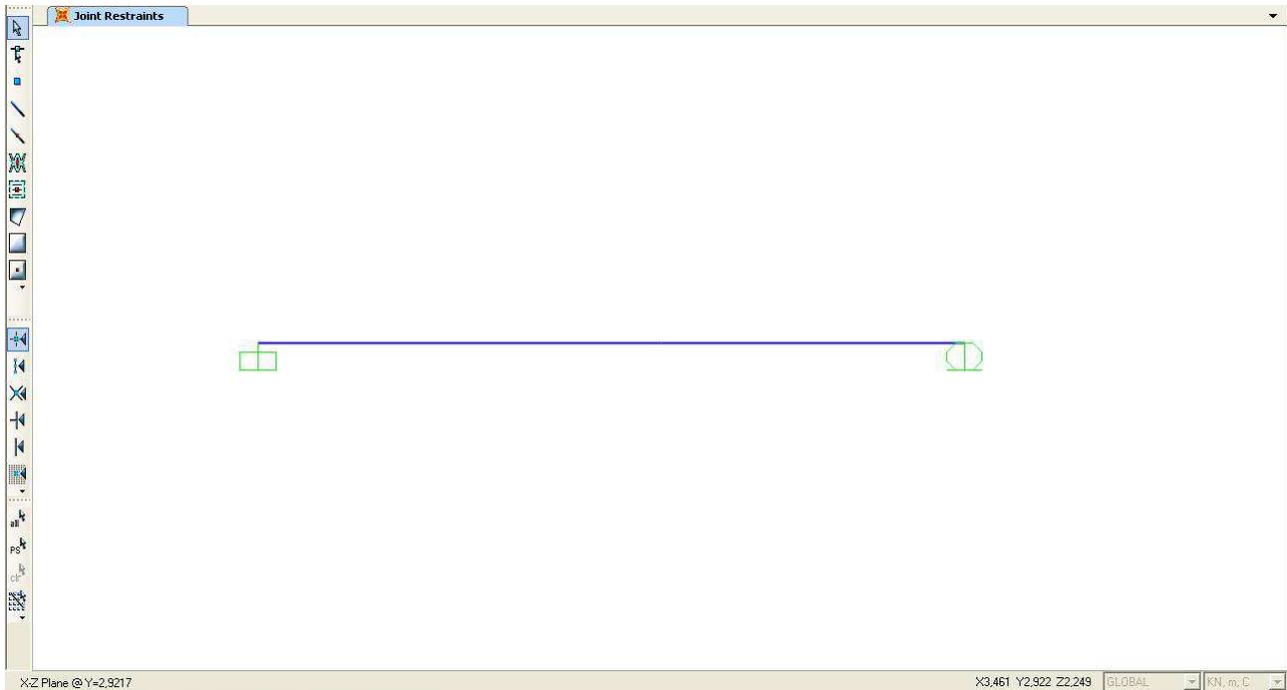


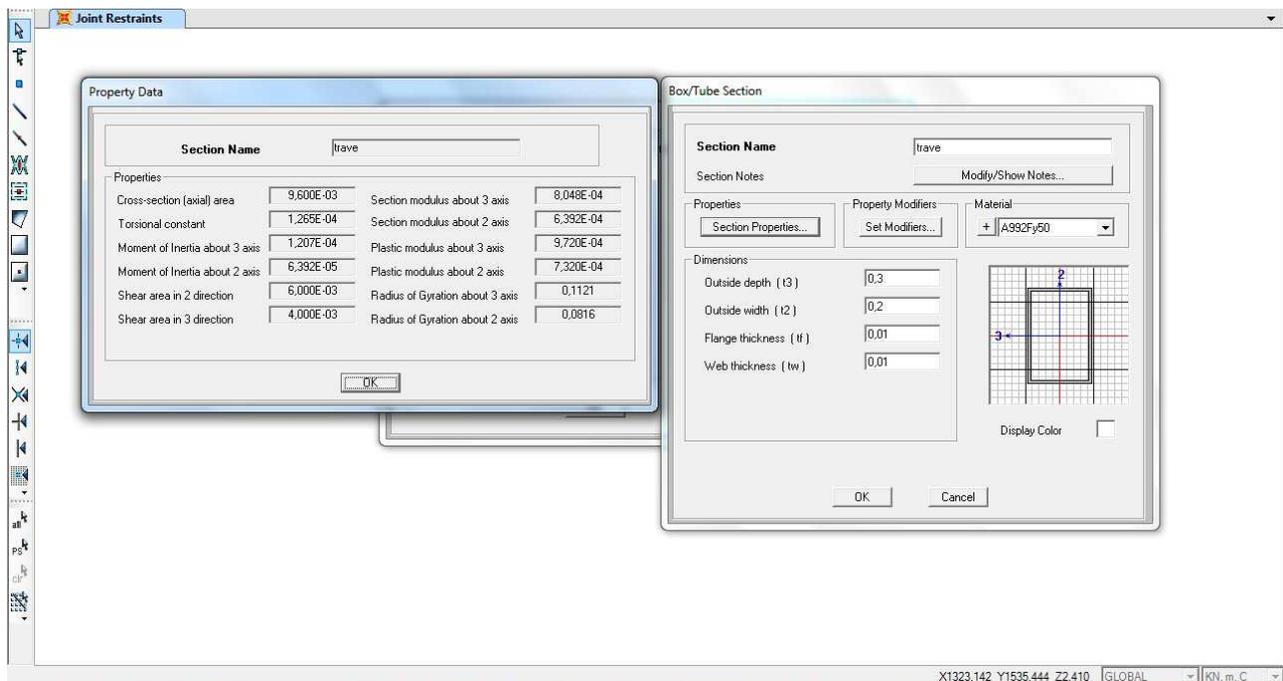
## ESERCITAZIONE 2\_STUTTURA IPERSTATICA

Verifica dei risultati della linea elastica su SAP.

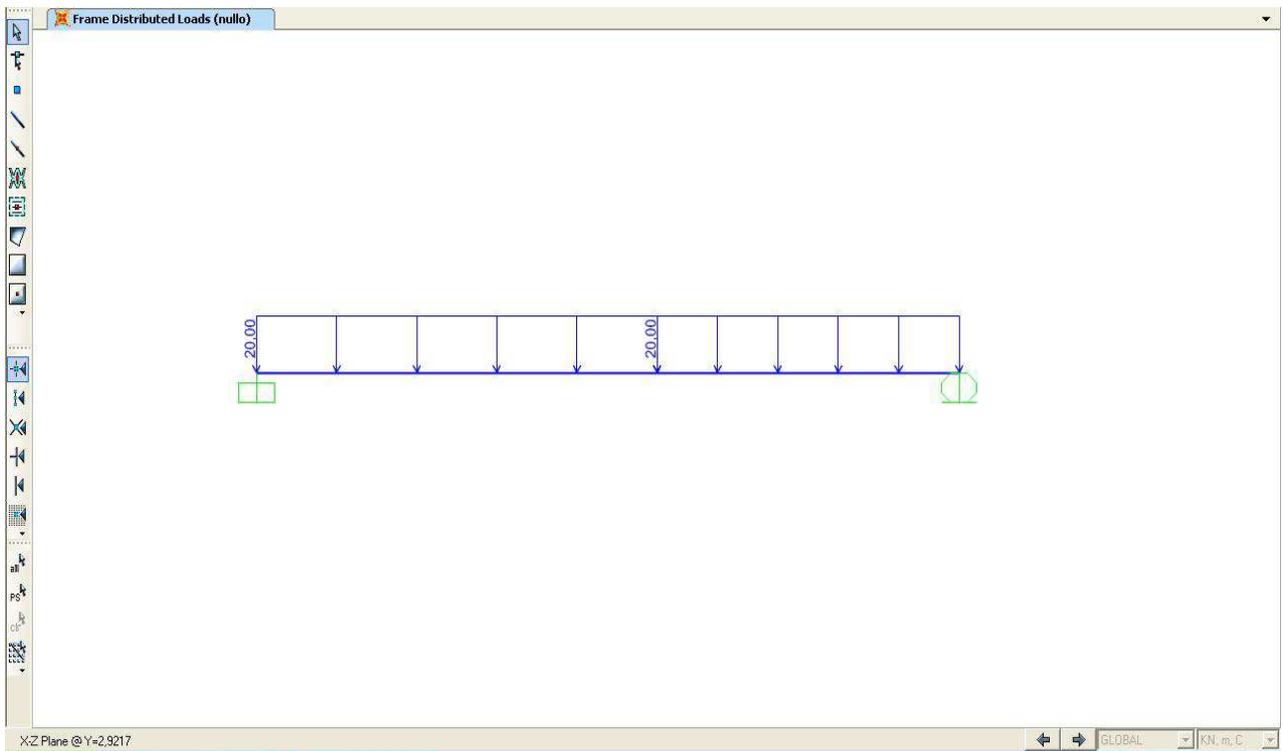
Dopo aver disegnato una trave lunga 5m, posiziono un punto a  $0,57L$  dove lo spostamento è max ed assegnare i vincoli.



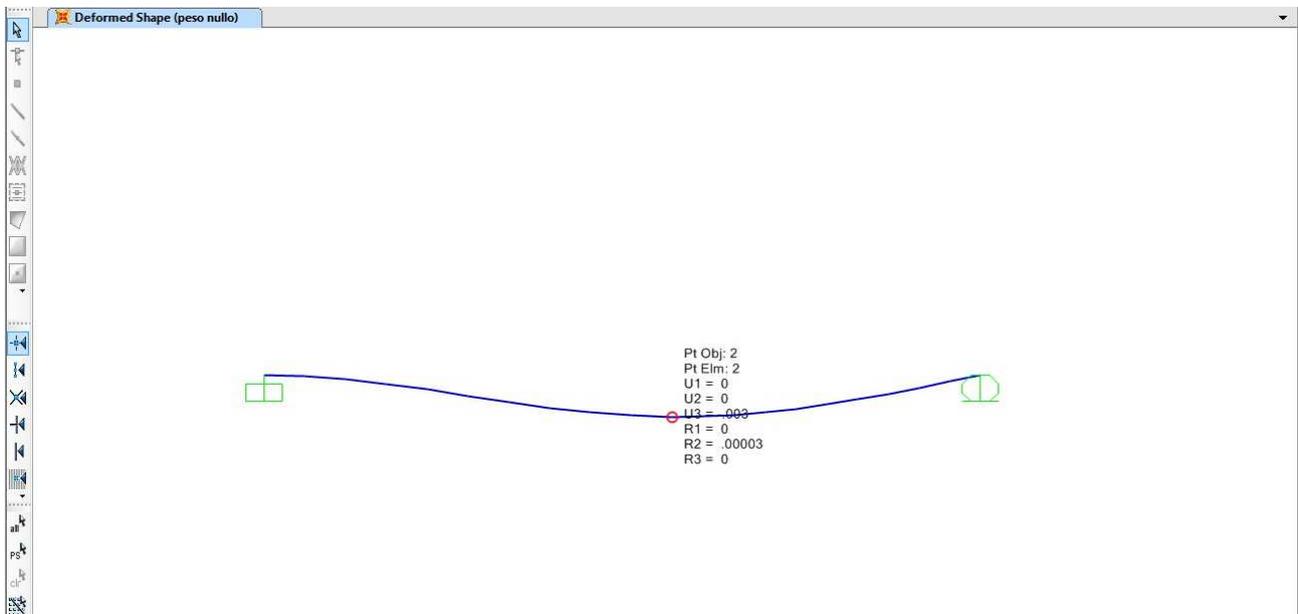
Dopo aver dato il carico nullo alla struttura, assegnare una sezione.



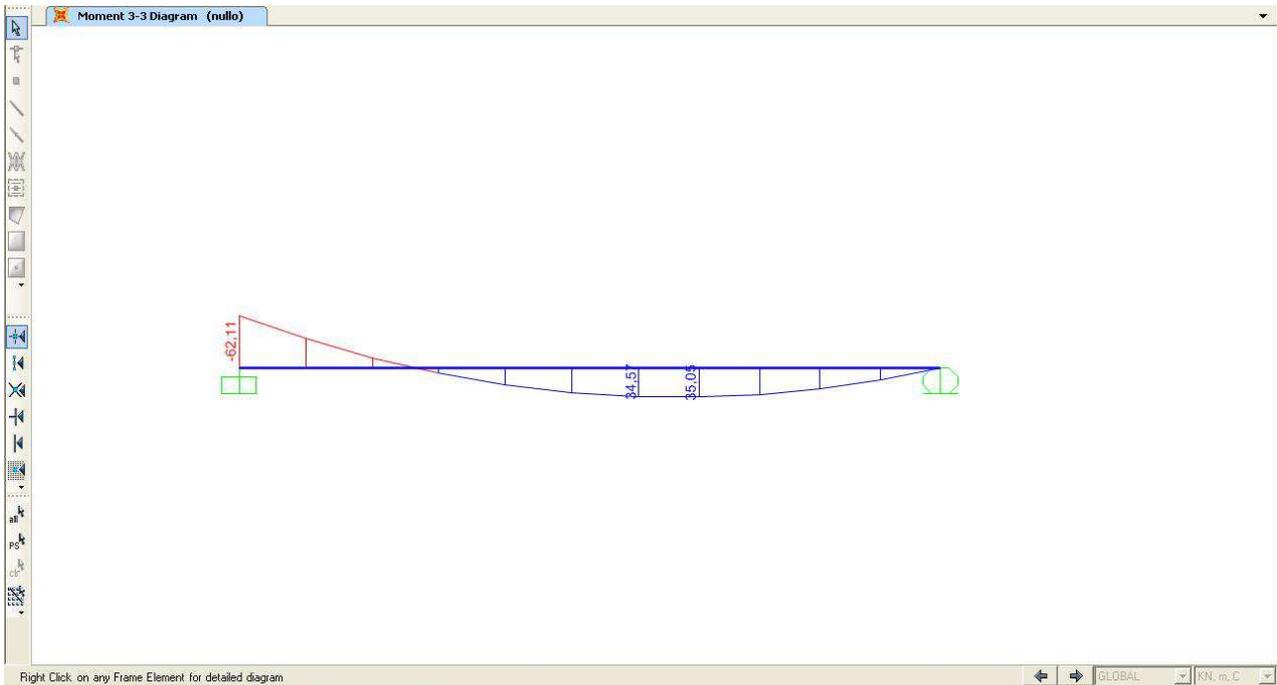
Assegnare il carico distribuito pari a 20 kn/m



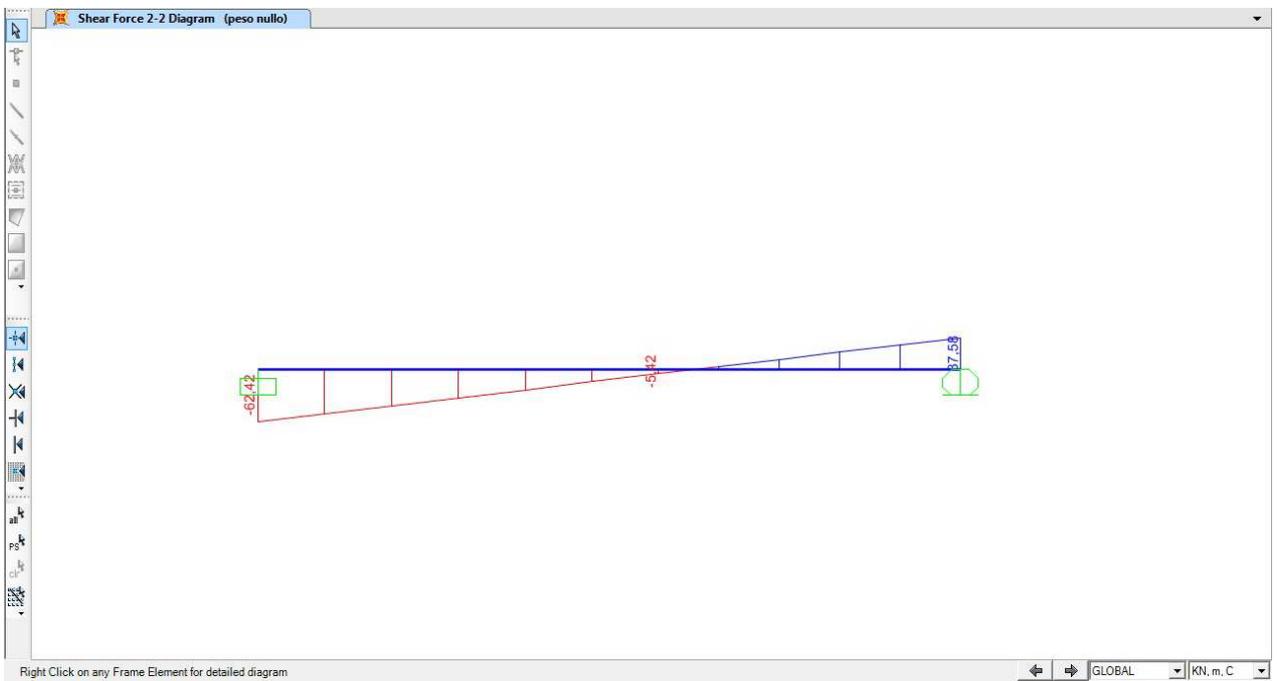
Lancio l'analisi della struttura e visualizzo la deformata



## Diagramma del momento



## Diagramma del taglio



Per vedere lo spostamento verticale guardo le tabelle (display > show tables > analysis results > joint displacement).

The screenshot shows a software window titled "Deformed Shape (peso nullo)". Inside, a "Joint Displacements" dialog box is open, displaying a table of results. The table has columns for Joint Text, Output Case Text, Case Type, and three displacement/rotation values (U1, U2, U3 in meters and R1, R2, R3 in Radians). The data for three joints is as follows:

Joint Text	Output Case Text	Case Type	U1 m	U2 m	U3 m	R1 Radians	R2 Radians	R3 Radians
1	peso nullo	LinStatic	0	0	0	0	0	0
2	peso nullo	LinStatic	0	0	-0.00296	0	0.000028	0
3	peso nullo	LinStatic	0	0	0	0	-0.002198	0

At the bottom of the dialog, it shows "Record: 1 of 3" and buttons for "Add Tables..." and "Done". The main window's status bar at the bottom left says "Ready" and the bottom right shows "Start Animation" and "GLOBAL KN, m, C".