

The first program that helps any kind of engineer in studying beams and Saint-Venant problems. The program calculates support pressures and stress graphics of many isostatic support solutions and any kind and combination of loads. There is moreover the opportunity to choose ideal or real beams and, for the last one, to choose material and section with their characteristics. A very easy interface will help you everywhere and will give you the chance to choose your solution and also to compare graphics with double display option and to save them.

Important: check to have copied all beams.zip files in the active folder!

Example:

Analyse a 20m iron fixed beams with a “double T” geometry and a concentrated load of 1000 N with an angle of 45° positioned at 15m from the left side (5 from the right side).

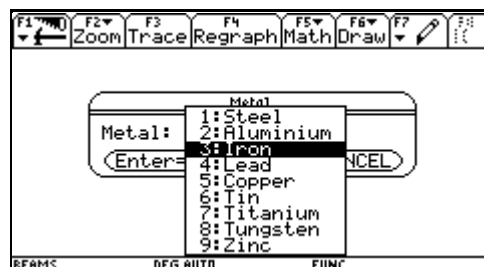
Enter Beams() and, when ready, press F1.

Enter 20 in length space and choose help both for metal and geometry:



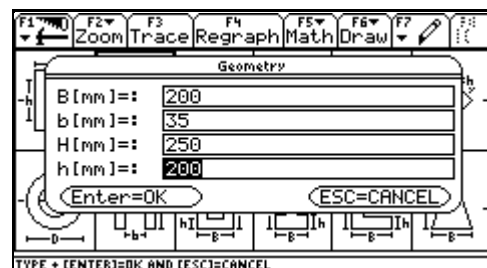
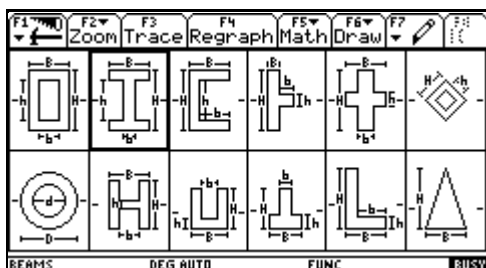
Enter

Select your metal (iron): the program automatically take the correct density value (you can modify it in a second time).



Enter

Now choose the geometry of the section your beam and enter its dimensions:



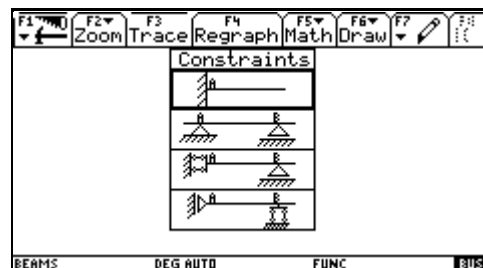
Enter

Now program data has automatically updated:

If you want to analyse an ideal beam just type 0 for Density and Section Area.

Enter

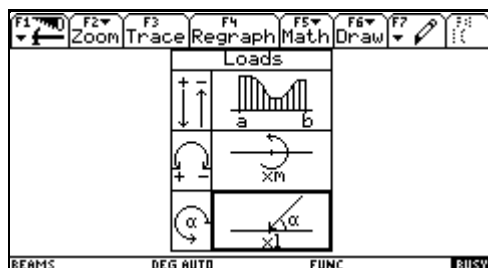
Press F2 to chose your supports:



Enter

Chose fixed side (for example the left side) and press Enter

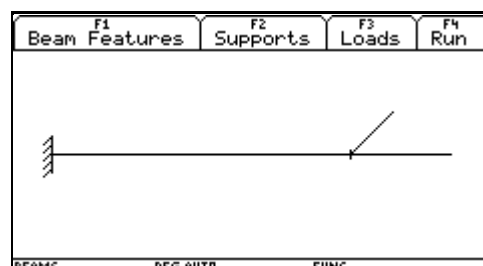
Press F3 to chose your load, select the lower window (concentrated loads) and enter load's data:



Remember that x_1 is from left side to right side of the beam. Press Enter

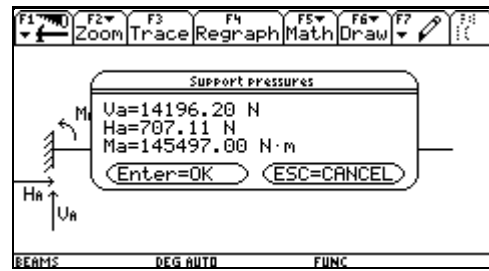
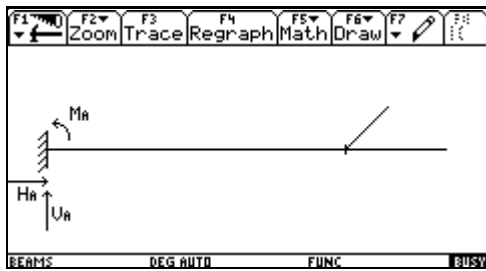
If you want you can use F3 another time to add how many loads you want.

Now your problem is completely defined: you can see the picture of the beam with supports and requested loads (remember that there is also the hided beams weight).

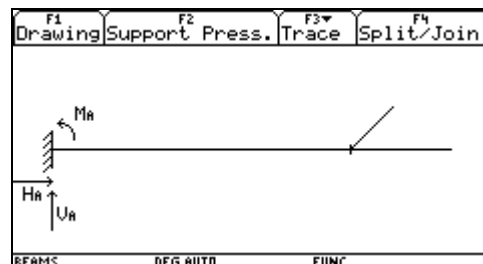


Press F4 to run the program.

The program shows first the forces in the supports and second their absolute values:



Press Enter to display the analysis home screen:



You can select:

F1: Redraw beam and forces after a graphic trace.

F2: Shows support absolute values.

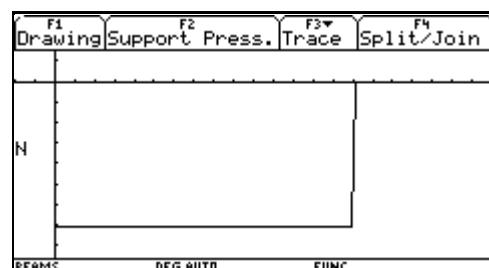
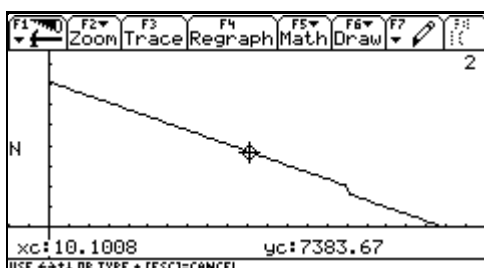
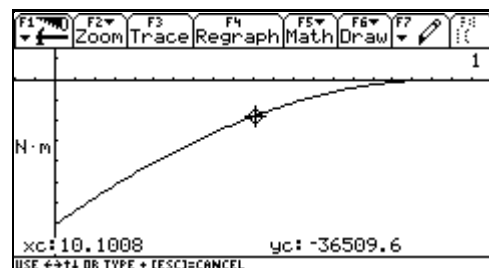
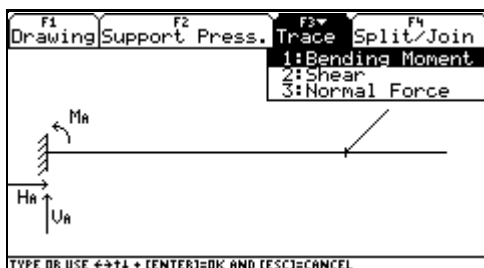
F3: Trace Bending Moment, Shear and Normal Force on the beam

F4: Allows to compare two graphics: Bending Moment & Shear, Shear & Normal Force etc.

Esc: Quit the program. You can chose if you want to save data or not.

Example: try to trace Bending Moment, Shear and Normal Force.

Press F3 and chose alternately 1, 2 and 3



This program has been already used many times without problems. If you find any bug first assure you to have selected the English language in the Mode and not to have translated the code with any program. If the problem persists, please, let me know.

For a better and faster answer, please, enclose some screenshot of the bug: entered inputs, expected outputs, error messages, erroneous code line, Mode setting... it will help me very much!

My address is paolosilingardi@interfree.it; write **TI-Program** as Object of e-mail!

**IN ORDER TO PREVENT SPAMMING, E-MAIL WITHOUT THE CORRECT OBJECT
WILL BE AUTOMATICALLY DELETED!**

You can find all my programs at this address:

<http://www.ticalc.org/archives/files/authors/44/4458.html>.

Remember to vote this program in the site!

Paolo Silingardi