AxisVM Structural Analysis Software

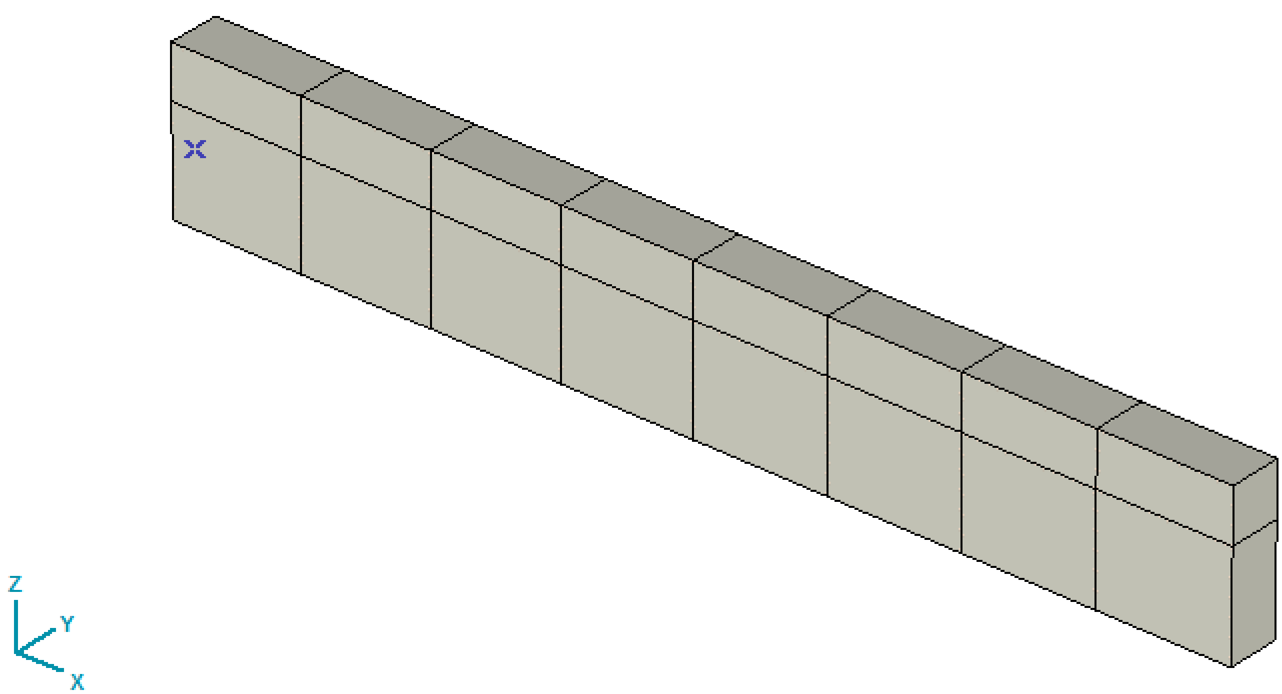
November 2005

Link Elements Tutorial

This short step-by-step tutorial will answer some questions on “How to use the link elements”. The tutorial is based on an example of modeling a semi-composite layered beam, and assumes that the user has AxisVM modeling experience with the beam and rib finite elements. Floor-wall connections can be modeled in a similar way.

Introduction

Assume two wood beam members interconnected by connectors (nails or screws) that provide inter-layer force transfer. The resulting two-layer beam exhibits a semi-composite behavior, since the interlayer slip can not be in general fully prevented. The interlayer slip can be close to zero or can develop freely corresponding to a fully-composite and non-composite behavior, respectively. Depending on the number and type of the connectors, the connectors can be considered uniformly distributed over the length of the semi-composite beam (case A) or they can be considered discrete connections (case B).



To model the connection between the two layers, line-to-line (L-L) link elements (case A) and L-L and node-to-node (N-N) link elements (case B) will be used. In both cases the L-L links will ensure the conform deformation of the two layers.

Modeling the two beam layers

The example structure (see figure below) is plane and is only loaded in its plane, so can be modeled as 2D. The X-Z global plane was chosen (arbitrary), so the degrees of freedom (DOF) of all the nodes will be constrained as “fcfcfc” where “f” and “c” correspond to free and constrained DOF, respectively. The beam is simply supported. The top layer is 20 cm x 20 cm and the bottom layer is 20 cm x 40 cm cross-section madce of Douglas fir. A mid-span point load is applied.

Since L-L link elements (2x3-node elements) are intended to be applied to the model, the beam members of the two-layered structure should be modeled by rib elements (3-node beams). Please see the figure below for model configuration.

Modeling the inter-layer connection

Case A (uniformly distributed connector modeling)

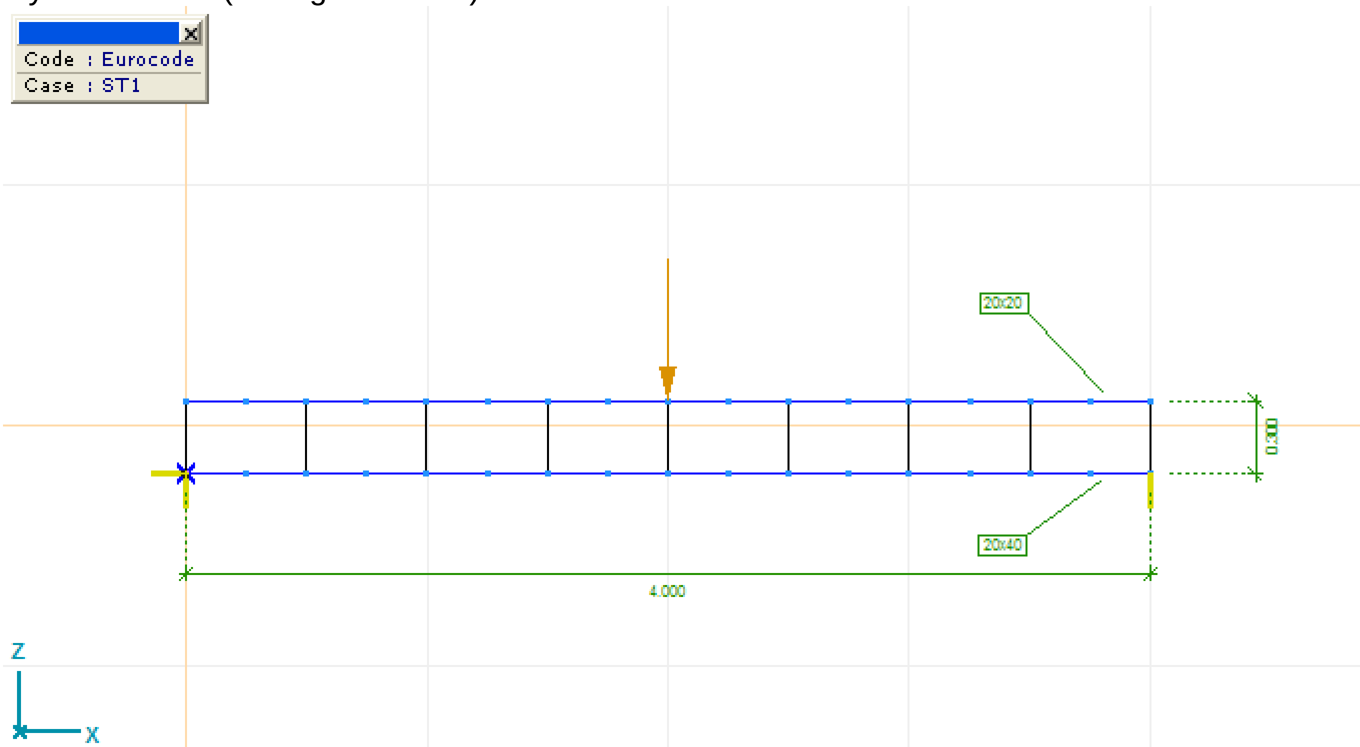
The L-L link elements have six stiffness components. In this example, by coincidence, the local system of the L-L links is parallel with the global coordinate system. In this case the horizontal, K_X, (slip) stiffness, is provided by the L-L link elements, uniformly distributed over the length, and its value is usually based on experimental investigations (slip tests).

The following stiffness components have to be defined:

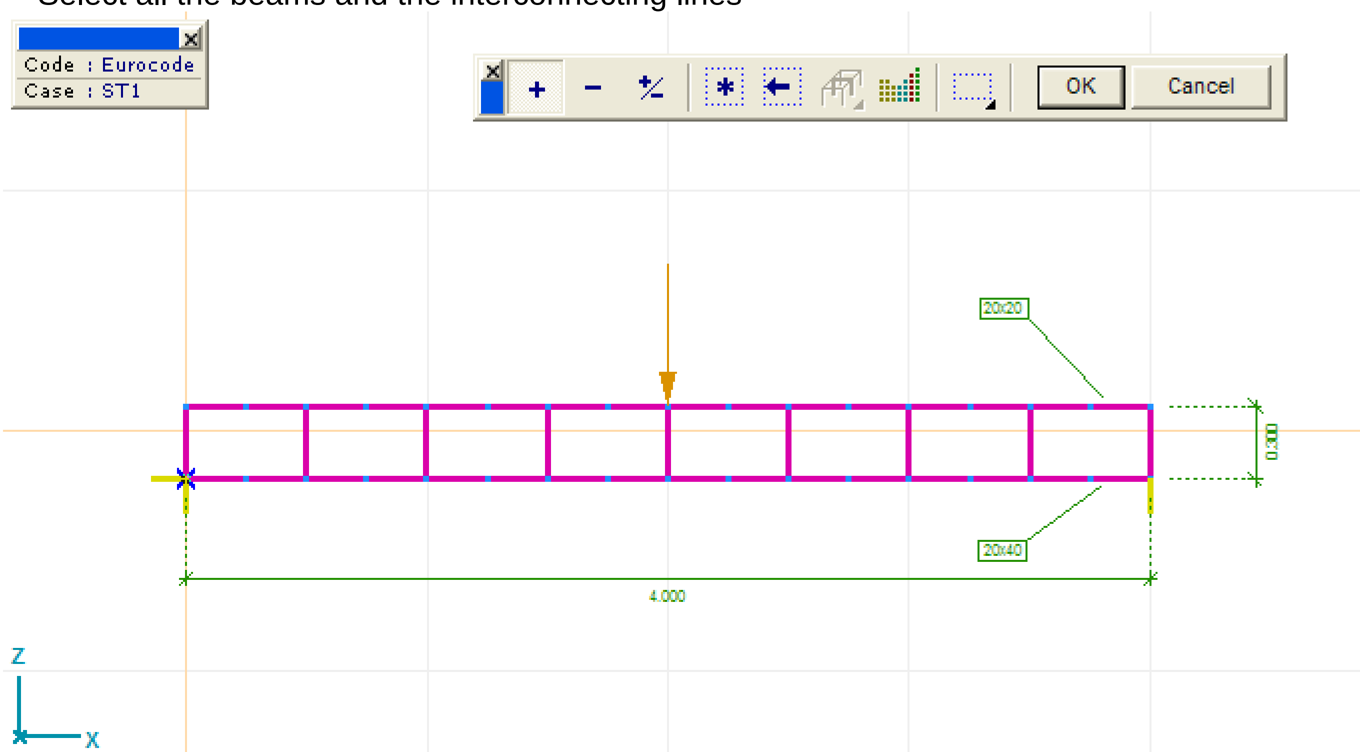
- K_X: in this case is the horizontal (slip) component.
  - A zero value would result in a non-composite behavior (not the case, since there are a number of connectors installed). A zero value will also introduce a rigid body mechanism, since the two layers will allow the two wood members to slide on each other. Therefore, to avoid such a singularity, a finite low value should be entered even in the cases when “non-composite” behavior is desired.
  - An infinite (very large) value would result in a fully composite behavior (can be close if there are “numerous” and “rigid” connectors).
- K_Z: in this case is the vertical component. The stiffness entered here should be a very large value.

The steps of defining the L-L link elements

a) Connect the corresponding end nodes of the rib elements of the two layer with lines (see figure above)

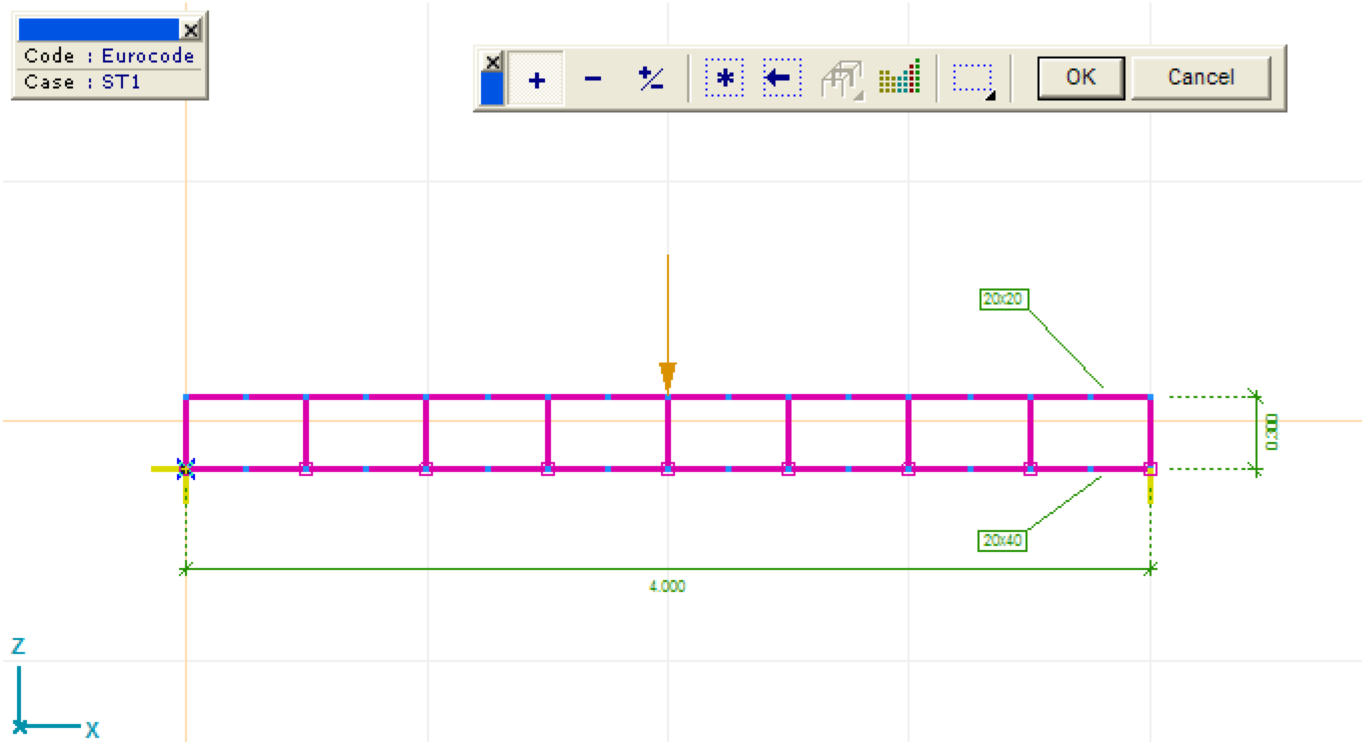


b) Select all the beams and the interconnecting lines

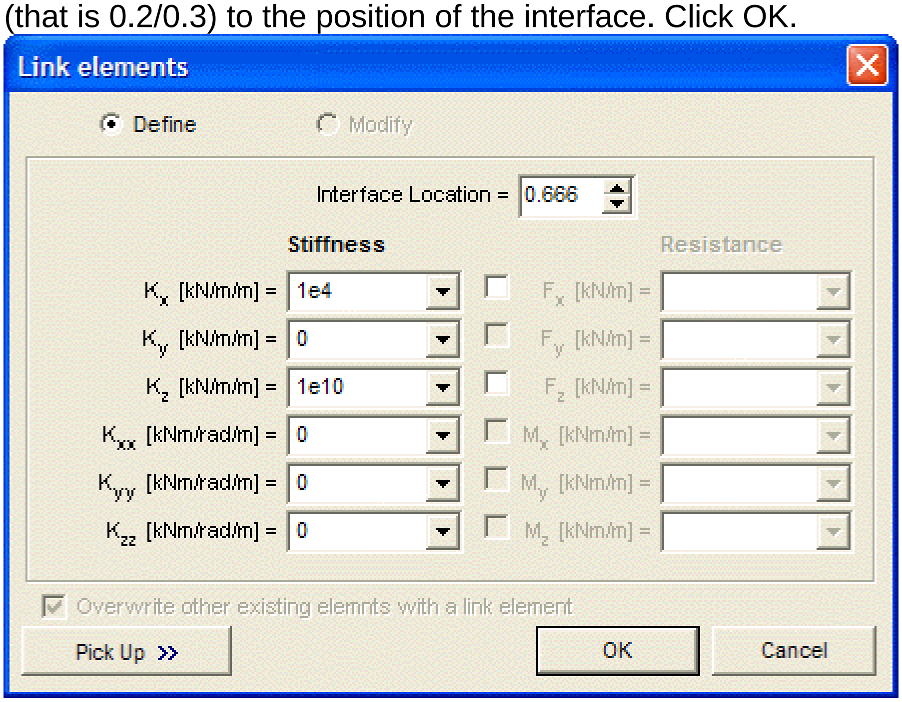


c) Click on small arrow in the bottom-right corner of the Links icon and select the L-L link icon.

d) Select all the nodes on the lower beam



e) Enter the following stiffness values K_X=1E4 and K_Z=1E10. Enter 0.666 (that is 0.2/0.3) to the position of the interface. Click OK.



f) Click OK.

Case B (discrete connector modeling)

The N-N and L-L link elements have six stiffness components. In this example, by coincidence, the local system of the L-L links is parallel with the global coordinate system. The local system of the N-N elements is chosen as parallel with the global system of axes. In this case the horizontal (slip) stiffness, is provided by the N-N link elements, and its value is usually based on experimental investigations (slip tests).

The following stiffness components have to be defined for the L-L links:

- K_Z: in this case is the vertical component. The stiffness entered here should be a very large value.

The following stiffness components have to be defined for the N-N links:

- K_X: in this case is the horizontal (slip) component.
  - A zero value would result in a non-composite behavior (not the case, since there are a number of connectors installed). A zero value will also introduce a rigid body mechanism, since the two layers will allow the two wood members to slide on each other. Therefore, to avoid such a singularity, a finite low value should be entered even in the cases when "non-composite" behavior is desired.
  - An infinite (very large) value would result in a fully composite behavior (can be close if there are "numerous" and "rigid" connectors).

The steps of defining the L-L link elements could be the following (similar as above except for step j):

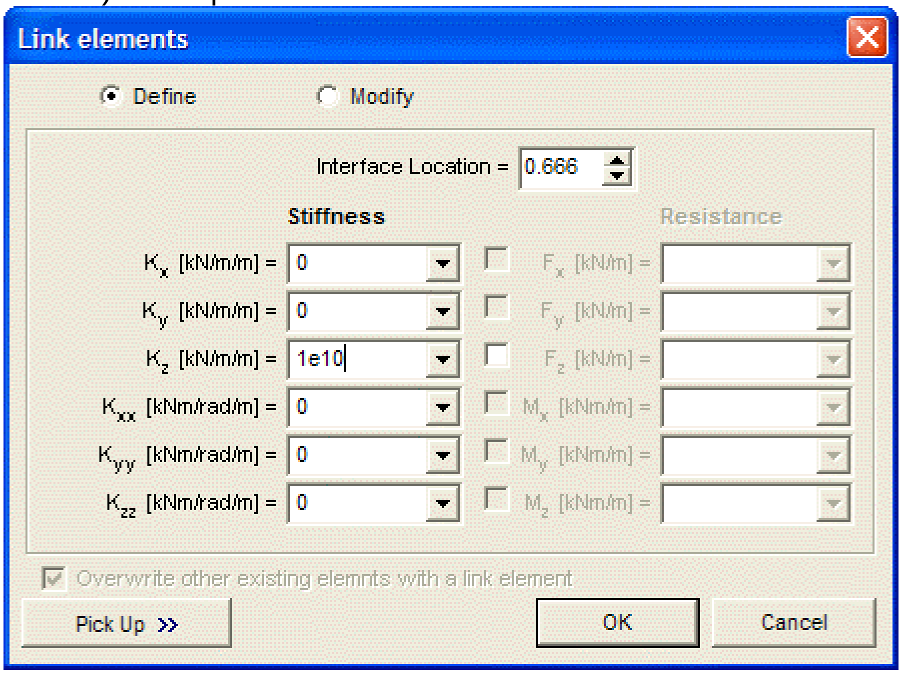
a) Connect the corresponding end nodes of the rib elements of the two layer with lines

b) Select all the beams and the interconnecting lines

c) Click on the small arrow in the bottom-right corner of the Links icon and select the L-L link icon.

d) Select all the nodes on the lower beam

e) Enter the following stiffness value K_Z=1E10. Enter 0.666 (that is 0.2/0.3) to the position of the interface. Click OK.



f) Click OK.

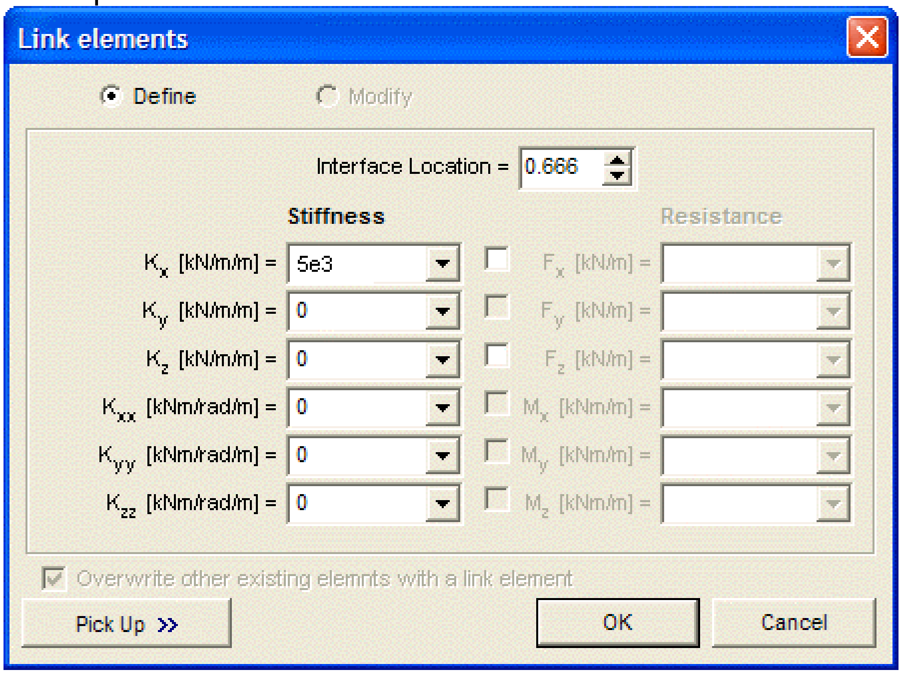
The steps of defining the N-N link elements could be the following:

g) Select the interconnecting lines where connectors are placed

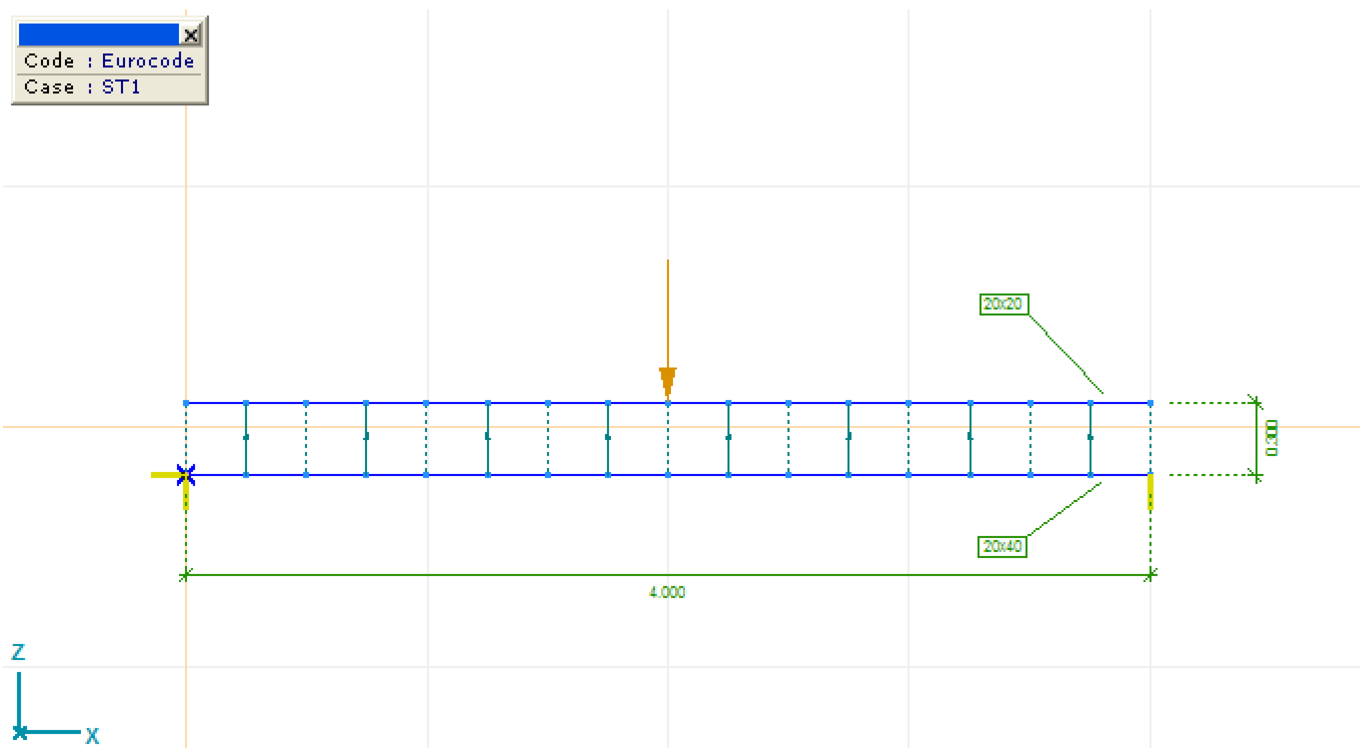
h) Click on the small arrow in the bottom-right corner of the Links icon and select the N-N link icon.

i) Select all the nodes on the lower beam

j) Enter the following stiffness value K_X=5E3. Enter 0.666 (that is 0.2/0.3) to the position of the interface. Click OK.



k) Click OK.



Conclusion

By varying the horizontal (slip) stiffness component of the corresponding type of link elements the non-composite, semi-composite, and full-composite multilayer beams can be studied.