

Introduction

This program calculates capacity loads and performs code checks to the Working Stress versions of the AISC and API codes.

The editions of the design codes used are:-

API RP 2A WSD 22nd Ed 2014.

ASTM A36 9th Ed 1989.

The section shapes accepted are symmetrical I, channel, RHS and CHS sections. Of these, tubular sections are checked to API, all others are checked to AISC.

Units

The units used are :-

kN for force or load

kN-m for bending moment

MPa (N/mm²) for stress

mm for dimensions, displacement

Conversion factors to US units are;

Force 1 kip = 4.45 kN

1 kN = 0.225 kip

Moment 1 kip-in = 0.113 kNm

1 kNm = 8.85 kip-in

Stress 1 ksi = 6.895 MPa

1 MPa = 0.145 ksi

Length 1 inch = 25.4 mm

1 mm = 0.039 inch

The codes use American nomenclature in which the elastic section modulus is $S_{x,y}$ and the plastic modulus is $Z_{x,y}$. This application has been written for engineers more used to British codes, in which the elastic section modulus is designated $Z_{x,y}$ and the plastic modulus is $S_{x,y}$. The British nomenclature is used however the terms 'elastic' and 'plastic' are included for clarity.

Error messages

Should incorrect data be entered on the main screen then an error message will appear and the program will not recalculate.

To clear this message simply correct the input data, the message will disappear and recalculation will proceed.

Warning messages

Warning messages are given for some conditions (e.g. high l/r values).

They do not prevent recalculation.

Beam analysis

A simple 2-D frame analysis is available to analyse a single span beam, for various end conditions.

To access this, click on the 'Analyse a beam' button at the top of the section where the applied loads are entered.

The beam is internally subdivided into 100 elements and the moments, displacements etc. are reported at the ends of these elements.

Entering Data

The program displays a number of screens, however the first screen is the main one and it is on this screen that most data are entered.

Enter data in the input boxes. Some already contain preset default data (e.g. Cmx etc.) however these can be overwritten. Leaving a box blank is the same as entering zero.

To enter your own section, (eg plate girder etc), click on the 'Input section data directly' button beneath the section selection box.

The 'View more results' button will show another screen which contains more details of the section properties and stresses etc.

The code check equations are only activated after (any) applied loads are entered.

Definitions

Grade

Once the steel grade is selected (default is grade 43) the program uses the provisions of BS4360 to set the yield stress (F_y) for all subsequent calculations.

The selection is based on the thickness of the flange elements for bending about the x-x axis, except for the case of RHS sections, bent about their y-y axis, in which case the web, (which is now a flange) thickness is used.

F_y

Whilst the yield stress is automatically selected (see Grade, above) this selection can be overwritten by entering a value for F_y in the F_y input box.

The limits for user defined F_y are $1 < F_y < 1000$ MPa.

It is possible that at high F_y values (possibly set while editing F_y input), the selected section might be classified as slender for x-x bending, thus triggering an error message, which sets an internal flag and stops further processing.

To clear this flag, either change F_y or select any section or 'click' the 'Input section data directly' button.

C_b (AISC only, Chapter F1.)

Modifier for allowable stress. Default value is 1.

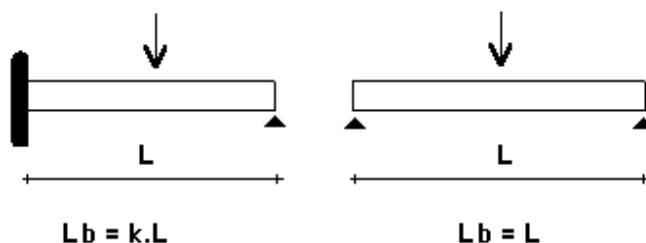
The code sets an upper limit of 2.3.

C_{mx} , C_{my} (AISC Chapter C1. API Section 3.3.1d)

Modifiers for the beam and column (x-x, y-y) bending moments and have default values of 0.85.

L_b beam (AISC Chapter F1.)

The actual unbraced length of the compression flange (of I or [section). It is also used as the effective torsional buckling length ($K_z.L$) in chapter E4.(b).



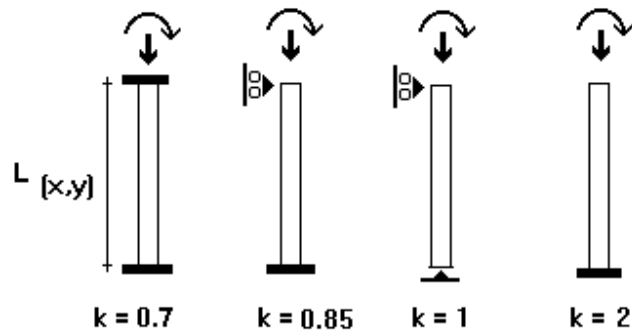
L_e for column buckling

The effective length of columns, $K_{x,y}.L$, used for calculating column buckling stresses ($I_e/r(x,y)$ etc.).

It is the actual buckling length, and includes any effective length factors.

The figure below shows some typical column effective length factors

$$L_e [x,y] = k \cdot L [x,y]$$



Allowable stress factor (API Section 3.1.2)

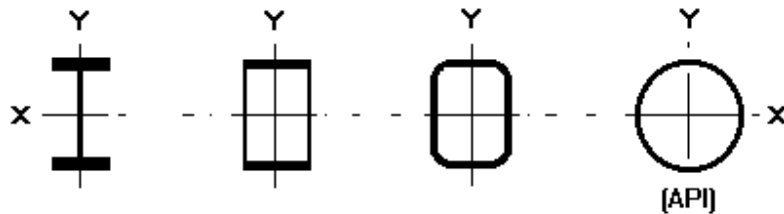
The allowable stresses (bending, axial, shear) are multiplied by this factor. Normally it is 1.0, but for some short term loads (e.g. wind) the allowable stress is often increased and this is catered for by increasing this factor, usually to 1.33.

The default (and minimum permitted) value is 1.0.

Code check failure is recorded when the combined stress ratio (often referred to as UF or IR) is > 1.0.

Bending moments

The bending axis x-x is always assumed to be the major axis.



Moment x-x is the moment about the major axis.

Moment y-y is the moment about the minor axis.

For tubulars, the applied moments can be entered individually as M_x , M_y , the vector summation is accounted for in the code check equations.

Shear Force

Shear(y) is the shear load applied in the direction of the minor axis, i.e. in direction of web.

For tubulars shear(y) should be entered as the vector sum of the applied (x & y direction) shears.

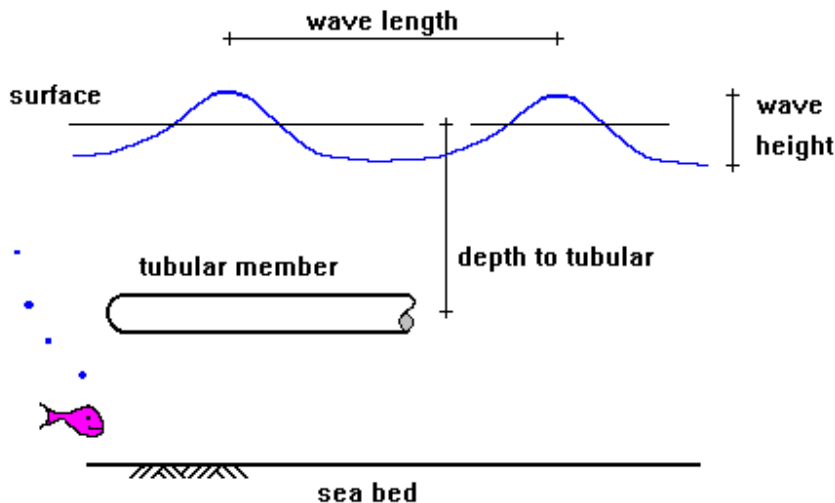
Load capacities

The safe loads shown are the maximum individual loads or bending moments that the member can carry using the grade and effective length values entered.

API calculation assumptions

Hydrostatic loading

The general dimensions for hydrostatic loading are given below.



Applied limit state loads

The applied loads are the factored column axial load and beam bending moment from general frame action (if any) that will act on the tubular in conjunction with the hydrostatic loading. It is not the local moment caused by the hydrostatic load.

The stresses from this loading are calculated as

$f_a = \text{Axial load} / \text{area}$

$f_b = \text{Moment} / Z$

The moment is the vector sum of M_x and M_y .

Member axial buckling length

The effective length used for calculating column buckling strength ($l/r(x,y)$ etc.).

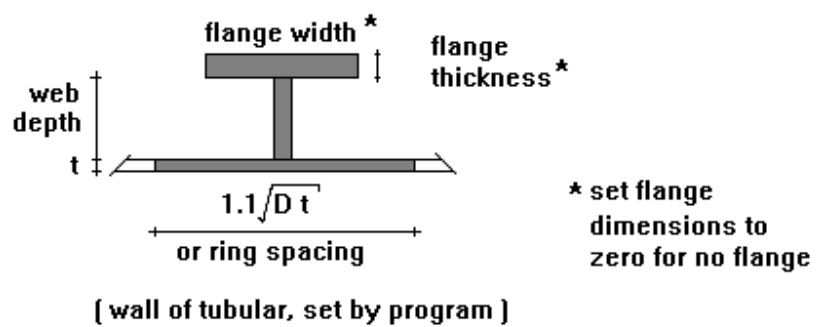
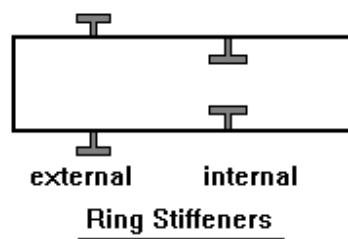
It is the actual buckling length, and includes any effective length factors.

Enter the column length appropriate for the applied axial and moment loads.

Ring dimensions

The program assumes that the tubular wall will act as a flange of breadth $1.1(D.t_w)^{0.5}$ and thickness t_w . This is the inner or outer flange according to the ring type selected (internal or external). The flange dimensions asked for are those of the fabricated ring. For a ring stiffener with no flange, simply leave the flange dimension input boxes blank or enter zero.

The ring dimensions are shown below.



Wave length

The wave length ,L, for Airey waves is given by the general formula

$$L = \left\{ \frac{g T^2}{2\pi} \right\} \cdot \tanh\left(\frac{2\pi d}{L} \right)$$

This is normally solved from tables, but in the absence of tables the following approximations can be used.

Shallow water	$d / (g T^2) \leq 0.0025$
$L = T (g d)^{0.5}$	

Deep water	$d / (g T^2) > 0.08$
$L = g T^2 / (2\pi)$	

Where

g = gravitational acceleration (9.81m/s/s)

T = wave period (seconds)

pi= 3.1416

d = water depth (m)

Section data file

The section data files are ASCII files which contains the section dimensions and properties.

These files should be in the same directory as this application's .exe file, and are :-

SECT_UK.DAT	British sections
SECT_NOR.DAT	British Steel Nordic range of tubulars
SECT_US.DAT	larger US W sections
SECT_EU.DAT	European sections
SECT_NZ.DAT	NZ and Australian sections

The default file is usually either SECT_UK.DAT or SECT_US.DAT . If you select "Other" from the section selection menu or should the program not find the selected file then an 'open file' dialogue box will appear from which you should select a file to be used.

The default files should not be amended. If you wish to add sections, or create a new data file, then take a copy of a sections file (to another name of your choosing) and then edit the new file using an ASCII editor. The use of an editor which uses proportional font is not recommended as the program reads fields and unless your data are in the correct fields, the program might give erroneous results.

To get the program to use your new file, select the "Other" option on the section selection menu and then use the file selection dialogue box.