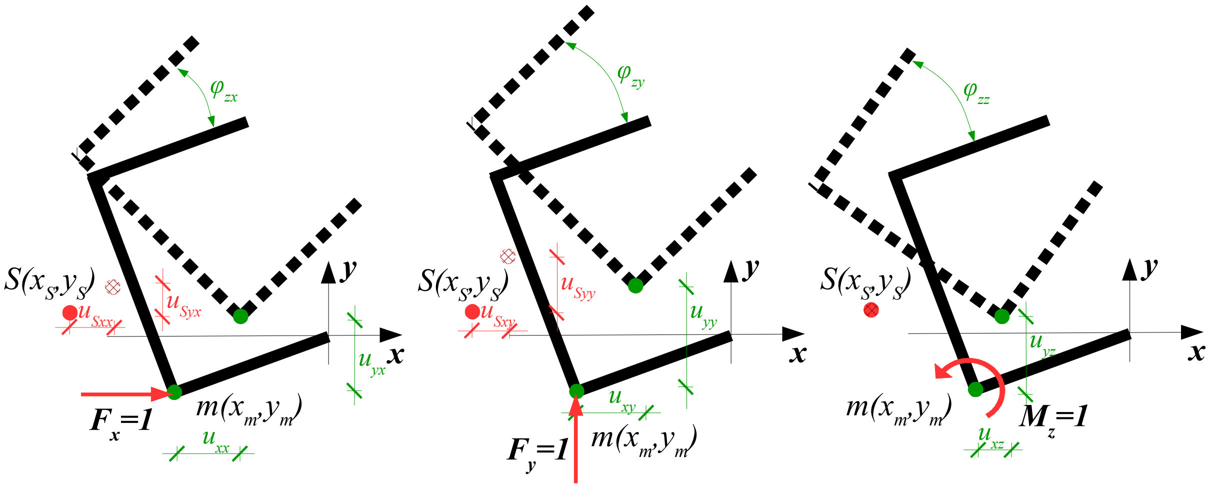

Figure 1.4.6.2.1 – The displacements according to the unit loads (forces and moment) acting on the selected key node of the diaphragms

Based on the displacements due to unit loads at the selected key node (forces and moment, see Fig. 1.4.6.2.1) the global coordinates of the shear center of a storey can be calculated with the following formulas (the global z coordinate is known, because it is lying in the plane of the defined diaphragm):

$$x_S = x_m - \frac{\varphi_{zy}}{\varphi_{zz}} \qquad \text{(Eq. 1.4.6.2)}$$

$$y_S = y_m + \frac{\varphi_{zx}}{\varphi_{zz}} \qquad \text{(Eq. 1.4.6.3)}$$

1.4.6.3 The calculation of the idealized bending stiffnesses in the principal rigidity directions

The usage of the idealized bending stiffness and principal rigidity directions is optimal by high-rise buildings where the bracing system provided mostly with shear walls. The first step is the calculation of the translations of the shear center due to the unit loads on the selected key node at the top (highest) diaphragm (see Chapter 1.4.6.2 and Fig. 1.4.6.3.2). The distances between the shear center and the selected key node are:

$$\Delta x = x_S - x_m \qquad \text{(Eq. 1.4.6.4)}$$

$$\Delta y = y_S - y_m \qquad \text{(Eq. 1.4.6.5)}$$

The translations at the shear center due to the unit forces on the selected key node (according to the rigid diaphragm, see Fig. 1.4.6.2.1):

$$u_{Sxx} = u_{xx} - \varphi_{zx} \Delta y$$ (Eq. 1.4.6.6)

$$u_{Syx} = u_{yx} + \varphi_{zx} \Delta x$$ (Eq. 1.4.6.7)

$$u_{Sxy} = u_{xy} - \varphi_{zy} \Delta y$$ (Eq. 1.4.6.8)

$$u_{Syy} = u_{yy} + \varphi_{zy} \Delta x$$ (Eq. 1.4.6.9)

Based on the reciprocal relations the result of Eq. 1.4.6.7 will be equal to the result of Eq. 1.4.6.8. Based on these values the calculation of the principal translations is the next step. The matrix of the translations of the shear center according to the unit forces at the selected key node:

$$\underline{\underline{U}} = \begin{bmatrix} u_{Sxx} & u_{Sxy} \\ u_{Syx} & u_{Syy} \end{bmatrix}$$ (Eq. 1.4.6.10)

The eigenvectors of this matrix (Eg. 1.4.6.10) gives us the idealized principal rigidity directions. With the eigenvalues of this matrix we can calculate the idealized bending stiffness of the structure as well if we assumed it as a vertical cantilever building with a fixed end at the bottom.

The principal translations (eigenvalues of Eq. 1.4.6.10) at shear center:

$$u_1 = \frac{u_{Sxx} + u_{Syy}}{2} + \sqrt{\left(\frac{u_{Sxx} - u_{Syy}}{2}\right)^2 + {u_{Sxy}}^2}$$ (Eq. 1.4.6.11)

$$u_2 = \frac{u_{Sxx} + u_{Syy}}{2} - \sqrt{\left(\frac{u_{Sxx} - u_{Syy}}{2}\right)^2 + {u_{Sxy}}^2}$$ (Eq. 1.4.6.12)

The order of the principal translations are the following:

$$u_1 \geq u_2$$ (Eq. 1.4.6.13)

The directions of the rigidity axes are the eigenvectors of Eq. 1.4.6.10. Based on these eigenvectors the direction of the strong axis (see Fig. 1.4.6.3.1):

$$\alpha_1 = \arctan \frac{u_1 - u_{Sxx}}{u_{Sxy}}$$ (Eq. 1.4.6.14)

The direction of the weak axis (see Fig. 1.4.6.3.1):

$$\alpha_2 = \arctan \frac{u_2 - u_{Sxx}}{u_{Sxy}}$$ (Eq. 1.4.6.15)

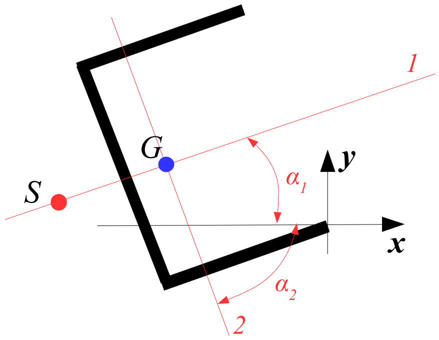

Figure 1.4.6.3.1 – The principal rigidity directions

If the height of the cantilever structure (see Fig. 1.4.6.3.2) is assumed „*H*" then the idealized bending stiffness in the principal rigidity directions can be calculated. The idealized bending stiffness around the weak axis:

$$u_1 = \frac{H^3}{3EI_2} \quad \text{and based on this} \quad EI_2 = \frac{H^3}{3u_1} \qquad \text{(Eq. 1.4.6.16)}$$

The idealized bending stiffness around the strong axis:

$$u_2 = \frac{H^3}{3EI_1} \quad \text{and based on this} \quad EI_1 = \frac{H^3}{3u_2} \qquad \text{(Eq. 1.4.6.17)}$$

These formulas are based on the deflection of a cantilever due to a point load at the end according to pure (uniaxial) bending.

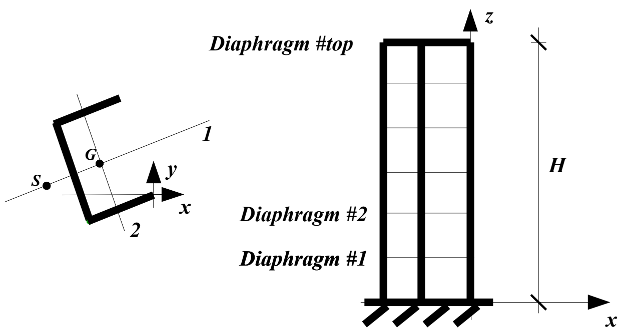

Figure 1.4.6.3.2 – The assumed fixed end at the bottom cantilever for the idealized bending stiffness calculation