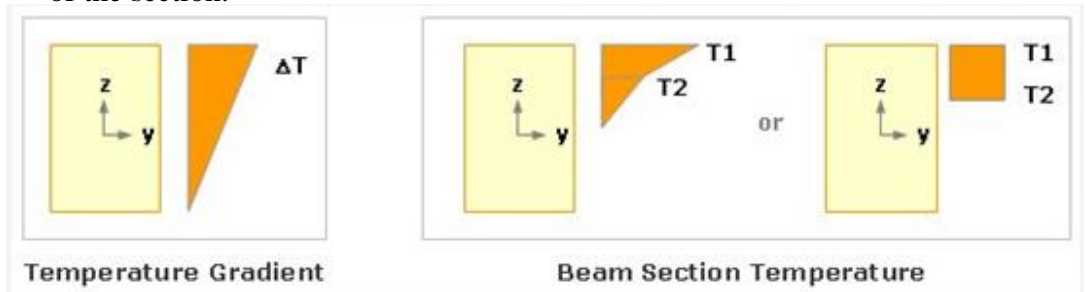




## Can you provide me with Thermal Analysis Guidance on Midas Civil?



In Midas Civil, temperature load can be assigned using  $\Delta T$ , which will make a linear temperature progression for “General” section, while “PSC/Composite” section can use different temperature up to 20 times on each direction (y and z) of the section.

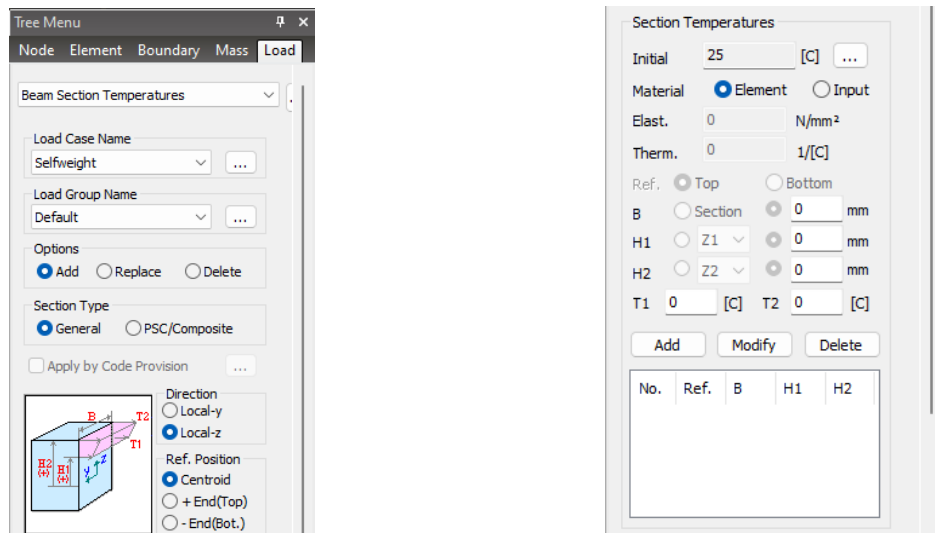


Temperature Gradient

Beam Section Temperature

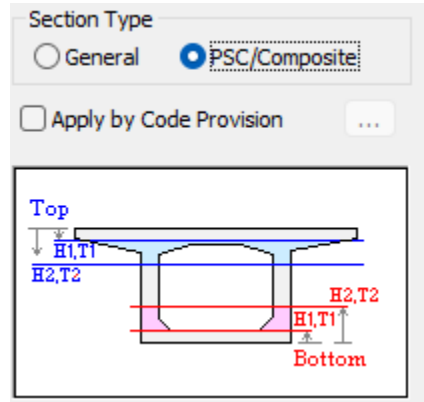
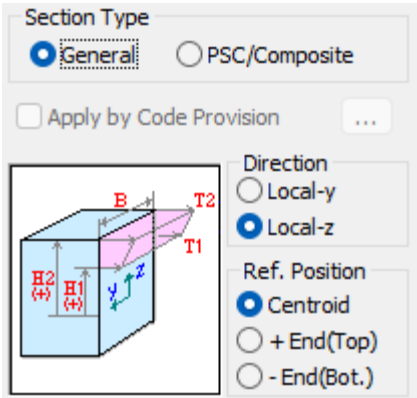
Assigning the temperature load require you to click Load > Temp/Prestress> Beam Section Temp.

After assigning your predetermined load case and load group (if any), If you have already assigned the material property, go ahead to click the "Element", otherwise click "input" if you want to specify certain elasticity modulus and thermal expansion coefficient determined.



You can choose direction (Top or bottom) of the reference point in which you will apply the B, H1, and H2 (you can set them manually on the right field, or set predetermined Shear Check position (Z1, Z2, and Z3) distance in Material Properties>Section), T1 and T2 for temperature value at H1 and h2 respectively, and click "Add".

Keep in mind that if you use PSC/ Composite section type, you must use “PSC section” as model, similar to if you are using “user-made” section you should use General Section type like seen below. Then you can determine the direction on y or z direction with the reference position of temperature load’s area.



After assigning the section, adding the temperature, selecting the element’s area to assign load, go ahead and apply. You can check the load tables, perform analysis, and get the results.

