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 "Diaphragm calculation" 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.

There are two different options for the diaphragm calculation:

- – Rigid membrane

- – Fully rigid

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.

1.4.6.1.1 Rigid membrane behaviour

The finite element nodes which are lying in the region of a 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.
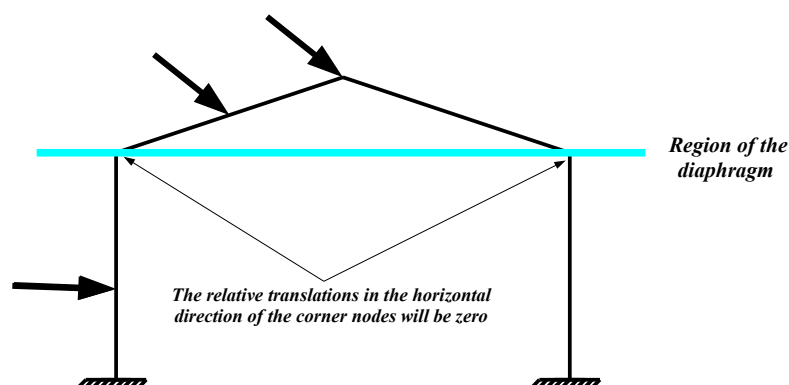

Figure 1.4.6.1.1 – The behaviour of the diaphragm

If there are plates, bars, trusses, etc. in the regions 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 region 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 region of the diaphragms the **rigid membrane 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 region 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.
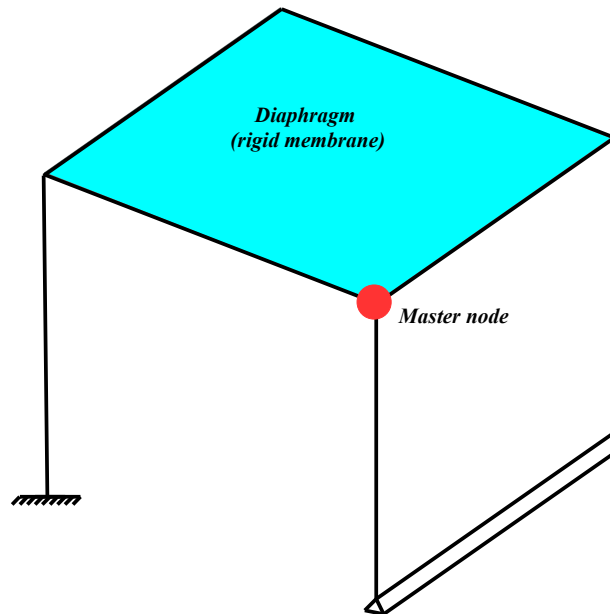
Figure 1.4.6.1.2 – The behaviour of the membrane rigid diaphragm

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.

1.4.6.1.2 Fully rigid behaviour

The finite element nodes which are lying in the region of a diaphragm will work together. In this case **the relative translations and relative rotations of those nodes will be zero.** With other words the diaphragm will work as a classic rigid body.

If there are plates, bars, trusses, etc. in the regions of the diaphragms (**without eccentricities**) the membrane (normal) and bending stiffnesses of those elements are theoretically infinite, which can be imagined as an infinite rigid body, therefore the **in-plane forces** (membrane forces) and **out-of-plane forces** (bending moments) **equal to zero** in those elements which are lying in the region of the diaphragms (**without eccentricities**) because **there are no membrane or bending strains in them** (e.g. for shell elements see Eq. 1.4.30). The same analogy is true by trusses, beams and edge connections which are lying in the diaphragms.

$$
\begin{bmatrix} \varepsilon_{x'} \\ \varepsilon_{y'} \\ \gamma_{x'y'} \\ \kappa_{x'} \\ \kappa_{y'} \\ \kappa_{x'y'} \\ \gamma_{x'z} \\ \gamma_{y'z} \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \quad \text{thus:} \quad \begin{bmatrix} n_{x'} \\ n_{y'} \\ n_{x'y'} \\ m_{x'} \\ m_{y'} \\ m_{x'y'} \\ q_{x'z} \\ q_{y'z} \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \qquad \text{(Eq. 1.4.30)}
$$

If eccentricities are applied in plates, bars which are lying in the region of the fully rigid 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 degrees of freedom). In this case the diaphragm calculation is impossible and an error message will be send to the users.

The following warnings are appropriate for each diaphragm behaviour:

> WARNING: The diaphragms need to be horizontal. The support elements and 3D solid elements are not able to connect to a diaphragm because 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 region 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 region of the diaphragm if they ought to be work together as a rigid membrane or a fully rigid body.
>
> If eccentricities are applied in plates, bars which are lying in the region of the diaphragms, the eccentricities of those elements will be ignored during the diaphragm calculation!
>
> Due to the infinite membrane rigidity or fully rigid behaviour of the diaphragms and the lack of some or all internal forces in the elements which were placed in the region 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:

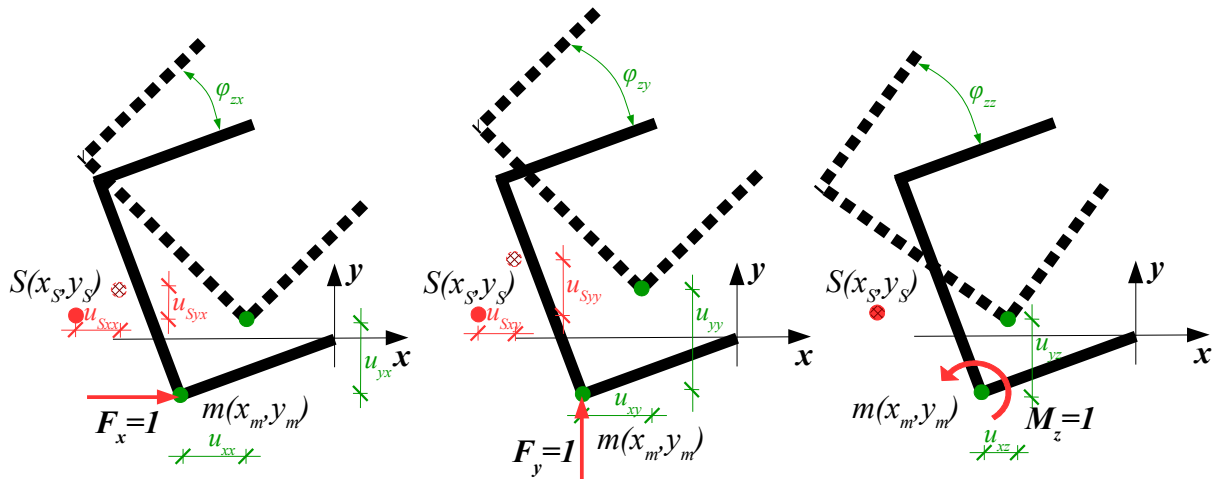$$x_m; y_m \hspace{6cm} \text{(Eq. 1.4.6.1)}$$

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 region of the defined diaphragm):

$$x_S = x_m - \frac{\varphi_{zy}}{\varphi_{zz}} \hspace{5cm} \text{(Eq. 1.4.6.2)}$$

$$y_S = y_m + \frac{\varphi_{zx}}{\varphi_{zz}} \hspace{5cm} \text{(Eq. 1.4.6.3)}$$

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:

$$\Delta x = x_S - x_m \hspace{5cm} \text{(Eq. 1.4.6.4)}$$

$$\Delta y = y_S - y_m \hspace{5cm} \text{(Eq. 1.4.6.5)}$$

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):

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

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

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

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

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:

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

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:

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

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

The order of the principal translations are the following:

$$u_1 \geq u_2 \qquad \text{(Eq. 1.4.6.13)}$$

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):

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

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

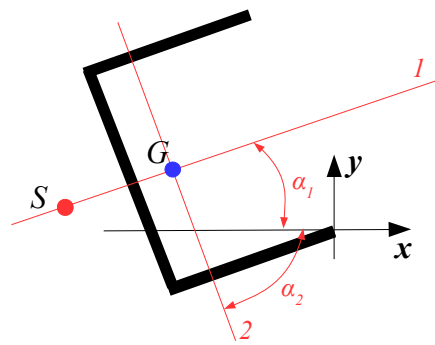$$\alpha_2 = \arctan \frac{u_2 - u_{Sxx}}{u_{Sxy}} \qquad \text{(Eq. 1.4.6.15)}$$

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:

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

The idealized bending stiffness around the strong axis:

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

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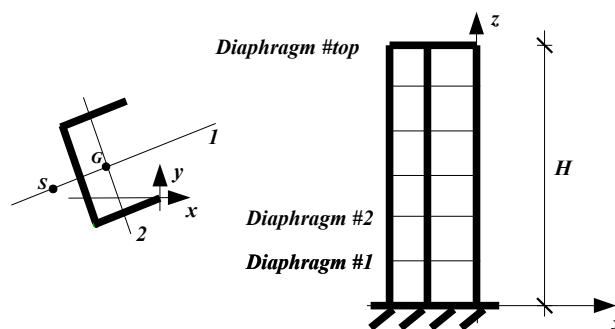

Figure 1.4.6.3.2 – The assumed fixed end at the bottom cantilever for the idealized bending stiffness calculation

1.4.7.3 Calculation considering plastic behavior of structural elements

In **FEM-Design 3D Structure** there is a plastic calculation option by the setup of load combinations. If this option is adjusted, then in the relevant load combination the following structural elements will work with their plastic limit forces and moments (if they are set):

– Point supports (1 DOF, translation or rotation; plastic limit force [kN] or moment [kNm]);

– Point support groups (6 DOF, translations and rotations; plastic limit forces [kN] and moments [kNm]);

– Line supports (1 DOF, translation or rotation; plastic limit specific force [kN/m] or specific moment [kNm/m]);

– Line support groups (6 DOF, translations and rotations; plastic limit specific forces [kN/m] and specific moments [kNm/m]);

– Surface supports (3 DOF, translations; plastic limit specific forces [kN/m^2]);

– Point connections (6 DOF, relative translations and relative rotations; plastic limit forces [kN] and moments [kNm]);

– Line connections (6 DOF, relative translations and relative rotations; plastic limit specific forces [kN/m] and specific moments [kNm/m]);

– Surface connections (6 DOF, translations and rotations; plastic limit specific forces [kN/m^2]);

– Trusses (1 DOF, relative translation; plastic limit force [kN]);

– Edge connections by shell elements (6 DOF, relative translations and relative rotations; plastic limit specific forces [kN/m] and specific moments [kNm/m]).

In this list we indicated the number of degrees of freedom (DOF) of the elements and the types of adjustable plastic limit forces and/or moments.
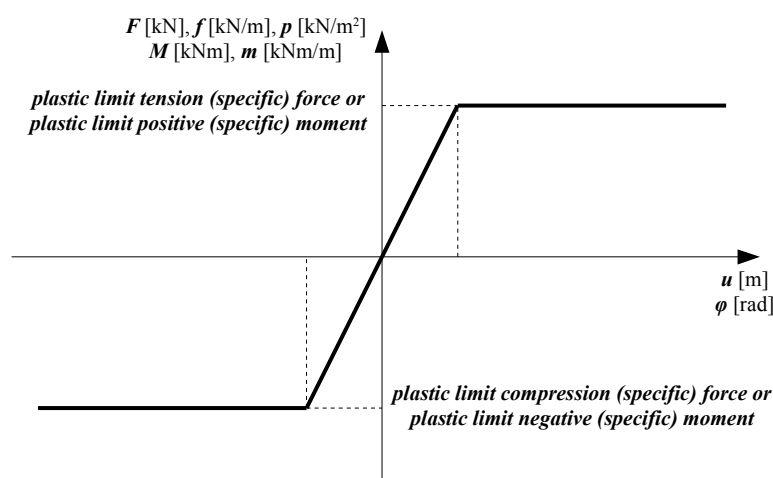

Figure 1.4.7.3.1 – The behaviour of the plastic structural elements linear elastic, perfectly plastic model

In mechanical point of view the plastic behavior is a linear elastic, perfectly plastic behavior (see Fig. 1.4.7.3.1). In **FEM-Design 3D Structure** there is an option to set the mentioned limit forces or moments by the plastic behavior elements (indicated above).

WARNING: The adjusted **plastic limit forces or moments are independent from each other**. With other words there are no interactions between the plastic limits regarding to the DOF of the elements. However the detach behavior is available parallel with the plastic behavior.

All kinematic loads, such as support motion loads, temperature loads, stress loads, etc. will be ignored during the plastic calculations on the specific elements (see the former list) where the plastic limits were adjusted. At the end of a plastic limit calculation a warning message appears if such load types were adjusted in the specific load combination.

Due to numerical stability during the finite element calculation the tangents of the plastic parts have 10^{-5} order of magnitude slope value (see Fig. 1.4.7.3.2).
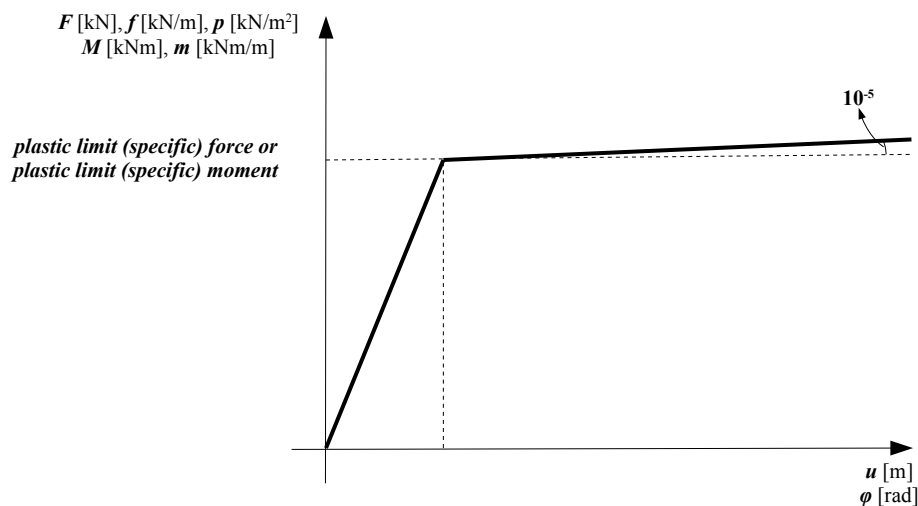

Figure 1.4.7.3.2 – The behaviour of the plastic structural elements
linear elastic, perfectly plastic model

The mathematical solution method to solve the plastic FE calculation in this case is the implicit Newton-Raphson method. The solver works with load steps and there are several iterations in one load steps to reach the correct solution (see Fig. 1.4.7.3.3).
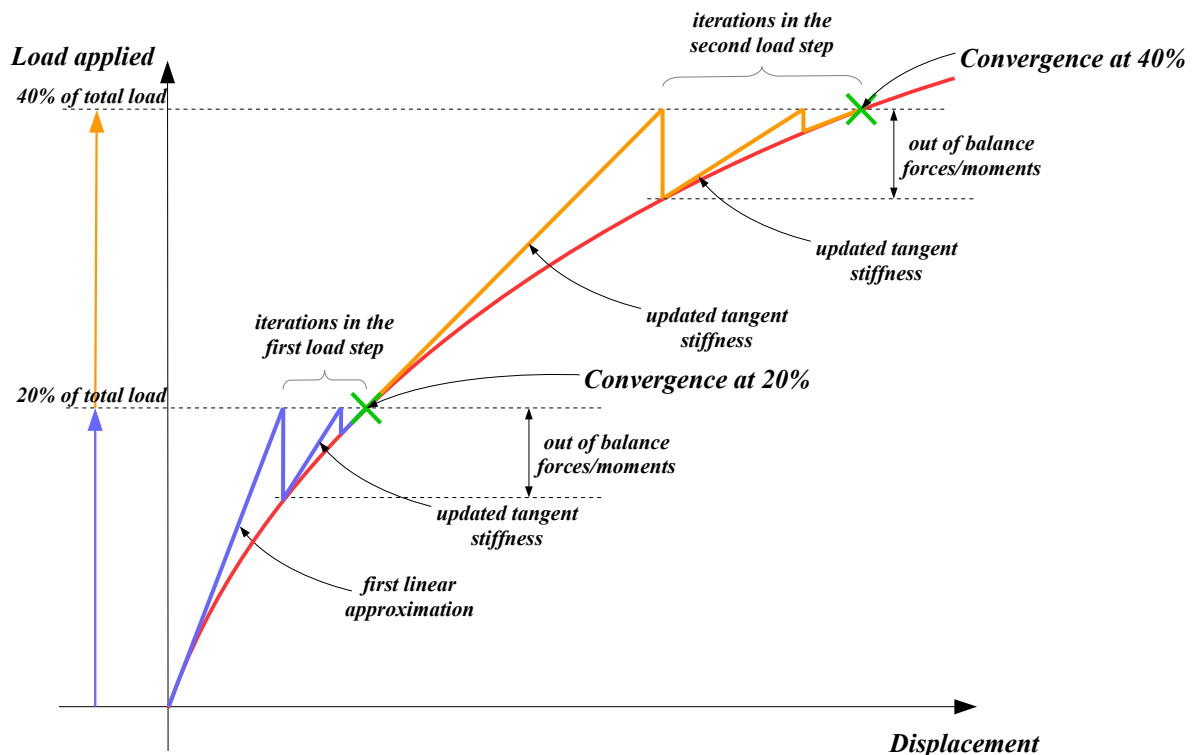
Figure 1.4.7.3.3 – A schematic draw about the applied Newton-Raphson iteration method

In the **Calculation options** the user can adjust the following **control parameters of the plastic analysis**:

– Default load step in % of the total load: this value is the amount of loads in the first linear approximation in every load step (see Fig. 1.4.7.3.3). The default value is 20%.

– Minimal load step [%]: this value is the minimum reduced size of the load step. If there is no convergence in one load step with the default load step value we reduce the amount of the current load step to the half size but it can't be less than the adjusted minimal load step. The default value is 2%.

– Maximum equilibrium iteration number: this number is the maximum number of the iterations in one load step (see Fig. 1.4.7.3.3). The default value is 30 pcs. but in a highly nonlinear model this value should be increased.

The convergence criterion (see Fig. 1.4.7.3.3) is reached when 10^{-3} times the L2 norms of the applied force vector and applied moment vector are smaller than the L2 norms of the unbalanced force vector and unbalanced moment vector.

> WARNING: In a highly nonlinear behavior structure it can occur that according to the plasticity of the structural elements the originally statically determinate structure become a statically overdetermined (e.g. due to plastic hinges, etc.).

If it is occured the structure not able to bear the total amount of the applied load level. The Newton-Raphson solver will stop at the last converged load level. The state of this last converged load level will depend on the control parameters of the plastic analysis (see the former paragraph).

The last converged load level will be indicated after a plastic load combination calculation and also in the titles of the results which ones belong to a plastic load combination.

The calculation time of a highly nonlinear structure could increase exponentially compared to a regular linear calculation.