SMA

Shape and Material Archive

Version 1.0

Guide

Designed for

BricsCAD®

By:

CASTALIA

http://www.castaliaweb.com
Via Pinturicchio, 24
20133 Milan - Italy
staff@castaliaweb.com

Revision 1, February, 6, 2019

Shape and Material Archive - BricsCAD
Application Guide

Created: 07/02/2019 in Milan

Producer:
Castalia srl

Author:
Ing. Paolo Rugarli

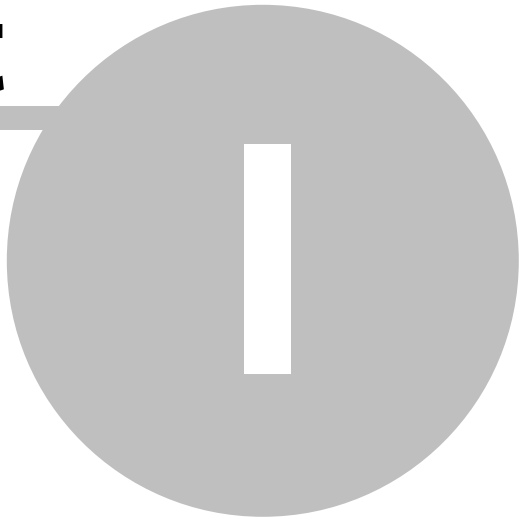
Guide by:
Ing. Paolo Rugarli

Web:
www.steelchecks.com

Assistance:
staff@castaliaweb.com

Table of contents

Part

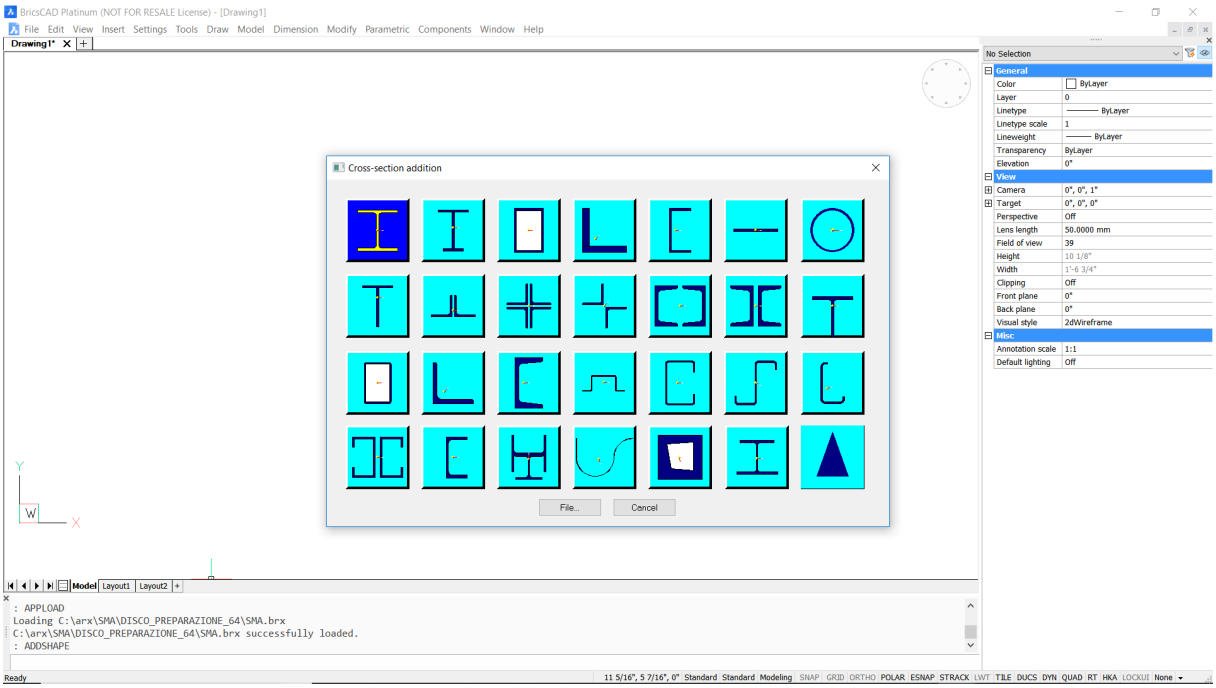


1 Introduction

1.1 Welcome

Welcome to Shape and Material Archive (SMA) BRICSCAD© application. SMA has been designed to work under the BricsCAD© shell, as an extension application.

SMA is a quick and easy tool, used to access structural engineering cross-sections, add them as a block in your current document, and choose them according to a number of design criteria. So, SMA is not only a very wide cross-section archive (more than 19,000) but it is also a tool to properly choose them for typical structural analysis problems.



New cross-sections, not available in archive, may be designed inside SMA

SMA may be also used to study the best dimensions of a cross-section, so as to reach some specific design goal.

SMA has been developed by Castalia srl, an Italian Engineering firm active since 1991, an owner of a number of structural applications developed internally (mainly Sargon and CSE).

SMA is easy to install and use. It's your every day tool to refer to structural cross-sections (European, USA, Indian, Russian, and more) using your preferred units.

Enjoy!

1.2 License agreement

It is agreed as follows:

Castalia s.r.l. grants the use to the concessionaire no. 1 copy of data elaboration program SMA.BRX hereinafter referred to as "program".
The program is composed of DLLs and other files, software security, the license agreement and all the support materials delivered in the form of electronic documents in various formats.

The grant is regulated by the following conditions:

1) Castalia srl, by agreement with the authors, has and maintains the exclusive rights of copyright on the program, on the manual and on all the written material accompanying the program. This program is protected by copyright law and copyright of Italy, by the provisions of the international treaties and all applicable national laws. The program is composed of different modules (DLLs) each of which is and shall remain the property of the authors and not of the Concessionaire.

2) The Concessionaire has the right to use one copy of the program in a single processor. The concessionaire will not be able to use the Program in more than one computer or terminal at the same time.

3) Neither the Program nor a copy thereof may be sub-licensed to third parties.

4) The program may not be modified or incorporated in other programs, converted, decoded, decompiled, disassembled or subjected to any process aimed at its conversion in the source program.

5) In case of non-compliance with the conditions referred to above, this concession contract will be terminated due to the fault of the concessionaire, who will have to return uninstall the program, deleting all files from his/her hard disk, without the right to any reimbursement, not excluded the compensation for further damages and possible prosecution.

6) The Program is provided "as is". Castalia s.r.l. and authors, despite the fact that the program has been subjected to careful checks, declines all responsibility in the hypothesis that the results of the processings obtained with the use of the same proves to be affected by errors or failures of any kind, understanding that the Concessionaire is in any case obliged to check the results of the processing/preparation.

7) Castalia srl guarantees that the product and the media on which the software is provided are substantially devoid of significant defects for a period of three (12) months from the date of delivery of the product.

8) Requests for warranty replacements must be made within seven days from the date of observation of the defect, accompanied by satisfactory evidence and precise indications.

9) Except as set forth above, there is no other warranty, statement or condition concerning the product or services or performance of Castalia srl or authors, expressed or implied, including (and beyond) the implied warranty of ability to accomplish a specific task.

10) The responsibilities of Castalia srl and of authors for damage to the Concessionaire or to any other party for any cause, can never exceed the price paid for the product unit that has caused the damage. In no case, Castalia or authors will be responsible for any damage caused by the failure to fulfill its obligations on the part of the Concessionaire, or for any loss of data or profits, savings, or any other consequential or incidental damages or for any claim based on actions of third parties.

11) In case of any dispute, the place of jurisdiction is Milan.

1.3 Installation

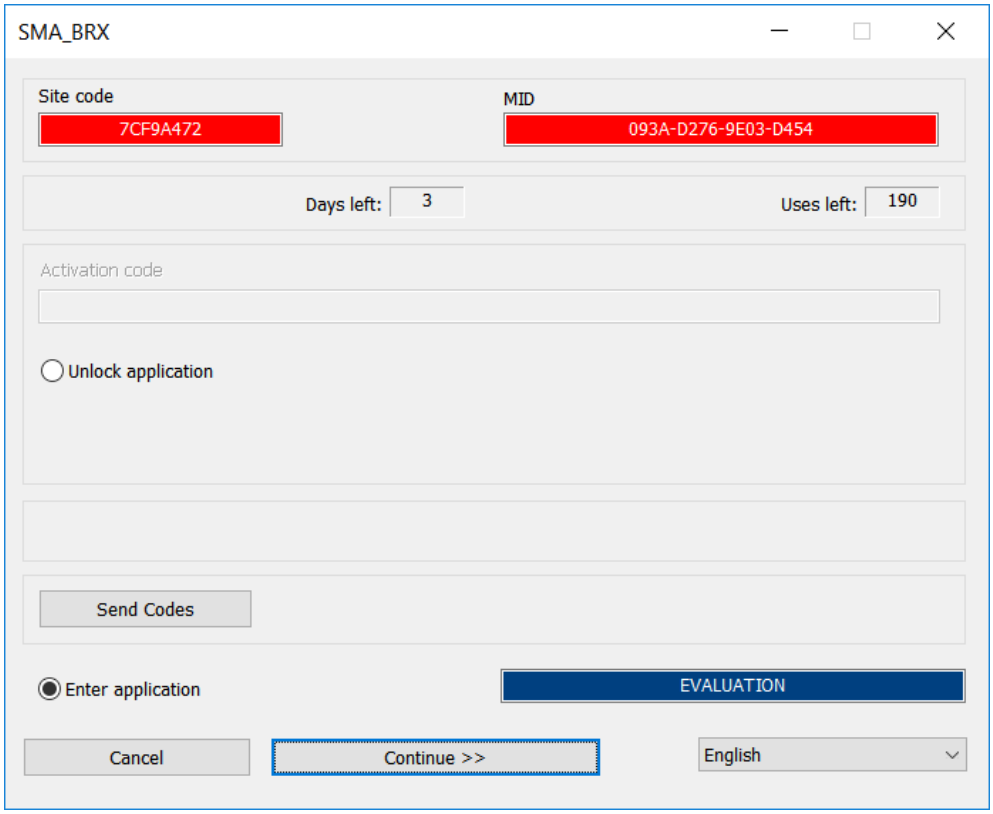
To install SMA, it is sufficient to create a folder named for example "C:\SMA", and copy the extracted files from the initial zip file. Alternatively, copy the zip file in the folder and extract the files therein. The folder can be also called in another way or be positioned in another folder as sub-folder.

SMA.BRX can now be loaded as every BRICSCAD APP by executing the BRICSCAD Command **Tools-Load Application**. As soon as the application is loaded, an activation dialog opens (see below).

In no way CLASS4 interferes or can interfere with the system. The registry is not modified and no particular privileges are required to work with the program files.

The program can operate for 10 days and 200 executions without activation code. After this period, it is essential to activate the program 13. It is recommended to activate the program immediately after the purchase.

As long as the program is not activated, you can run it within the limits explained. By choosing "*Enter appplication*" and then "*Continue*" the program starts.



See: How to activate the program 13.

1.4 Activation

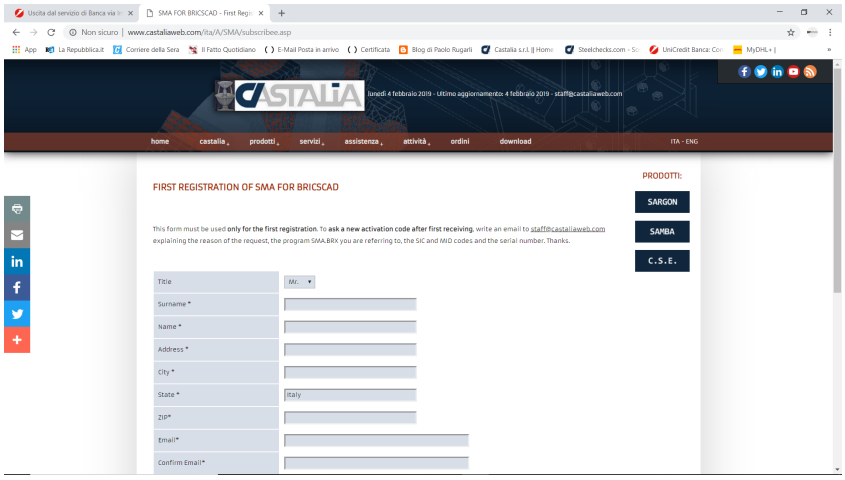
Once the program is installed 11 it is possible to run it without activation for 10 days or 200 executions. After this period, it is essential to activate the program to enable it. It is recommended to activate the program immediately after the purchase. .

To activate the program, you must connect to the following web page, that you can automatically open by clicking the button "*Send codes*":

www.castaliaweb.com/ita/a/sma/subscribee.asp

The page is shown in the following figure. It displays a form in which, it is necessary to provide a series of data, part of which are referring to address, while another part is constituted by three codes: SIC, MID and SN.

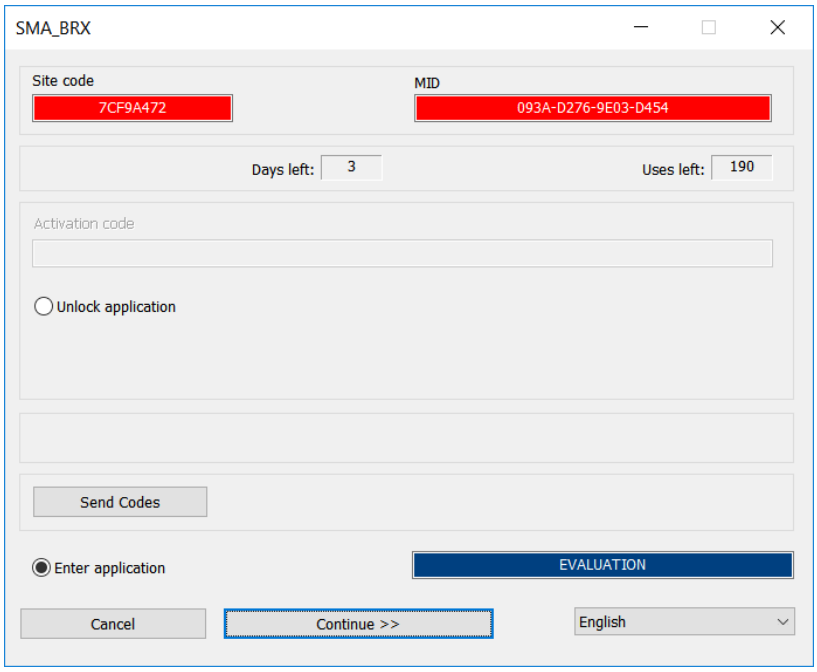
The user implicitly provides consent to the processing of data which will be retained by Castalia srl solely in order to inform users of successive versions of the product or similar products. The data provided will not be provided to third parties.



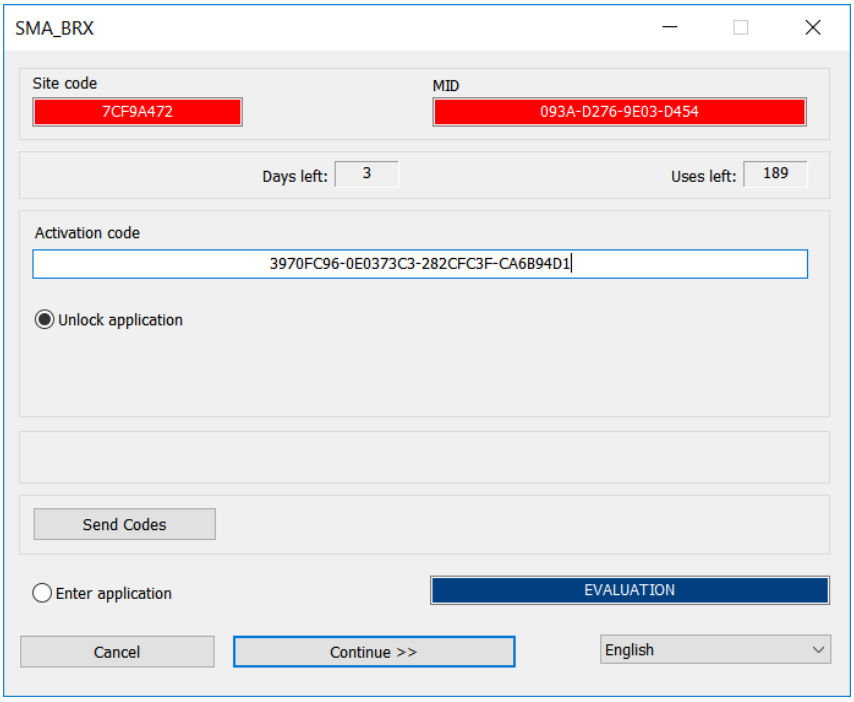
The serial number (SN) is given by the producer or by the retailer with the purchase of the program. The SIC codes and MID are supplied from the startup screen of the

program itself: it is advisable to bring them into the form using copy and paste to avoid mistakes: the codes must be reproduced exactly. It is recommended also to keep such codes stored in a file for greater safety.

Once completed and sent the form, you must wait for one or two working days, then you will receive by e-mail to the address indicated, the "Activation Code" (that will also be stored). The activation code is necessary to activate the program, according to the modalities now clarified.



As long as the program is not activated, you can run it within the limits explained (10 days), by choosing "*Enter application*" as in the previous window and then "*Continue*". When you will register, you must choose "*Unlock application*" and insert the "*Activation Code*" received by e-mail (always better using copy and paste), then "*Continue*".



At this point, the program is activated and can be run indefinitely on the computer selected. In order to let you transfer the license from a computer to another, **save the activation code you received**.

The web page indicated above is used only for the first activations. If for various reasons, you need a new activation code, you need to write an email to staff@castaliaweb.com specifying the reasons for the request and indicating (with copy and paste), the new SIC codes and MID corresponding to the new installation. Don't forget to explain it's for SMA for BRICSCAD application.

1.5 License transfer

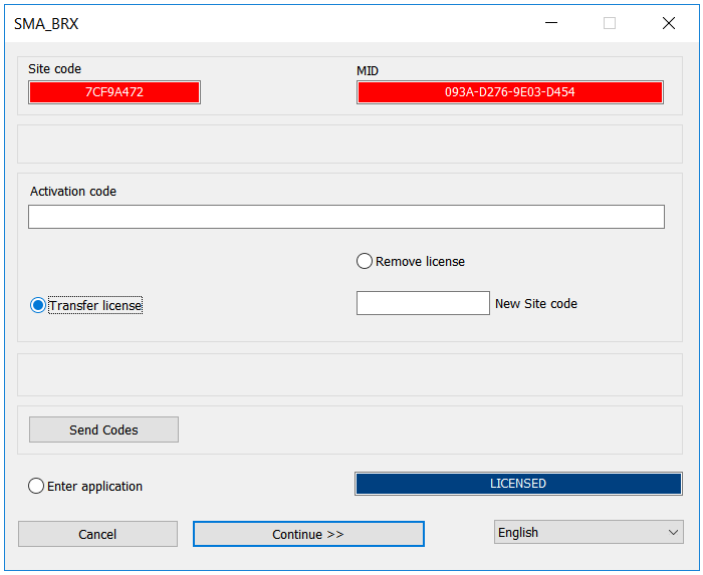
The program works normally on a certain computer. You can transfer the license from one computer to another using the following procedure.

1) The program is installed and activated at the moment on the computer A.

2) Install the program on a computer B.

3) Run the program on your computer B and take note of the SIC codes and MID.

4) Start the program on your computer A, and choose "Transfer license", as in the dialog in the following figure.



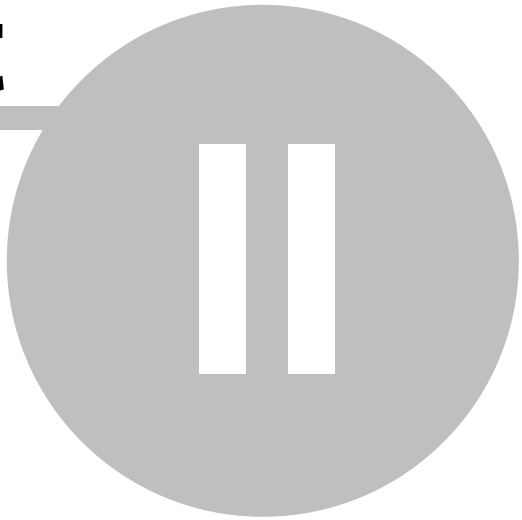
5) At this point enter the activation code of the computer A, **the one you saved when you installed the software in computer A**, and then enter the code "New SIC" (new SIC), that is the SIC code of the computer B.

6) Press the button "Continue".

7) At this point, the program is no longer executable on the computer, but you will receive an activation code for the computer B. All you have to do is enter this activation code on the computer B to complete the unlocking of the program on the computer B.

8) The license is passed from computer A to computer B.

Part



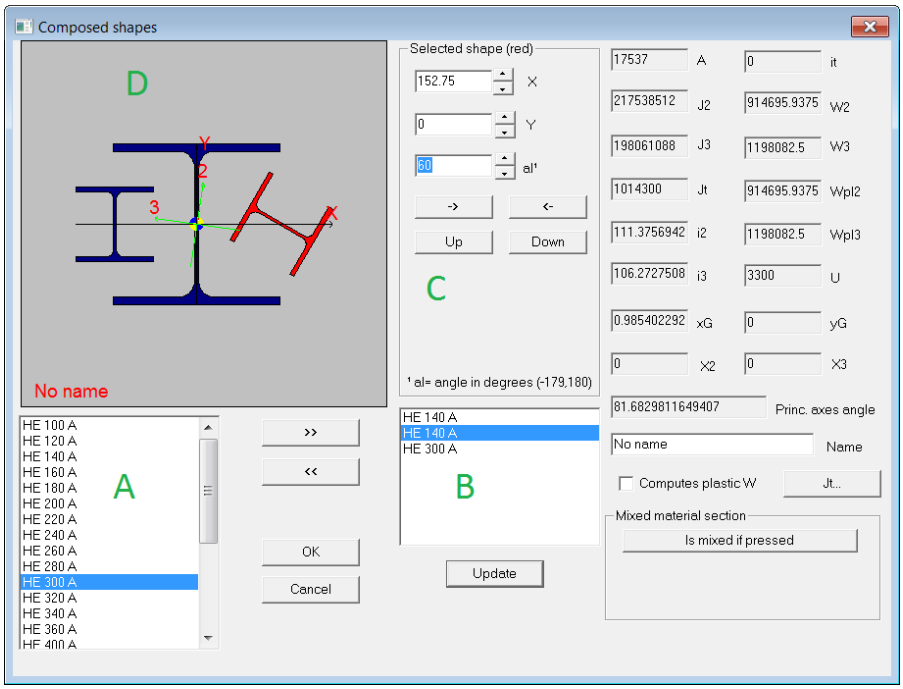
2 Overview

2.1 SMA: a quick and easy tool

SMA, Shape and Material Archive, is an application designed to work as an extension of BricsCAD©. It is an easy and quick software that extends the capability of BricsCAD©, by providing a set of tools very useful to engineers and architects.

As it is well known, steel structures, all over the world, use special steel components usually prismatic, and with special cross-sections. The cross-section shape is optimized for a given design goal, and each cross-section is supplemented by a number of very important numerical data, that are used for the structural checks. These checks are a complex issue, and involve resistance, stability and deformation. Each Nation, broadly speaking, has its own steel structure practice, so, all over the world there is a huge number of standard cross section. These standard cross-section have very well defined sizes (e.g. depth, width, thicknesses and corner radii), that should be strictly respected in order to be compliant with the standards.

So, for this reason, it is very useful to have the availability of a huge number of these cross-sections, supplemented by their numerical data useful for the analysis and check.



Example of composed cross-section assisted design

SMA not olny provides the cross-section, but it is also a tool to select them properly according to design needs, and allows the interactive modification of free-shaped cross-sections, in order to reach some needed numerical property (e.g. section bending modulus, second area moment and so on). A number of typical structural problems are tackled by specific filters, with the aim of extracting from the archive of all the cross-sections, only those that will comply with the design criteria specified. Albeit this may not be considered a full check for all the possible failure modes (which is usually done in a structural analysis program) it is a very useful tool in order to pre-select or also possibly choose, the proper cross-section for a given design need.

SMA provides a very huge cross-section archive, exposing to BricsCAD users a number of functionalities that SMA has in common with the structural software produced by Castalia srl, and mainly directed to steel structure designers.

At a reasonable cost, SMA provides a very useful tool, for the every day work of engineers and architects.

2.2 The parents of SMA: Sargon and CSE

The kernel of SMA is a set of DLLs (dynamic link libraries) that are exactly the same used by the two main applications that the producer of SMA, Castalia srl, has developed since the end of the 80s: Sargon and CSE.

This means that the kernel of SMA has been successfully used since 1991, in true-world structural engineering both in Italy and abroad. So, even if SMA as a BRX application for BricsCAD© has been released in 2019, the numerical and programmatic kernel of SMA has been developed, used and tested since many many years (from 1991 onward).

Sargon is a huge finite element program mainly oriented to steel structures. It has linear and non linear solvers (geometrical and material non linearities) , and a comprehensive set of finite elements, including truss, beam, thin and thick plate-shell, membrane, solid, spring, beam on elastic soil, plate on elastic soil, beam with semi-rigid connections, both in elastic and plastic range. Sargon has a number of executables related to it, which are able to check steel structures according to several standards: Eurocode 3, British Standard, AISC both ASD (allowable stress design) and LRFD (load and resistance factor design).

CSE (Connection Study Environment) is an application fully related to the design, study and check of steel connections. CSE has been the first commercial

software to deal with the problem of generic connections in 3D, allowing the automatic creation of plate-shell finite element models of components for their stress-analysis under general conditions of stress (since 2008). The extended research carried on for developing CSE led to the publication of the book Steel Connection Analysis, written by Paolo Rugarli, who is also the author of SMA.

2.3 The cross-section and material archive

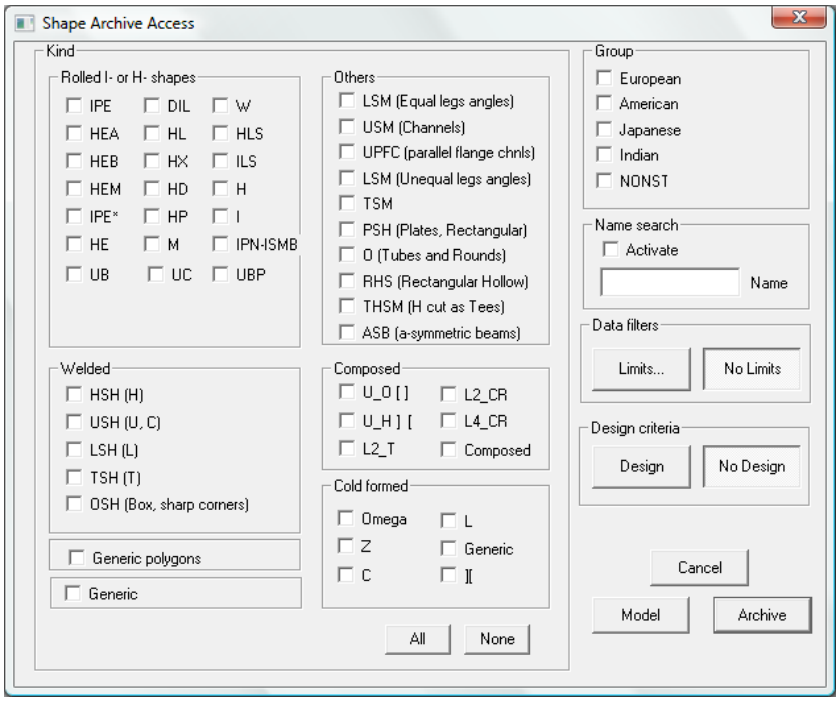
SMA is furnished with a huge cross-section and material archive: more than 19,000 different cross-sections may be referenced.

The archive is the same used by the parents of SMA 20. This is embedded into the file *archive.sma* which is in the same folder of the BRX application "SMA.BRX". In order to access the cross-section and material archive, it is necessary to load it 40 if this has not been loaded yet in the current working session. This is done by simply choosing the *archive.sma* file, in the folder where it has been placed. Ideally SMA might read also other archives, in the SMA format. These might be created using the application SAMBA (Shape And Material Brisk Archive). This means that by using SAMBA you might extend the archive your SMA application is using virtually with no limit. SAMBA, in fact, enables the user to also load cross-sections and materials from a text file, so allowing a quick enlargement of the archive or its customization for special design purposes (e.g. special sets of welsed or cold-formed cross-sections).

The file embeds the cross-sections and materials in binary, proprietary format.

The cross-sections totally available in the file *archive.sma* are 19066 and the materials are 140. These include elements from USA, Europe, UK, India, Russia, Japan, Italy, and other Countries.

Among the 19,066 cross sections available are the following:



- IPE, HEA, HEAA, HEB, HEM, UPN, UPFC, UB, UC, UBP, W, M, HP, HD, HX, HL, DIL, ILS, Angles (equal and unequal legs, L, GOST, LR, ISA...), Channels (GOST, U, UR, C, CH, ...), RHS, SHS, CHS (O, PIPE, HSS, ...), ASB, T cut from I or H rolled, double angles (_||_, and +) , four angles (+) double channels ([ ] ands][ ), cold formed (C, L, Z, Hat, lipped or not, and free form), composed, polylines-made et cetera.

2.4 Choice of the cross section by filters

Accessing a so huge cross-section archive needs some tool in order to *extract* only a subset of them, so that we can pick what we need from a reduced set.

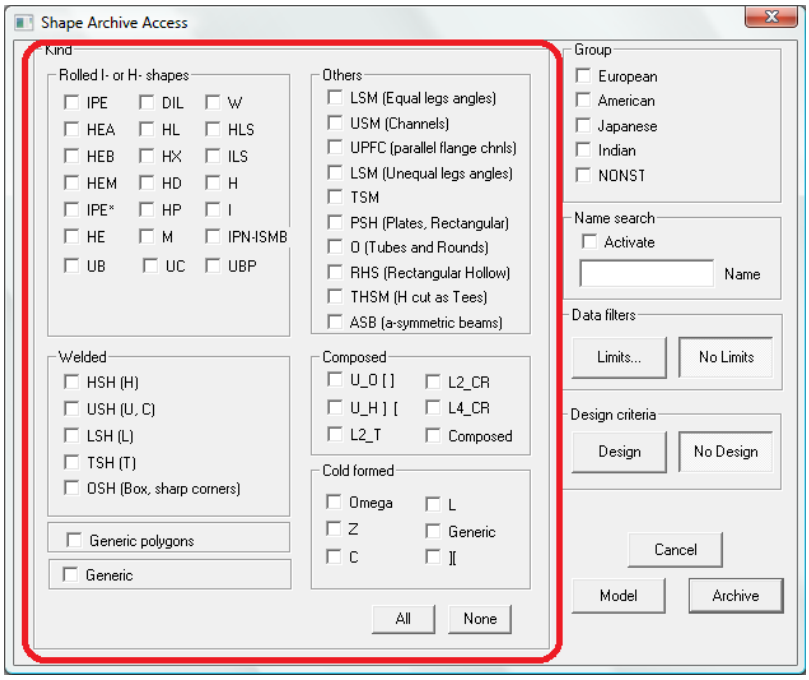
In order to do that, some filters 89, in the SMA terminology, must be set up.

Filters may be based on:

- type 23: only the cross-sections of specified types will be extracted.
- design criteria 25: only the cross-sections satisfying user-defined design criteria will be extracted.
- data limits 27: only cross-sections that have some specific feature like area or second area moment within user-defined ranges will be extracted.
- name 30: only cross-sections matching the name (using wild cards) specified by the user, will be extracted.
- group 32: only those cross-sections belonging to the group specified (based on geography), will be extracted

If no filter is specified, all the cross-sections will be extracted sorted by increasing area.

2.4.1 Type



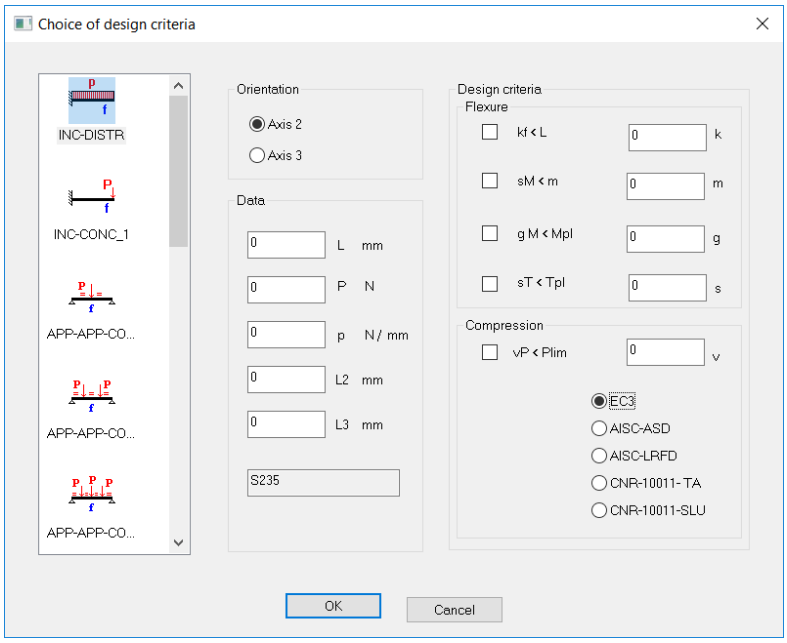
The available types (or "kinds") are:

- **Rolled I or H cross-sections (round corners SM):**
  - IPE
  - HEA
  - HEB
  - HEM
  - IPE*
  - HE
  - UB
  - DIL
  - HL
  - HX
  - HD
  - HP
  - M
  - UC
  - W
  - HLS
  - ILS
  - H
  - I
  - IPN-ISMB
  - UBP
- **Welded or sharp corners (SH)**
  - HSH (H with unequal flanges)
  - USH (C or U)
  - LSH (L)
  - TSH (T)
  - OSH (box with unequal flanges and equal webs)
- **LSM (rolled angles equal legs)**
- **USM (rolled channels tapered flanges)**
- **UPFC (parallel flange rolled channels)**
- **LSM (unequal legs)**
- **TSM (rolled Tee with tapered web)**
- **PSH (rectangles or plates)**
- **O (CHS if thickness is lower than half diameter, Round if thickness = half diameter)**
- **RHS (Rectangular and Square rolled hollow sections, round corners)**

- **THSM (Tee cut by rolled I- or H-)**
- **ASB (asymmetrical beams, rolled I- or H- with unequal flanges)**
- **Composed**
  - U_O: two rolled channels [ ]
  - U_H: two rolled channels][
  - L2_T: two rolled angles _||_
  - L2_CR. two rolled angles +
  - L4_CR: four rolled angles +
  - Composed: these are generic cross sections obtained by assembling more elementary cross-sections
- **Cold Formed**
  - Omega: also called "hat" cross-sections
  - Z: with or without lip
  - C: with or without lip
  - L: with or without lip
  - Generic: free form cold-formed cross-sections
  -][: two cold formed L assembled as][
- **Generic polygons: these are cross sections defined by closed polygons (may contain holes)**
- **Generic: these cross-sections do not have a graphical representation and may not be used in SMA for BricsCAD©.**

2.4.2 Design Criteria

Design filters are very useful. They have been conceived, many years ago, as a quick tool to help engineers and architects to decide which could be the best cross-section to use for a given structural member.



The structural configurations considered are the following:

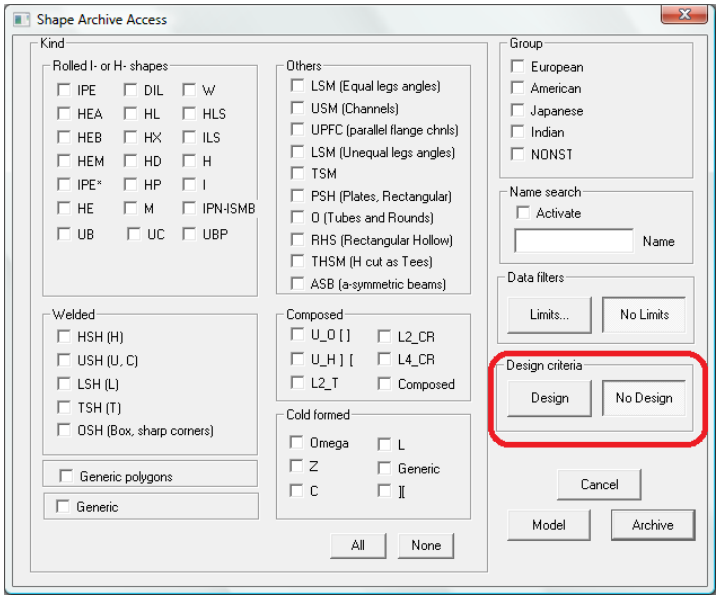
- simply supported beam
- cantilever
- continuous beam 2 equal spans
- continuous beam 3 equal spans
- continuous beam 4 equal spans
- Euler column
- Double clamp beam
- Semi-rigid support simple beam

Each structural configuration may appear loaded in different ways (e.g. one concentrated load at mid, two equal concentrated load at thirds, and so on). The user may choose one structural configuration and one loading mode. He/she will then input the span (or the effective length for the column) and the load.
Then, it is possible to ask to SMA to extract only those cross sections that, when considering the structural configuration assumed, and with the given load, will satisfy the design criteria. These are:

- simple and continuous beams
  - limitation of deflection f as a fraction of span
  - limitation of Von Mises stress s
  - limitation of applied bending moment

- limitation of applied shear
- column
  - limitation of the applied axial force

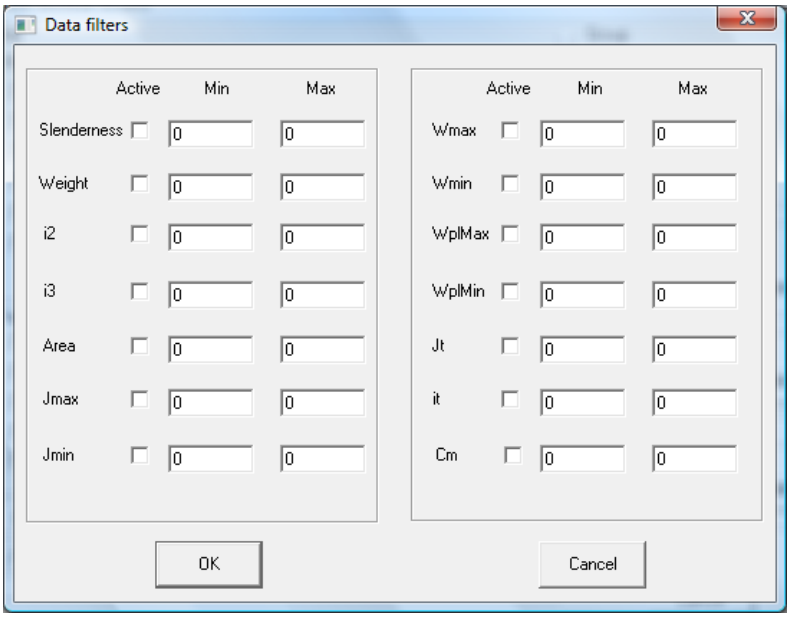
The limitation of the axial force for column may be done according to several possible standards. Cold-formed specific requirements (e.g. effective area) are not taken into consideration.



To access to the dedicated dialog the **Design** button must be pressed, from the Shape Archive Access dialog 89.

2.4.3 Limits

Limits filters are useful if the user - as it frequently happens for experienced engineers - already has a value in mind for one or more relevant numerical features of the unknown cross-section: for instance the section modulus, or its second area-moment. The user will then specify a numerical range that the required feature will have to respect. This range must be entered taking into consideration the current units 100.



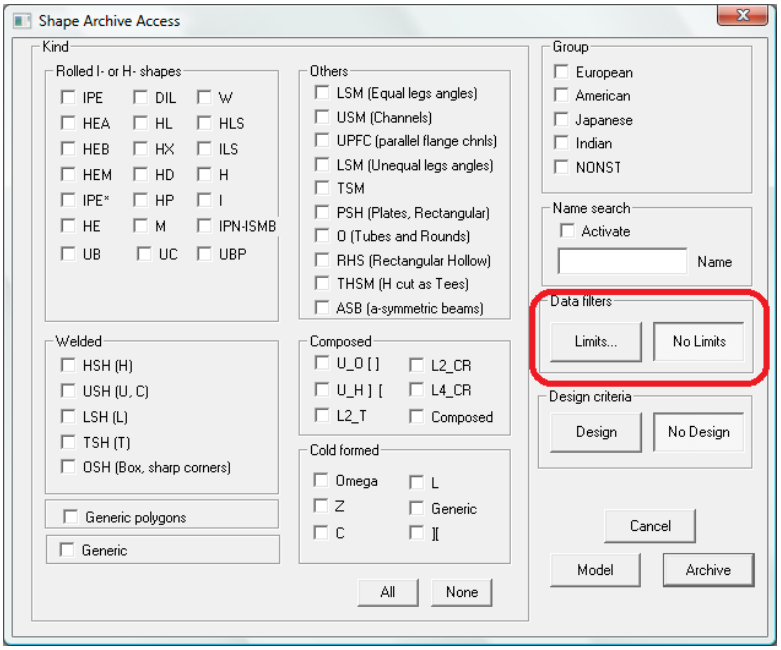
The relevant features are:

- Slenderness (this may not be used in SMA as it depends on the member length, unknown. It is kept for compatibility with future versions, it's a pure number)
- Weight: this is the weight per unit length of member assuming steel as material (force per unit length)
- Strong axis inertia radius i2 (length)
- weak axis inertia radius i3 (length)
- Area (length^2)
- Maximum second area moment Jmax (length^4)
- Minimum second area moment Jmin (length^4)
- Maximum flexural section modulus Wmax (length^3)
- Minimum flexural section modulus Wmin (length^3)
- Maximum plastic flexural section modulus Wmax (length^3)

- Minimum plastic flexural section modulus Wmin (length^3)
- Torsional constant Jt (length^4)
- Torsional radius, it (length)
- Warping constant Cm (length^6)

Each filter must be activated ticking on the relevant check box in the **Active** column. A minimum and a maximum values must be entered, using the current units (e.g. for second area moment if the current unit is cm, the numbers entered will be interpreted as cm^4).

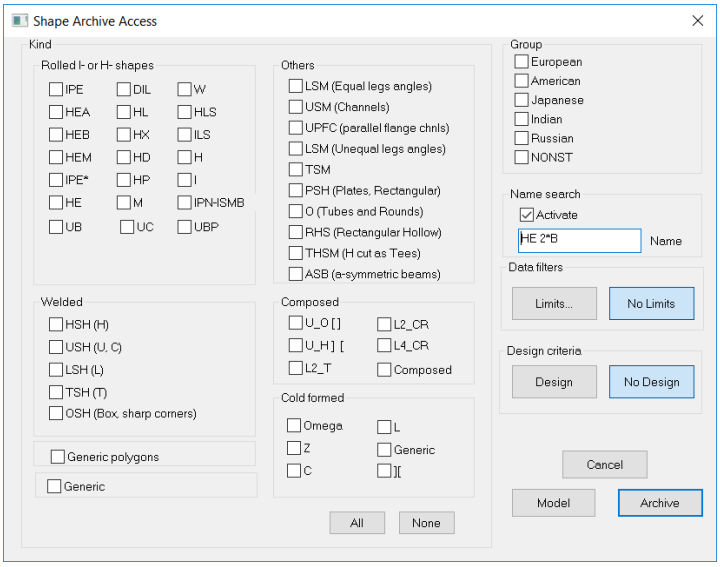
To access to the dedicated dialog the **Limits** button must be pressed, from the Shape Archive Access dialog 89.

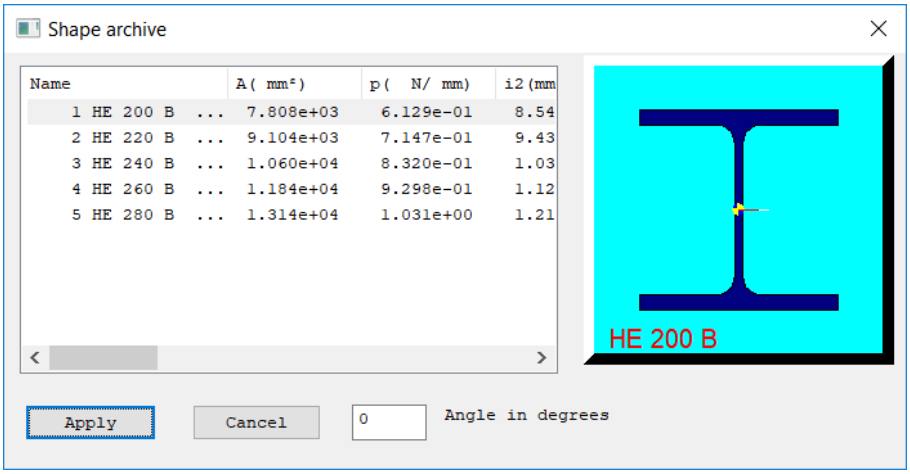


2.4.4 Name

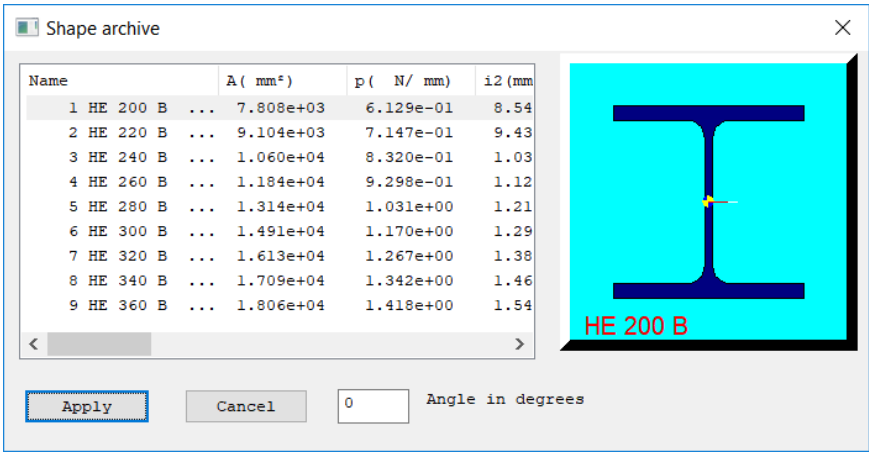
Sometimes we need a specific cross-section , or a specific subset of cross-sections, having a part of their names in common (names may not be duplicated). To do so, we may specify a search string, also using wild characters.

For instance, if we would like to extract all HEB section having 200 mm depth ore so, we may ask for the cross-sections "`HE 2*B`" (these are as in the next picture). Please notice the need of the blank space after "HE" and before 2.

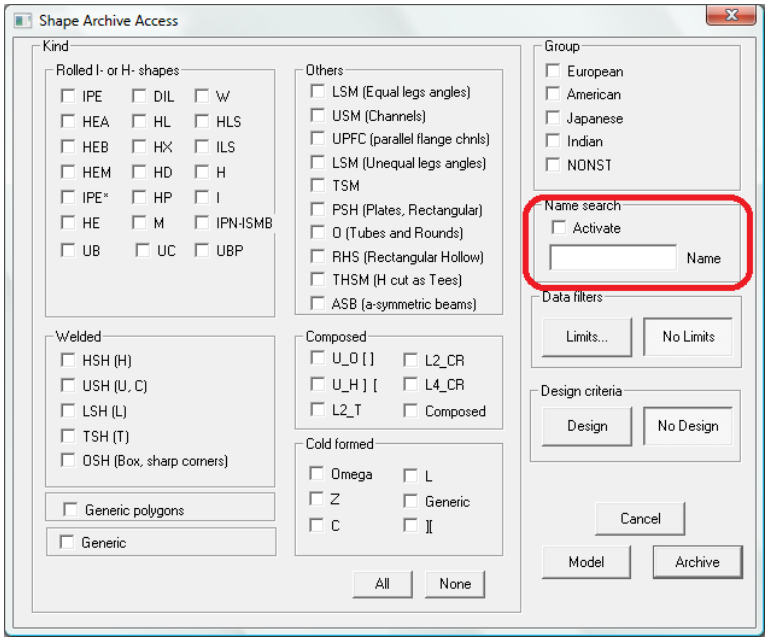




Search strings may be separated by semicolons. For instance searching for "`HE 2*B;HE 3*B`" we get:



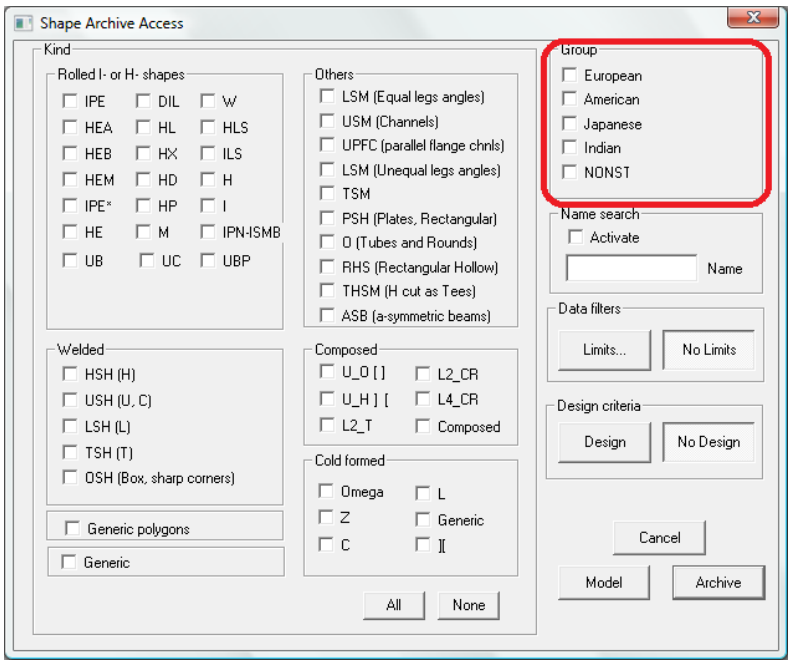
To set a name-filter, the **Name-search** controls should be used in the [Shape Archive Access dialog](#) 89.



2.4.5 Group

Group filters are useful to restrict the set of cross-sections to those of a given Nation or Continent.

To set a name-filter, the **Group** controls should be used in the Shape Archive Access dialog 89.



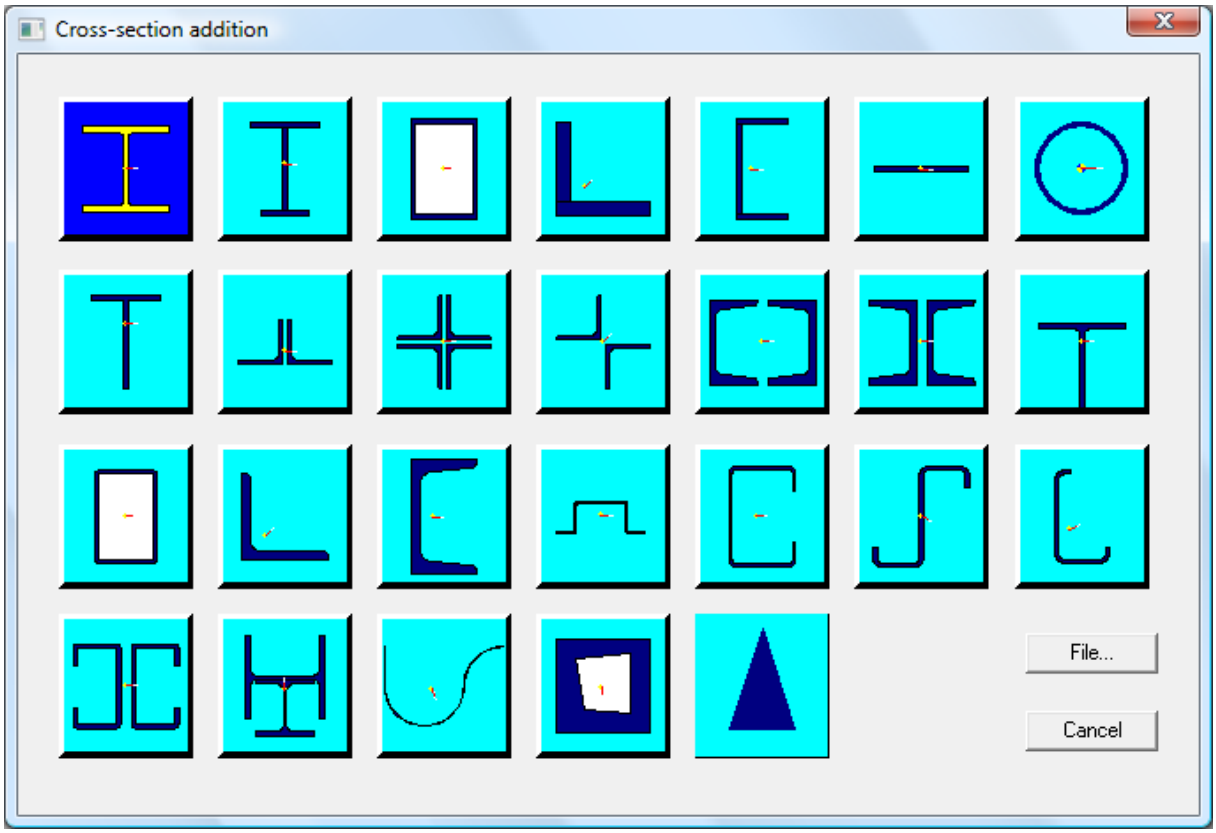
2.5 Addition of a new cross-section

Sometimes you wish to reference a new cross section, that is not available in the database. SMA lets you do that, providing all the tools you might need in order to design cross-section dimensions so that the final cross-section properties are compliant with you design goal (area, and so weight, flexural modulus, second are moment, plastic flexural modulus are some of the cross-section feature you may have a look at, while choosing the best dimensions for your need).

If you need a new cross-section, with specific type 23, sizes and name, you may enter the command ADDSHAPE 40: this will enable you to define, study and add to the current BricsCAD© document, the block you need.

2.5.1 Types available

When considering the addition of new cross section, we are asked to specify the type by this dialog using bitmap-buttons.



The meaning of the buttons is strictly related to the type of the cross-section SMA manages. This is a very huuge set.

They are (row by row, from left to right):

- Rolled I- or H-
- Welded (or reinforced concrete) sharp corners I- or H-
- Box (sharp corners, usually welded)
- Welded (or reinforced concrete) sharp corners L

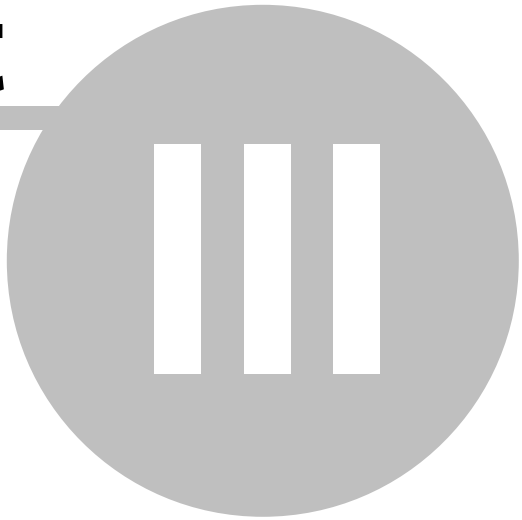
- Welded (or reinforced concrete) sharp corners C
- Rectangles
- CHS or Round

- Welded (or reinforced concrete) sharp corners T
- Double rolled angles
- 4 rolled angles
- 2 rolled angles
- 2 rolled channels
- 2 rolled channels
- Tee cut from rolled I- or H-

- Rhs or Shs with round corners (rolled)
- Rolled angles
- Rolled channels with tapered flanges
- Cold formed hat
- Cold formed C (with or without lip)
- Cold formed Zed (with or without lips)
- Cold formed L (with or without lips)

- Doubel cold formed C
- Totally generic composed (n cross-sections assembled together)
- Totally generic made up by polygons
- Totally generic (no graphical representation, unused in SMA)

Part



3 Commands

SMA has the following available commands which enable the user to get the needed data in the most suitable and quick way:

ADDSHAPE 40 to define a cross-section by dimensions, and add it as a block in current BricsCAD© document. The cross-section will also be added to a (non permanent) vector.

CURRMAT 77 to review and possibly modify curent material properties

GETMAT 84 to choose one material from the archive as current

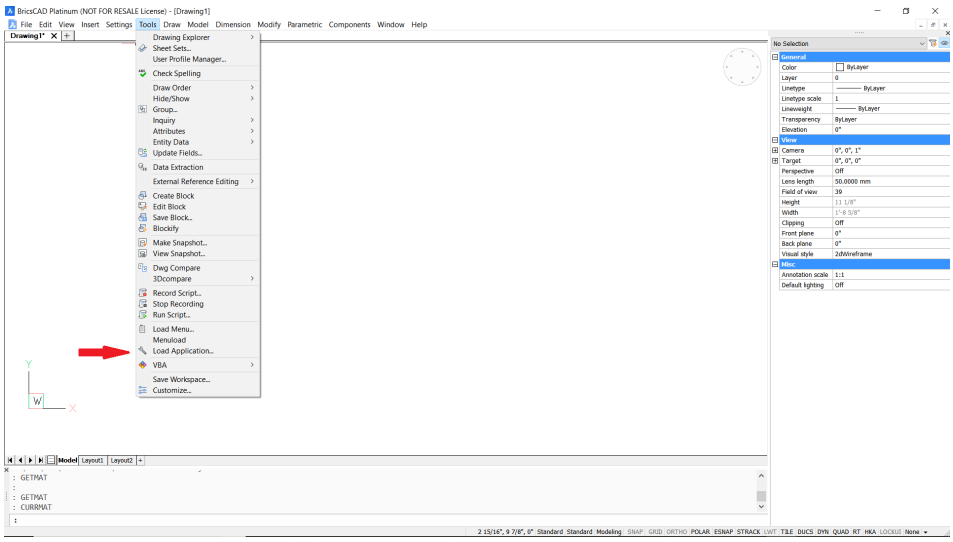
GETSHAPE 87 to pick a cross-section from the archive, using proper access filters. The cross-section will be added as a block to the current BricsCAD© document.

SETUNIT 100 to set the current units (default: mm, N). These are used to input and output numerical data, and to scale the sizes in the added block

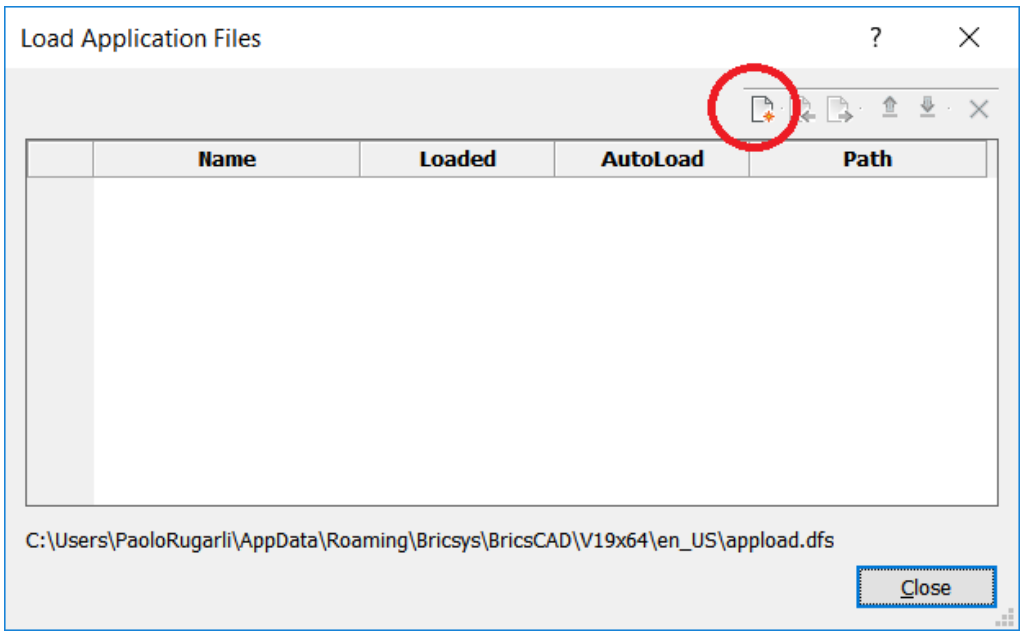
The commands may be entered by the command prompt once the SMA application has been loaded 36.

3.1 Loading of SMA application

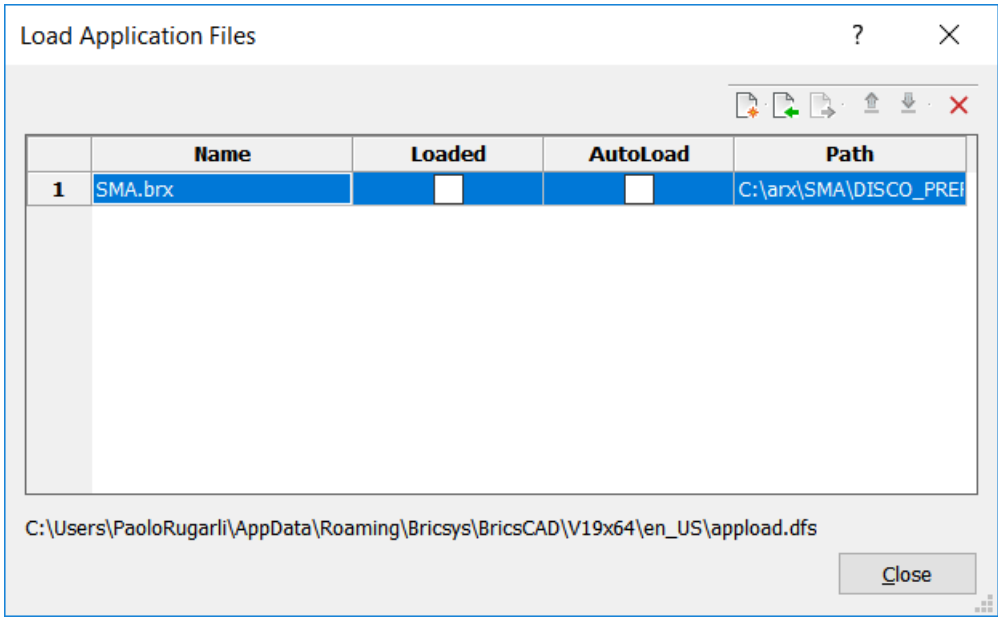
In order to load the SMA application, running BRICSCAD© 64 bit, the command "APPLOAD" must be given from the command prompt. As an alternative, using the menu, APPLOAD command is executed through the menu **Tools**, and the command **Load Application** (see pic below).



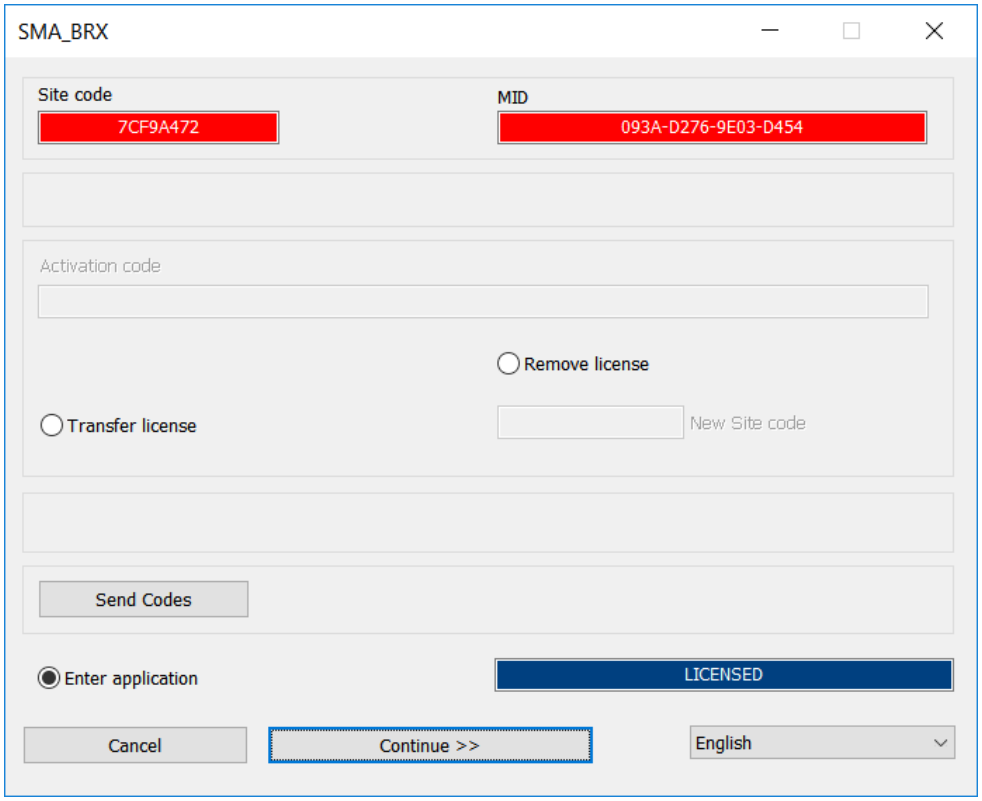
Then, the following dialog appears:



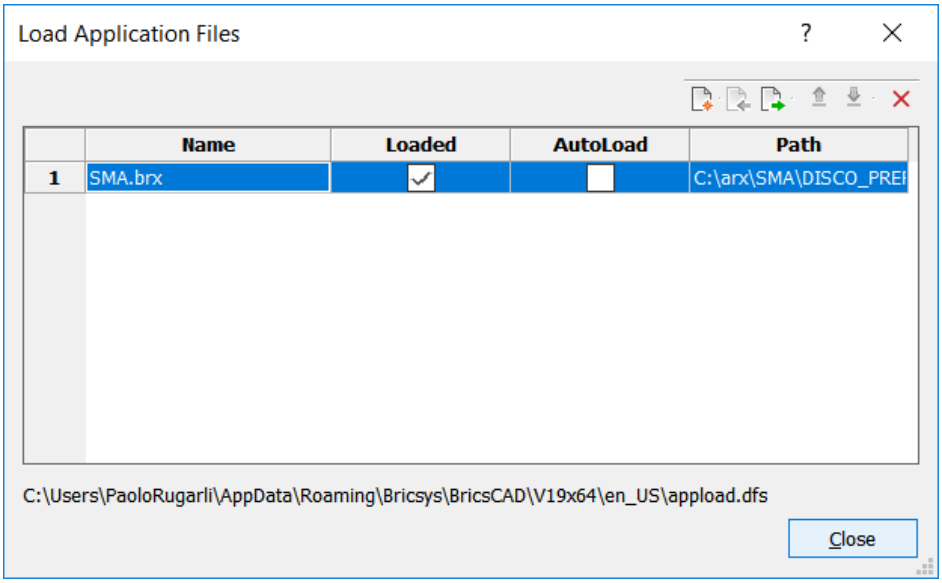
Clicking the circled item you are prompted to choose the proper folder, where all the SMA files have been saved, and pick the SMA.BRX application file. This will lead you to the following situation:



Now, by ticking over the check box in the column "**Loaded**" you will ask to load the application. "**Autoload"** is useful if you wish BRICSCAD© to directly always load the application at start. Now, you have to enter in the application by clicking the Continue button (if the application has already been activated, see activation 13):



Finally the command prompt will confirm the application is successfully loaded, and you will be in this condition:



You may close the dialog and enter the command you need, from the command prompt.

Keep in mind that by default, at start, the units are (mm, N) and derived units. You may change the units at any time executing the command SETUNIT.

If the SMA application is not loaded, this may depend on the following reasons:

1. You are using a 32 bit version of BRICSCAD©.
2. Not all the needed DLLs are available in the folder where the SMA.BRX file is placed.
3. File are corrupted or damaged for some reason.
4. Some file has been deleted.
5. BRICSCAD© version you are using is not supported (version should be ≥ 19).

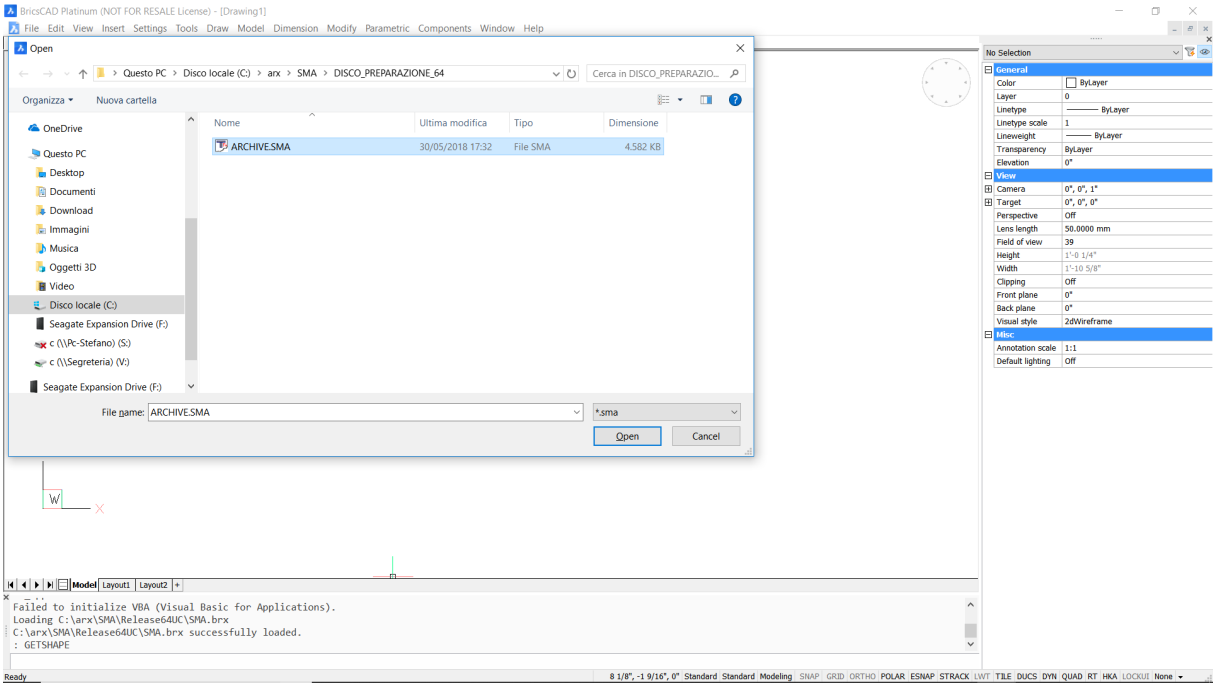
Other possible reasons are by far less probable and would require a specific investigation.

3.2 ADDSHAPE

Enter the following string in the BRICSCAD command prompt:

ADDSHAPE

If the cross-section and material archive has not been loaded yet, you are prompted to choose the "archive.sma" file, which is the database file received with SMA, and in the same folder of SMA.BRX.

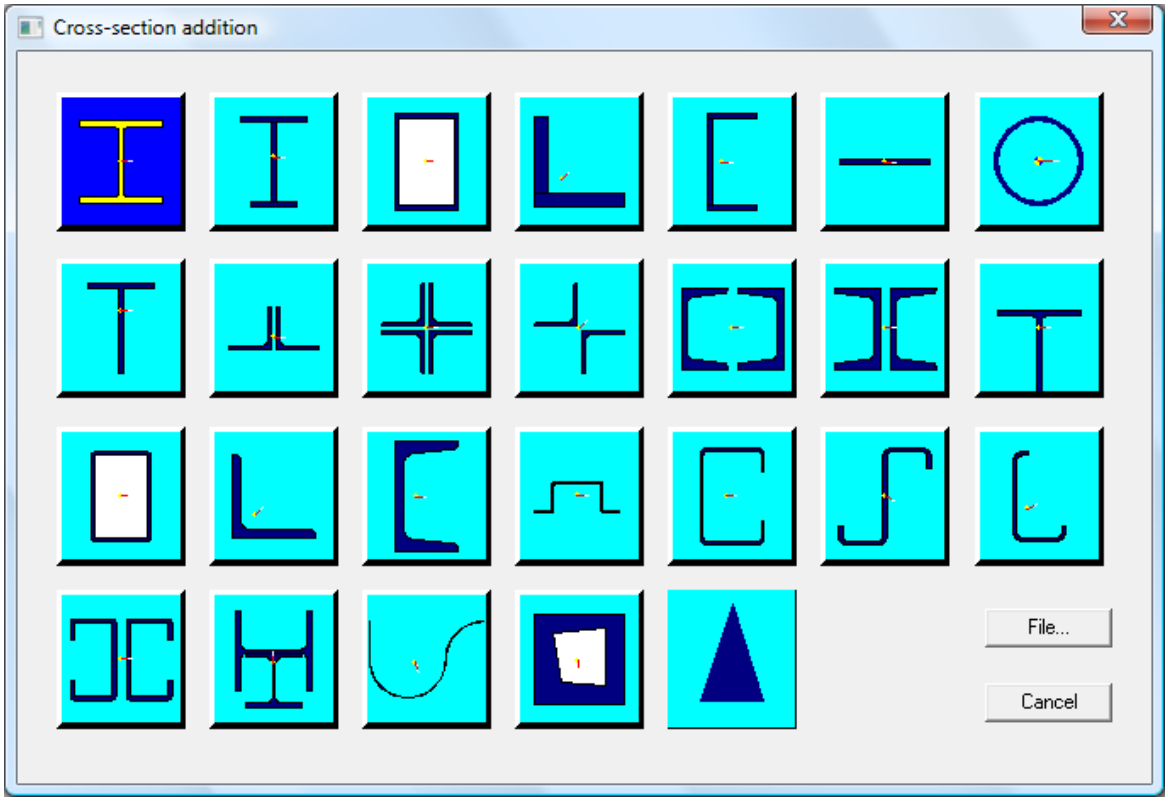


This file embeds all the cross-sections and materials of the existing archive (more than 19,000 cross sections, USA, UK, European, Indian, Russian, and more).

By this command, a proper dialog appears 42. This dialog is used to choose the type 33 of the cross section you wish to add. Once you choose it, a proper dedicated dialog opens, depending on the choice you made.

This command is used to design a cross-section that will be later added to the current BricsCAD© document as a block, using the units currently selected. The newly added cross-section is also added to a local vector of "model" cross-sections, that may be later chosen if needed.

3.2.1 Shape addition (dialog)



In this dialog box, a cross-section kind 42 must be chosen, then its name and sizes will be defined in a proper dialog box. Finally, the insertion dialog box 100 will appear.

3.2.2 Shape types (dialogs)

Cross sections are available in a range of types (welded, rolled, cold-formed and composite) and shapes (H-, L-, U-, C-, Z- or Omega-sections, solid or hollow circular, solid or hollow rectangular, generic, etc.).

New sections can designed and added as a block in the current BRICSCAD document (supplementing the 19,000+ sections which can be found in the archive).

The various different dialog boxes relating to the sectional forms are listed below. Similar types of dialog are grouped together.

Rolled, welded and cold-formed sections or "standard" form 43

The archive also contains IPN- and TSM-sections (rolled T-sections with tapered sides).

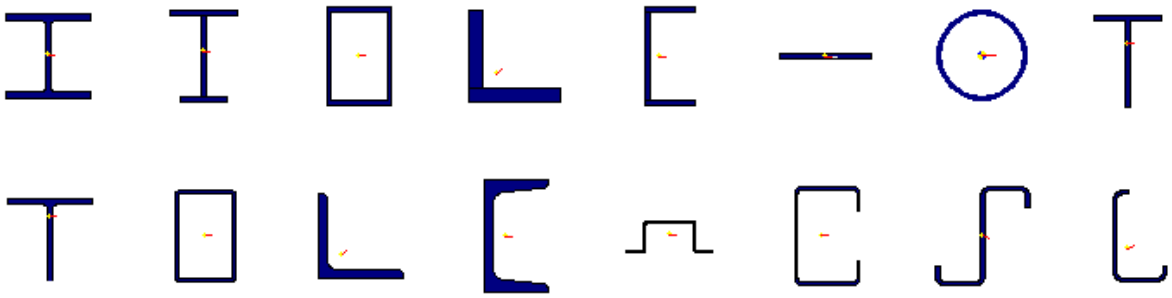
3.2.2.1 Standard shapes (dialogs)

This part of the documentation provides a general description of a number of dialog boxes which have features in common. These dialogs are used to define a new section or display its properties. When a new section is being added, the boxes with editable values (the profile dimensions and name) appear in white, with all the others in grey. In enquire mode, all the boxes are greyed out.

The two letters "SM" stand for "smooth" [corners] and are used for rolled cross sections. The two letters "SH" stand for sharp [corners] and are used for welded cross sections. "CF" stand for cold formed.

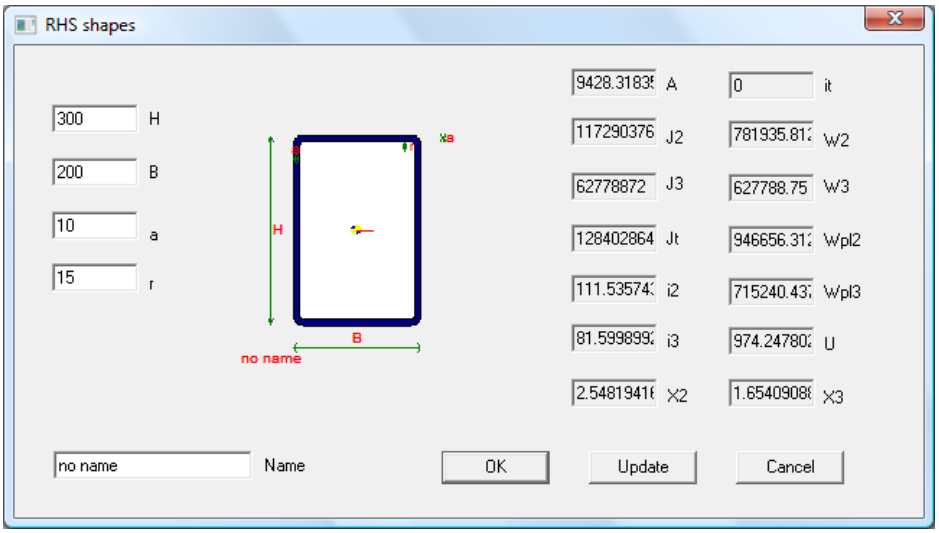
The following profiles use a dialog box which is similar to that which we are about to describe:

- welded sections: H-, L-, C-, T- and box sections
- rolled H-, L-, C- and T-sections, and hollow rectangular sections
- flat or rectangular sections
- hollow or solid circular sections
- cold-formed L-, C-, Z- or Omega sections



]

Consider, for example, the dialog box for hollow rolled rectangular sections, which appears as shown below.



The section dimensions appear on the left: if the profile is being created, these boxes are white and can be edited; press **Update** to update the image and the numerical data on the right to reflect the changes made.

The parameters to be specified in this case are the height *H*, width *B*, thickness *a* and external radius *r*. The dimensions will be different for other types of profile; simply consult the image to see which of the profile dimensions are being referred to.

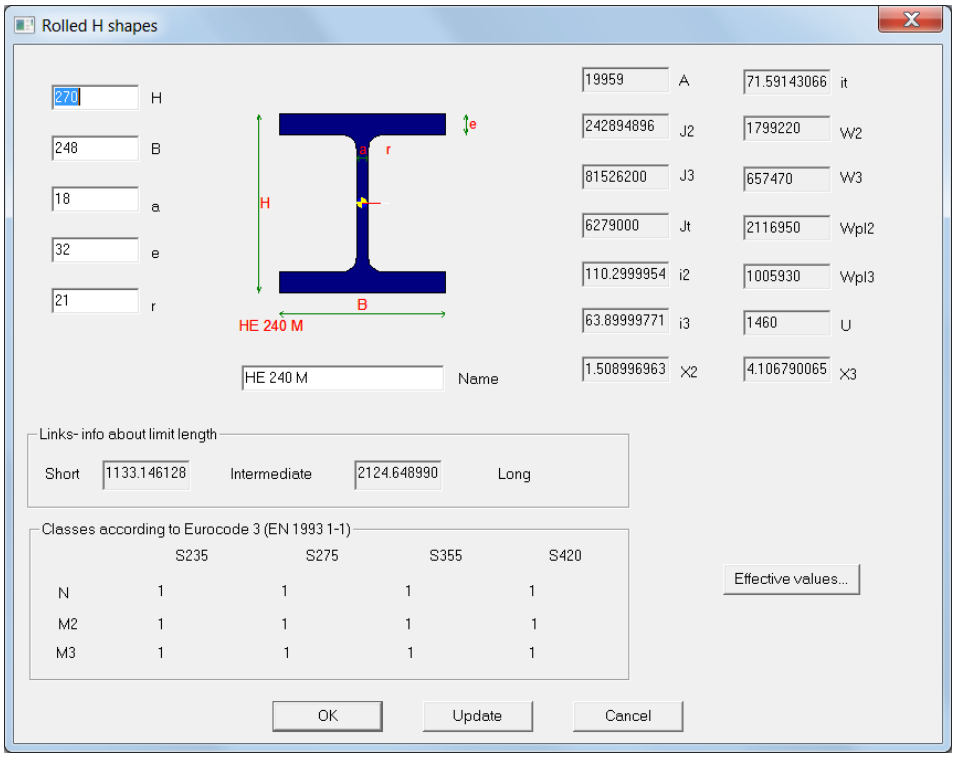
The section name is shown underneath (this can be edited if the profile is a new one being added).

All the dimensions are interpreted in terms of the active units.

The sectional properties are shown on the right of the image, and are computed automatically from the data entered. Press **Update** to refresh the values on the basis of the current dimensions. Additional information may be computed automatically for certain kinds of profile.

To confirm the insert of a new section as a block in the current BRICSCAD© document, click **OK**.

3.2.2.2 Dialog: H Rolled cross-sections



This dialog allows you to define the dimensions of a generic rolled I- or H- cross section and to study its properties. In the case of standard profiles HEA, HEB, HEM and IPE dimensions are not editable.

To the right of the drawing section all the typical cross-section data are listed. To the left of the drawing section are listed the editable dimensions.
The **Refresh** button allows you to update data with a gray background based on data with a white background (free parameters).
The button **Effective values** refers to the dialog in which the effective properties of the profile are calculated. This feature is not available in SMA.

There is also information on the lengths for short and intermediate seismic links according to what is specified in the Eurocode 8 and the typical classes according to EC3.

Dimensions

These sections are identified by the following parameters:

h	total height
b	width of the flange
a	thickness of the web
e	thickness of the flange
r	curvature radius of the fillet

The following necessary inequalities are applicable :

* All the dimensions must be greater than 0;
* h >= 2e+2r
* b >= 2e+2r

The moment of torsional inertia does not include the effects of the warping (secondary twist).

The principal axes of inertia and properties of the gross section

The principal axes y and z of the section are called respectively 2 and 3. The moment of inertia J2 is therefore the moment of inertia of the section with respect to the y axis. In the figure are represented, the barycenter and the main axis 2 (y); the axis 3 (z) is perpendicular to the axis 2.

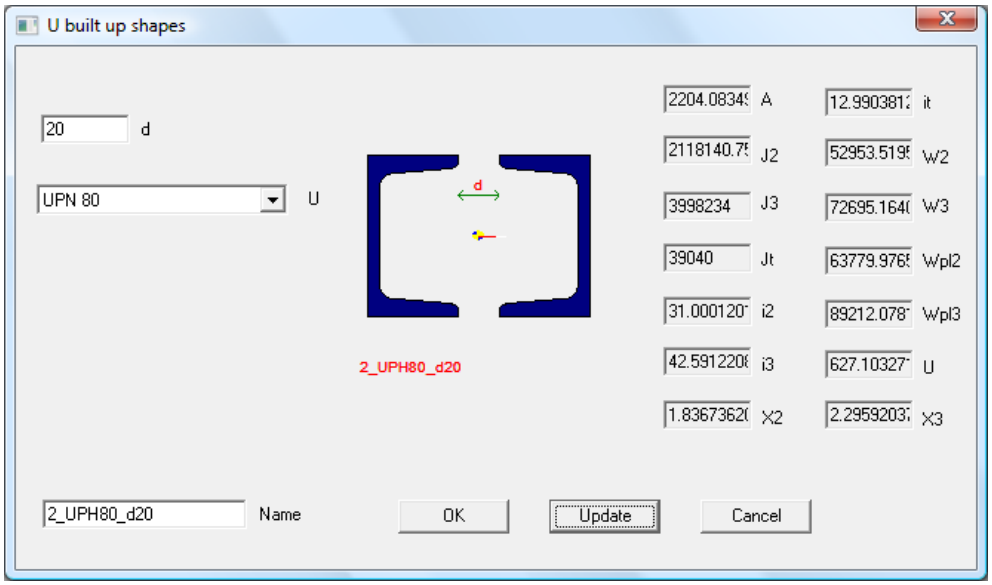
A	total area
J2	moment of inertia with respect to the principal axis 2
J3	moment of inertia with respect to the principal axis 3
Jt	moment of torsional inertia
i2	radius of inertia with respect to the principal axis 2
i3	radius of inertia with respect to the principal axis 3
X2	shear factor for shear parallel to principal axis 2
X3	shear factor for shear parallel to principal axis 3
it	radius of inertia
W2	elastic modulus of resistance to bending around axis 2
W3	elastic modulus of resistance to bending around axis 3
Wpl2	plastic modulus of resistance to bending around axis 2
Wpl3	plastic modulus of resistance to bending around axis 3
U	surface of painting

3.2.2.3 U or L composed shapes (dialogs)

These dialog boxes are used for composite U-sections (rolled or cold-formed) and rolled L-sections. As detailed below, they look different for different types of composite section.

Here it must be chosen the cross section and the distance between the identical cross-sections. All the dimensions are interpreted in terms of the active units.

All the cross-section in the archive with the proper type are listed in the drop-down list. They are ordered by area.



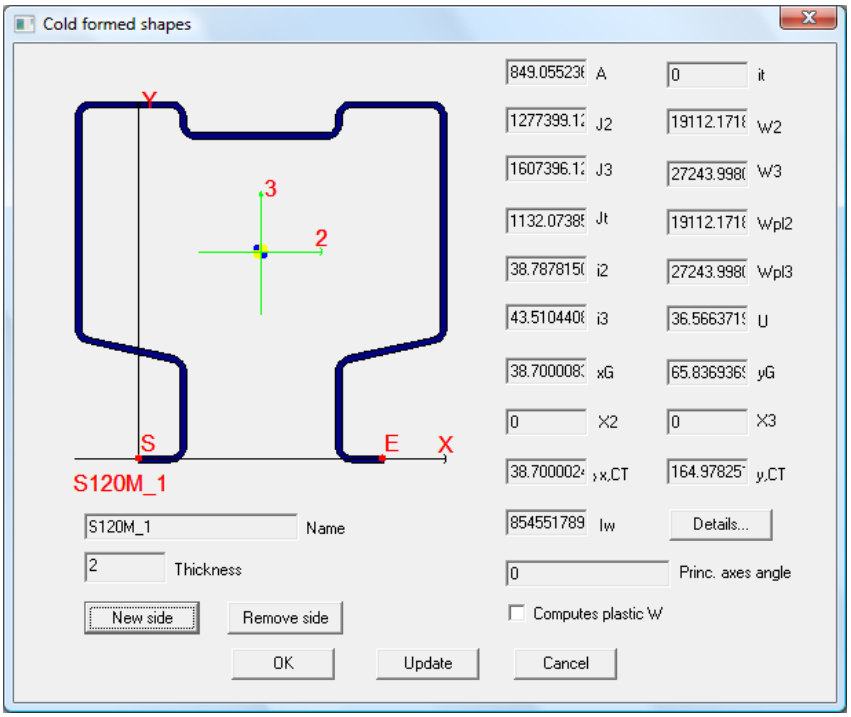
Example: two [ ] channel bars

There are six different types of composite section associated with these dialog boxes:

2 channels, [ ]
2 channels,] [
2 cold-formed sections,] [
2 angles forming a Tee _||_
2 angles in a cross formation +
4 angles in a cross formation +

In all cases, the basic section is shown together with the distance between the sections and the properties of the section as computed automatically.

3.2.2.4 Cold formed shapes (dialog)



This dialog box is used to describe generic cold-formed sections. First of all, the section thickness is specified, then the sides are added and finally a name is chosen. The thickness may be altered after adding the sides; to do so, simply edit the value and click the update button.

All the dimensions are interpreted in terms of the active units.

The **New Side** button is used to insert a side and gives access to the corresponding dialog box 50 (for selecting the type of the side - straight or circular - and the means of adding the new side).

The **Remove Side** button deletes the last side added.

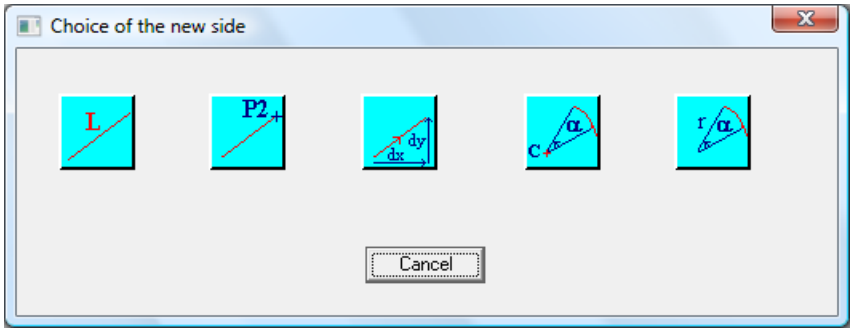
The computational quantities are updated each time a side is added. Sides can also represent holes.

Sections can be open or closed. A closed section ends at precisely the same point where it started. The torsional moment of inertia of a closed section is computed using Bredt's formula.

After adding all the sides, the user can tell the program to calculate the plastic moduli, which can be done by ticking the relevant box and then clicking the **Update** button. This triggers an iterative loop.

The **Details** button gives access to another window which lists some important properties, computed as per appendix C of EN-1993-1-3.

3.2.2.4.1 Choice of the new side (dialog)

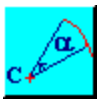


With this dialog box, the user can choose which type of side to add, and the numerical criterion for doing so. Each image button corresponds to a different selection, as below:

adds a straight-line side tangential to the last side added. Only the length of the new side needs to be given (through a dedicated dialog box 52).

adds a straight-line side. The absolute coordinates of the new point need to be given (through a dedicated dialog box[52]).

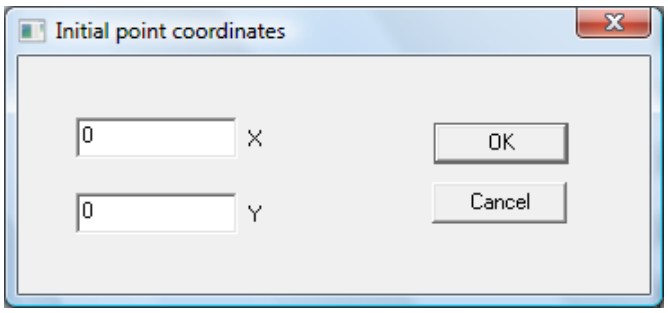
adds a straight-line side. The coordinates of the new point need to be given relative to the previous one (through a dedicated dialog box[53]).



adds a circular side. The coordinates of the centre and the arc angle in degrees need to be given (through a dedicated dialog box[54]).

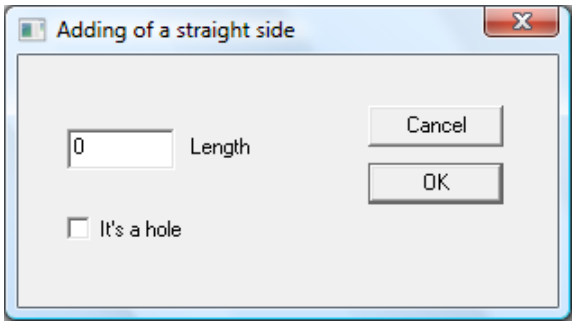
adds a circular side. The mean radius and the arc angle of the new side need to be given (through a dedicated dialog box[55]).

3.2.2.4.1.1 Initial point coordinates (dialog)



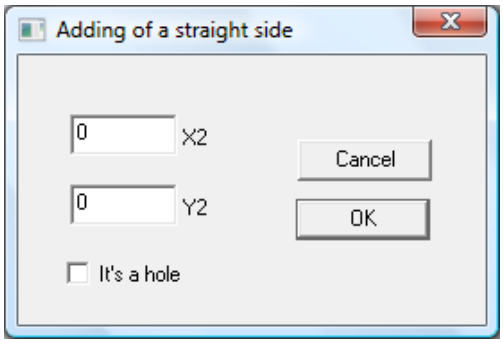
When the first side of a cold-formed section is added, the program prompts for the coordinates of the initial point, i.e. the first extremity of the side being added. This dialog box is used to enter the coordinates of the initial point (the active units of measure apply).

3.2.2.4.1.2 Adding of a straight side (dialog)



This dialog box is used to input the length of the new side in the active units, and whether or not this side represents a hole. The side will be added so as to be continuous with the last side introduced. If no sides have yet been added, the button which gives access to this dialog box is inactive.

3.2.2.4.1.3 Adding of a straight side (dialog)

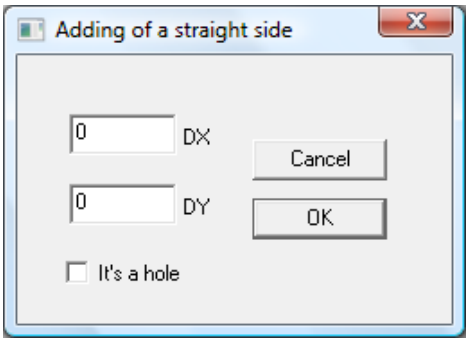


This dialog box is used to input the coordinates of the second extremity of the new side, in the active units, and whether or not this side represents a hole. The first extremity of the new side coincides with the second extremity of the last side added. If no sides have yet been added, the system prompts for the coordinates of the initial point instead (via the Coordinates-of-initial-point 51 dialog box).

Where the new side forms a cusp with the previous one, the program may accept or not the side, however it can automatically add a circular-arch connecting edge to generate a continuous transition between the new and old sides, thus automatically eliminating the cusp without the user needing to concern themselves with the problem. In this case the user is prompted to specify the internal radius of the edge to be introduced – see Internal radius of curvature 55.

It is also possible to accept cusps.

3.2.2.4.1.4 Adding of a straight side (dialog)



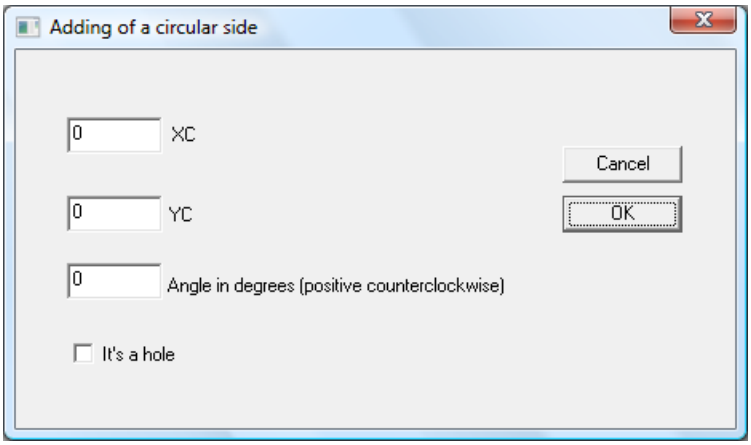
This dialog box is used to specify the coordinates of the second extremity of the new side (in the active units) relative to the last extremity added, and whether or not this side represents a hole. The first extremity of the new side coincides with the second extremity of the last side added. If no sides have yet been added, the system prompts for the coordinates of the initial point instead (via the Coordinates-of-initial-point 51 dialog box).

Where the new side forms a cusp with the previous one, the program may accept or not the side, however it can automatically add a circular-arch connecting edge to generate a continuous transition between the new and old sides, thus automatically eliminating the cusp without the user needing to concern themselves

with the problem. In this case the user is prompted to specify the internal radius of the edge to be introduced – see Internal radius of curvature 55.

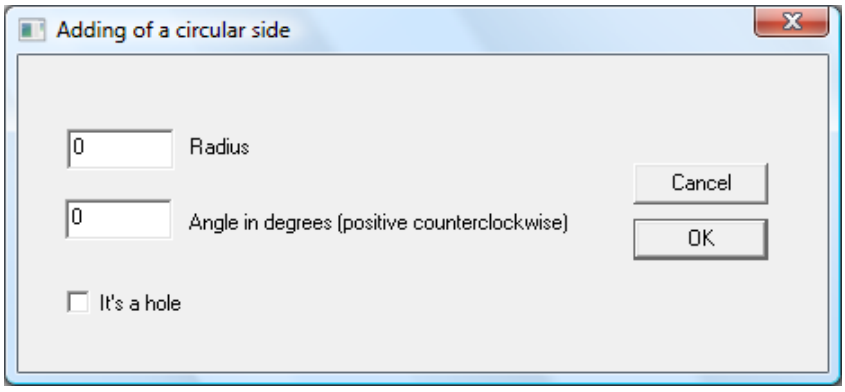
It is also possible to accept cusps.

3.2.2.4.1.5 Adding of a circular side (dialog)



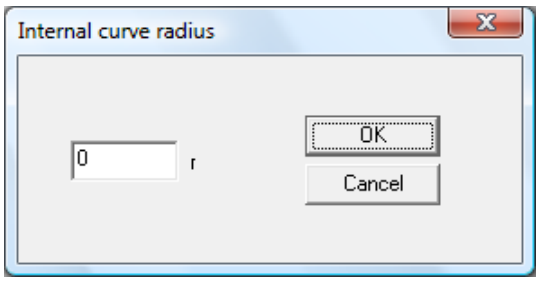
This dialog box is used to input the coordinates of the centre of the circle to which the new side belongs, the arc angle of the new side, and whether or not the side to be added represents a hole. The arc angle is expected in degrees and is positive if measured anticlockwise. The side will be constructed so as to be tangential to the previous side added. If no sides have yet been added, the system prompts for the coordinates of the initial point instead (via the Coordinates-of-initial-point 51 dialog box) and assumes that the initial tangent is horizontal. The coordinates of the centre are interpreted in terms of the current units of measurement.

3.2.2.4.1.6 Adding of a circular side (dialog)



This dialog box is used to input the radius of the circle to which the new side belongs, the arc angle of the new side, and whether or not it represents a hole. The arc angle is expected in degrees and is positive if measured anticlockwise. The side will be constructed so as to be tangential to the previous side added. If no sides have yet been added, the system prompts for the coordinates of the initial point instead (via the Coordinates-of-initial-point 51 dialog box) and assumes that the initial tangent is horizontal. The coordinates of the centre are interpreted in terms of the current units of measurement.

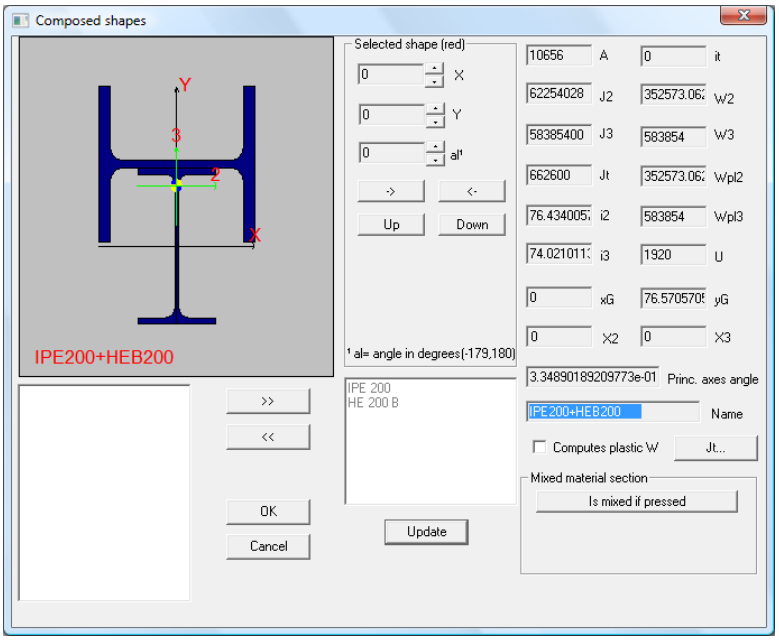
3.2.2.4.1.7 Internal curve radius (dialog)



When a straight-line side is added which forms a cusp with the previous side, SMA prompts the user for whether or not a circular edge should be added in order to

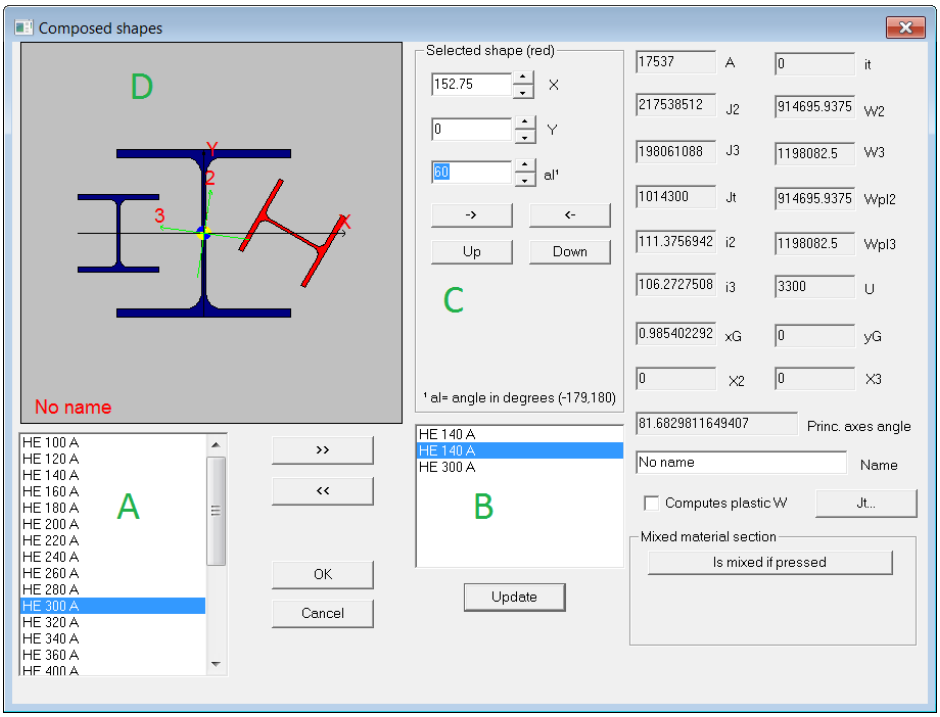
restore continuity. If the user confirms, the programs then prompts for the internal radius of the arc to be added, using this dialog box. The data must be supplied in the current units.

3.2.2.5 Composed shapes (dialog)



This important dialog box is where composed shapes data are input, and is therefore a true working environment. The dialog box is also used to output shape information.

When the background of a given field is white, this means that datum is editable, if the background is gray datum is read only (it depends on parameters or the dialog box is used in inquire mode).



Meaning and use of controls

Control A (see image)

Here all available shapes are listed, that is all the shapes which can be used to create the new section. The available shapes are those *extracted* when the command was executed. Therefore before executing command you must extract at least the shapes you plan to use in order to create the new shape. Among available sections there are composed and cold formed shapes, so you can have composed by cold formed and composed by composed.

Control B (see image)

Here are listed the sections used to create the new shape, that is the shapes chosen until now. The selected shape (blue row) is the one painted in red in the full drawing. If a cross section is used more than once, it will appear more than once in this list.

Button >>

It is used to add the selected shape in control A to shapes in control B: that is to add a new composing shape.

Button <<

It is used to remove the selected shape from control B.

Controls in C area (see image)

These controls are used to move the selected shape in control B. Controls "**X**", "**Y**" stands for X and Y coordinate of selected shape center, while "a" is used to rotate the shape (a is the angle between selected shape x reference axis and composed shape X reference axis). Button "**->**", "**<-**", "**Up**" and "**Down**" are used to translate selected shape so as to search for tangent sides. For instance clicking "**->**" the selected shape will be moved on the right until one of its sides gets tangent to one of the other shapes. Move direction depends on the button choice. Using this command is very useful because it allows a fast and precise move of composing shapes, one relative to the others.

"Compute plastic W " control

If this control is active (a tick is applied) computations will include the iterative procedure used to compute plastic W. We suggest to activate the control only once you've finished to set the shapes, otherwise shapes movement will be lowered down. After you've applied the tick you must press **Update** button. If this computation is not required plastic moduli are set equal to elastic ones.

Update button

It is used to update computed data after a change which does not imply automatic computation. If, for instance, you manually modify data reported inside controls "X", "Y", and "a", or you modify the "compute plastic W" control value, you must press **Update** to get correct values.

How composed shapes are added

Once you've extracted composing shapes, if you choose to add a composed shape you'll get this dialog box.

Composing shapes are chosen among those of control A and put in control B (even more than once), using control ">>". Now you select each shape one by one, and position it in the right place using controls in C area. Buttons "**->**" "**<-**" "Up" and "Down" are particularly useful, 'cause usually shapes are mutually tangent . If you wish to change a shape added in control B you first have to remove it, and then you add the replacing shape.

If during sections moving you cross a physically not allowable layout, "OK" button gets grayed, to avoid the addition of meaningless shapes.

Moving shapes, do keep into account that shape coordinates are their center coordinates with respect to global reference axes.

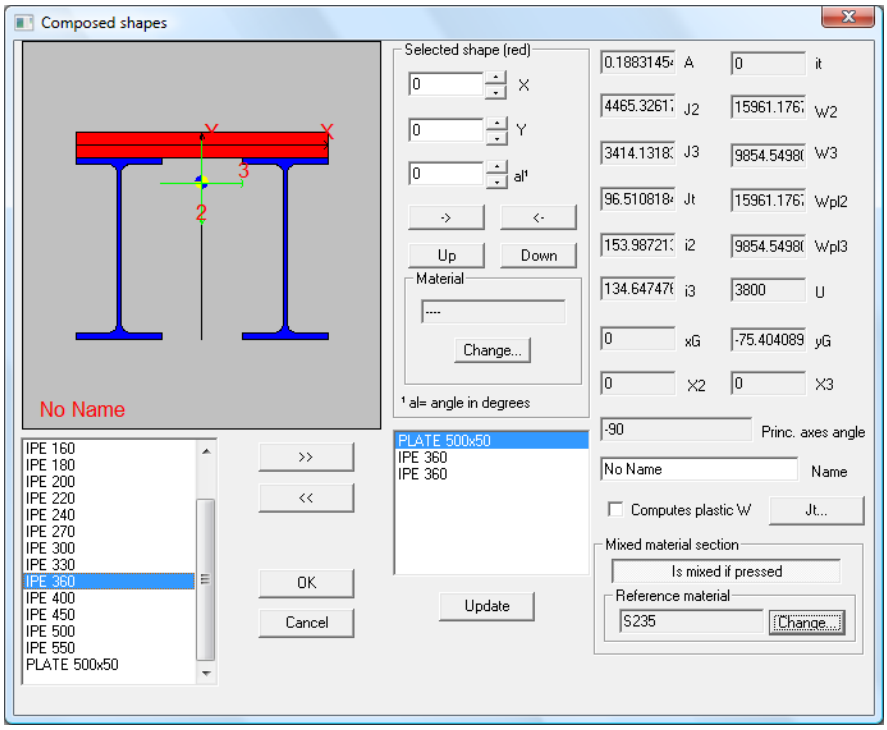
If you wish detailed information about composing shapes you can double click on the shape (both in control A and B).

Once you've got the desired section you place a tick in "Compute plastic W " and press the **Update** button. Plastic moduli are update and are – obviously – always greater than elastic ones.

Plastic moduli computation is not a trivial problem. It may happen that the algorithm does not converge: you will get a message. This usually happens when plastic neutral axis crosses regions where sharp curves are present. Usually in sections having one center line this does not happen.

As to symbol meaning see details 70

MIXED MATERIALS COMPOSED CROSS_Sections



This range of sections has been further extended to include mixed sections. In essence, the sections that make up the composite shapes may optionally be allocated a material, which in general may be different for each.

A reference material then needs to be selected for the final composite section (the material to which all the computational quantities will be homogenized).

To compute the area, the center of gravity and the moments of inertia, the homogenization coefficient given by $K_{el}=E/E_r$ is used, where E_r is the Young's modulus of the reference material.

The elastic section moduli are computed such that multiplying W **by the yield stress of the reference material f_{yr}** gives the moment of first plasticization of the section, i.e. the moment at the elastic limit (at the first point to yield, regardless of the material of which it is made).

$$M_{el} = W * f_{yr}$$

In practice

$$W = Min\{ (J_{om} * K_{pl}) / (d K_{el})\}$$

where:

- J_{om} is the moment of inertia, homogenized for the main axis considered;
- K_{pl} is the ratio between the yield stress of the material at the point considered and the yield stress of the reference material;
- K_{el} is the ratio between the Young's modulus of the material at the point considered and the Young's modulus of the reference material;
- d is the distance of the point considered from the main axis considered.

In practice, the computation of the plastic section moduli involves calculating the plastic W values by homogenizing the areas with the factor K_{pl}, so that multiplying W_{pl} **by the yield stress of the reference material** gives the moment of full plasticization of the section.

$$M_{pl} = f_{yr} * W$$

Note that there is no need for any of the sections to be made of the reference material, and therefore homogenization can be carried out against any material.

The modulus of elasticity and yield stress data is taken from the archive, without any multiplication coefficient being applied.

The “theory”

The “generic composite” section type now enables us to define mixed sections, namely sections that are made up of various other sections, each of which being made of a different material, and where all are assumed to be fully bonded, with the plane sections maintained.

Mixed steel/concrete and steel/wood sections can thus be specified, with various types of concrete or other materials of choice.

All the properties of the section will be standardized against an equivalent material, respect to which the various component parts will be homogenized. There is no need for any of the component sections to be made of the reference material.

The materials of the component sections are sourced from the archive, hence they must be present in it. Where:

n is the number of sections present

E_r is the stretch modulus of the reference material

E_i is the stretch modulus of the generic material at generic point i

s_{yr} is the yield stress of the reference material

s_{yi} is the yield stress of the generic material at the generic point of reference, i

$K_{eli} = E_i/E_r$

$K_{pli} = s_{yi} / s_{yr}$

We have

$$A = \sum_{i}^{n} \int_{Ai} K_{eli} dA$$

$$S_x = \sum_{i}^{n} \int_{Ai} y K_{eli} dA$$

$$S_y = \sum_{i}^{n} \int_{Ai} x K_{eli} dA$$

$$x_g = S_y / A$$

$$y_g = S_x / A$$

$$I_x = \sum_{i}^{n} \int_{Ai} y^2 K_{eli} dA$$

$$I_y = \sum_{i}^{n} \int_{Ai} x^2 K_{eli} dA$$

$$I_{xy} = \sum_{i}^{n} \int_{Ai} xy K_{eli} dA$$

from which the principal axes and the moments of inertia J_2 and J_3 about them can be derived using the standard methods.

As for the section moduli W, these are defined such that, when multiplied by the yield stress of the reference material, they bring the section, at some point on it and for some material, to its first yield.

In practice, given the point P_i of material i, and where d is the distance from the main axis considered, we have

The stress at the point as if it were made of the reference material:

$$s_r = Md/J$$

The homogenized stress (the true stress of the actual material at that point):

$$s_i = MK_{eli}d/J$$

The limit condition applies when this stress is equal to the yield of material "i", and thus:

$$MK_{eli}d/J = K_{pli}s_{yr}$$

The moment that achieves this value is given by:

$$M = (K_{pli}J/dK_{eli})s_{yr}$$

Therefore the modulus of the mixed section is given, as the point i varies, by the minimum value of:

$$W = \min\{K_{pli}J/dK_{eli}\}$$

With this assumption, the moment at the section's elastic limit is given by the usual formula $M = Ws_{yr}$.

As regards the plastic moduli, these are obtained by homogenizing the areas with K_{pl} rather than K_{el}, using the formula:

$$M_{pl}=W_{pl}s_{yr}$$

Thus the moment of full plasticization of the section is obtained by multiplying W_{pl} by the yield stress of the reference material.

At this point it is worth making a few remarks about how this data is used.

With the mixed sections, we need to proceed as follows. The sections must be attributed to the elements in the usual way, although in order for the use of these sections to be meaningful, the **only** material that must be allocated to them is the reference material, i.e. the material that the sectional properties have been homogenized against.

That said, the mixed sections may be used interchangeably with the others, and the elastic behaviour of the mixed beams can be correctly modelled, in order to obtain stresses and displacements that are consistent with the theory.
In terms of the results that follow on from this, we need to make the following observations.

The stress values (N/A), (M/W) and (N/A+M/W) are meaningless, in that the stress obtained is an ideal, homogenized stress, i.e. it is the stress that would apply at the point of first plasticization if this were made of the reference material, which in general is not going to be the case.

An indirect estimate of the level of utilisation of the section in the elastic phase is given by the following dimensionless quantity:

$$sfr = \left|\frac{N}{N_{el}}\right| + \left|\frac{M_2}{M_{2el}}\right| + \left|\frac{M_3}{M_{3el}}\right| = \left|\frac{N}{A_{el}\ \sigma_{yr}}\right| + \left|\frac{M_2}{W_{2el}\sigma_{yr}}\right| + \left|\frac{M_3}{W_{3el}\sigma_{yr}}\right|$$

where the W values are those computed by the program (and already described above) as elastic Ws, while Ael is a homogenized area defined so as to give the axial action of the first plasticization when multiplied by syr.

Given that

$$N = \sum_{i=1}^{n} \int_{Ai} (\varepsilon E_i) dA = \varepsilon \left(\sum_{i=1}^{n} \int_{Ai} E_i dA \right) = \varepsilon E_r \left(\sum_{i=1}^{n} \int_{Ai} K_{eli} dA \right) = \varepsilon E_r A$$

where A is the homogenized area computed by the program, the axial action of the first plasticization is obtained by requiring that the normal stress at the generic point is equal to the yield stress, taking the minimum axial action:

$$\varepsilon = \frac{N}{E_r A}$$

$$\sigma_i = E_i \frac{N}{E_r A} = \sigma_{yi}$$

from the above, the following result applies at the generic point

$$N = A \frac{K_{pli}}{K_{eli}} \sigma_{yr}$$

hence

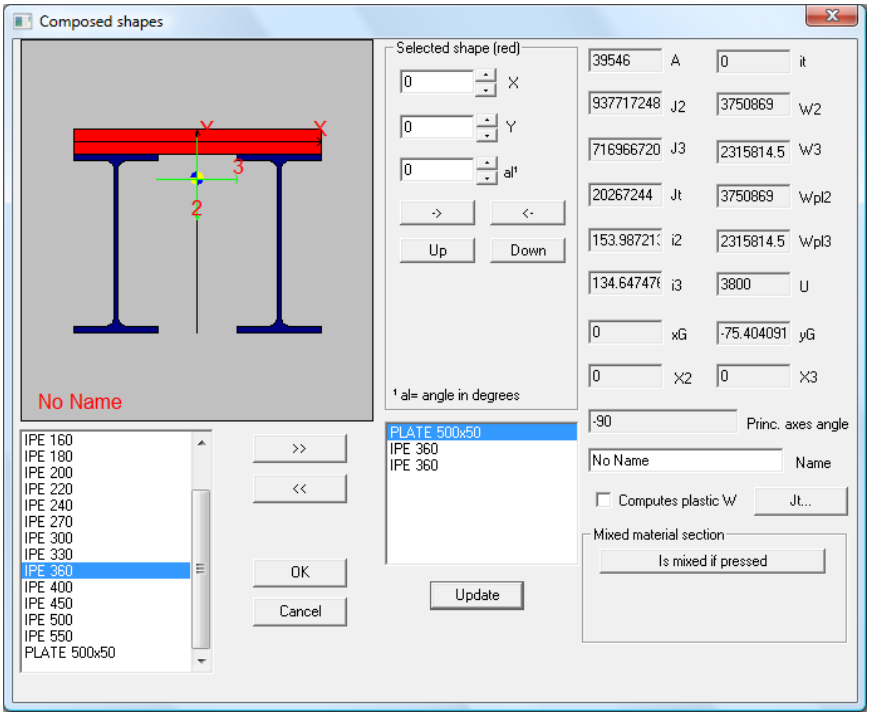
The axial action of full plasticization is obtained instead by the following relation:

$$A_{pl} = \sum_{i}^{n} \int_{Ai} K_{pli} dA$$

We can now use these results to define a coefficient of plastic utilisation, as follows:

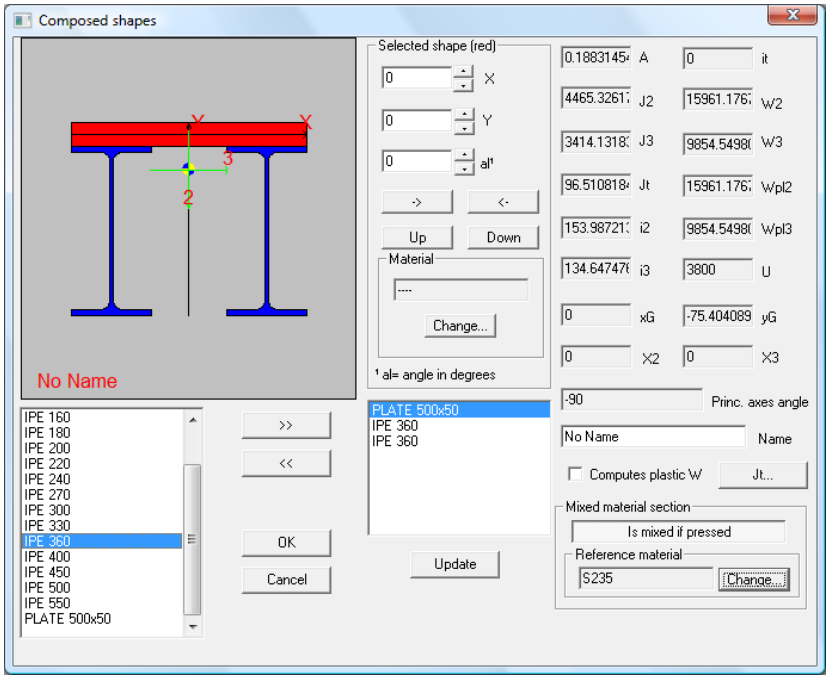
$$sfr = \left|\frac{N}{N_{pl}}\right| + \left|\frac{M_2}{M_{2\,pl}}\right| + \left|\frac{M_3}{M_{3\,pl}}\right| = \left|\frac{N}{A_{pl}\sigma_{yr}}\right| + \left|\frac{M_2}{W_{2\,pl}\sigma_{yr}}\right| + \left|\frac{M_3}{W_{3\,pl}\sigma_{yr}}\right|$$

MIXED SECTIONS – USER NOTES

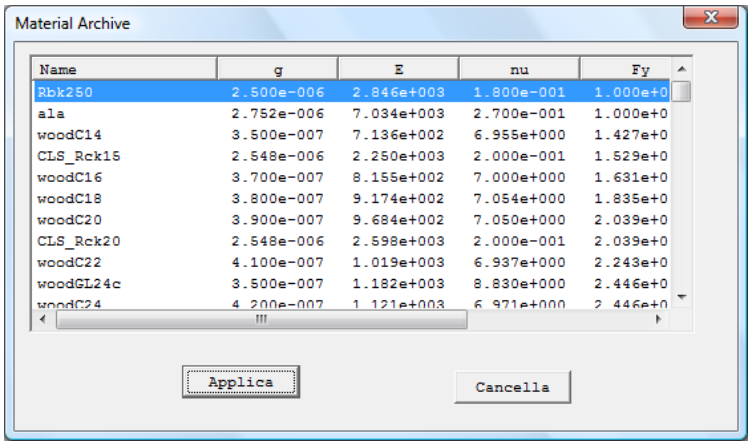


In practice, the controls have been relocated to make room for a new "gateway" button, "Is Mixed if pressed". Initially the button is un-pressed, and the dialog box appears as shown in the figure above. When the button is pressed, the dialog box appears as seen below, and the controls are enabled to provide the functionality needed to define:

- The reference material for the overall section (the Change button in the panel at the bottom).
- The material of which the generic section selected and shown in red is composed (the Change button in the panel containing the translation and rotation controls).



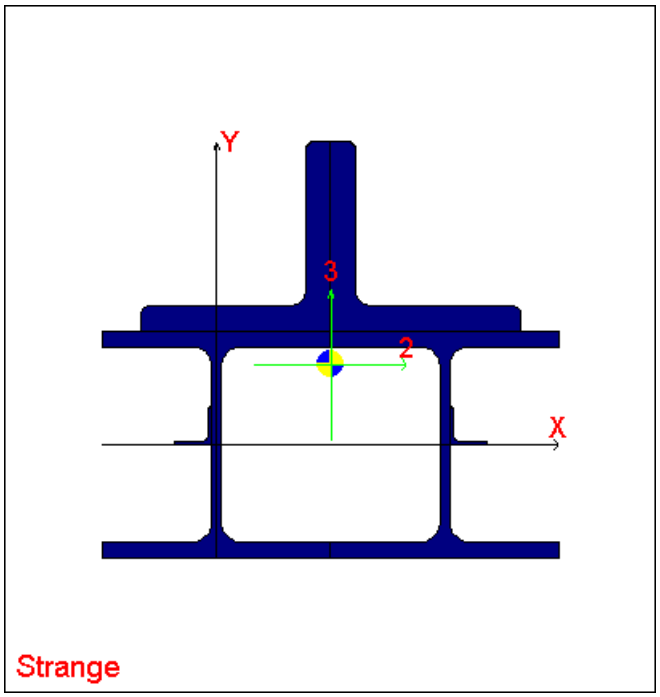
The reference material and the material of each individual section that makes up the overall section will need to be defined by selecting from the list of materials in the archive in the usual way, using the dialog box shown below.



Both the component sections and the component materials must already be present in the archive before the command is run.

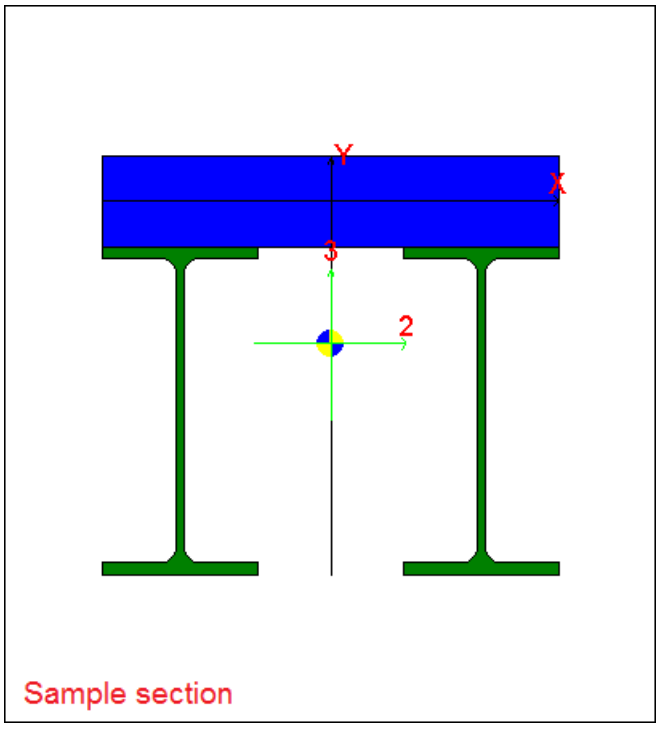
To compute the plastic moduli, tick the corresponding box and click on the Update button.

3.2.2.6 Generic composed shapes



SMA can describe generic composite shapes. This means that you describe sections obtained by collecting an arbitrary number of shapes, defining each shape position in plane. (shape center position and rotation angle).

It is also possible to define composite cross-sections of composite cross-sections.



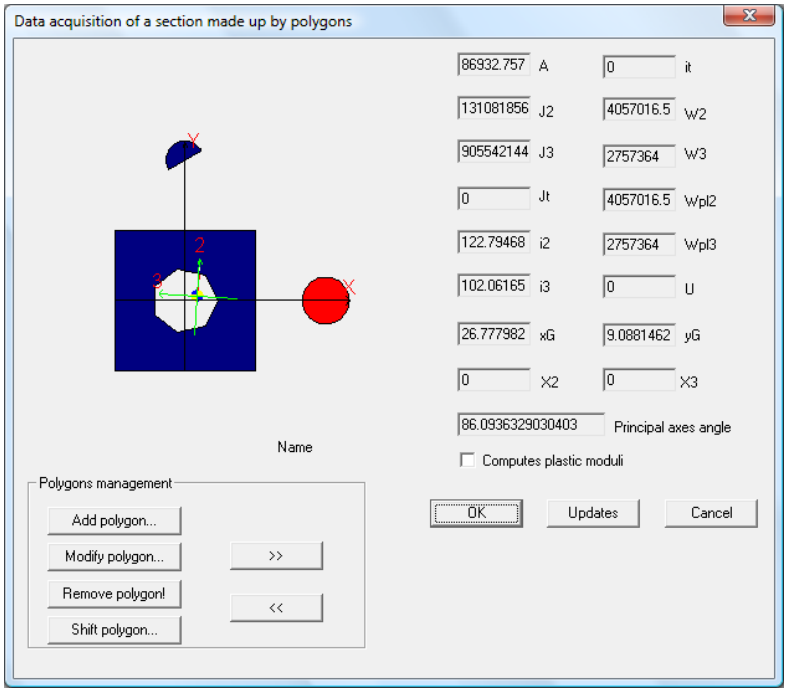
Mixed sections can be handled (those with an arbitrary number of materials) as particular generic composite shapes.

3.2.2.7 Data acquisition of a section made up by polygons (dialog)

This important dialog box is the point of input for the sections which are composed of combinations of generic polygons, and is therefore a tool of some significance. This dialog box is also used to provide information on the profile.

When the field relating to a given quantity has a white background, it can be edited; otherwise, when greyed-out, it is non-editable (as the quantity is derived from the previous ones).

If the dialog box is opened for information purposes only, all the quantities are greyed-out and cannot be edited.



A polygon can be added by clicking the "**Add polygon**" button, which brings up an additional dialog box[74] in which a polygon can be defined. When closing the dialog box, the user is prompted to specify whether the polygon represents a filled or an empty object.

The "**>>**" and "**<<**" buttons are used to select one of the polygons which make up the section, and enable the user to move from one to another. The currently selected polygon is shown filled in in red.

The "**Modify Polygon...**" button allows the user to re-edit the currently selected polygon, by reaccessing the dialog box which is used to define an individual polygon.
The "**Remove Polygon**" button deletes the currently selected polygon.
The "**Translate Polygon**" button enables the user to specify a translation vector to apply to the currently selected polygon.

To compute the plastic W values as well, tick the "**Compute Plastic Ws**" box and then click **Update**.

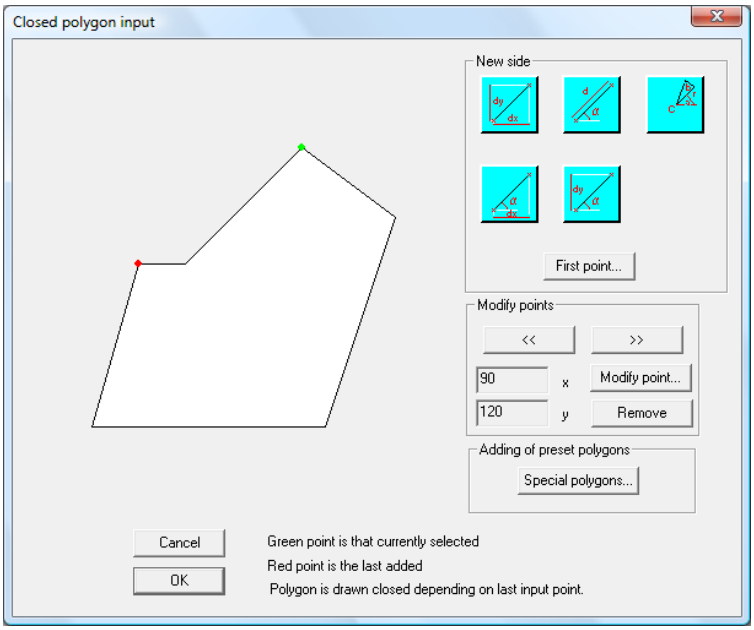
A unique name must be chosen for the section to distinguish it from those already present in the Archive.

Dimensions are given in the active units of measurement.

Given the generality of the problem, the torsional moment of inertia can not be computed automatically by the program (a differential equation on the partial derivatives over the domain would need to be solved when only the primary torsion would be available). It is therefore the user's responsibility to assign reasonable values for the torsional moment of inertia and the radius of gyration.

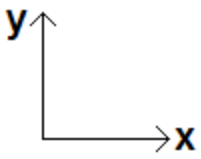
Nevertheless, the computation of the plastic section moduli is carried out automatically by the program, provided that there is a tick in the dedicated box. This initiates an iterative process which enables the plastic section moduli to be evaluated.

3.2.2.7.1 Closed polygon input (dialog)



This dialog box enables the form of a generic polygon plate to be defined.

Clicking on **First point** brings up another dialog box in which the X- and Y-coordinates of the first point must be specified, in the active units of measurement. The reference system of the polygon is shown by the following figure.



There are various ways of determining the next point:

Inputting DX and DY  – this requires the distance in relative coordinates to be specified between the new point and the last one input (shown in red).

Inputting an angle and a distance  – requires the distance from the last point and the angle (in degrees) between the new side and the horizontal.

Inputting an arc  – requires the coordinates of the centre of the arc, the initial and final angles (in degrees), the radius and the number of subdivisions into straight-line segments. The coordinates and radius must be specified in the active units of measurement.

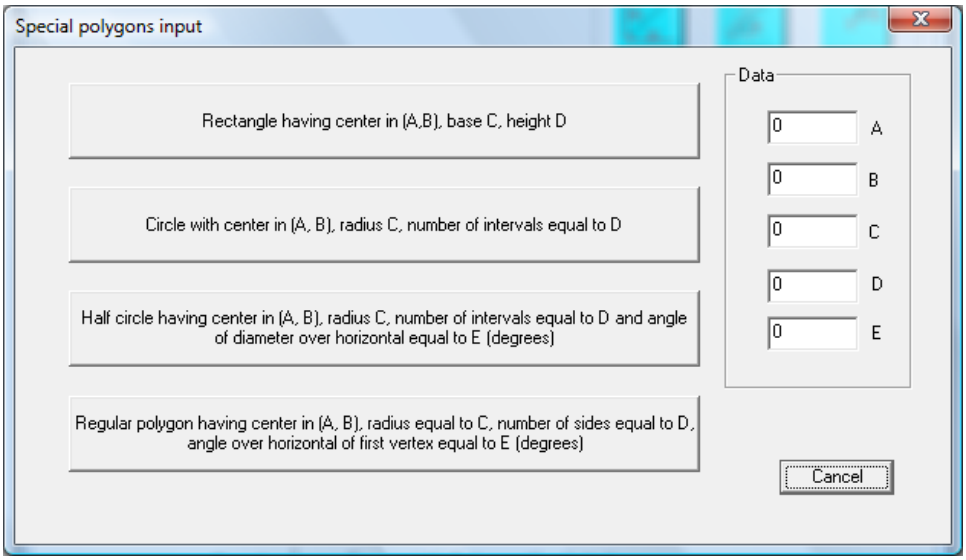
Inputting an angle and a distance (projected DX)  – requires an angle (in degrees) and the projection of the new side along the X-axis, in the current units of measurement.

Inputting an angle and a distance (projected DY)  – requires an angle (in degrees) and the projection of the new side along the Y-axis, in the current units of measurement.

Once all the necessary points have been added, the polygon is confirmed by clicking *OK*.

The current point is shown in green. The "**<<**" and "**>>**" arrows are used to change the current point, which can be removed (using the **Remove** button) or modified (with the **Modify point...** button), which provides access to the dialog box with its coordinates.

The **Special polygons...** button brings up a further dialog box which allows whole polygons to be added by specifying some of their parameters. These polygons are added to others which have been prepared earlier, and may themselves be modified by changing their points.



The following polygons may be added:

- Rectangle, given the coordinates of the centre (A and B), the base (C) and height (D)
- Circle with centre (A, B), radius C and number of intervals D
- Semicircle: in addition to the data in the previous bullet point, the angle of the diameter above the x-axis must also be specified (in degrees)
- Regular polygon of centre (A, B), radius C, number of sides D and angle above the horizontal of the segment joining the centre with the first vertex equal to E (in degrees)

3.3 CURRMAT

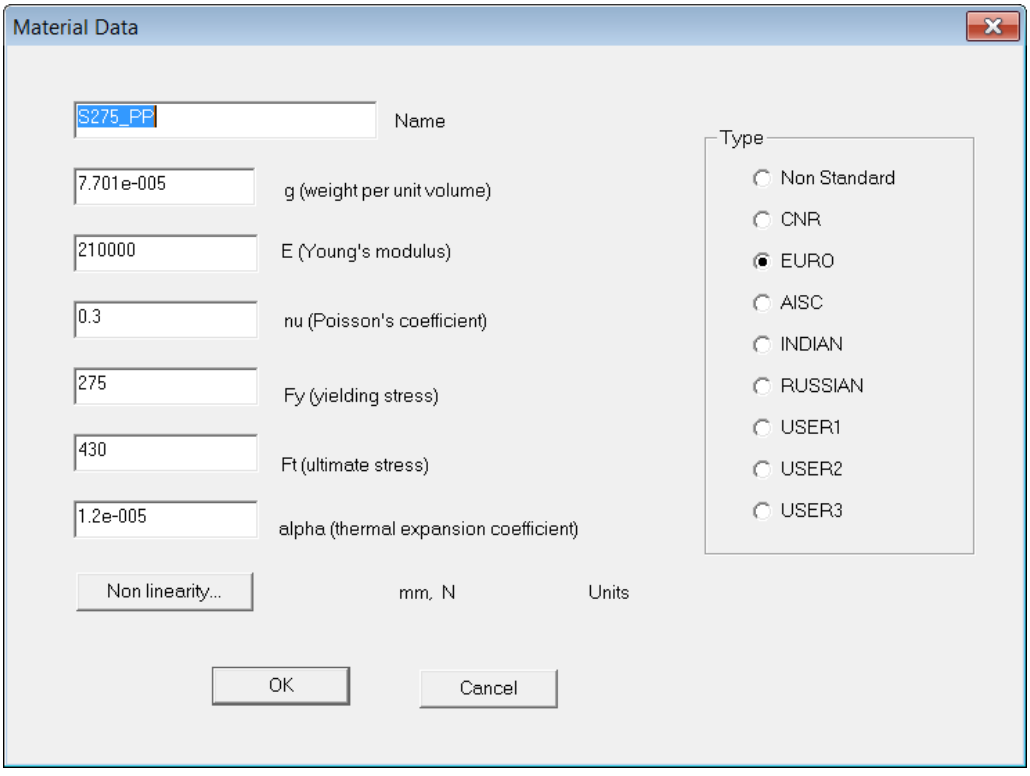
Enter the following string in the BricsCAD© command prompt:

CURRMAT

This command allows the modification and viewing of the current material properties thanks to a dedicated dialog box 77. The current material is only useful in order to properly drive the working mode of design filters 96 when accessing the cross-section database. Default material is European S235 (yield at 235 MPa).

3.3.1 Material data (Dialog box)

MATERIAL DATA (DIALOG BOX)



This dialog box is used to add a new material to the archive, to modify an existing material and to get information about a material.

If fields have a grey background, they can not be modified.

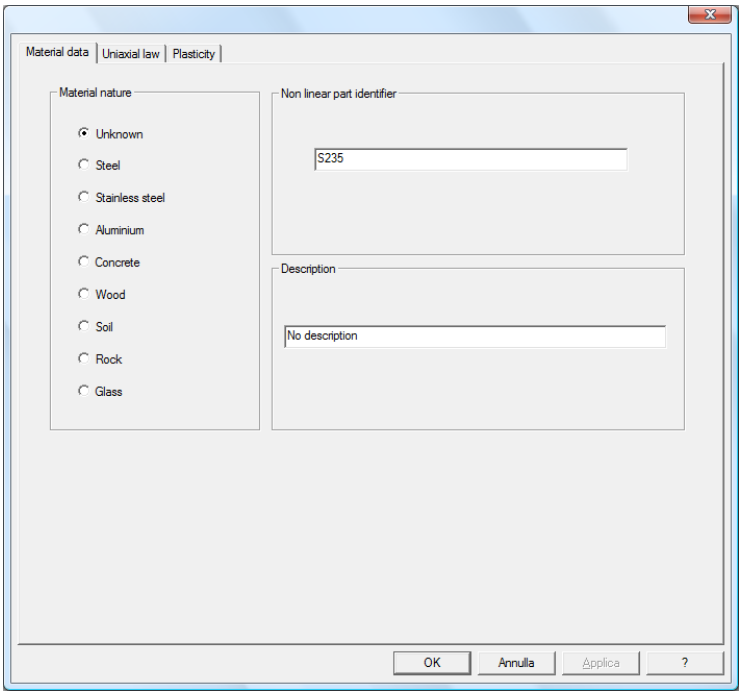
The meaning of symbols is the following:

- Name name of the material (max 20 characters)
- g weight per unit volume, in current units of measure
- E Young's modulus, in current units
- nu Poisson's ratio
- Fy yielding stress or elastic limit, in current units
- Ft ultimate stress, in current units
- Alpha thermal expansion coefficient, in current units

"Non linearity" button opens a property sheet 80 where nonlinear data for current material can be defined.

3.3.1.1 Material data (Property page)

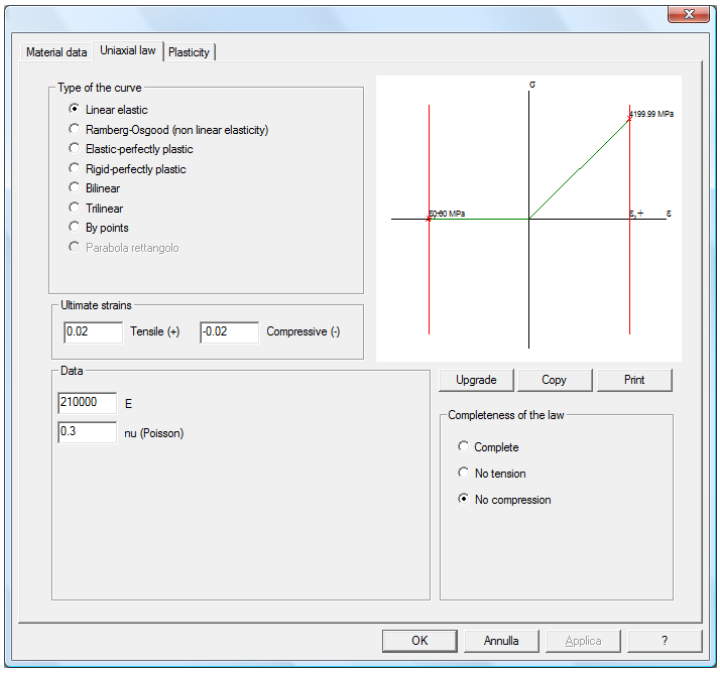
MATERIAL DATA (PROPERTY PAGE)



In this property page it is possible to assign to the material a nature (steel, wood, concrete, etc.), an identifier of nonlinear part and a description of the material itself.

This data is not needed in SMA, but is kept for outward compatibility.

3.3.1.2 Uniaxial law (Property page)



This property page is used to define the uniaxial constitutive law for the material. Linear elastic, non-linear elastic, elasto-plastic and rigid-plastic constitutive laws are available. Each law may be complete, no-tension or no-compression. We will look at all these laws in detail below.

This data is not needed in SMA, but is kept for outward compatibility.

The image to the right is updated when the button underneath is clicked, based on the parameters defined. The image can be copied to the clipboard or printed directly.

Ultimate strains

All the constitutive laws use positive (tension) and negative (compression) ultimate strains. If the material is linear elastic, it nevertheless has a limit, i.e. when its ultimate strains are reached: at these points, the material "breaks", the tension drops immediately to zero and the element ceases to respond (if it is the Gauss point of an element, this point is no longer taken

into account in the calculation). Hence, to simulate a material of unlimited elasticity, high strains must be set. The recommended value is 0.02, which corresponds to 210000x0.02=4200MPa for a typical steel. If the material is plastic, the plastic flow is interrupted when its limit strain is reached and it starts to break.

Completeness

All the constitutive laws can optionally be set up as no-tension or no-compression. During load reversal, if the ultimate strain has not been reached when the load is reversed, the material can begin to react again; if, on the other hand, the ultimate strain has been reached, then the material is broken, and reversing the load will not change this.

Data

This section is subject to change, as different parameters are required based on the type of curve chosen.

Type of curve

Appropriate parameters for the curve selected will be required in the Data section. The available curves are as follows.

- ***Linear elastic (unlimitedly elastic)***

The "unlimitedly" part is bound up with the amplitude of the ultimate strains. The modulus of elasticity, E, and the Poisson coefficient, nu, must be defined.

- ***Ramberg-Osgood (non-linear elasticity)***

This law is specified via the following parameters:

E	modulus of elasticity in linear phase
ν	Poisson coefficient
K	see formula
n	see formula

The Ramberg-Osgood law is a non-linear elastic law described by the following equation:

$$\varepsilon = \frac{\sigma}{E} + \left(\frac{\sigma}{K}\right)^{\frac{1}{n}}$$

- ***Elastic-perfectly plastic***

This law is specified via the following parameters:

E modulus of elasticity in linear phase

σ_y yield stress

ν Poisson coefficient

The plastic flow occurs without hardening.

- ***Rigid-perfectly plastic***

This law is specified via the following parameters:

σ_y yield stress

ν Poisson coefficient

Numerically, the infinite stiffness is simulated by E= 1.e12.

- ***Bilinear (elasto-plastic)***

This law is specified via the following parameters:

ν Poisson coefficient

ε_1 deformation at yield

σ_1 stress at yield

ε_2 ultimate deformation

σ_2 stress at ultimate deformation

As extrapolations are not permitted, it is a good idea to equate the extreme deformation to the ultimate strain. Except where the no-tension or no-compression option has been selected, it is assumed that the law is symmetrical.

The type of hardening may be isotropic, kinematic or mixed.

- ***Trilinear (elasto-plastic)***

This law is specified via the following parameters:

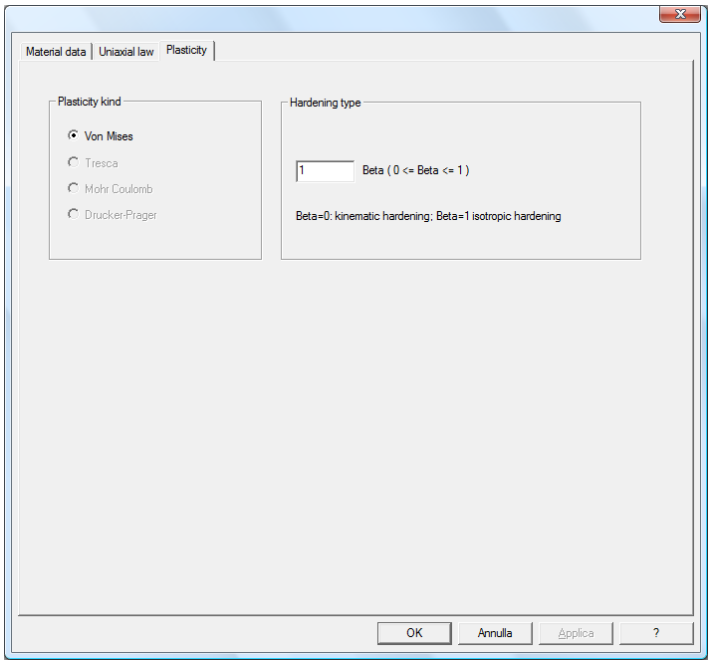
ν	Poisson coefficient
$\varepsilon 1$	deformation at yield
σ_1	stress at yield
ε_2	intermediate deformation
σ_2	stress at intermediate deformation
ε_3	extreme deformation
σ_3	stress at extreme deformation

As extrapolations are not permitted, it is a good idea to equate the extreme deformation to the ultimate strain. Except where the no-tension or no-compression option has been selected, it is assumed that the law is symmetrical.
The type of hardening may be isotropic, kinematic or mixed.

- ***By points (elasto-plastic)***

In this case, in addition to the Poisson coefficient (nu), a value pair (ε, σ) is given for each point defined. The origin must always be included. The points are in increasing order of deformation (first the maximum negative deformations, then the negative ones, then the origin, then the positive deformations, followed by the maximum positive deformation).
As extrapolations are not permitted, it is a good idea to equate the extreme deformation (positive and negative) to the ultimate strain.
The type of hardening may be isotropic, kinematic or mixed.
To insert a new point, make suitable entries in the "Eps" and "Sigma" fields, then click the "Add" button. A point with the selected coordinates will be added to the end of the list. To insert a point between other existing points, select the desired row and click "Insert" instead of "Add". To remove a point, select the corresponding row and click "Delete".
NB: the by-points law may be complete, no-tension or no-compression; complete laws must be symmetrical.

3.3.1.3 Plasticity (Property page)



In this property page it is possible to define the kind of material plastic locus and the kind of work hardening (β=0 kinematic, β=1 isotropic).

This data is not needed in SMA, but is kept for outward compatibility.

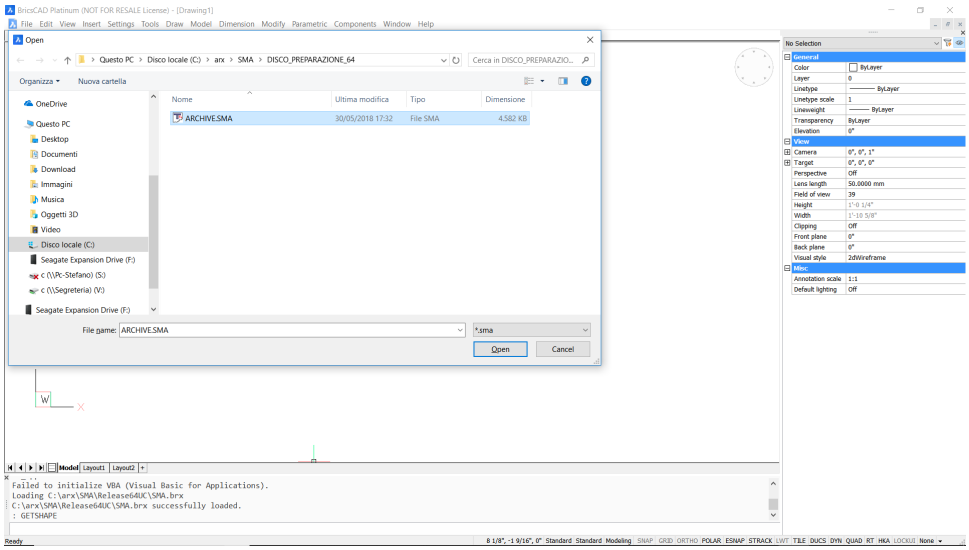
3.4 GETMAT

Enter the following string in the BricsCAD© command prompt:

GETMAT

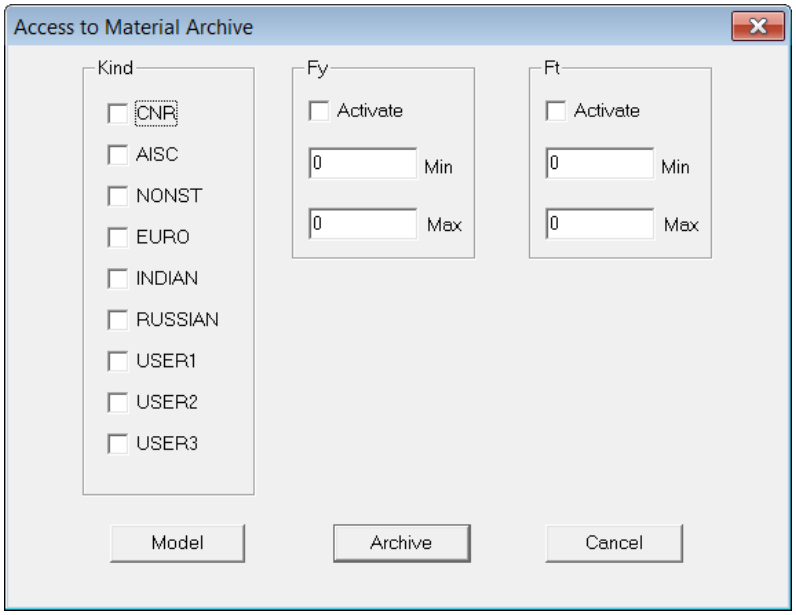
This command allows the choice of a material as current material, via a dedicated dialog box 85. The choice of the material is only useful in order to properly drive the working mode of design filters 96 when accessing the cross-section database. Default material is European S235 (yield at 235 MPa).

If the cross-section and material archive has not been loaded yet, you are prompted to choose the "archive.sma" file, which is the database file received with SMA, and in the same folder of SMA.BRX.



This file embeds all the cross-sections and materials of the existing archive (more than 19,000 cross sections, USA, UK, European, Indian, Russian, and more).

3.4.1 Access to material archive (dialog)



By clicking on **Model,** the user can select a material from those already present in the FEM model; the **Archive** button, on the other hand, provides access to an online archive of materials.

If the user opts to work with the online archive, the results can be filtered so as to display only those materials which meet the requirements set in the **Type**, **Fy** and **Ft** fields.

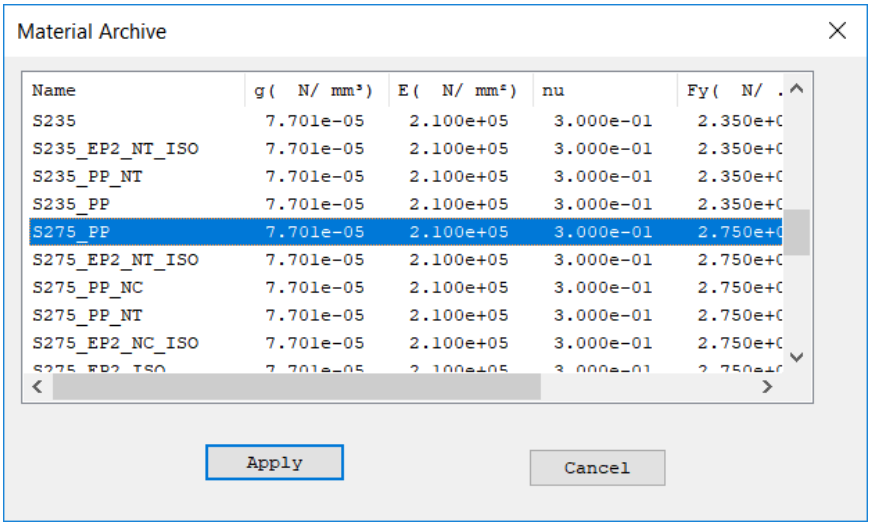
Type: tick one or more boxes to extract only the materials of the selected types.

Fy: if the **Active** box is ticked and the desired values have been entered in the **Min** and **Max** fields (in the current units of measurement), only those materials will be extracted whose yield stress falls within the Min-Max range.

Ft: if the **Active** box is ticked and the desired values have been entered in the **Min** and **Max** fields (in the current units of measurement), only those materials will be extracted whose ultimate stress falls within the **Min**-**Max** range.

Once the parameters have been set up and the archive selected, the user is presented with a further dialog box 87 containing the list of the materials which meet the chosen criteria.

3.4.2 Material archive (dialog)



This dialog box presents a list of the materials. Pick one and click **Apply** or, alternatively, double-click on a material to select it and apply it to the selected elements.

The suffix used in the standard material archive have the following meaning:

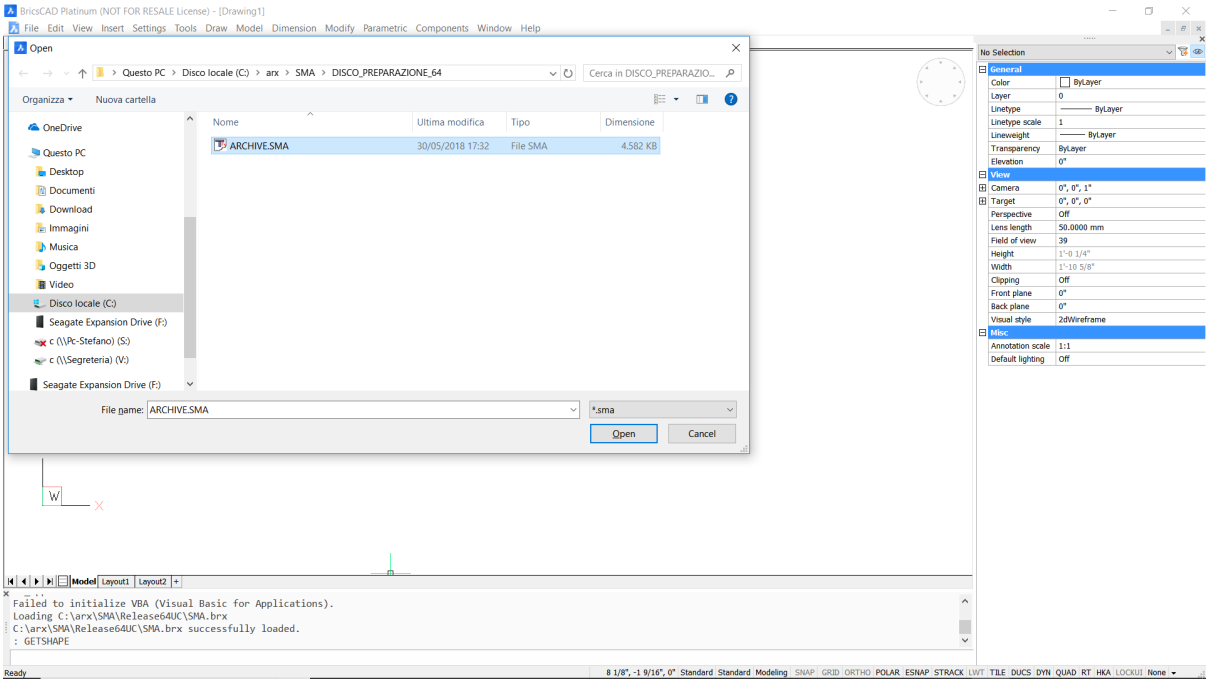
- PP: perfectly plastic
- NT: no tension
- NC: no compression
- EP2: elastic-plastic (hardening) two branches
- ISO: isotropic hardening

3.5 GETSHAPE

Enter the following string in the BricsCAD© command prompt:

GETSHAPE

If the cross-section and material archive has not been loaded yet, you are prompted to choose the "archive.sma" file, which is the database file received with SMA, and in the same folder of SMA.BRX.



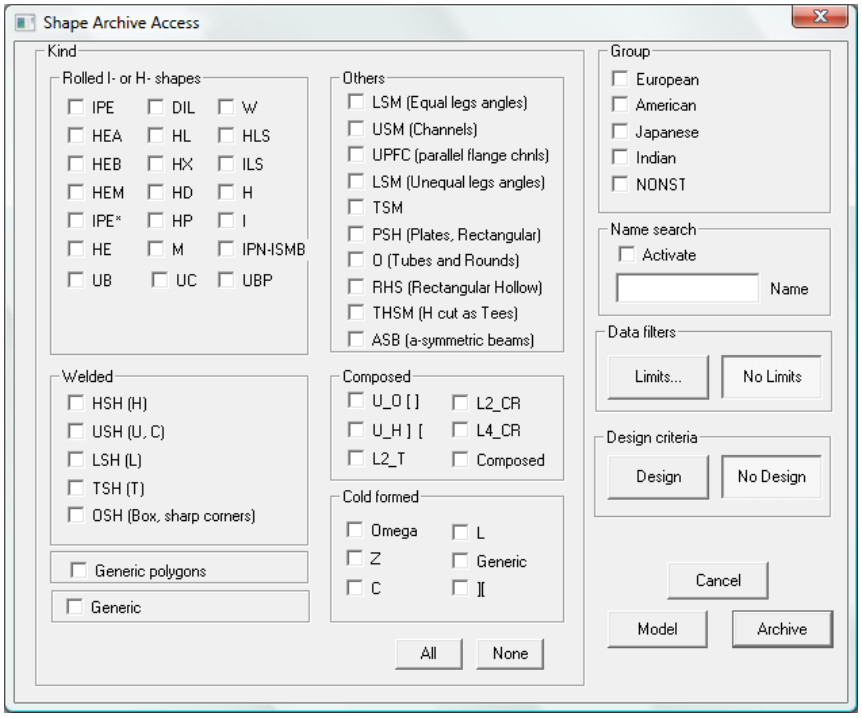
This file embeds all the cross-sections and materials of the existing archive (more than 19,000 cross sections, USA, UK, European, Indian, Russian, and more).

By this command, a proper dialog appears 89. This dialog is used to choose some criteria to be used to extract from the cross-section archive a subset, that will be later used to pick up one specific cross-section. The final aim of this command is twofold:

1. To add a block to the current BricsCAD© document, fully describing the cross section, using the currently active units.
2. To add to the "model" cross-section vector this cross-section, so that it may be later referenced.

The command may also be used to extract a subset of the available +19,000 cross sections so that some design criteria is satisfied 96 by those extracted cross-sections. So, it is also a tool usable to help designing main structural elements.

3.5.1 Shape archive access (dialog)



This dialog box enables the filters to be defined for extracting a subset of the cross-sections from the archive. Once the filters have been set up, only those cross-sections will be extracted from the archive which meet these filter criteria.

This dialog box contains tick boxes and buttons. The tick boxes fall into three categories: type, group and name search.

Group: there are four available group filters: European, American, Japanese and Non-standard.

Type: this has five panes: Rolled H shapes, Welded, Other, Composites and Generic.

Rolled H shapes: this groups together the rolled H type cross-sections (HEA, HEB, HEM, IPE, HE, ILS, UC, UB, UBP, etc).

Welded: this contains welded H sections (HSH), U sections (USH), L sections (LSH), T sections (TSH) and box sections (OSH). Concrete cross-sections of the same shape are also included under this heading, even though they are not "welded".

Other: this contains channels (UPN or UNP, UPFC), angles with equal or unequal legs (LSM), T-sections (TSM or THSM), plates (PSH), rectangular sections (PSH), tubes (O), rounds (O) asymmetric beams (ASB).

Composite: contains the composed cross-sections. L2_T are angles composed to form a T. L2_CR are two angles composed to form a cross. L4_CR are four angles composed to form a cross. Generic composites are cross-sections obtained by arbitrarily joining a certain number of elementary or composite cross-sections.

Cold formed: this groups consists of C, L, Z and W cross sections, along with completely generic ones.

Generic: this consists of the sections identified by moments of inertia and area only.

Name search: this is used via a tick box called "Activate" and a text box called "Name". If the box is ticked, the character string in the Name field will be used to filter the cross-sections. The string may contain semi-colons (";") to separate different criteria, and the asterisk character ("*") as a wildcard standing for "any character (none, one or many)". Some example search strings are shown below:

PLT*	all names beginning with PLT
PLT*; HE*100*	all names beginning with PLT and all HE*100* sections
*PLT	all names ending in PLT

The name filter can be used to extract sections of categories not provided among the standard filters by type. If sections called PIPPO 100, PIPPO 120, etc., are added, since no standard "PIPPO" type filter is available, a search string like "PIPPO*" can be used instead.

Only those sections will be selected which meet at least one of the type criteria and at least one of the group criteria and the name search criterion. If no group filter is specified, the group will be ignored. If no type filter is specified, the type will be ignored. If the "Activate" box is not ticked, the name filter will be ignored. If there are no filters to take into account, the search results will be determined by the other standard filters only (if any are specified). If no other filter has been specified either, all sections will be extracted from the archive (not recommended).

The dialog box contains various buttons besides the tick boxes.

The **Limits** button allows other filters to be selected, i.e. the filters on the numeric properties of the cross-sections, in an additional dedicated dialog box (Data filters 94). These filters are additional to the ones discussed above.

The **No Limits** button removes all data filters.

The **Design** button allows extra filters to be selected (those based on design criteria), via another dialog box provided for the purpose (Choice of design criteria 96). These filters are additional to the ones discussed above.

The **No Design** button removes all design-based filters.

The **Model** button enables the user to browse through the cross-sections added so far to the current BRICSCAD© document.

So, to sum up, there are filters by **group**, by **type**, by **name**, on the **quantitative data** and on the **design criteria**.
A cross-section that satisfies at least one of the active group criteria is included by the group filter.
A cross-section that satisfies at least one of the active type criteria is included by the type filter.
A cross-section that satisfies at least one of the name criteria (if activated) is included by the name filter. Filters which have not been activated do not exclude any cross-sections.
A cross-section that satisfies all the specified data/quantity criteria is included by the data/quantity filter.
A cross-section that satisfies all the specified design criteria is included by the design filter.

Filters are cumulative (more than one can be specified): the sections extracted are those which meet all the filter criteria (by type, group, name, data/quantity and

design criteria) which have been specified in this dialog box and in the Data filters 94 and Choice of design criteria 96 dialog boxes, without exclusion.

If no filters have been specified, the entire archive is returned (not recommended).

Examples:

To extract all the IPE and HEA cross-sections, and no others, tick IPE and HEA to activate the filter. Ensure all other boxes are not ticked. Click the **No Limits** and **No Design** buttons to ensure that there are no other active filters. Click **Archive**.

To extract only those IPE cross-sections of area greater than 1000 mm2. **Tick IPE** only, click the **Limits** button, tick against **Area**, enter 1000 as the minimum value (it is assumed at this point that the unit of length is the mm) and a very large number as the maximum value. Click **OK** to come back to this dialog box, and click **Archive**.

To extract welded I-sections called HSU....., **tick HSH** only, activate the **name search** filter and enter the 4-character string shown here within the inverted commas: "**HSU***".

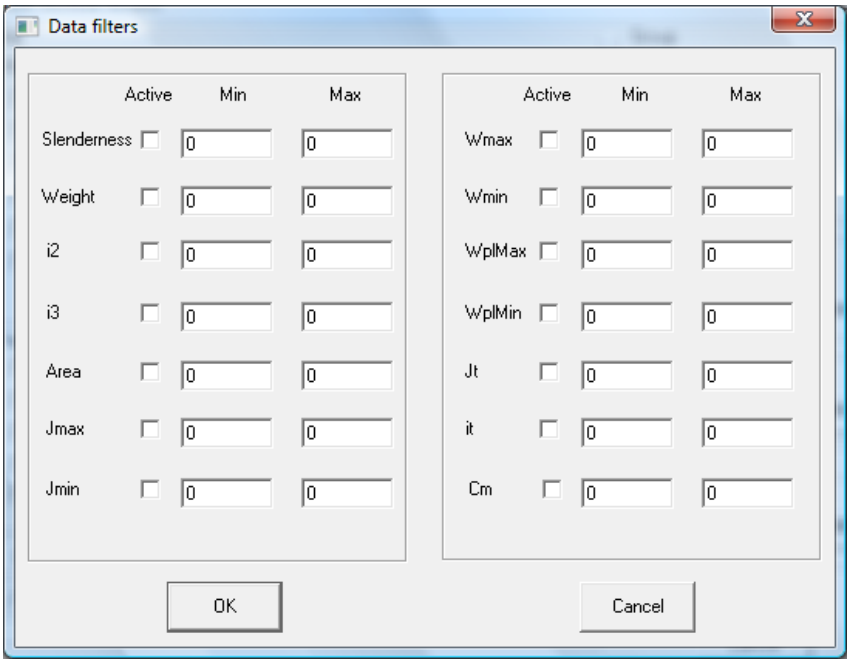
To extract welded I-sections called HSU....., plus welded I-sections called HSD...., **tick HSH** only**, activate the name search** filter and enter the following string between the inverted commas as shown: "**HSU*;HSD***".

Suppose we need to design a 4-metre span doubly-supported beam with a mid-span load of 20t. The design requires a deflection of less than 1/500 of the span

and a stress due to flexure of less than 1600 Kg/cm2. A HEA, HEB or HEM section is required.

In this case, the **HEA, HEB and HEM type filters** should be activated, and then the **Design** button pressed. In the design dialog, the structural model must be selected with the **doubly-supported beam with mid-span load**, the **span** and **load** must be specified (in the units of measure currently in use), then the **deflection criterion will need to be activated**, entering 500 as the value for **k** and the limit stress as the value for **m**. Then click the **Archive** button and a dialog box 100 will appear.

3.5.1.1 Data filters (dialog)



This dialog box is accessed via the **Limits** button on the Cross-section Archive 89 dialog box.

This dialog box enables the numeric filters to be defined for extracting a subset of the cross-sections from the archive. Maximum and minimum values can be set (in the active unit of measurement) for each quantity described in the dialog box (**Slenderness**, **Weight**, etc.). To activate a filter, the corresponding tick box in the "Active" column then needs to be ticked. The tick indicates that the filter is active.

The meanings of the symbols should all be clear, except for **Cm** (warping constant), **Jt** (torsional moment of inertia) and **it** (radius of gyration).

Jmax: maximum second area moment

Jmin: minimum second area moment.

Wmax: maximum section modulus

Wmin: minimum section modulus

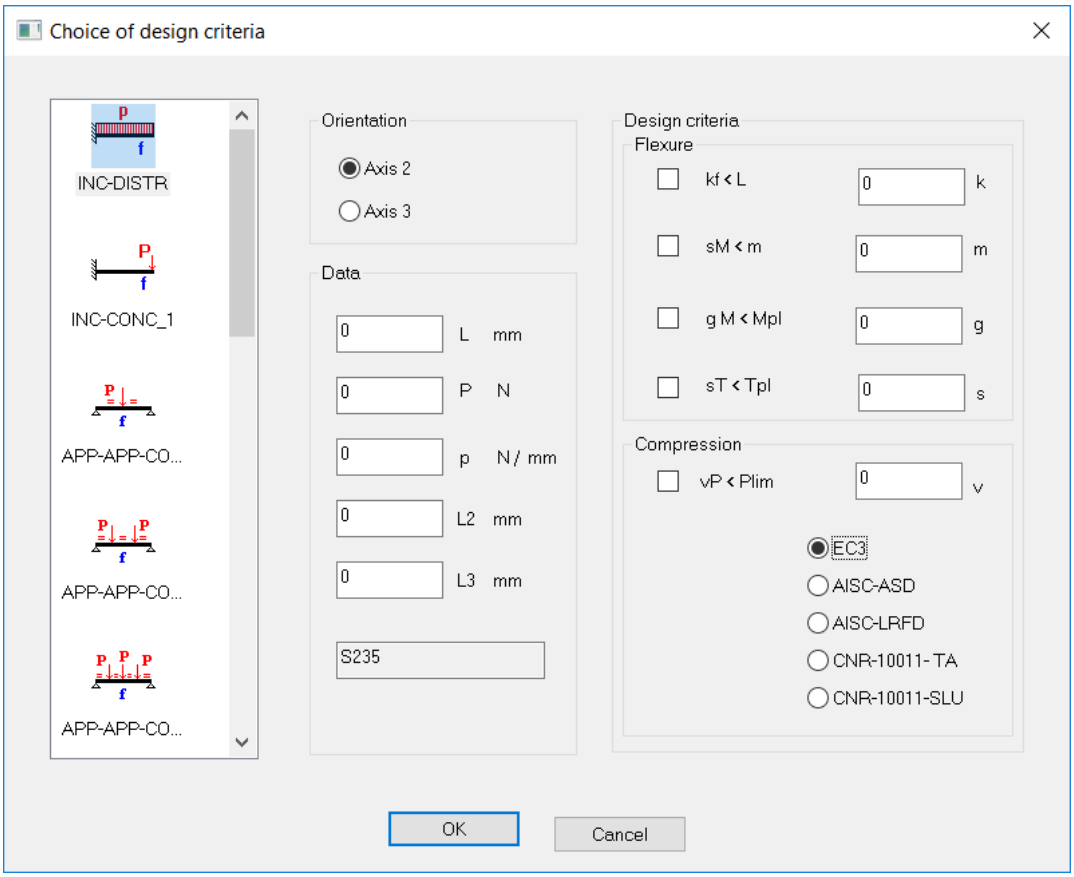
WplMax: maximum plastic section modulus

WplMin: minimum plastic section modulus

If a filter is activated, then necessarily only the sections complying with it will be extracted from the archive. The numeric values are interpreted against the units of measurement which are active at the time the command is run.

The slenderness filter and the weight filter assume that the length of the element(s) to which the cross-section is to be applied is known. These two filters are reserved for use in conjunction with the Sargon code. When using SAMBA, these two filters must not be activated.

3.5.1.2 Choice of design criteria (dialog)



This dialog box enables a certain number of design criteria to be set for application to a structural model.

Firstly, the appropriate structural model for the beam or column to be designed is selected. The available models are displayed clearly in the left hand pane for the user to browse through. A model can be selected by clicking on it. The available models include the beam with springs at each extremity having a moment distribution such that the moment in the span is equal to pL^2/10, whereas the moment at the supports is equal to pL^2/40. This intermediate distribution between support and clamp can be used to address the case of frames, where the node is elastically constrained by the rest of the structure (naturally the “exact” moment distribution depends on the real value

of the spring constant: the distribution proposed here is an average which is often used in engineering practice).

Once the above has been attended to, the length (**L** or **L2** and **L3**) and load data are filled in. If the structural model entails a distributed load, the load **p** must be specified; if the model envisages one or more concentrated loads or an axial load, then load **P** must be specified. Both the span and the loads **p** and **P** are interpreted against the active units of measurement. Load **P** is a force, while load **p** is a force per unit length.

For the compression model, it is not L which needs to be provided, but rather **L2** and **L3**, i.e. the two free deflection lengths relative to the main axes 2 and 3. If the compression model is selected, length L is ignored. If, however, any other model is chosen, then L2 and L3 are ignored.

In order that the design criteria can be validly used, a material must have already been selected. If the material has not been selected, the system will not permit the user to exit from the dialog box without first having inactivated all the design criteria.

The material selected is common to all elements (beams and trusses) selected when the **Sections** command is run. If the sections selected have no material assigned, or if they have different materials, then no material will be selected.
The name of the currently selected material is shown in the box under the data about the spans and the loads. If this box is blank, no material is currently selected.

Columns

If the structural model is that of a column, the criterion pertaining to the point load **vP < Plim** must be determined, by specifying the standard used to evaluate the stability curve and the safety factor v (v=1.2 indicates a safety factor equal to 1.2). If a column is selected, all the other criteria must be inactivated.

The Plim (limit load) value is computed by using the stability curves defined by each standard.

The available standards are CNR10011-88 on allowable stresses (TA) and ultimate limit states (SLU), the AISC-89 standards on allowable stress design (ASD), and Eurocode 3.

The calculation uses the formulae established by each standard, so as to verify that the design satisfies the criterion. **In examining the stability situation, the maximum slenderness is always taken into consideration, which is computed using the free deflection lengths given, independently of the specified axis (2 or 3).**

Beams

If the structural model is that of a beam (under various constraints and loads), the following design criteria can then be specified:

kf < L

The deflection must be less than L/k, where k is a design value defined by the user. The deflection value taken into consideration is always the maximum deflection value, in keeping with the pre-selected model (of loads and constraints). The modulus of elasticity is that of the material selected. The moment of inertia used is that of the axis specified (2 or 3).

sM < m

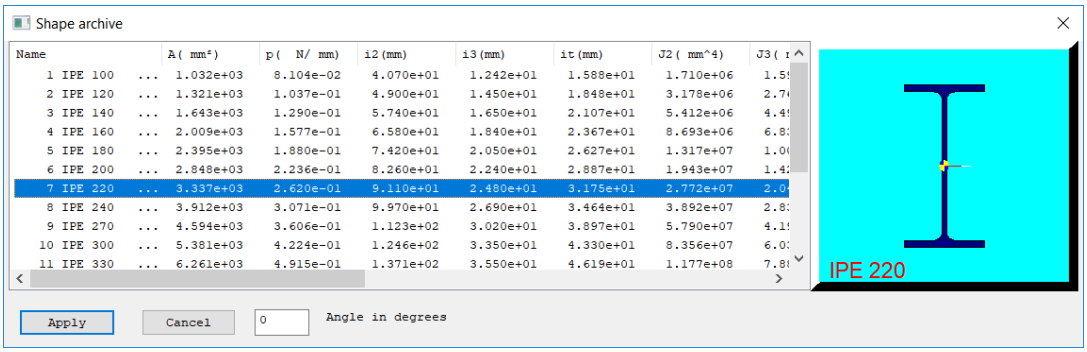
The maximum normal stress due to flexure (sM) must be less than the limit value (m) specified by the user. The normal stress is evaluated on the basis of Saint-Venant's theory, taking the flexure to be perpendicular to the axis specified (axis 2 – the stronger – or axis 3). The moment is the maximum moment on the beam, even if continuous.

gM < Mpl

The maximum moment read on the beam multiplied by the safety factor g must be less than the plastic limit moment, evaluated using the yield stress fy of the material selected, and the plastic section modulus of the axis specified (2 or 3).

If the diagram selected is that of a beam, the design criteria relating to columns must be deactivated.

3.5.1.3 Shape archive (dialog)



This dialog box is used for browsing the cross-sections extracted from the archive, selecting one that will be added as a block to the current BRICSCAD© document. It is also used to apply a previously defined new cross-section. To apply the section, select **Apply**.

By pressing the image-button detailed information may be got.

The angle data is not currently used in SMA app.

3.6 SETUNIT

Enter the following string in the BricsCAD© command prompt:

SETUNIT

This command enables the currently active units of measurement to be changed, via a dedicated dialog box. After the execution of the command, all the data output by SMA, and all the data you should input to SMA, will be considered coherent with the current units. Derived units (e.g. second area moment, or load per unit length) will use coherent unit system depending on length and force units.

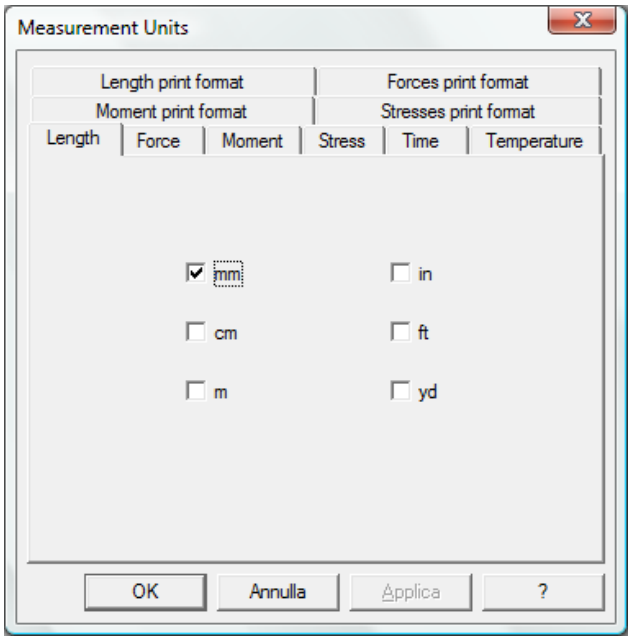
When the block referring to a cross section is added to the current BricsCAD© document, it is added rescaling according to the current units.

For instance, if the current units are "mm" and you add a rectangular cross-section with h=200 and b=100, this means you are adding a cross section having h=200 mm and b=100 mm. The cross-section block will have a containing box of 200x100.

But if instead you have currently active "cm", to add the same cross-section you will input h=20 (cm) and b=10 (cm). The containing box of the block added will be 20x10.

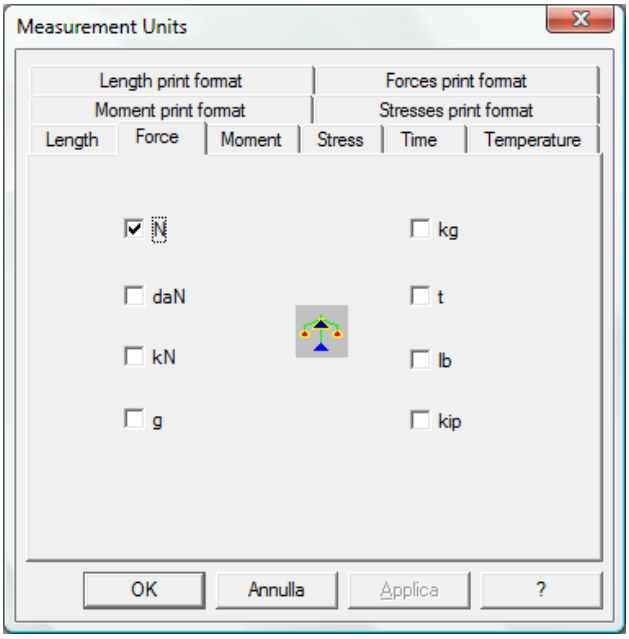
So, basically, you should tune your units coherently with the units you are using in your BricsCAD© document, in order to be sure no scaling is applied.

3.6.1 Units - Length (dialog)



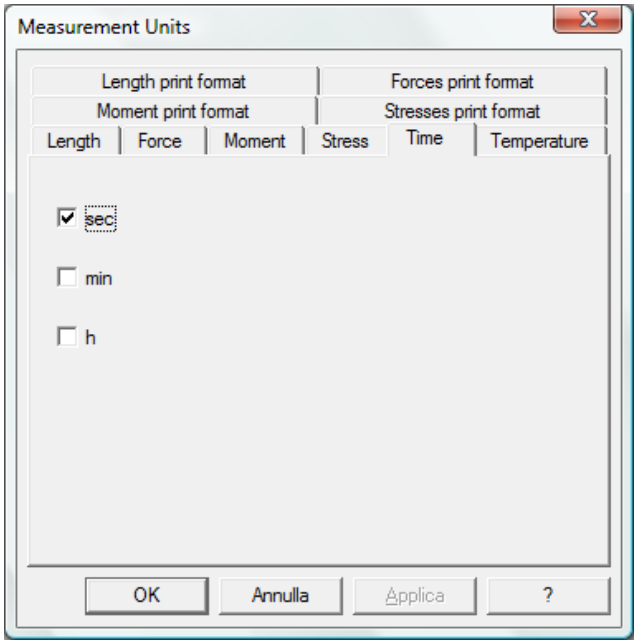
This dialog box is used to select the current units of measurement for lengths.

3.6.2 Units - Force (dialog)



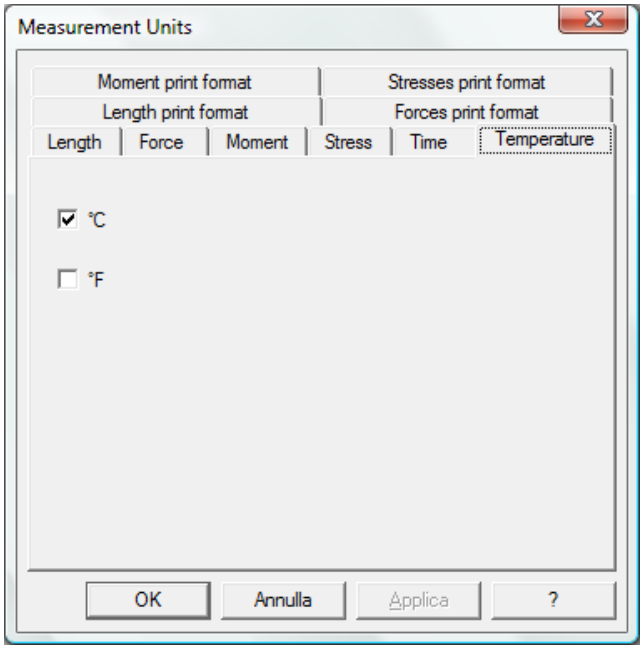
This dialog box is used to select the current units of measurement for forces.

3.6.3 Units - Time (dialog)



This dialog box is used to select the current units of measurement for time.

3.6.4 Units - Temperature (dialog)



This dialog box is used to select the current units of measurement for temperature.

Index

http://www.castaliaweb.com http://www.steelchecks.com
Via Pinturicchio, 24
20133 Milan - Italy
staff@castaliaweb.com