Software Instructions

Structural Mechanics

Instructions for using BeamBoy

V2

| Created By:	Louis-Philippe Rouillard	Date:	2/12/13
Reviewed By:		Date:	

EIT ENGINEERING INSTITUTE OF TECHNOLOGY

Instruction
For using the software BeamBoy

1. Initial Setup

*To access the remote labs, use HTML5
*Electromeet HTML5 instructions are available in Moodle.*

- The software Beamboy is available on
  - Remote lab 1
  - Remote lab 2 and
  - Remote lab 4
- Open BeamBoy
  - The icon is located on the desktop
- The following window appears:
  - Make sure you select Metric units
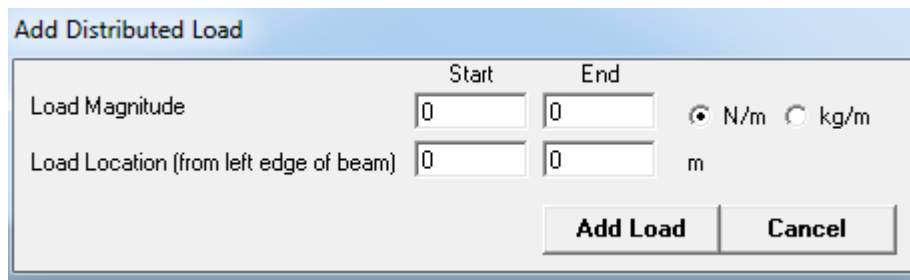 - Enter the Beam length
  - Click continue to confirm
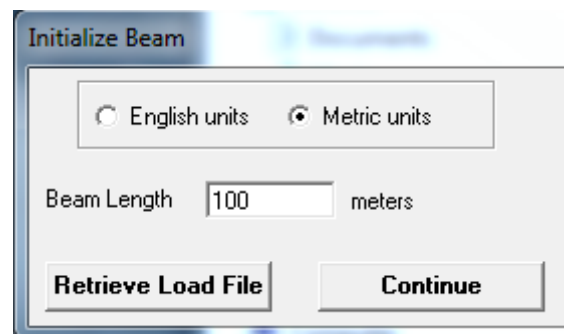
Note: if you make an error in any and you want to edit the beam length or the units, select "**Beam**" then "**Reinitialize**" and follow the steps to edit any entry.

2. Designing the beam

- To add support,
  - Press **Ctrl+F4** or select '**Feature**', '**Add Feature'** then '**Add Support'**
  - Select the type of support (single or cantilever support)
  - Enter the location of the support
  - then press "**Add Support**"
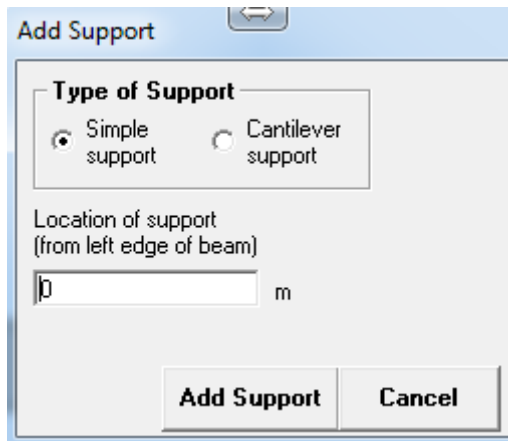
- To place a distributed
  - Press **Ctrl+F2** or select '**Feature**', '**Add Feature'** then '**Add Dist. Load'**
  - Enter the load magnitude and select the desire units
  - Enter the location of the load
  - Select "**Add Load**" to confirm

- Place a point load
  - Press **Ctrl+F1** or select '**Feature**', '**Add Feature'** then' **Add Point Load'**
  - Enter the load magnitude and select the desire units
  - Enter the location of the load
  - Select "**Add Load**" to confirm
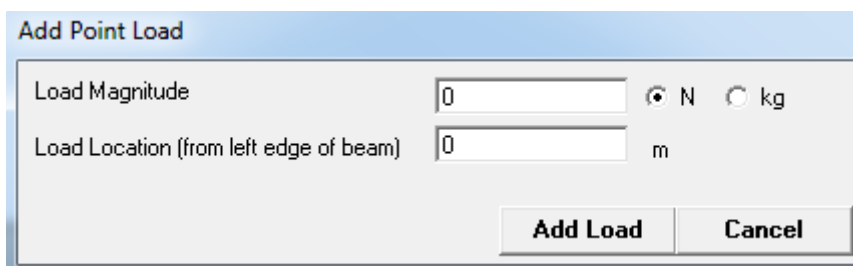
- To choose a cross section for the beam,
  - select "**Beam**" from the menu then "**Properties**"
  - Press "**Choose**"
  - Select the suitable cross--section
  - Press "**Use X-X Values**"
  - Press "**Continue**" to confirm

Note: if you make an error in any of the following entries, you can edit them by selecting **"Feature"** menu and then **"Edit Feature"** and follow the steps to edit an entry.

3. Calculation

- To calculate
    - o Press **Ctrl+R** to calculate or select **"Calculate"** then "**Calculate Stress and Deflection"**
    - o This will generate the calculations screen and the moment, bending stress and deflection graphs will appear.