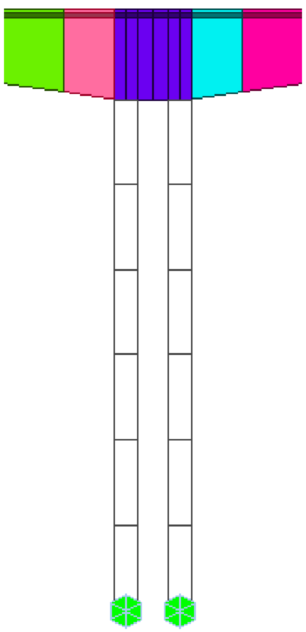
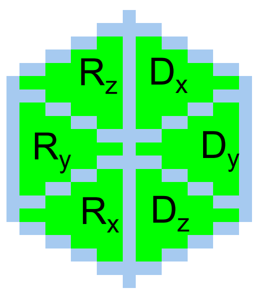
Midas Civil Self-Learning

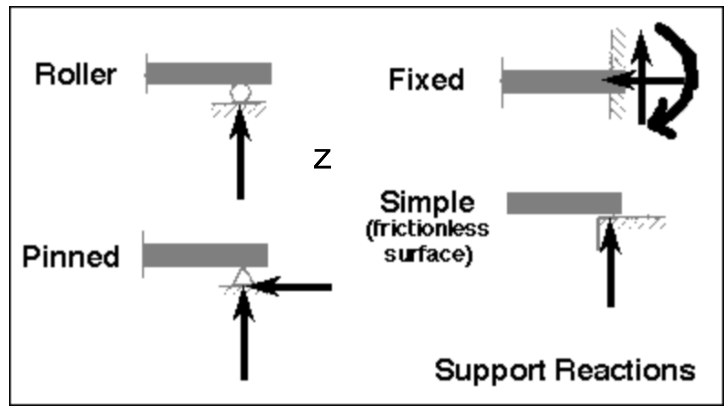
Supports in midas Civil

Supports (Full Fixity/Full Release)

In order to be able to analyze a structure, it is first necessary to be clear about the forces that can be resisted, and transferred, at each level of support throughout the structure. The actual behavior of a support or connection can be quite complicated. So much so, that if all the various conditions were considered, the design of each support would be a terribly lengthy process. And yet, the conditions at each of the supports greatly influence the behavior of the elements which make up each structural system.







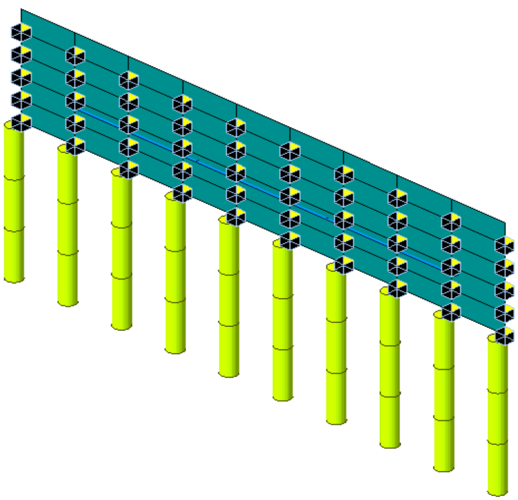
	D_x	D_y	D_z	R_x	R_y	R_z
Rolled	✔	✔	X	✔	✔	✔
Fixed	X	X	X	X	X	X
Pinned	X	X	X	✔	✔	✔

Allowed DOFs for various boundary conditions

X - Restrained
✔ - Allowed

Point Spring Support (Partial Fixity/Partial Release)

Point Spring Supports on an Abutment



Spring Supports

Point spring supports are used to model the supports with finite stiffness. For example, a mat foundation on soil can not be modelled properly by using normal supports. Spring supports with the spring stiffness calculated based on the soil properties are necessary to model the non-linear behavior of the soil.

Spring Supports can be of four types:

a. Linear
b. Compression Only
c. Tension Only
d. Multi Linear

Surface Spring Support

Enter spring stiffness per unit supporting area of planar or solid elements to create elastic spring supports. Elastic Link elements may be created simultaneously. This function is mainly used to define several elastic supports on surfaces represented by the modulus of spring. For example, if the user wishes to define elastic supports for subgrades of foundations or underground structures, subgrade springs will be automatically entered at each node represented by concentrated stiffness. This function enable the user to specify the surface or line stiffness without having to worry about the discretization (sizes) of elements, which is automatically taken care of by the program.

The function is mainly used to consider elastic support conditions of subgrades in the analysis of foundations or underground structures.

Rigid Link vs Elastic Link

Rigid link constraints geometric, relative movement of a structure, where degrees of freedom of subordinated nodes called Slave Nodes are constraint by a reference node called Master Node. The relative movements of the master node and slave nodes are such as if they are interconnected by a three-dimensional rigid body. In this case, relative nodal displacements are kept constant. While an Elastic link connects two nodes to act as an element, where the user defines its stiffness.

Rigid Type of Elastic Link vs Rigid Link

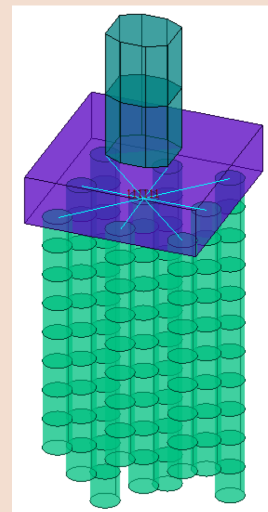
- Rigid Type of Elastic Link and Rigid Link are similar in that both are used to simulate rigid behavior. However, the user must be cautious in using them because their internal processes are different in the program. In case of Rigid Type Elastic Link , the element stiffness is automatically calculated based on the working model, assigning a large stiffness value of magnitude 105~108 times the stiffness of neighboring elements.

- Such exceptionally large stiffness may cause a numerical error because of the relatively large stiffness of the link element. Therefore, when the model contains an element, which has large stiffness to replicate a rigid action, it is recommended that Rigid Link be used rather than Rigid Type Elastic Link. Rigid Link geometrically constrains the relative movements between the Master and Slave Nodes without being affected by large stiffness of other members.

Force Output

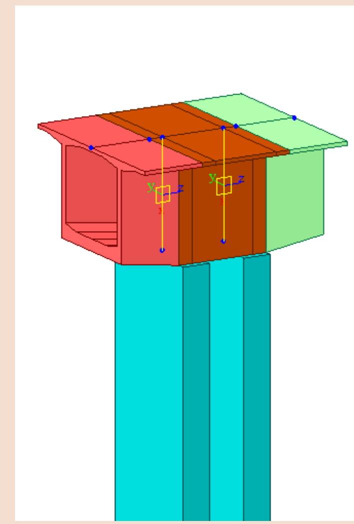
- midas Civil displays forces in an elastic link in post-processing. Therefore, an elastic link should be preferred for modeling a bearing if you want to check forces in bearing in post-processing mode.

Rigid Link Usage



Rigid Link can be conveniently used to connect pile cap to top of pile columns

Elastic Link of Rigid Type



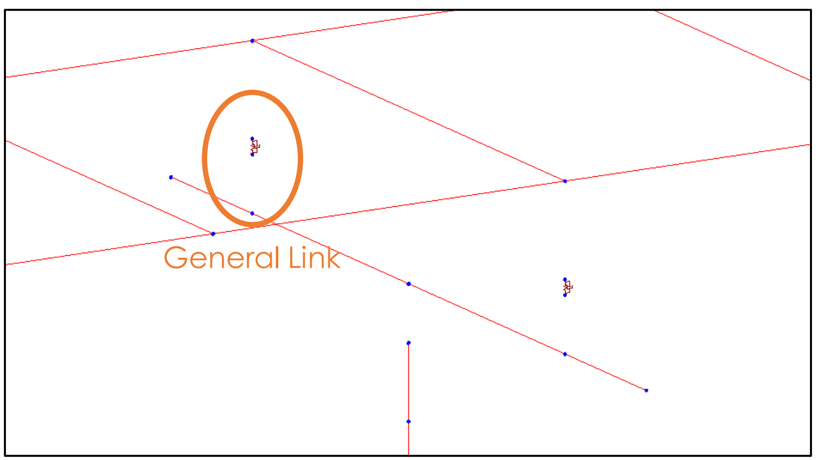
Elastic Link of Rigid Type can be used to connect structural elements to ensure monolithic behavior.

General Link & General Spring

General Links

General Links can be used to formulate different type of boundary conditions such as:

- Viscoelastic Damper
- Gap
- Hook
- Hysteretic System Lead Rubber Bearing
- Isolator Friction Pendulum System Isolator
- Triple Friction Pendulum System Isolator



General Springs

General spring is normally used to reflect the support stiffness of piles. Pile support stiffness related to each degree-of-freedom are considered in the structural analysis.

In typical bridge structures, analysis models do not include the pile footings. Instead, boundary springs are assumed to exist at the bottom of foundation or at the pile caps. The following general stiffness offers true stiffness of pile elements.

$$\begin{bmatrix} \frac{EA}{L} & 0 & 0 & 0 & 0 & 0 \\ & \frac{12EI_z}{L^3} & 0 & 0 & 0 & \frac{6EI_z}{L^2} \\ & & \frac{12EI_y}{L^3} & 0 & -\frac{6EI_y}{L^2} & 0 \\ & & & \frac{GJ}{L} & 0 & 0 \\ & & & & \frac{4EI_y}{L} & 0 \\ Symm. & & & & & \frac{4EI_z}{L} \end{bmatrix}$$

Such springs can be modelled using General Springs in midas Civil.

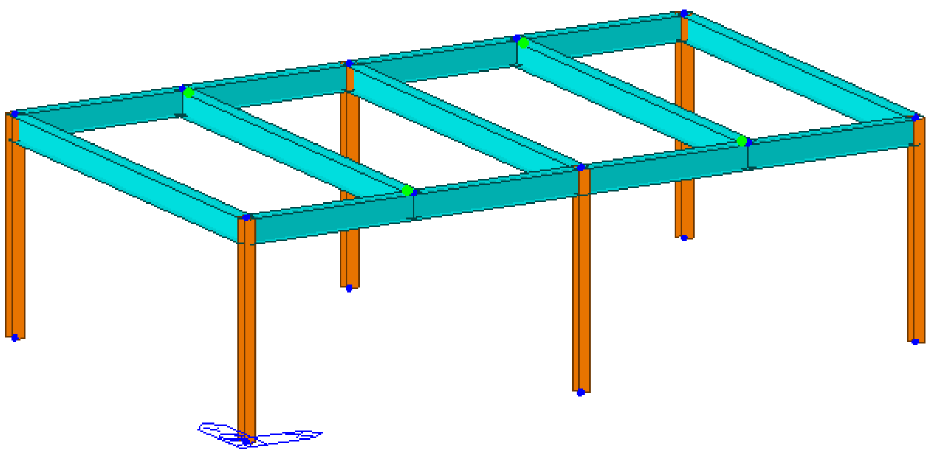
Releases – Beam end vs Plate end Release

An end release will allow either or both ends of a beam element to rotate about or translate along one or more of the local axes of the beam. A beam end release can be either **relative** or **value**.

Relative
You need to enter the stiffness ratio. For example, 0 represents completely released condition and 1 does fully fixed condition.

Value
Enter true magnitudes of stiffness values to define Partial Fixity.

A release can be in F_x, F_y, F_z, M_x, M_y, M_z or M_b direction.

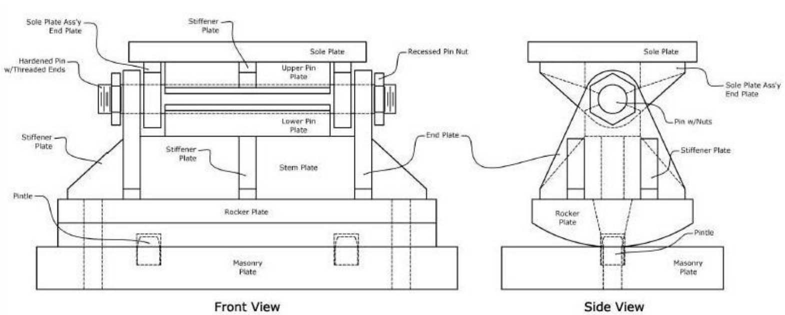


For example, if a My, and Mz is released for a beam on i-end and j-end, it will behave as a truss element.

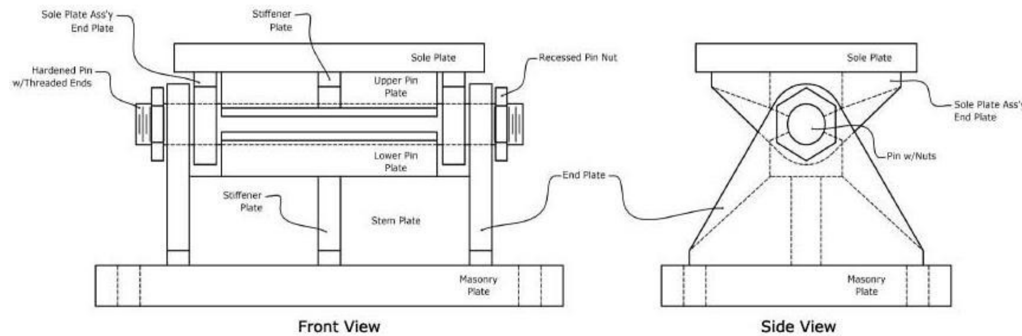
Bridge Bearing Simulation in midas Civil

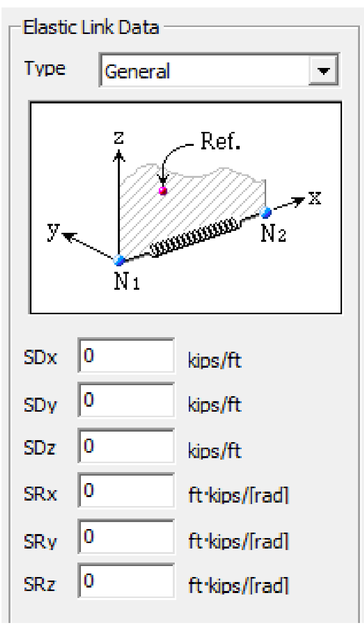
Rocker Bearing

Expansion Rocker Bearing



Fixed Rocker Bearing





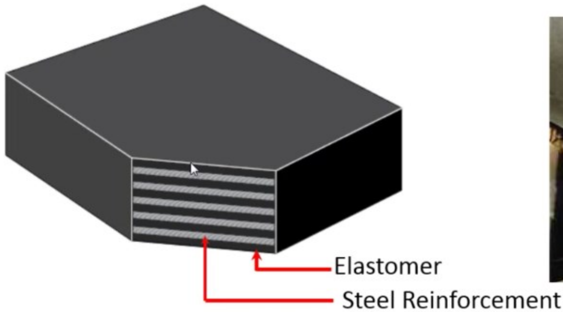
To model a Rocker Bearing in midas Civil, Elastic Link can be used.

For example, for a simple bearing , the following set of parameters might be ideal:

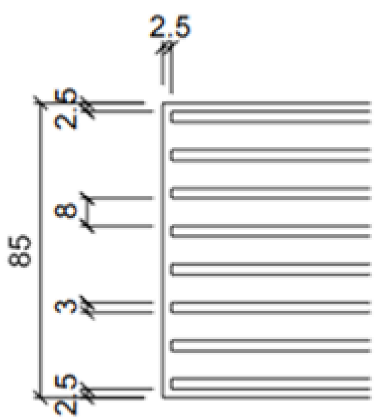
- A General type of elastic link
- SDx = 10e12 kips/ft
- SDy = 10e12 kips/ft
- SDz = 10e12 kips/ft
- SRx = 10e12 kips/ft
- SRy = 10e2 kips/ft
- SRz = 10e12 kips/ft

Elastomeric Laminated Bearing

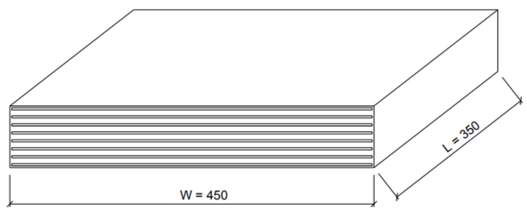
Steel Reinforced Elastomeric Bearing



Stiffness of an Elastomeric Bearing



Dimensions of an elastomeric bearing

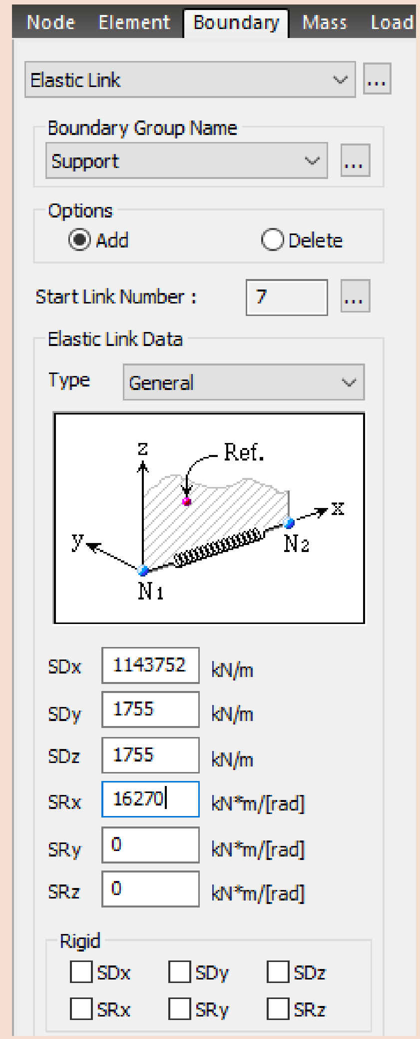


Stiffness Calculation Elastomeric Bearing

$$K_H = k_{eff} = \frac{G_{eff} A}{H_r} = \frac{680x0.1575}{0.061} = 1755kN/m$$

$$K_V = \frac{E_c A}{H} = \frac{617263x0.1575}{0.085} = 1143752kN/m$$

$$K_\theta = \frac{EI}{H_r} = \frac{617263x0.0016}{0.061} = 16270kNm/m$$



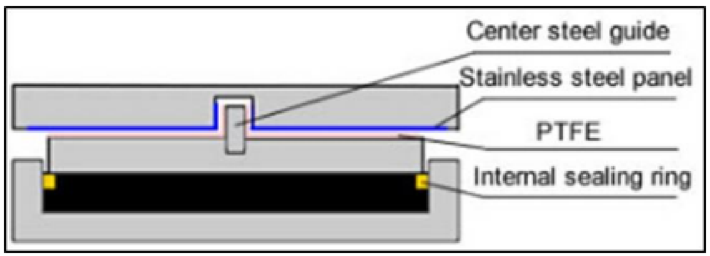
Elastomeric Bearing can be simulated using the Elastic Links in midas Civil.

K_H stiffness value needs to be entered as SD_y and SD_z i.e. horizontal stiffness values of elastic links

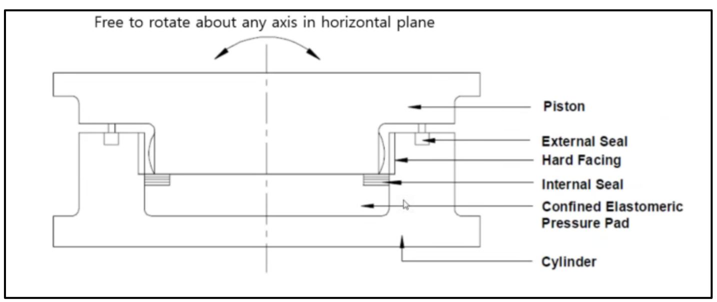
K_V stiffness value needs to be entered as SD_x i.e. vertical stiffness value of elastic links in MIDAS Civil.

Pot Bearing – Uni-directional & Multi-directional

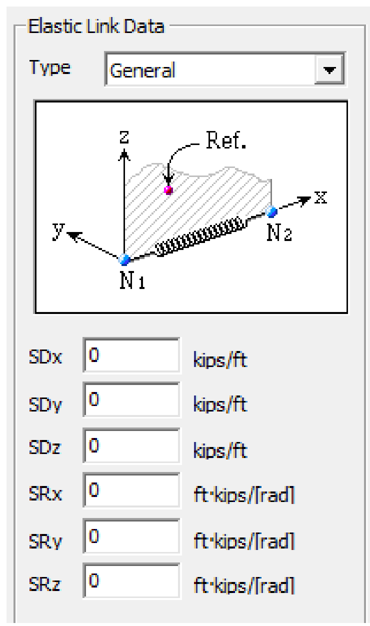
Free sliding pot bearing can move in any direction and therefore cannot accommodate any horizontal force. On the other hand, the central guided pot bearing can accommodate the horizontal force that is perpendicular to central steel guide as shown in the figure.



Guided Pot Bearing



Fixed Pot Bearing



Types of POT PTFE Bearings

- Fixed
- Free sliding type (can translate in both horizontal direction)
- Sliding type with guides (can translate in guided horizontal direction)

For guided bearings (like POT-PTFE bearing), high values for SD_y and/or SD_z could be assigned to simulate high lateral bearing stiffness.

In addition for multi-directional bearing, high value of SD_z can be assigned while keeping normal values for SD_x & SD_y.