

M_{cr} Calculation software

David Brown and Ibrahim Fahdah of the SCI introduce the new design tool on www.steelconstruction.info

Most designers working in accordance with the Eurocodes will have met the elastic critical buckling moment, M_{cr} . This moment is needed as an essential step on the way to calculate the lateral torsional buckling (LTB) resistance of a member. The non-dimensional slenderness $\bar{\lambda}_{LT}$, which is needed to calculate the LTB resistance is given by:

$$\bar{\lambda}_{LT} = \sqrt{\frac{W_y f_y}{M_{cr}}}$$

M_{cr} can be determined by calculation, using the following expression:

$$M_{cr} = C_1 \frac{\pi^2 E I_z}{L^2} \sqrt{\frac{I_w}{I_z} + \frac{L^2 G I_t}{\pi^2 E I_z}}$$

This expression is straightforward to use, but is limited to uniform members with “fork end” supports and loads assumed to be applied at the shear centre. Other more involved expressions are available that deal with loads not applied at the shear centre (stabilising or destabilising loads) and different end conditions. The only real uncertainty is the value of C_1 , which depends on the shape of the bending moment diagram. For linear and non-linear bending moment diagrams, expressions are available to calculate C_1 ^[1].

Many real design situations become complex. An irregular loading pattern may make the bending moment diagram quite unorthodox. Some loads may be applied at the shear centre, whilst others on the same member may be applied elsewhere within (or outside) the section depth. Restraints may be intermittent, to one flange, or the other, or outside the section depth, at different locations along the member. The numerical expressions for M_{cr} are unable to deal adequately with these types of situations and software must be used.

Previous articles in New Steel Construction have discussed the use of software^[2,3] and a design tool to calculate M_{cr} has been available on steelconstruction.info for a considerable period. The design tool on steelconstruction.info has been replaced with a new software, substantially increasing the scope, features and output; this article offers advice on how the software should be used.

Interface

Users will note that the interface looks markedly different from other design tools available on the steelconstruction.info website. The new software has a wider scope, with many opportunities for the user to define sections, beams, loading and restraints, which demands a flexible and comprehensive interface (Figure 1). In general, users work from one tab to the next, saving data at each step. The saving of data is important, as it triggers a refresh of any graphic, a re-ordering of any tabulated data and makes subsequent tabs available.

Although the software has been developed to assess beams, it may also be used to analyse steel members subjected to bending and axial loading.

Input Data

The first step when using the software is to create a project, which is a collection of one or more beams. Material types and cross sections are

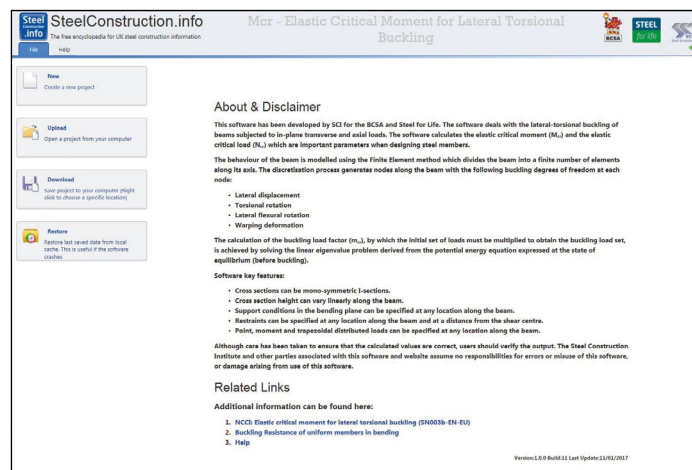


Figure 1: M_{cr} software interface

defined for the project, which are then used when beams are added to the project. Because the software allows a number of different beam arrangements (sections, length, loading, restraints etc) to be defined and analysed at one time, perhaps grouped because they are all part of one single project, a number of sections might be selected at this stage.

The “Section Definitions” tab, shown in Figure 2, allow users to add + modify ✎ and delete ✖ cross sections. The software allows users to add standard sections (UB and UC) or non-standard sections defined by their dimensions or their properties.

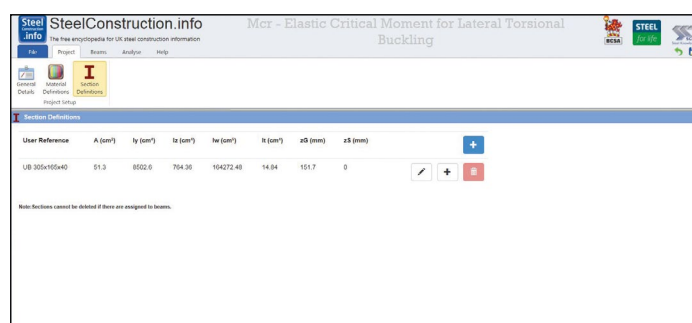


Figure 2: Sections Definitions – initial view

Users then visit the “Beams” tab (Figure 3) to work with the beams in the project. Beams may be added, duplicated or deleted. The currently selected beam is indicated in the drop down box at the top left corner of the tab. By default the remaining input tabs for a beam are greyed out. These tabs will be enabled as the user visits them working from the left to the right.



Figure 3: Beams input tabs

[1] Calculating the C_1 factor for lateral torsional buckling. NSC Nov/Dev 2013
 [2] Getting the best out of LTBeam, NSC, May 2009
 [3] Use of LTbeamN, NSC, January 2015

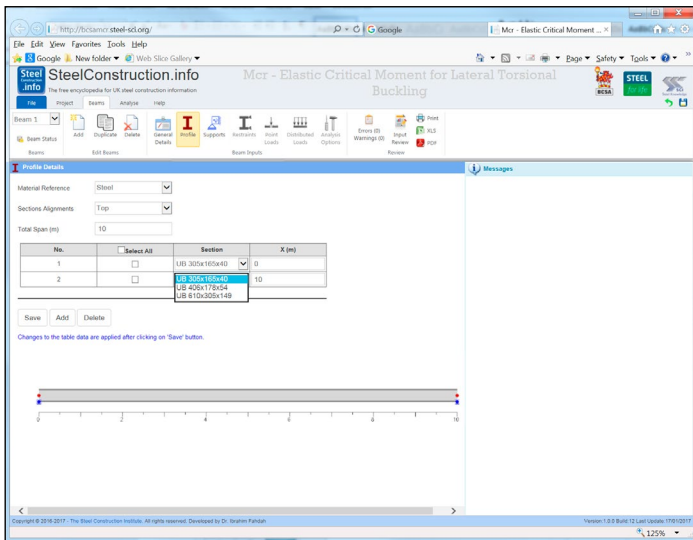


Figure 4: Profile input

Figure 4 shows the “Profile” input. This screen shows the length of beam (the default is 10 m) and invites the user to define the cross-section at each end of the beam. Only the cross-sections previously defined are available. The data should be saved, which will update the graphics. In general, the cross-section will be identical at each end of the beam. The software allows a beam to be made up a number of different sections, using the “add” option. This may be used if part of the beam is tapered and part has a uniform cross-section. Tapered beams must be defined by dimensions, so the cross-section at each end of the section must be defined and selected on this “Profile” input. Only the depth of the beam can be varied on a tapered beam. The use of cross-sections defined by dimensions and a beam comprised of two sections allows the user to investigate beams which are tapered over part of their length, as shown in Figure 5.

Supports may now be selected, with default locations at each end of the beam. By default, both supports (shown in blue) are pinned and fixed vertically, one end is fixed laterally. These defaults can be changed by the user, and intermediate supports added if required.

Restraints may now be added. The default restraint condition is of “fork

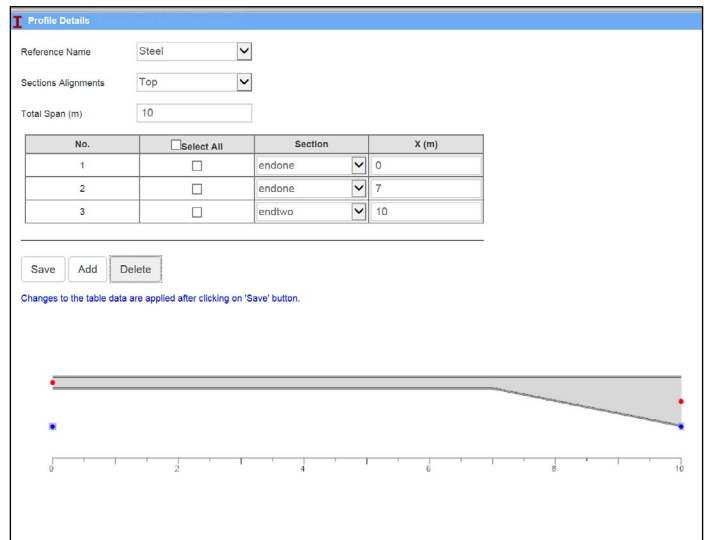


Figure 5: Part tapered beam

ends” at each end of the beam, which is laterally fixed, warping free and the restraints are located at the shear centre. All of these defaults may be changed and additional restraints may be specified. Normally, restraints might be discreet ‘point’ restraints, but the user can select a length of continuous restraint if required. Restraints may be specified at any point with respect to the top of the section, bottom or shear centre. Figure 6 shows a beam with intermediate restraints (red dots), one above the top flange, and one on the top flange. Note that saving the data is always required to refresh the graphics.

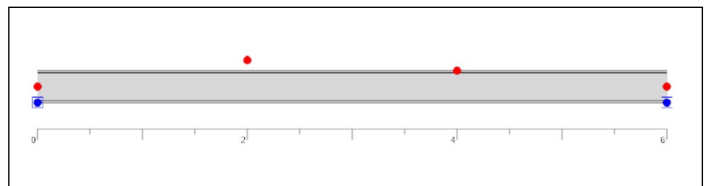


Figure 6: Illustration of restraint options

Nationwide delivery of all Structural Steel Sections

RAINHAM



Phone: 01708 522311 Fax: 01708 559024

MULTI PRODUCTS ARRIVE ON ONE VEHICLE

Having amended (or at least viewed) and saved the restraint data, loading data is now available. The default inputs are empty, so loads are introduced by the “add” button. Point loads, point moments, axial loads, distributed loads and varying loads may all be added, with the position of load application anywhere within (or outside) the section depth. This allows users to deal readily with destabilising (above the shear centre) and stabilising (below the shear centre) loads.

Figure 7 shows some of the possibilities available to the designer when defining loads.

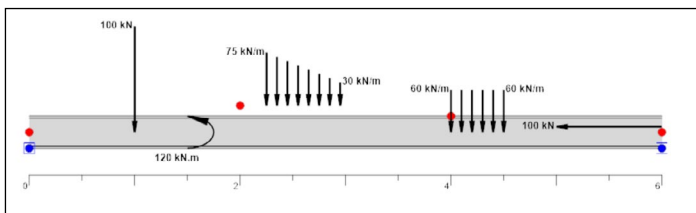


Figure 7: Load arrangements

The last view on the beam inputs group is “Analysis Options” view. The analysis options of a beam allow the user to vary the number of the finite elements used to represent the beam – more elements may improve the accuracy of the result marginally. The default of 100 elements is recommended.

The options to ignore bending effects and ignore axial effects demand some explanation. When verifying beams using expressions 6.61 and 6.62 of BS EN 1993-1-1, the lateral torsional buckling resistance is determined assuming there is no axial load, and the axial resistance is determined assuming there is no bending; the interaction is accounted for within the expressions. The opportunity to ignore one type of load allows the user to readily determine the values of M_{cr} and N_{cr} in the absence of axial force and bending respectively. The values of M_{cr} and N_{cr} will be different if calculated with both types of load applied simultaneously.

One warning

It may be very helpful to use this software to determine N_{cr} and thus the buckling resistance of a beam under axial load – particularly if the restraints are to one flange only, or perhaps to both flanges but at different locations along the beam. This version of the software only

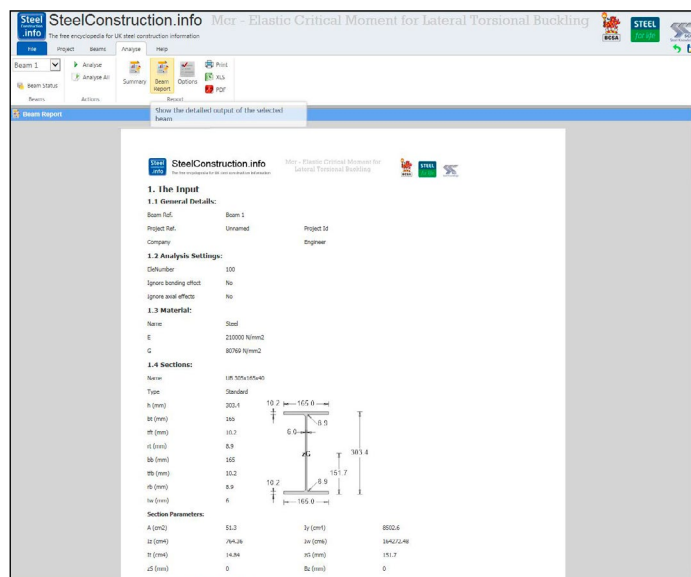


Figure 8: Detailed beam report

reports the value of N_{cr} in the minor axis direction. It would be possible to restrain the beam in the minor axis so effectively that major axis flexural buckling becomes the critical behaviour – which is not investigated by the software.

Finally, the analysis and output!

The “Analyse” tab allows the user to move to the analysis stage. Users can carry out the analysis and review the analysis results for the currently selected beam, or all beams in the project. The default report is a simple summary of M_{cr} and N_{cr} values, but much more detail can be obtained by selecting the “Beam Report” (Figure 8). The reports can be exported to pdf, or Excel, or printed. The reported “buckling factor” is the amplifier by which the actual loads must be multiplied to obtain the elastic critical buckling moment. As a trivial example, a beam of 6 m span and 20 kN/m UDL results in a midspan bending moment of 90 kNm. If the value of M_{cr} is 109.9 kNm, the buckling factor is $109.9/90 = 1.221$. A buckling factor less than 1 indicates an immediate problem – a larger beam is required.

GRADES S355JR/J0/J2

STEEL

Head Office: 01708 522311 Fax: 01708 559024 Bury Office: 01617 962889 Fax: 01617 962921
email: sales@rainhamsteel.co.uk www.rainhamsteel.co.uk

Beams • Columns
Channel • Angle
Flats • Uni Flats
Saw Cutting
Shot Blasting
Painting • Drilling
Hot & Cold Structural
Hollow Sections

RAINHAM STEEL Proud sponsors of BOXNATION Channel of Champions