



**GIVE MORE VALUE
TO YOUR TIME**

SOFTWARE AND SERVICES FOR PRECAST AND ENGINEERING



PREF

**CONSTANT HEIGHT
PRESTRESSED BEAMS**

USER'S MANUAL

Table of contents

1. FUNCTIONALITY OF THE PROGRAM.....	5
1.1. CALCULATION SCHEME.....	5
1.2. RULES.....	5
1.3. GEOMETRIC FEATURES	7
2. System requirements - Windows® release.....	9
3. MOVING DATABASE ON Windows 7 – 8 - 10	9
4. PROGRAM'S INSTALLATION.....	11
5. Initial SET UP.....	14
6. SETTINGS	14
6.1. SET SEISMIC PARAMETERS.....	15
6.2. CRITERIA.....	15
6.2.1. GENERAL tab	16
6.2.2. FIRE tab	16
6.2.3. TRANSP/LIFT. tab.....	17
6.2.1. STIRRUP DESIGN tab	17
7. CHANGE LANGUAGE.....	18
8. AUTOMATIC UPDATES.....	20
9. MAIN WINDOW	21
9.1. NEW PROJECT ON FILE	22
9.2. OPEN PROJECT FROM FILE	22
9.3. DATABASE PROJECTS	22
9.3.1. TO CREATE A NEW PROJECT	23
9.3.2. TO OPEN AN EXISTING PROJECT.....	23
9.3.3. UPDATE BEAM.....	23
9.3.4. SAVE PROJECT AS	23
9.3.1. DELETE PROJECT / DELETE JOB.....	23
10. BEAM DATA.....	25
10.1. NOMENCLATURE	25
10.2. FEATURES	26
10.3. LINE LOADS	27
10.3.1. LOADS WHEN CAST IN PLACE IS PRESENT	28
10.3.2. LOADS WHEN THERE ISN'T A CAST IN PLACE	28
10.4. WEIGHTS of floors to automatically calculate G1.....	29
10.5. LOCAL VERIFICATION	30
10.6. COEFFICIENTS	30
10.7. AUTOMATIC DESIGN OF STRANDS	31
10.8. SHEAR on support	31
10.9. BUTTONS	32

11.	CAST IN PLACE.....	33
12.	POINT LOADS	34
12.1.	SPACING	34
13.	MATERIALS	35
13.1.	DEFAULT MATERIALS.....	36
14.	COORDINATES	37
15.	STRANDS	38
15.1.	Graphic area.....	39
15.1.1.	ZOOM.....	39
15.1.2.	STRAND REPRESENTATION IN GRAPHIC AREA.....	40
15.2.	TOOLS WINDOW.....	41
16.	REBARS.....	44
16.1.	ZOOM AND GRAPHIC.....	45
16.1.	REBAR REPRESENTATION IN GRAPHIC AREA	46
16.1.	TOOLS WINDOW.....	47
17.	POST-TENSED STRANDS.....	50
17.1.	ZOOM AND GRAPHICS	51
17.2.	GRAPHIC AREA	52
17.1.	TOOLS WINDOW.....	53
17.2.	Insert/edit of cable:	55
18.	CALCULATIONS	59
18.1.	PARTIAL PRESTRESS VERIFICATION	62
18.2.	BUTTONS FOR REPORTS	64
18.3.	EXPORTATION	64
19.	VERIFICATION WITH LONG OVERHANGS (SUPPORTS)	66
20.	STIRRUP DESIGN	67
20.1.	The process of design	68
20.1.1.	SHEAR REBARS	68
20.1.2.	STIRRUPS	68
21.	SERVICEABILITY VERIFICATON IN ANY SECTION.....	71
22.	Research Spans and Capacities	73
23.	SEISMIC VERIFICATION	75
23.1.	ASSOCIATE SEISMIC PARAMETERS TO JOBS.....	77
24.	FIRE VERIFICATION	78
25.	LEFT CUT	81
	<i>Verification of Gerber Support in the ends</i>	<i>81</i>
26.	SNOW - WIND LOADS	82
27.	UTILITY	83
27.1.	CONVERSION OF UNIT OF MEASURES to convert lengths	83
27.2.	STRANDS AREAS TABLE.....	84
27.3.	REBARS AREAS TABLE	84

27.4.	CALCULATION OF REBARS AREAS	84
28.	MENU BAR.....	85
28.1.	File	85
28.2.	Input Data	85
28.3.	Calculations	85
28.4.	Tools	85
28.5.	Settings.....	86
28.6.	Cambia database	86
28.7.	?.....	86
29.	DATA MANAGEMENT	87
29.1.	SAVE - OPEN – DELETE projects	87
29.1.1.	Save	87
29.1.1.	OPEN	88
29.1.1.	DELETE.....	88
30.	“GEOMETRY” EDITOR	89
30.1.	What is it for:.....	89
30.2.	FILE MANAGEMENT	89
30.3.	GEOMETRY EDITOR FEATURES	90
30.4.	INPUT OF NEW SECTIONS.....	91
30.4.1.	SECTION BY TRAPEZIUMS	91
30.4.2.	SECTION BY POINTS	93
30.4.3.	PARAMETRIC SECTION	97
30.4.4.	SECTION BY DXF.....	99
30.5.	GRAPHIC AREA	101
30.6.	STRANDS	102
	INSERTING STRAND BY RETICULUM	104
	INSERTING STRAND BY COORDINATES	105
	INSERTING STRAND BY GRID	105
30.7.	REBARS.....	106
30.8.	SETTAGGI	108
30.9.	MENU'	108
30.10.	TOOLBAR	110
30.11.	VERTICAL TOOLBAR.....	111

1. FUNCTIONALITY OF THE PROGRAM

The program verifies and checks iteratively prestressed beams of sections that are SYMMETRICAL respect to the vertical axis. The beams may have a streamlined section (equal along the whole length), or can have the different shape of the ends. The sections can be hollow, with one or more holes (not hollow-core slabs, for which there is the special program).

The program is divided into two modules:

- In "**GEOMETRTY**" you store the formwork of the beam (sections and the available positions of the strands and the reinforcement).
- In "**CALCULATIONS**" are inserted the specific data required for checking the beam to be calculated, using one of the formwork geometries stored in the module geometry.

To perform a calculation first you have to store the section with the geometry program, then you run the actual calculation with the calculation program. Each calculated beam saved with all its data is called "Project."

For "project" we mean a particular calculation performed with one of the beams in the archive.

1.1. CALCULATION SCHEME

The constraint is simple support with the ability to have overhangs on the ends. It can have a cast in place to bring overloads, also the casting can be of complex shape.

Starting from left support, the prestressed area of the beam is verified in a series of sections up to the right support, and the section most subject to bending is highlighted.

The losses of tension are calculated in the centre of gravity of the strands contained in the lower 2/3 of the beam. The calculation at limit states is executed in an exact manner based on diagrams of breakage of the strands supplied by the manufacturer of the strands.

In each section the programs checks sigma main traction and compression on the centre of gravity and shows the minimum distance where the stirrups should be placed. Superior and inferior reinforcement is always calculated, to absorb the tractions as requested from Regulations.

The program will also verify the transitional phases: the draft, lifting, lifting / transport after a storage period.

The support section, in the absence of overhang, is considered not prestressed.

There is the possibility to calculate a section chosen from the user, with the control of main tau and sigma of traction on the whole height of the section. Finally, it is possible, choosing a particular reinforcement, find the diagram of use, of course this calculation does not consider the concentrate loads.

1.2. RULES

The calculation of the beam follows Limit States according to Eurocode2, as permitted by NTC 2018 (user can still calculate with the old DM 14/01/08).

The calculation of the beam follows the Allowed Stresses and D.M. 14/2/92, as permitted by D.M. 9/1/96.

NAMES

LCTOT	= total beam length
LC	= effective length
SBS	= Left Overhang
SBD	= Right Overhang
TP	= Shear at support for the beam weight only
TPP	= Shear at support due to permanent overloads bore from beam
TPfase	= Shear at support due to permanent overloads bore from beam and cast in place
TAA	= Shear at support due to dominant variable overloads
QkApp	= Shear at support due to second variable overloads
MPA(i)	= Moment due to beam weight only, in i-section
MPPA(i)	= Moment due to permanent loads bore from beam only, in i-section
MPfase(i)	= Moment due to permanent overloads bore from beam and cast in place , in i-section
MPnonDE(i)	= Moment due to permanent overloads not completely defined bore from beam and cast in place , in i-section (or beam only if there's no cast in place)
MAA(i)	= Moment due to dominant variable overloads , in i-section
MPqk(i)	= Moment due to second variable overloads , in i-section
TPA(i)	= Shear due to beam weight only, in i-section
TPPA(i)	= Shear due to permanent overloads bore from beam, in i-section
TPfase(i)	= Shear due to permanent overloads bore from beam and cast in place , in i-section
TnonDE(i)	= Shear due to permanent overloads not defined bore from beam and cast in place (or beam only if there's no cast in place) , in i-section
TAA(i)	= Shear due to dominant variable overloads, in i-section
TPqk(i)	= Shear due to second variable overloads, in i-section
H1	= beam height
Ac	= area of the concrete beam section only
PERI	= Perimeter of the beam shape
DimNo	= Nominal size of the beam = $2 \times Ac / PERI$
Jba	= Moment of inertia centre of gravity section only concrete
A1C	= Area of the beam concrete + homogenized steel
AT	= Area prestressed steel
ATI	= Area prestressed inferior steel
KB	= distance of the centre of gravity of prestressing steel and the upper edge of the section
Ypp	= Distance of the centre of gravity of only prestressing inferior steel and the upper edge of the section
YS	= Distance of centre of gravity of the section and the upper edge of the beam itself
WS	= modulus of resistance respect to the upper edge of section
WB	= modulus of resistance at the height of the centre of gravity of the strands
WI	= modulus of resistance at inferior edge of section
Wg	= modulus of resistance at the height of the centre of gravity of the inferior strands
Jg	= moment of inertia of centre of gravity of section
Sc	= First moment of area of the upper part at the centre of gravity of the section with

respect to the horizontal axis passing through the centre of gravity itself.

Rck	=	Conc. Characteristic Cubic Strength of beam at 28 days
GammaC	=	Safety factor γ_c for beam Concrete = 1.4 or 1.5
fck	=	Characteristic Cylinder Compressive Strength = $R_{ck} \times 0.83$
fcd	=	Design Concrete stress = f_{ck} / γ_c
fctm	=	Mean Value of tensile Strength = $0.30 \times (f_{ck})^{2/3}$
fcfm	=	Mean Value of bend. Strength = $f_{ctm} \times 1.2$
Rckj	=	Conc. Characteristic Cubic Strength of beam at initial loading
Ec	=	Elastic Modulus of beam concrete = $22000 * ((f_{ck} + 8) / 10)^{0.3}$
Abar	=	width section in the centre of gravity of the beam
AN	=	width at the support section (input datum)
AMIN	=	midspan section (input datum)
HR	=	Total height cast in place
YO	=	Distance of beam section centre of gravity + cast and the upper edge of the beam itself
JO	=	moment of inertia and centre of gravity beam section + cast in place
fpk	=	Characteristic strength Stabilized strands with low relaxation = 1860 N/mm ²
γ_c	=	Prestressing steel safety factor = 1.15
fp1k	=	1670 N/mm ²
fsd	=	$fp_{1k} / 1.15$
TESO	=	Initial stress in the tendons, input in Kg/cm ²
Es	=	Prestressing steel elastic Modulus = 195 kN/mm ²
EASUEC	=	Ratio between Prestressing steel elastic Modulus and beam concrete elastic Modulus
Coef1= γ_{G1}	=	Coefficient for the calculation of the ultimate moment to be applied to defined permanent loads
COEFg2= γ_{G2}	=	Coefficient for the calculation of the ultimate moment to be applied to not defined permanent loads
Coef2	=	Coefficient for the calculation of the ultimate moment to be applied to accidental loads
$\gamma_{Q1} = \gamma_{Q2}$	=	
Psi02 = Ψ_{02}	=	Multiplier seconds accidental overloads for rare stress
Psi1 = Ψ_{11}	=	Multiplier dominant variable overloads to frequent stress
Psi2 = Ψ_{21}	=	Multiplier dominant variable overloads to almost permanent stress
Psi12= Ψ_{12}	=	Multiplier seconds variable overloads to frequent stress
Psi22 = Ψ_{22}	=	Multiplier seconds variable overloads to almost permanent stress
ν	=	0.6
If $f_{ck} > 60$ then: $\nu = 0.9 - f_{ck} / 200 \geq 0.5$.		

1.3. GEOMETRIC FEATURES

The geometric characteristics of the midspan section and the ends are calculated in an exact way, because whatever the method by which it was created the section in the Geometry module (for trapezoids, points, DXF import...), the program draws the perimeter by points, and then calculates areas, and static moments of inertia **with respect to the horizontal axis passing through the upper edge of the section.**

From the characteristics referred to the upper edge later in the calculation are derived all the other parameters required for checks.

We use the following variables for the calculation

B1, C1, D1, B4, C4, P, Q, prod1, prod2, prod3, prod4

Nupu = total number of the points of the beam section (close figure).

yy(l) = Y of the point number i

xx(l) = X of the point number i

A1 = 0: B1 = 0: C1 = 0: D1 = 0: B4 = 0: C4 = 0

For i from 1 to nupu - 1

P = yy(l + 1) - yy(l)

Q = xx(l + 1) - xx(l)

A1 = A1 + (yy(l) + yy(l + 1)) * Q

B1 = B1 + (yy(l) ^ 2 + yy(l) * yy(l + 1) + yy(l + 1) ^ 2) * Q

C1 = C1 + (yy(l) ^ 3 + yy(l) ^ 2 * yy(l + 1) + yy(l) * yy(l + 1) ^ 2 + yy(l + 1) ^ 3) * Q

prod1 = xx(l) * yy(l + 1) - xx(l + 1) * yy(l)

prod2 = xx(l) + xx(l + 1)

prod3 = yy(l) + yy(l + 1)

prod4 = xx(l) * yy(l + 1) + xx(l + 1) * yy(l)

D1 = D1 - prod1 * (prod2 * prod3 - 0.5 * prod4) / 12

Do this loop for every i

For i from 1 to nupu - 1

Q = yy(l + 1) - yy(l)

B4 = B4 + (xx(l) ^ 2 + xx(l) * xx(l + 1) + xx(l + 1) ^ 2) * Q

C4 = C4 + (xx(l) ^ 3 + xx(l) ^ 2 * xx(l + 1) + xx(l) * xx(l + 1) ^ 2 + xx(l + 1) ^ 3) * Q

Do this loop for every i

Where ^ means elevate and * means multiply.

Set

Ac area of the sole concrete section in cm²

Mxc First moment of area in cm³ respect to superior edge of section

Jxc Second moment of area in cm⁴ respect to superior edge of section

Ac = A1 / 2

sx = B1 / 6

JX = C1 / 12

YS = sx / AC

JB = JX - AC * YS ^ 2

Mxc = AC * (H1 - YS)

Jxc = JB + AC * (H1 - YS) ^ 2

YS = H1 - YS = distance from centroid of the sole beam from sup. edge of beam

Numtref = number of strands in matrix

NT(l) = area of strand number i

dtY(l) = Y of strand number i

dtX(l) = X of strand number i

Numeroferri = number of total rebars

Areafe(l) = area of rebar number i

Yfe(l) = Y of rebar number i

Xfe(l) = X of rebar number i

Set

AT = 0 ; ati = 0 ; KTX = 0 ; KTY = 0 ; Ai = AC ; MX1 = MXC ; JX1 = JXC

We find the areas and First and Second moments of area of the beam, strands and rebars

For i from 1 to numtref

AT = AT + NT(l)

KTX = KTX + dtX(l) * NT(l)

KT = KT + dtY(l) * NT(l)

If dtY(l) < H1 / 2 then do: ati = ati + NT(l) e KTI = KTI + dtY(l) * NT(l)

Repeat this loop

2. System requirements - Windows® release

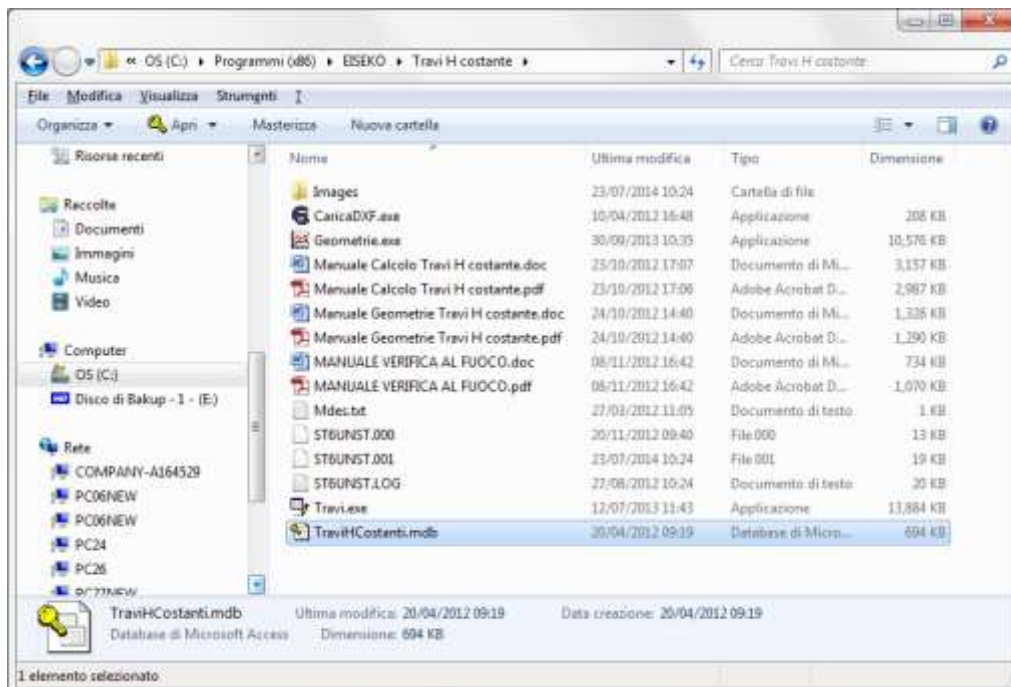
You can install this program on **Windows**, any release from Windows XP on.

On Windows Vista 7, and newer versions it's necessary to login as administrator, or move the working database in the directory "Documents" (C:\Users\Public\Documents), where everybody has administrator privileges (or you can save data on txt file instead of database: see "[DATA MANAGEMENT](#)").

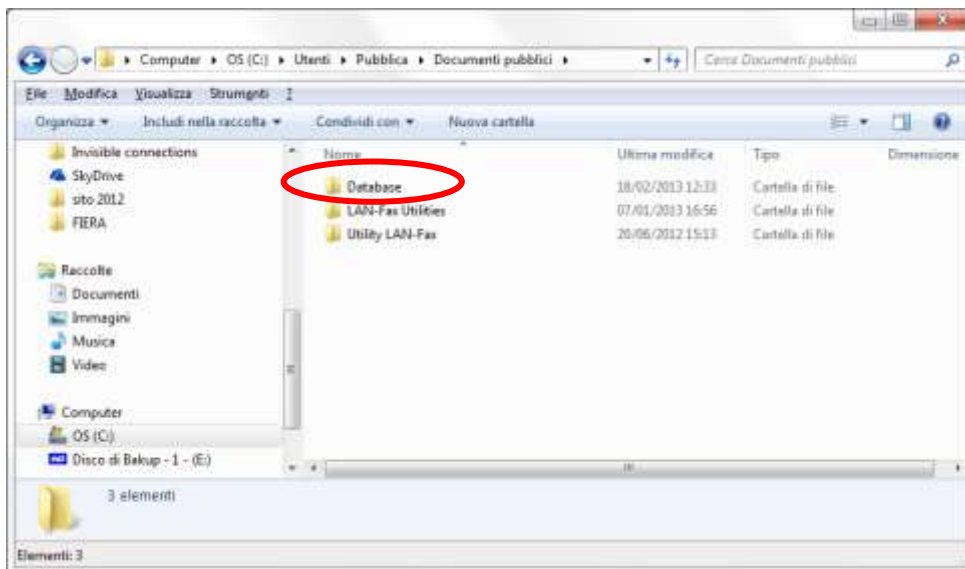
3. MOVING DATABASE ON Windows 7 – 8 - 10

Procedure (TO BE DONE AFTER INSTALLATION):

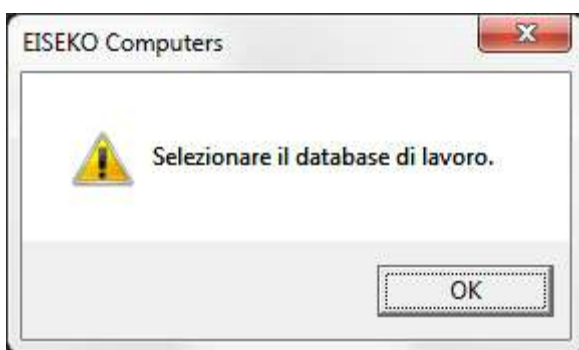
1. Open the program's directory C:\Program Files (x86)\EISEKO\Travi H Costante (or it you changed it during installation, that one).



2. Open the directory C:\Utenti\Pubblica\Public documents.
3. Create in the latter a new directory "Database" or "Database Eiseko", it will be used to store all the databases of all the Eiseko's programs.



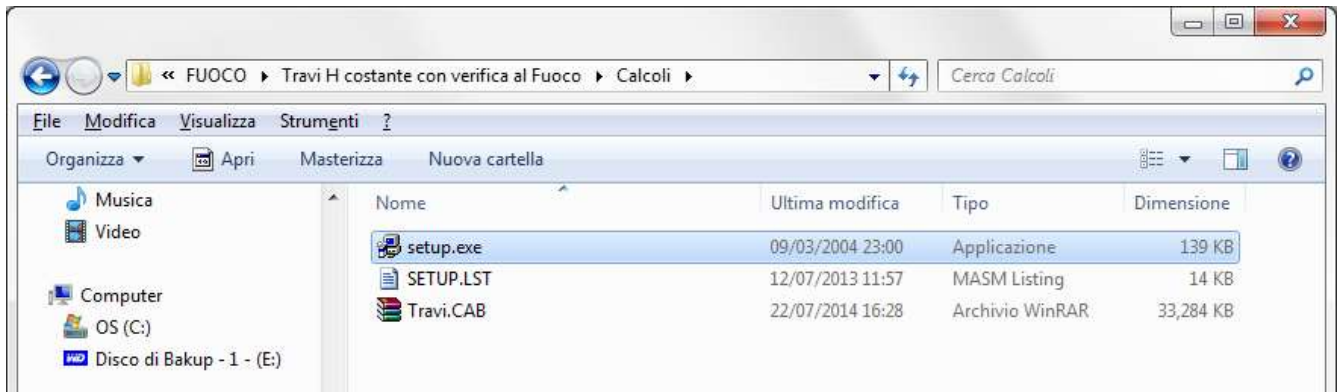
4. Move the database dragging and dropping it from program's directory to data directory "Database", or cut and paste it. We recommend not to leave the database in the standard program's directory.
5. After moving the database, when you run the program, it will ask you for the database.



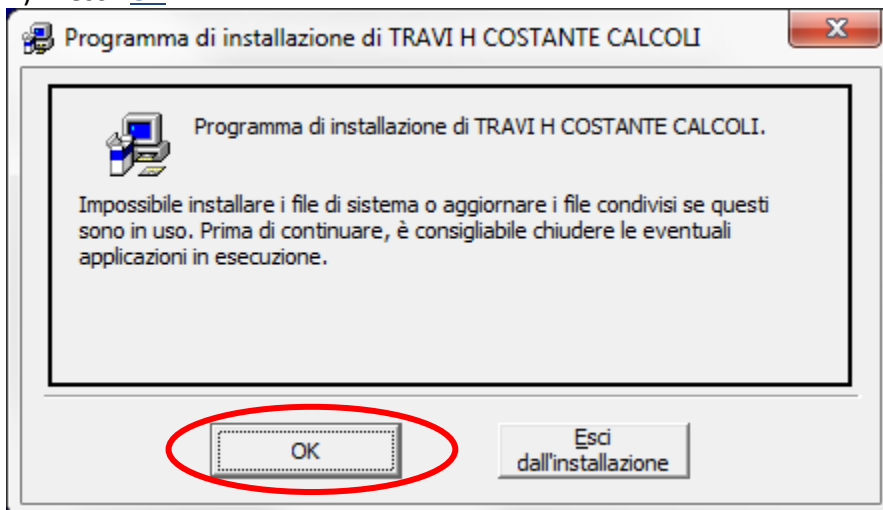
6. Press "OK"
7. Select the database "TraviHCostanti.mdb" from the directory created: "C:\Users\Public\Documents\Database".

4. PROGRAM'S INSTALLATION

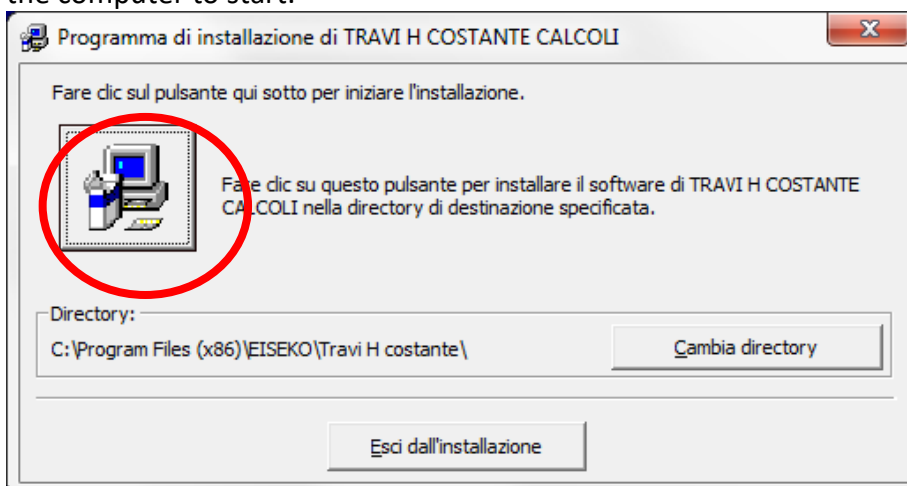
1) If you have a CD, enter the directory "TRAVI H COSTANTE", then "CALCOLI", and run setup.exe. If you download a zip installation file from internet: unzip it in any directory and run setup.exe. Repeat for directory GEOMETRIE.



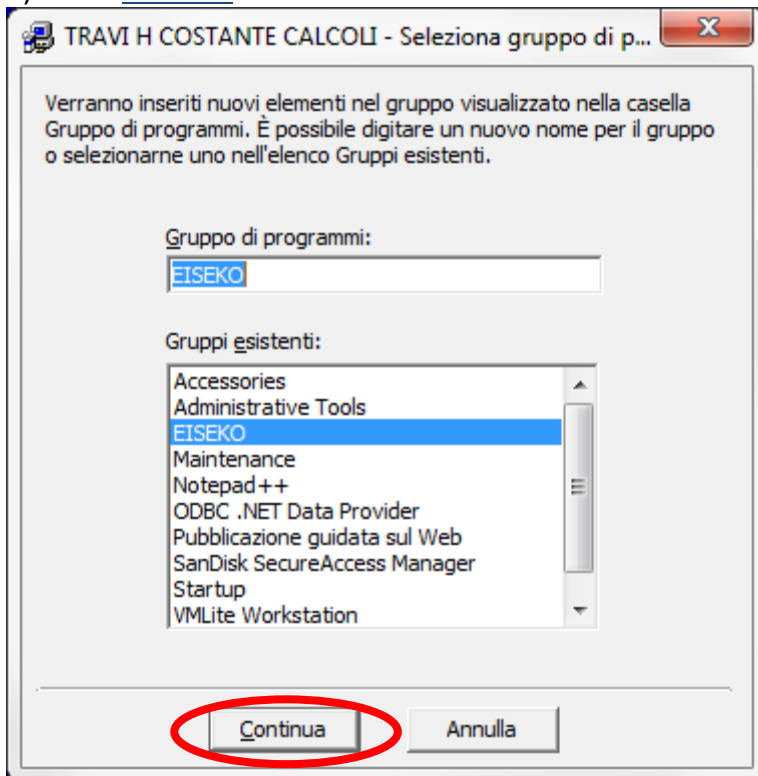
2) Press "OK"



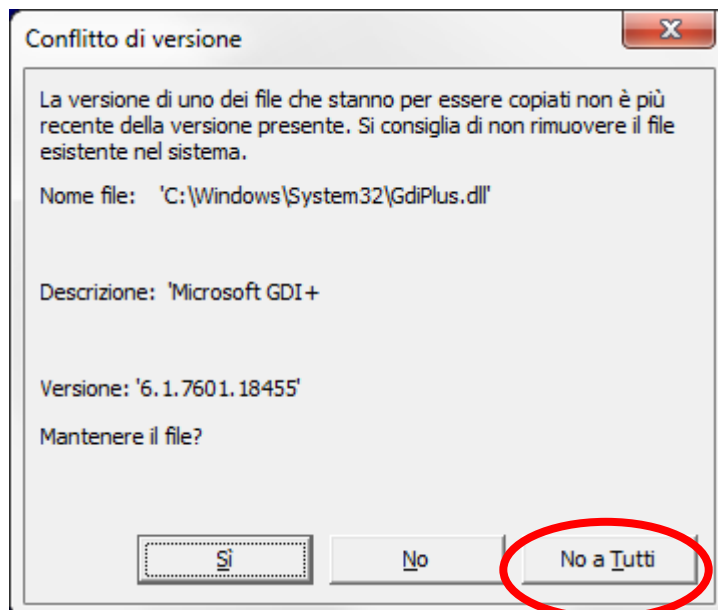
3) You can change the path for the program clicking on "[Change directory](#)". Click the button with the computer to start.



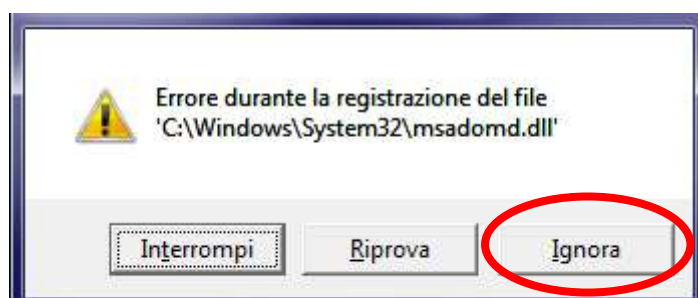
4) Press “Continue”:



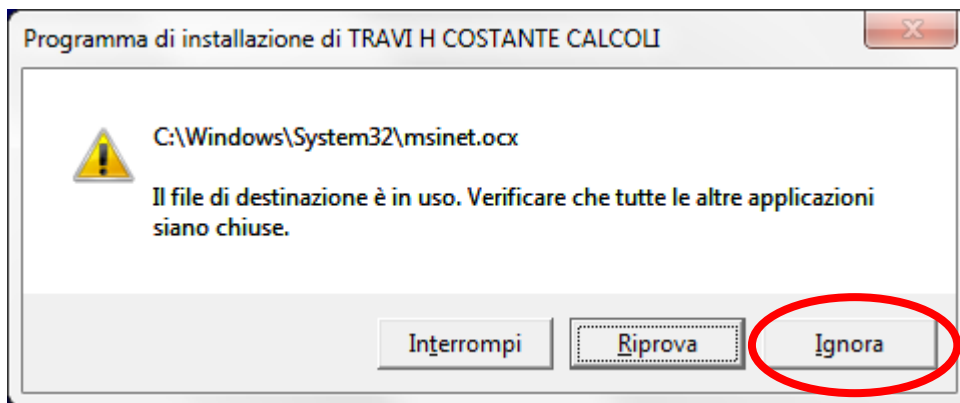
5) For any message regarding DLL (any dll):



Select “No to all” to grant update of dll.

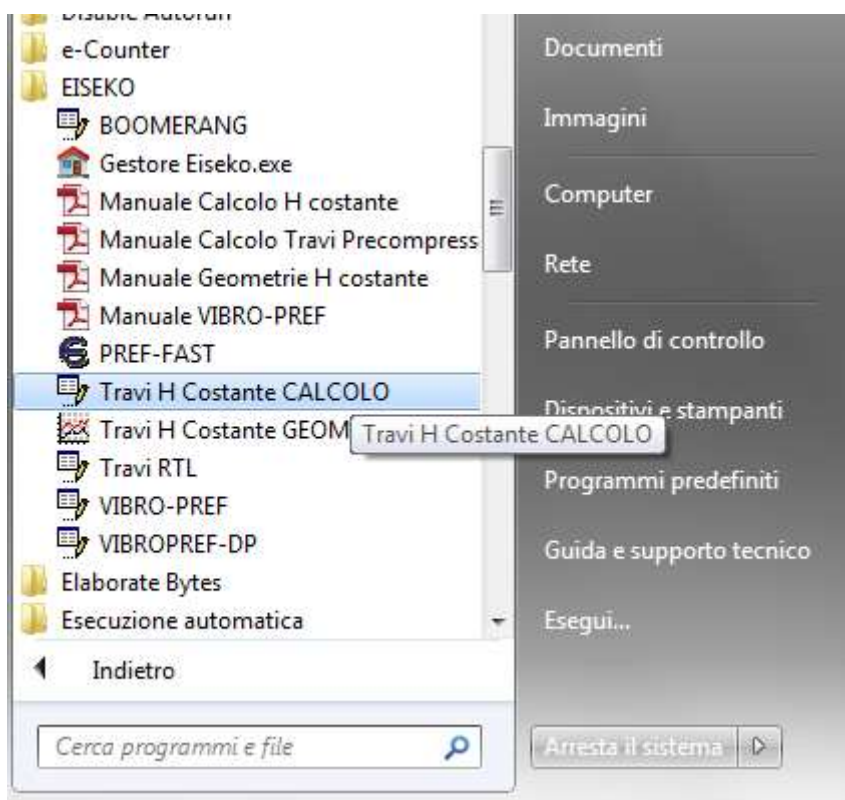


“Ignore” any error about dll (generally they are simply already registered).



Press “Ignore”.

6) To launch the program now select the menù Start on bottom left:
 “All the programs” – “Eiseko” – “Travi H costante CALCOLO”



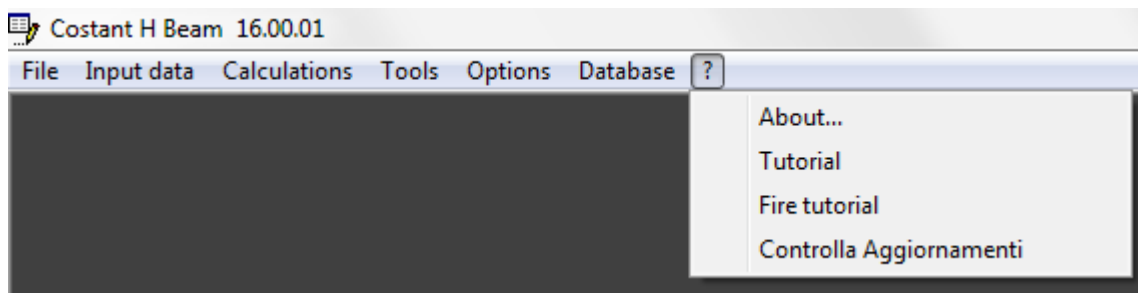
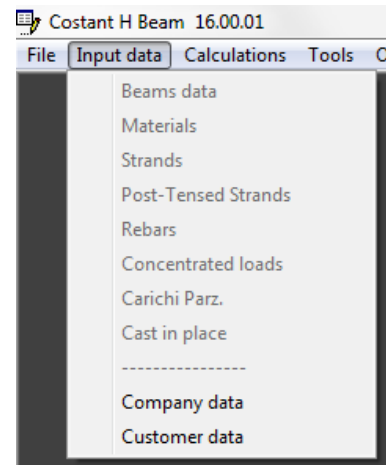
Or select “Travi H costante” from *Manager of the programs* Eiseko.

5. Initial SET UP

The first time you run the program set the name of your company: from menu "Input data".

"[Company data](#)" to set the data regarding your Company.

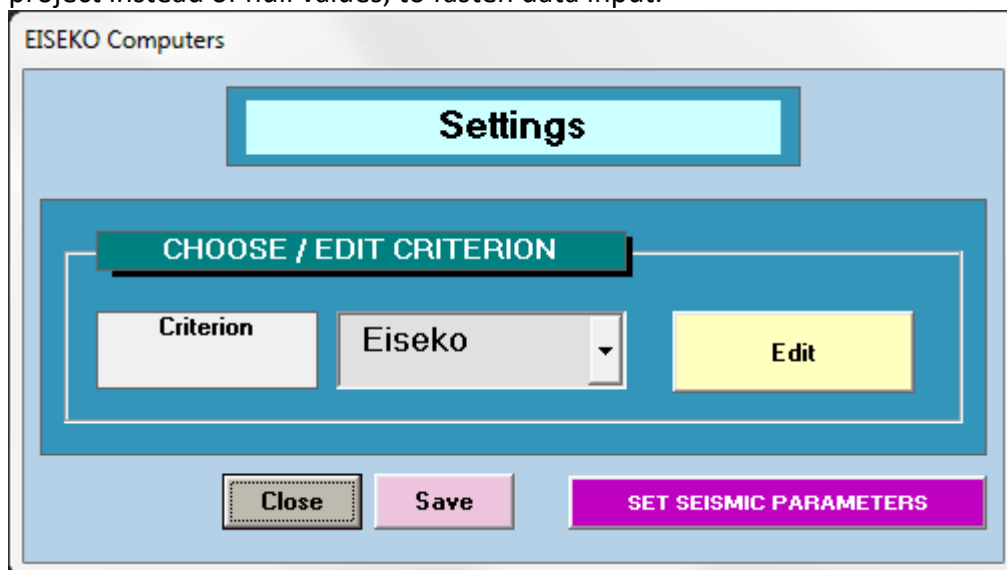
"[Customer data](#)" to set the data regarding your customer.



In "?" + "[About](#)" you can see the program release, the database you are using and last exportation. "[Tutorial](#)" opens this manual.

6. SETTINGS

From the main window press "[SETTINGS](#)" to set some default values you will use for new project instead of null values, to fasten data input.



6.1. SET SEISMIC PARAMETERS

You can set here all the seismic data related to the location of your job. See chap. 23.

The screenshot shows the 'SEISMIC PARAMETERS' dialog box. At the top, there's a title bar 'EISEKO Computers' and a main title 'SEISMIC PARAMETERS'. Below the title, there's a 'Job Order' dropdown menu set to 'nuovacomm'. Underneath, there are two buttons: 'DELETE LOCATION' and 'CHOOSE LOCATION', followed by a text field containing 'ROMAGNESE (PV)'. The main area contains several input fields: 'Location Latitude' (44.838), 'Location Longitude' (9.327), 'Use Class - Usage coef. - Cu' (II - 1), 'Structure factor' (1.5), 'Hor. Structure factor' (1.5), 'Topographic Category' (T1), and 'Subsoil Category' (B). At the bottom, there's a printer icon, a 'Close' button, and a 'Save' button.

6.2. CRITERIA

All “default data”, as a group, can be saved in a “CRITERION”: e.g., if an engineer works with more companies, he can save the different data for the different companies in more *criteria* saved with a name indicating the company. **Before** creating a new project select the Criterion of the company for which you are making the project. The use of the “Criteria” is particularly useful for lifting and transportation schemas, that always varies.

Select the Criterion from the combo box and click the button “EDIT” to set/modify the data:

The screenshot shows the 'CRITERION' dialog box. At the top, there's a title bar 'EISEKO Computers' and a main title 'CRITERION'. Below the title, there's a subtitle 'SET THE DEFAULT VALUES FOR THE NEXT NEW PROJECTS'. Underneath, there's a 'PROJECT CRITERION' dropdown menu set to 'Eiseko', followed by three buttons: 'Add Criterion', 'Copy Criterion', and 'Delete Criterion'. The main area is divided into four tabs: 'GENERAL', 'FIRE', 'TRANSP./LIFT', and 'STIRRUP DESIGN'. The 'GENERAL' tab is active, showing various input fields: 'Left Overhang' (0.1 m), '1st Sect. from support' (1 m), 'POST-TENSION' section with 'N. of Strands in cable' (3) and 'Single strand Area' (1.39), 'UNIT OF MEASURE' section with 'Italian Calculation' (Kg / cm²) and 'T.U. 2008 - EC2' (N / mm²), 'Length/Height Ratio' section with 'L/H - R' (35) and a warning message, and 'LOADS' section with 'm² def. spacing' (10 m) and 'm'. At the bottom, there's a printer icon, a 'Close' button, a 'Save' button, and a '?' button.

On top you can manage the criteria: you see the one you are working on and then you can add, delete, copy criteria.

On bottom you have the default data(of the criterion chosen on top) to modify, groped in tabs.

6.2.1. GENERAL tab

LEFT OVERHANG Default value for support: when you enter the beam length in “BEAM DATA” and press “Enter”, the program inserts automatically the left support here selected and the effective span (as total length minus left overhang x 2 (left + right), to fasten data enter.

1ST SECT. FROM SUPPORT: Only for beam with two sections, it’s the default value for the length of the end part of the beam, with different section from midspan. Same value assigned for left and right ends.

Insert sup strands in strands design You can automatically design the strands, once you entered the data of the beam: check the option if you want the program to design also superior strands in your section.

L/H = R Ratio (effective span / height) for the check of the input data. If the limit is exceeded the program will give you a warning.

UNIT OF MEASURE Select the unit of measure (just for Italian calculation).

LOADS m^2 / ml Select the type of data input for loads: meter or m^2 . If you select m^2 you will be prompted for a span.

6.2.2. FIRE tab

Set calculation rules, REI, mesh, temperature of stirrups for every REI standard and aggregate type. When you run the fire check, in the window “Fire”, the program will propose automatically these data and user can change them for every single project, salving them with the other data.

GENERAL	FIRE	TRANSP./LIFT.	STIRRUP DESIGN
Left Overhang	0.1	m	<input checked="" type="checkbox"/> Insert Sup Strand in Strands Design
1st Sect. from support	1	m	
POST-TENSION		Length/Height Ratio	
N. of Strands in cable	3	L/H = R 35	
Single strand Area	1.39	If the beam length is longer than R*H then a warning message will be displayed.	
UNIT OF MEASURE		LOADS m² / m	
Italian Calculation	<input checked="" type="radio"/> Kg / cm ² <input type="radio"/> N / mm ²	<input checked="" type="radio"/> m ² def. spacing 10 m	
T.U. 2008 - EC2	<input type="radio"/> N / mm ²	<input type="radio"/> m	

6.2.3. TRANSP/LIFT. tab

GENERAL		FIRE		TRANSP./LIFT.		STIRRUP DESIGN	
				LIFTING		TRANSPORT	
Length > m	Length <= m	L. Lift. m	R. Lift. m	Left (m)	Right(m)	2nd support	
2	10	1	1	1	1	0	
10	12	1	1	1	1	0	
12	15	1	1	1.5	1.5	0	
15	20	1	1	2	2	0	
20	25	1	1	2.5	2.5	0	
25	30	1.5	1.5	1	2.5	5	
30	35	1.5	1.5	1	3	6	

TABLE OF LENGTHS This table sets a schema of values for lifting and transportation supports: in the first two columns you set the minimum and maximum values of support for a determined range of length of the beam. For example in the table above we set that a beam with length from 2 to 12 m, shall have a support = 1m, while between 12 and 15 m support will be 1.5m, and so on. For example: if you have a beam length = 14 m (so it is in the range of the third row) it will have a lifting overhang = 1 m and a transportation overhang = 1.5m. If you have a beam length = 27 m (so it is in the range of the sixth row) it will have a lifting overhang = 1.5 m and a transportation overhang = 1m on left and 2.5 on right, with 2nd support = 5 m.

ADD ROW \ DELETE ROW \ DELETE ALL: these commands let you work on the table's rows.

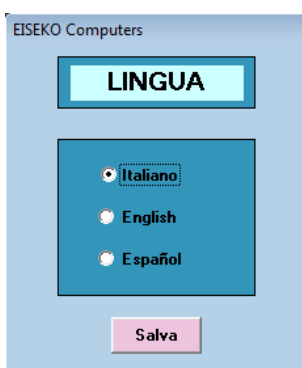
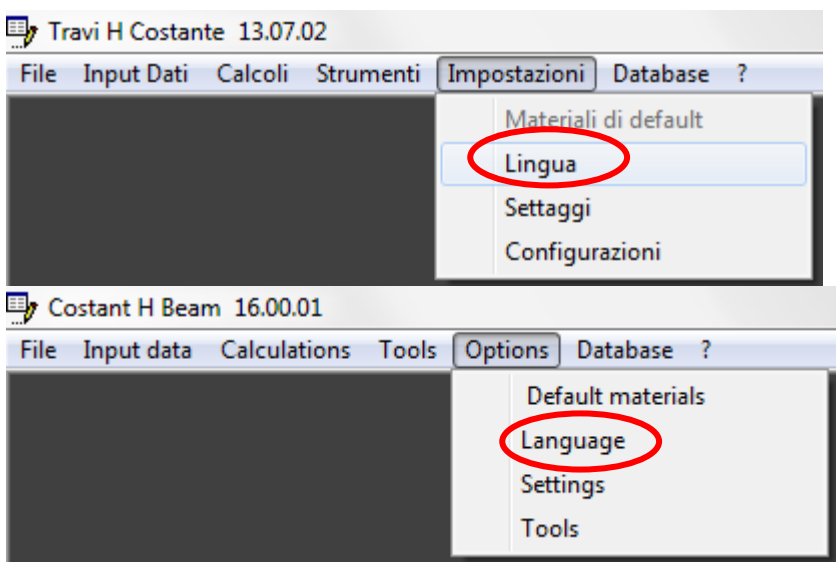
6.2.1. STIRRUP DESIGN tab

Set default data for the stirrup design (see dedicated paragraph).

GENERAL	FIRE	TRANSP./LIFT.	STIRRUP DESIGN
Field number	0	Stirrup Min Diameter	5
First Field length	0 cm	Stirrup Max Diameter	5
Vertical bar number	0	Inf.Rebar	
Stirrup spacing	0 cm	Number	0
Stirrup Diameter	5	Diameter	0

7. CHANGE LANGUAGE

To change the language used by the program to go to the Settings menu, Language:



Available languages are:

1. ITALIAN
2. ENGLISH
3. SPANISH



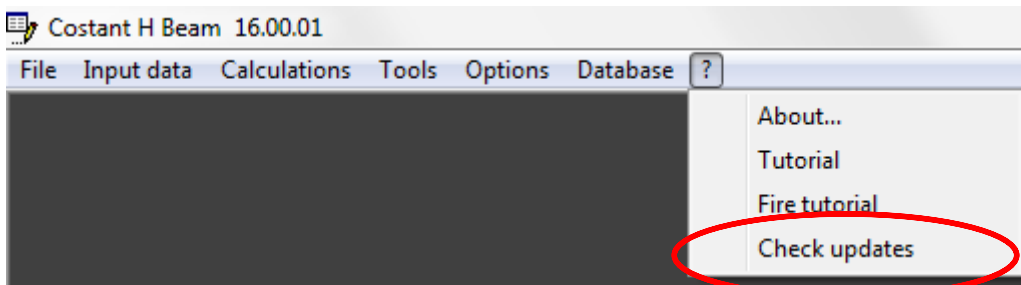
Select the desired language. The language is automatically updated for all open windows. All windows, reports and the program warning messages will be in the chosen language.

MODIFICATION EFFECTS ALL PROGRAMS of EISEKO COMPUTERS.

8. AUTOMATIC UPDATES



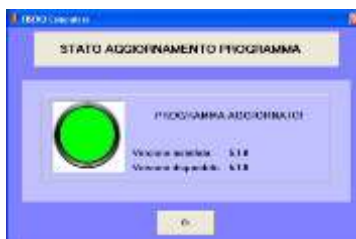
This window at startup warns if it is available on our website <http://www.eiseko.it/login/> a more recent version of the program that you are using



TO BE UPDATED

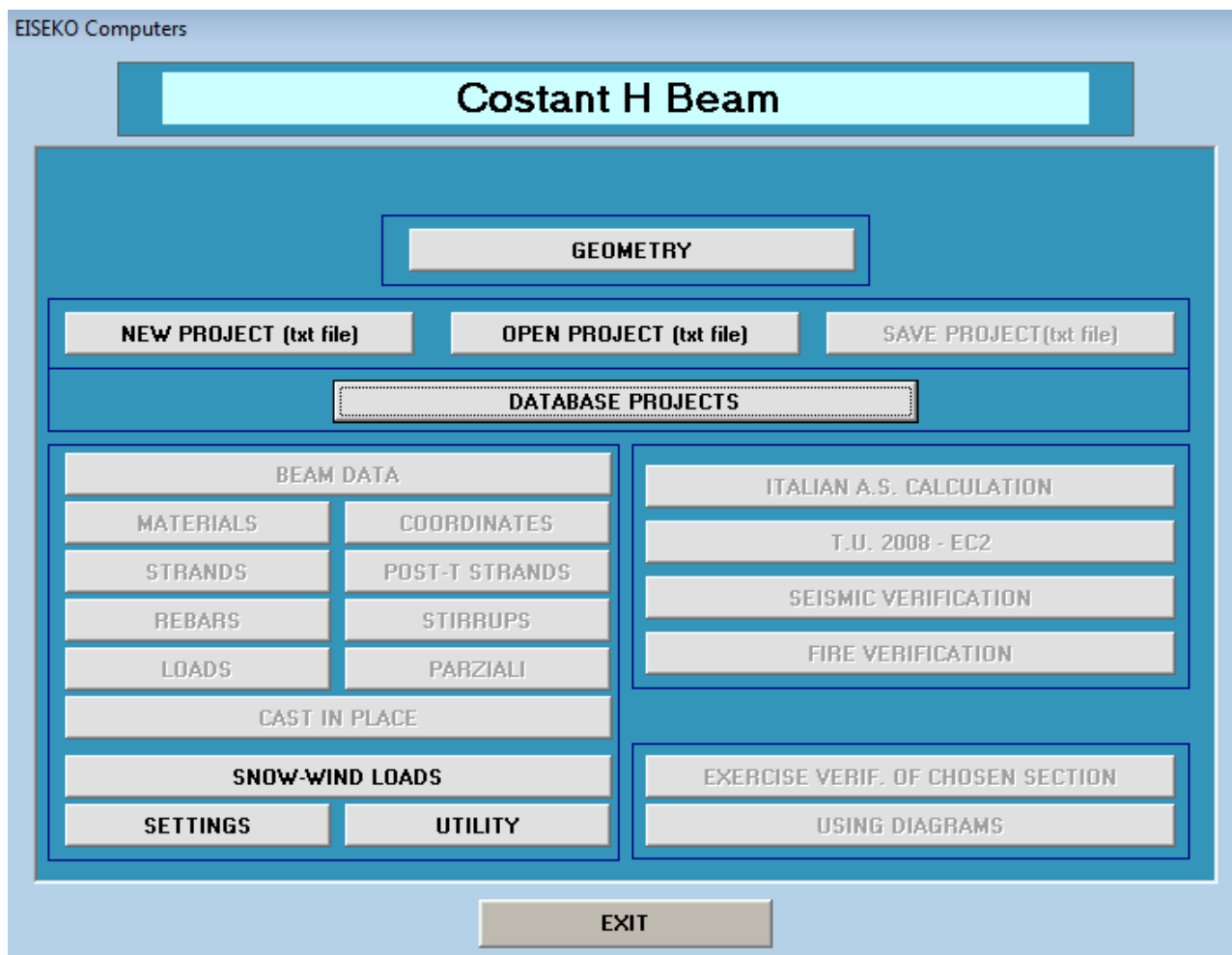


OK



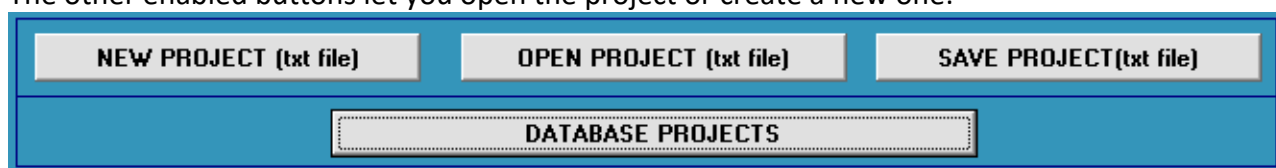
Click here to start update check

9. MAIN WINDOW



The button [“Geometry”](#) on top launches the Modulus “Geometry” to edit the sections of the beams (with strands and rebars).

The other enabled buttons let you open the project or create a new one:



To work on the database choose [“DATABASE PROJECT”](#), to work on the file [“NEW PROJECT \(txt file\)”](#) or [“OPEN PROJECT \(txt file\)”](#). For more details on this see [“DATA MANAGEMENT”](#).

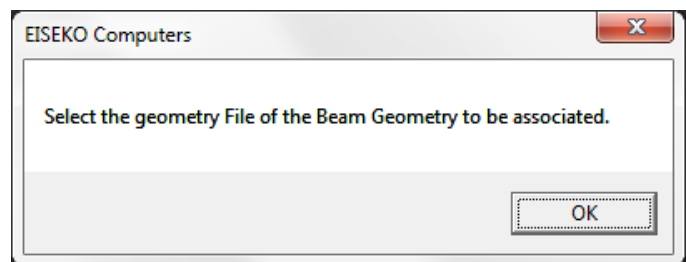
If you work on the database, you have all the beams and projects created with the calculation and geometry programs within the same file, which is managed automatically by the program and you don't need to worry about where to save it (you can see the name and path of the file in the

"Information" menu). You can still do operations like save more databases, use network database, change database. **WE RECOMMEND A REGULAR BACKUP OF THE DATABASE.**

If you work on files (save the data on text files) you can save the file where you want, locally or in the network. You have to work on files also in Geometry program, which will create the text files of the sections of the beams. Then, when you create a project, you will have to manually select the files to be associated with the beam from the directory where you saved it. **WE RECOMMEND A REGULAR BACKUP OF THE DATABASE.**

9.1. NEW PROJECT ON FILE

The program asks you to select the file with the geometry of the beam to be associated to the project, then open the standard Windows for opening the file. Select the correct file.



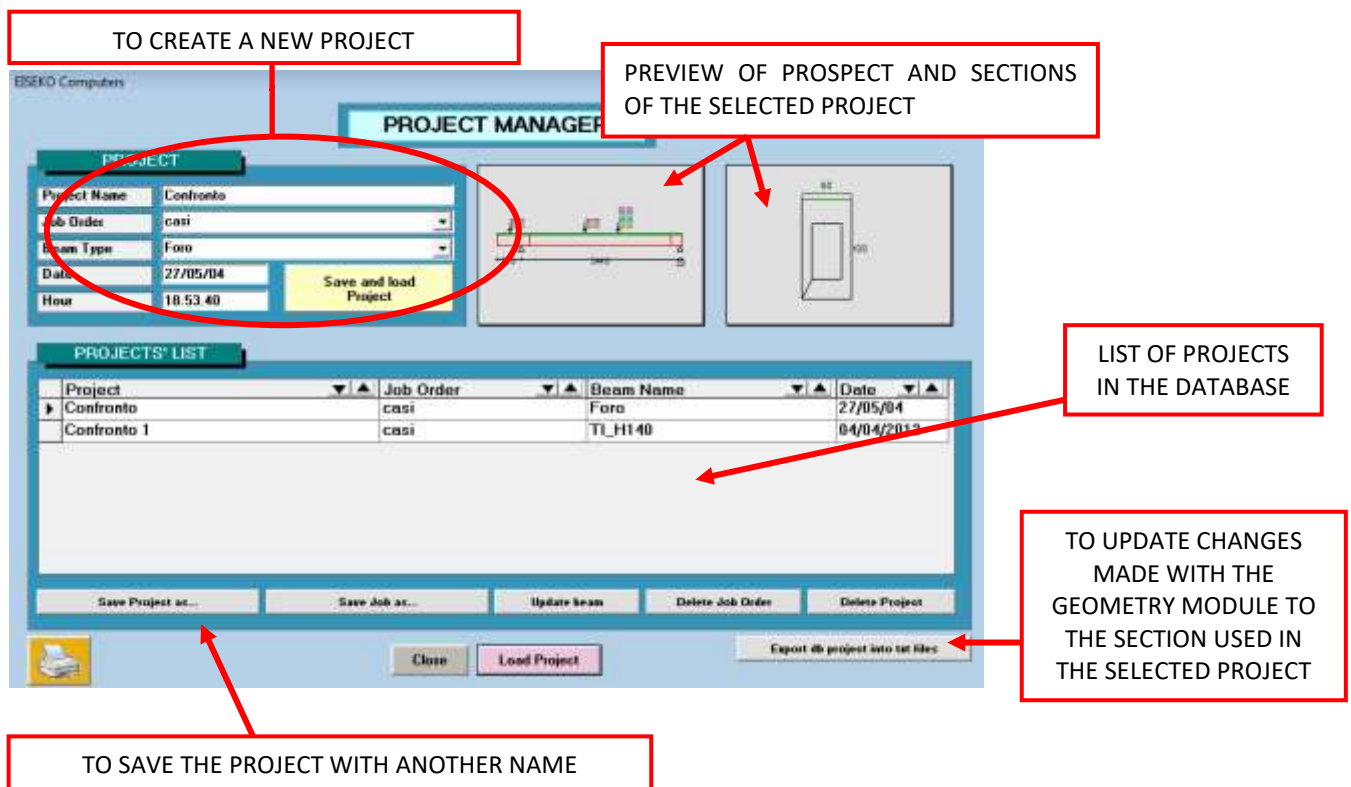
9.2. OPEN PROJECT FROM FILE

The program asks you to select the project file. It must be a project file created with this program, you cannot open other file types (not even other Eiseko programs).

9.3. DATABASE PROJECTS

It opens a window where you can create a new project, save a project with another name, delete projects or whole orders, all working on a single database.

If I select a project from the list, I can preview it.



The screenshot shows the 'PROJECT MANAGER' window with several components and annotations:

- TO CREATE A NEW PROJECT:** Points to the 'PROJECT' section where fields for Project Name, Job Order, Beam Type, Date, and Hour are entered, along with a 'Save and load Project' button.
- PREVIEW OF PROSPECT AND SECTIONS OF THE SELECTED PROJECT:** Points to the two preview windows on the right showing a 3D beam model and a 2D cross-section.
- LIST OF PROJECTS IN THE DATABASE:** Points to the 'PROJECTS' LIST table below.
- TO UPDATE CHANGES MADE WITH THE GEOMETRY MODULE TO THE SECTION USED IN THE SELECTED PROJECT:** Points to the 'Export db project into txt files' button at the bottom right.
- TO SAVE THE PROJECT WITH ANOTHER NAME:** Points to the 'Save Project as...' button at the bottom left.

Project	Job Order	Beam Name	Date
Confronto	casi	Foro	27/05/04
Confronto 1	casi	TL_H140	04/04/2019

JOB is a group of projects, usually referred to a customer, created to facilitate the management of projects within the database. As the name suggests, it is very useful to divide the calculations carried out for JOBS (orders), so that you can easily find them also after some time. NOTE: When you delete a JOB, you delete all the projects grouped in it.

9.3.1. TO CREATE A NEW PROJECT

On top left enter the necessary data:

the project name: unique name that identifies the project, must be different from all other existing ones in the database.

the name of the job: the job name for the project (the job is a group of projects, as explained above). To put the project in a job already created just select it from the drop-down list. Once selected, the list of projects is updated by displaying only the projects in the order selected. If the job does not exist, it is created automatically.

the beam name choose from the drop down list the section you already created with the Geometry module. Once selected, the list of projects is updated by displaying only the projects using the selected beam. You cannot insert new beams at this stage, the beam must have already been created with the Geometry module.

date and time: date and time are written automatically by the program.

Pressing "[SAVE AND LOAD PROJECT](#)" you create, save and load the project.

9.3.2. TO OPEN AN EXISTING PROJECT

To change a calculation already done just select it from the list and click the button "[LOAD PROJECT](#)", or double-click on the project list. The project could be edited and all changes will be saved automatically.

9.3.3. UPDATE BEAM

This button updates the beam of the project with the changes made with the Geometry module. With this calculation program you can change some parameters of the geometry of the inserted beam, or strands/rebars, however, if you were to make radical changes (change the sections, for example) will need to make changes from the program for the geometry, and then, to update the data of any projects already created with the modified beam, you will need to click the button "[UPDATE BEAM](#)". The project will remain intact with its values (Materials, Data Beam etc.) geometry will only be updated.

9.3.4. SAVE PROJECT AS

As this program automatically saves the data in the database, to save a project with a different name and then change it, you must use the button "[Save project as ...](#)" BEFORE CHANGING THE DATA.

9.3.1. DELETE PROJECT / DELETE JOB

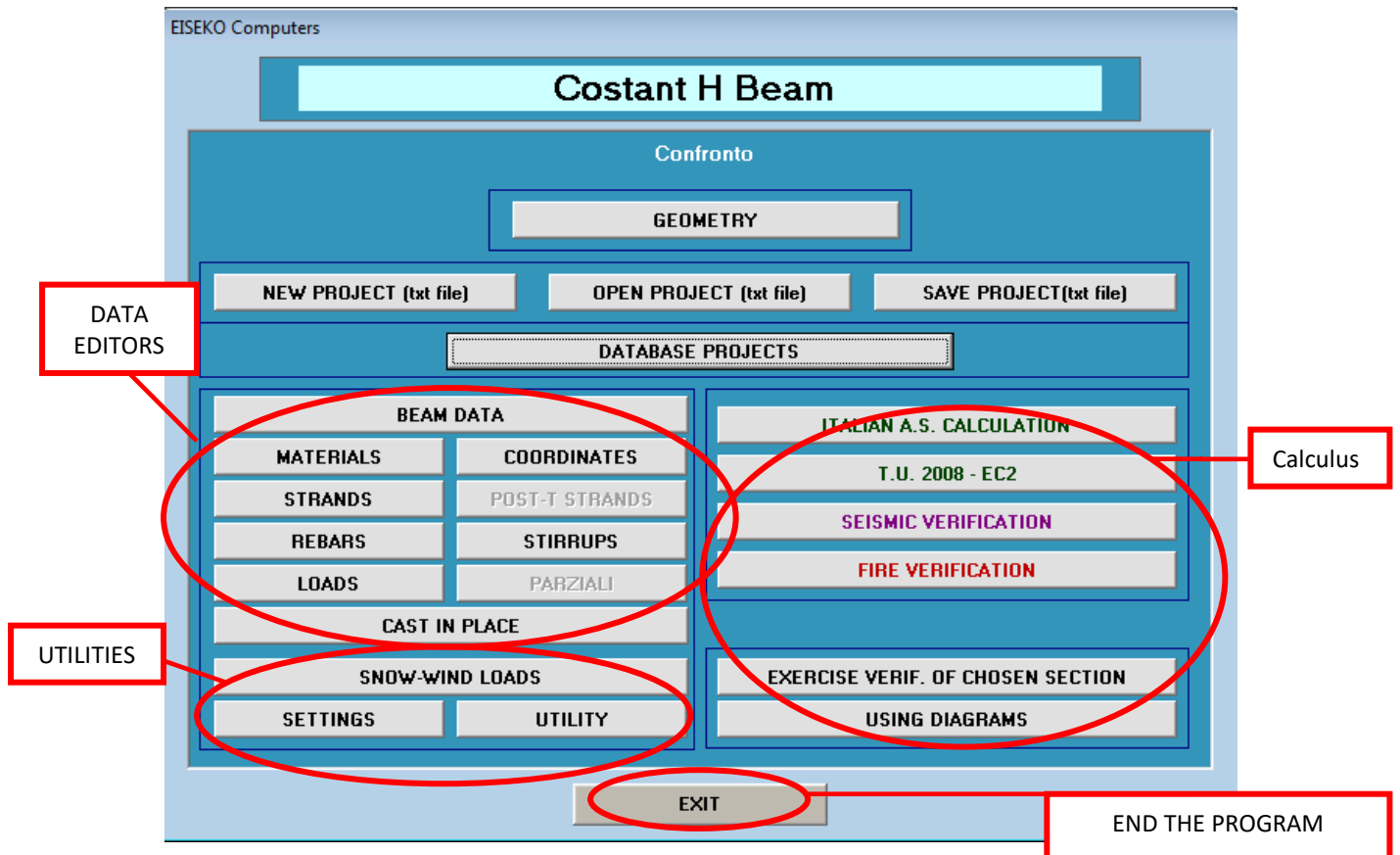
To delete a project select it from the list and click the button "[Delete project](#)".

YOU CAN NOT RECOVER DELETED PROJECTS, MAKE REGULAR BACKUP COPIES OF THE DATA.

To delete a job, select a project that was saved in that job from the list, and click the button "[Delete job](#)". If the job has no projects, create one new to delete the job. ***The elimination of the job will delete all the projects within it.*** To delete a job confirmation is required twice.

YOU CAN NOT RECOVER JOB ELIMINATED, MAKE REGULAR BACKUP COPIES OF THE DATA.

Once the project is loaded, the main window becomes:



We will analyse all the buttons in detail.

10. BEAM DATA

PROJECT: Confronto - BEAM NAME: Foro - JOB: casi

BEAM DATA Foro Width 50 Height 100 PP kg/m 038 PP kg/m² 84 L/N 13.71 < 35

Beam Length	16.8	m	Effective span	14.4	m
Left Overhang	2.2	m	Length of 2nd overhang Transp.	0	m
Lifting	Left 1 Right 1	m	Transport Overhang	Left 2 Right 2	m
Head section Min.Width	50	cm	Midspan section Min.width	20	cm
Support min. H for shear calculation	100	cm	Head length	Left 3 Right 1	m

Cast in place ☒ CIP Concentrate loads ☒ LOADS Partial loads ☐

loads m	L.Beam Spacing	10	m	R.Beam Spacing	10	m
loads m²	L.Floor beam length	10	m	R.Floor beam length	10	m

G1 fully def.perm.loads on beam only (CIP+others)	1800	kg/m	G1 fully def.perm.loads on beam and CIP	0	kg/m
G2 perm.loads NOT def. on beam + CIP	0	kg/m	G2 perm.loads NOT def. on beam + CIP	0	kg/m
Qk1 var.loads on beam and CIP	1500	kg/m	Category	D: shopping areas	
DOMINANT	150	kg/m²	Coefficients	ψ11 0.7 ψ21 0.6	
Qk2 variable loads	0	kg/m	Category	Other	
	0	kg/m²	Coefficients	ψ02 0.7 ψ12 0.7 ψ22 0.6	

LOCAL VERIFICATION

TOOTH	Tooth Load	0	kg	Base	0	cm	Height	0	cm
SUSPENSION	Susp. Load	0	kg	Incidence	0	cm	At tooth	0	cm²/m

Coef. SLU γG1 1.3 γG2 1.5 γQk1-γQk2 1.5

Relative humidity environment % 60 Rck 530 Rck1 420 Rck G C20/25

Exposure classes XC3 Moderate humidity or cyclic wet and dry

SHEAR VERIFICATION (6.2-EC2) NOT PRESTRESSED SECTION

EDIT ANGLE Strut Angle θ = 33°.69 Cotg θ = 1.5 θ Calculated = 0°

Close Save ITALIAN CALC T.U. 2008 - EC2 LEFT CUT SETTINGS STRANDS REBARS POST-TENSIONED STIRRUP DESIGN

Working title

10.1. NOMENCLATURE

BEAM LENGTH (m) = Total Length of the beam (that never coincides with the distance between the two theoretical support points).

EFFECTIVE SPAN (m) = Distance between the two theoretical support points of the beam.

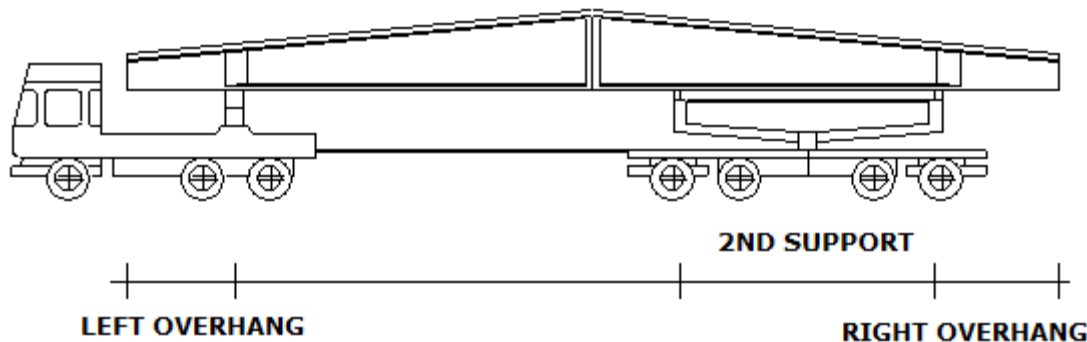
SBALZO SINISTRO (m) = Distance between the left point of support and the head beam to the left, if the beam has a big overhang also on the right must, the left must be the higher of the two.

END SECTION MIN. WIDTH (cm) = Used for the calculation of the min. stirrup area on ends and tau on support.

MIDSPAN SECTION MIN. WIDTH (cm) = Used for the calculation of the min. stirrup area along the beam. The program proposes min widths of the initial and midspan sections entered in the Geometry module. These values are editable.

Head LENGTH = If head section is different from midspan section, this value indicates the length of the part with the head section. These values must be positive and < 2 m. The length is the distance from the left end of the beam and point when the midspan section start.

Note: In case of one sole section, value is not requested (not even visible).



L TRANSPORT (m) = LEFT TRANSPORT OVERHANG = distance between left support on track and left beam end.

2nd support (m) = If beam is on a support like in the above picture, insert here the 2nd right support. If on track it is simply on 2 support, set this value to 0

R TRANSPORT (m) = RIGHT TRANSPORT OVERHANG (m) = distance between right support on track and right beam end.

L/R LIFTING (m) = Position of left and right lifting overhang, in meters.

MIN. SECTION HEIGHT for SHEAR (cm) = Height of section for the calculation of shear on support. The program propose the value entered in the GEOMETRY Module, user can change it.

10.2. FEATURES

DATA OF THE BEAM: MAXIMUM WIDTH AND MAXIMUM HEIGHT in cm, SELF WEIGHT in kg / m and kg / m2

PROGETTO: Confronto - NOME TRAVE: Foro - COMMESSA: casi

DATI TRAVE Foro Largh tot 50 H tot 100 PP kg/m 838 PP kg/m2 84 L/H 13.71 < 35

TO CHANGE THE SECTION FOR THE CALCULATION: JUST SELECT A DIFFERENT BEAM FROM THE LIST

L/H is the ratio span/height to check input data.

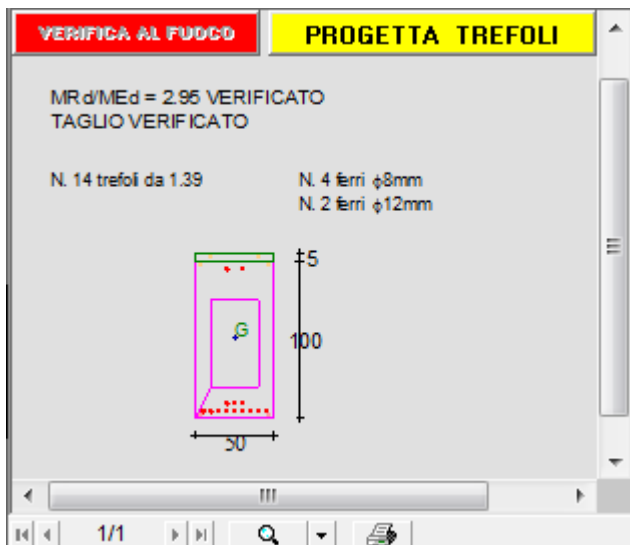
Lunghezza Trave	15.8	m	Luce di calcolo	14.4	m
Sbalzo Sinistro	2.2	m	Larghezza Rotta	0	m
Sollevamento	Sx 1 Dx 1	m	Trasporto	Sx 2 Dx 2	m
Larghezza minima sezione testata	50	cm	Larghezza minima sezione corrente	20	cm
Altezza MIN sez appoggio per taglio	100	cm	Lunghezza rinfresco	Sx 3 Dx 1	m

Getto in Opera ☒ GETTO Carichi Concentrati ☒ CARICHI Carichi Parziali ☐

TO INSERT THE CAST IN PLACE CHECK THE OPTION AND PRESS THE BUTTON

TO INSERT THE POINT LOADS CHECK THE OPTION AND PRESS THE BUTTON

VIEW THE PROSPECTUS WITH LOADS, CIP (IN GREEN), DIMENSIONS. Can be printed with the printer button on the top and change the zoom with the zoom button at the top or by double clicking the buttons of mouse: left -> zoom in; right click -> Zoom Out



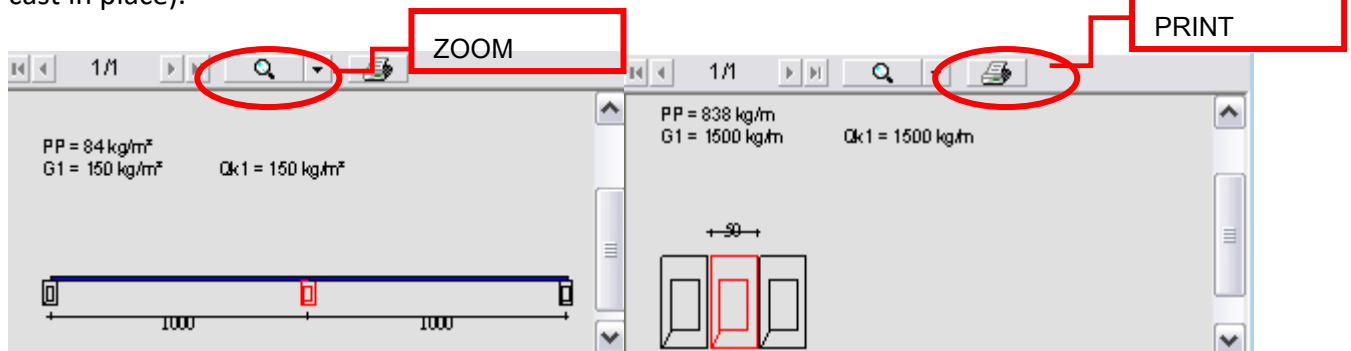
SCHEME OF THE BEAM: shows the midspan section, any cip, rebars, pre-tensioned strands, post-tensioned strands and the results of the checks at break and shear (checks only after it was made the calculation).

10.3. LINE LOADS

Loads may be entered by m or by m2.



Select the desired option: change the schema dynamically to the right (here it is never drawn the cast in place).



“[loads m²](#)”: In this case, enter the spans L / R and the length of the floor left / right. The loads m are calculated accordingly and used for the calculation. The weight of the beam is calculated by the program.

NOTE: Even if the geometry is provided, the program does not calculate the weight of the cast and it must be added to the loads.

10.3.1. LOADS WHEN CAST IN PLACE IS PRESENT

<input type="radio"/> loads m	L.Beam Spacing	10	m	R.Beam Spacing	10	m		
<input checked="" type="radio"/> loads m²	L.Floor beam length	10	m	R.Floor beam length	10	m		
G1 fully def.perm.loads on beam only (CIP+others)		1800	kg/m	G1 fully def.perm.loads on beam and CIP		0	kg/m	
		WEIGHT	180	kg/m²		0	kg/m²	
G2 perm.loads NOT def. on beam + CIP		0	kg/m					
		0	kg/m²					
Qk1 var.loads on beam and CIP		1500	kg/m	Category	D: shopping areas			
DOMINANT		150	kg/m²	Coefficients	Ψ11	0.7	Ψ21	0.6
Qk2 variable loads		0	kg/m	Category	Other			
		0	kg/m²	Coefficients	Ψ02	0.7	Ψ12	0.7
							Ψ22	0.6

G1 FULLY DEF. PERM. LOADS ON BEAM ONLY (CAST IN PLACE + OTHERS) = Loads supported by beam only when cast in place is present, with ultimate limit state they must be multiplied by γ_{G1} .

G1 FULLY DEF. PERM. LOADS ON BEAM AND CAST IN PLACE = Loads supported by beam and cast in place when cast in place is present, with ultimate limit state they must be multiplied by γ_{G1} .

G2 PERM. LOADS NOT DEF ON BEAM AND CAST IN PLACE = Loads supported by beam and cast in place when cast in place is present, with ultimate limit state they must be multiplied by γ_{G2} .

After permanent loads there are variable loads, you have to associate to every one of them a **category** of loads (tab. 2.5.I of TU 2008), to calculate corresponding Ψ . There is also a category "Other", to set your own Ψ .

Qk1 VAR. LOADS ON BEAM AND CAST IN PLACE = Variable dominant loads supported by beam and CIP when CIP is present, with ultimate limit state they must be multiplied by γ_{Qk1} .

Qk2 VARIABLE LOADS = Variable second loads supported by beam and cast in place when cast in p. is present, with ultimate limit state they must be multiplied by $\Psi_{02} \cdot \gamma_{Qk2}$.

10.3.2. LOADS WHEN THERE ISN'T A CAST IN PLACE

<input type="radio"/> loads m	L.Beam Spacing	10	m	R.Beam Spacing	10	m		
<input checked="" type="radio"/> loads m²	L.Floor beam length	10	m	R.Floor beam length	10	m		
G1 fully def.perm.loads		1800	kg/m			0	kg/m	
		WEIGHT	180	kg/m²				
G2 not fully def.perm.loads		0	kg/m					
		0	kg/m²					
Qk1 variable loads		1500	kg/m	Category	D: shopping areas			
DOMINANT		150	kg/m²	Coefficients	Ψ11	0.7	Ψ21	0.6
Qk2 variable loads		0	kg/m	Category	Other			
		0	kg/m²	Coefficients	Ψ02	0.7	Ψ12	0.7
							Ψ22	0.6

G1 FULLY DEF. PERM. LOADS = with ultimate limit state they must be multiplied by γ_{G1} .

G2 NOT FULLY DEF. PERM. LOADS = with ultimate limit state they must be multiplied by γ_{G2} .

After permanent loads there are variable loads, you have to associate to every one of them a **category** of loads (tab. 2.5.I of TU 2008), to calculate corresponding Ψ . There is also a category "Other", to set your own Ψ .

Qk1/ Qk2 VARIABLE LOADS = with ultimate limit state they must be multiplied by γ_{Qk1} .

10.4. WEIGHTS of floors to automatically calculate G1

G1 fully def.perm.loads 1800 kg/m

WEIGHT 180 kg/m²

You can store a group of frequently used floors to automatically calculate their weights, for the automatic calculation of G1, without having to consult every time the tables of weights.

Pressing the "[WEIGHT](#)" button the following window appears:

EISEKO Computers

WEIGHT OF ELEMENTS FOR AUTOMATIC CALCULATION OF G1

ID	Weight Name	Weight (kg/m²)	CIP H(cm)	Weight+CIP(kg/m²)	Permanent (kg/m²)	Total (kg/m²)
1	Esempio1-TT35	200	0	200	20	220
2	Esempio2-TT20	300	10	350	20	370

Selected = NONE

WEIGHT NAME: unique name to identify the type of floor.

WEIGHT: weight in kg/m² of sole floor.

CIP H: the height of cast in place (if present for the weight you are taking into account). The program calculates automatically the weight with 250 cm of CIP WIDTH, and write it in "WEIGHT+CIP".

PERMANENT: Enter the weight of permanent loads if present in kg/m².

The program will sum automatically all weights to calculate the total.

To set a weight to the beam data select the corresponding row (any cell of the row) and click the button "[SET SELECTED AS G1](#)".

In "BEAM DATA" it's possible to set directly a floor weight: click on the arrow on the right of the button "[WEIGHT](#)" and you get the list of the saved floors, select the name of the floor you want.

PESI 220 kg/m²

Esempio1-TT35

Esempio2-TT20

The weight is shown in "BEAM DATA" in the textbox G1 and in the graphic area with all details.

loads m L.Beam Spacing 10 R.Beam Spacing 10

loads m² L.Floor beam length 10 R.Floor beam length 10

G1 fully def.perm.loads 200 kg/m

WEIGHT 220 kg/m²

G2 not fully def.perm.loads 0 kg/m²

G3 variable loads 0 kg/m²

$G1 = 220 \text{ kg/m}^2$
 $G1 = 220 \text{ kg/m}^2 + (W + 200 \text{ CIP} + 0; \text{Permanent} = 20)$
 $Qk1 = 150 \text{ kg/m}^2$

To delete it just change the value directly in "BEAM DATA" or in the window "WEIGHTS" click the button "[Deselect weight G1](#)".

10.5. LOCAL VERIFICATION

The loads which are discussed in this paragraph are not additional to those included in the window of "BEAM DATA" but they only serve for the calculation of the stirrups required in the particular situations listed below.

Check of the load on a projecting tooth Enter the geometric data of the tooth and the load on it (on one of the two sides) with the WIDTH of its footprint. The program takes into account the distribution of the load and it shows the required stirrups.

LOCAL VERIFICATION									
<input checked="" type="checkbox"/> TOOTH	Tooth Load	8000	kg	Base	22.5	cm	Height	25	cm
<input type="checkbox"/> SUSPENSION	Susp. Load	0	kg	Footprint	15	cm	Af tooth	3.33	cm ² /m

Check of tooth and suspension. If the tooth is on the lower side of the beam (as it happens in the large majority of cases) there is an effect of suspension associated with the action of the load on the tooth, then in addition to the check of "TOOTH" we must add the check on "SUSPENSION". The program automatically inserts a double load than that on the tooth, since the suspension is viewed by the program as an agent on both sides of the beam web. The spread of the load is calculated for the tooth. The stirrup area for suspension is compared to that required by the calculations of the beam and the program prints the larger of two.

LOCAL VERIFICATION									
<input checked="" type="checkbox"/> TOOTH	Tooth Load	8000	kg	Base	22.5	cm	Height	25	cm
<input checked="" type="checkbox"/> SUSPENSION	Susp. Load	16000	kg	Footprint	15	cm	Af tooth	3.33	cm ² /m

Check suspension only. It is necessary in the case of load directly hanging on the web of the beam. Enter only the values of the suspended load and its footprint, in this case the program doesn't need to calculate the spread of the load. The stirrup area for suspension is compared to that required by the calculations of the beam and the program prints the larger of two.

LOCAL VERIFICATION									
<input type="checkbox"/> TOOTH	Tooth Load	0	kg	Base	0	cm	Height	0	cm
<input checked="" type="checkbox"/> SUSPENSION	Susp. Load	5000	kg	Incidence	50	cm	Af tooth	0	cm ² /m

10.6. COEFFICIENTS

Coeff. SLU	γ_{G1}	1.3	γ_{G2}	1.5	$\gamma_{Qk1}-\gamma_{Qk2}$	1.5
Relative humidity environment	%	60	Rck	530	Rckj	420
Exposure classes	XC3 Moderate humidity or cyclic wet and dry					

ULS Coeff.: ultimate limit state coefficients for perm. Fully def. loads γ_{G1} : used to calculate ultimate moment and shear, allowable value is 1.3.

ULS Coeff.: ultimate limit state coefficients for perm. Not Fully def. loads γ_{G2} : used to calculate ultimate moment and shear, allowable value is 1.5.

ULS Coeff.: ultimate limit state coefficients var. loads $\gamma_{Qk1}-Q_{k2}$: used to calculate ultimate moment and shear, allowable value is 1.5.

RELATIVE ENVIRONMENT HUMIDITY % = Environmental average humidity expected in the life of the structure (for the calculation of losses for shrinkage and creep).

EXPOSURE CLASS = is related to environmental conditions (for the limit states checks).

Rck – Rckj – RckG are Beam concrete Rck at 28 days - Concrete Rckj at initial time - Concrete in place Rck at 28 days – respectively, and are editable both here and in the "MATERIALS" window.

10.7. AUTOMATIC DESIGN OF STRANDS

The "**DESIGNS STRAND**" button allows you to design directly the minimum required strands as required for break verification. After entering beam data, click this button and the number of strands designed is shown above the section drawing. It still proceeds with control of all the checks at various stages.

After you run the design, in the beam schema in the lower right the checks are displayed (if shear is verified, if break is verified, and if there are values exceeding the limits in the table of calculation).

10.8. SHEAR on support

SHEAR VERIFICATION (6.2 EC2) NOT PRESTRESSED SECTION

EDIT ANGLE

Strut Angle $\theta = 33^\circ.69$ Cotg $\theta = 1.5$

θ Calculated = 0°

Shear Verification in the not-prestressed section on support, following the method “variable strut inclination method”: calculated angles are shown, to change them click the button “**EDIT ANGLE**”.

PROJECT: Confronto - BEAM NAME: Foro - JOB: casi

SHEAR VERIFICATION (6.2 EC2) NOT PRESTRESSED SECTION

☒ $v = 0.6 * [1 - f_{ck} / 250] \rightarrow f_{ywd} = f_{yk} / 1.15$ EC2 6.6 N
☐ $v = v_1$ $0.5 < v_1 \leq 0.6 \rightarrow f_{ywd} = 0.8 * f_{yk} / 1.15$ EC2 6.10.a N EC2 6.10.b N

<input type="radio"/> Strut Angle $\theta = 45^\circ.00$ Cotg $\theta = 1.0$	<input type="radio"/> Strut Angle $\theta = 26^\circ.57$ Cotg $\theta = 2.0$
<input checked="" type="radio"/> Strut Angle $\theta = 33^\circ.69$ Cotg $\theta = 1.5$	<input type="radio"/> Strut Angle $\theta = 21^\circ.80$ Cotg $\theta = 2.5$
<input type="radio"/> Strut Angle input $\theta =$ <input type="text" value="0"/> Cotg $\theta =$ <input type="text"/>	
<input type="radio"/> Strut Angle $\theta =$ calculated variable $1 \leq \text{Cotg } \theta \leq 2.5$ $\theta =$ <input style="background-color: yellow;" type="text" value="5.08"/>	

Close Save

Following the method “variable strut inclination method”, the user, as provided by EC2, can choose different angles for the Shear verifications of the strut between 45° and $21^\circ.80$.

Some values already set for convenience, can be chosen directly from the user as values to use in the calculation (45° , $33^\circ.69$, $26^\circ.57$, $21^\circ.80$), or you can enter the desired angle, or you can opt for the automatic angle calculation by the program ($\theta =$ calculated variable), in this case the value of the angle is such as to have simultaneously the breaking of concrete and steel.

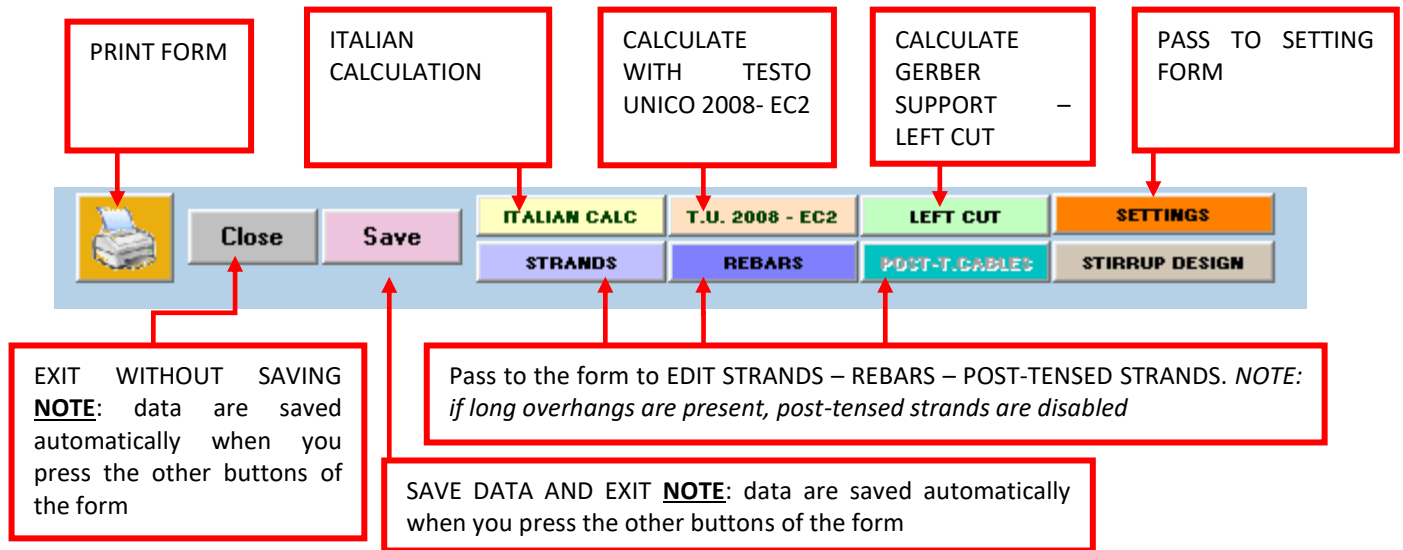
When you select the option for automatic calculation of the angle, the program, as from Rules, compares the calculated value with the minimum limits of $21^\circ.80$ and up to 45° : if the calculated value is less than the minimum then it would set equal to $21^\circ.80$, if it is higher than the maximum it is set equal to 45° .

The configuration shown above is the default, where Testo Unico and EC2 are equivalent if $\theta = 45^\circ$ a part from v of EC2.

You can also choose $v=v_1$, possible in EC2, in which it is overestimated the strength of concrete of 20 and simultaneously it is decreased the strength of steel by another 20% ($* f_{yk} f_{ywd} = 0.8 /$

1.15). It could be useful in the event that the concrete fails and you cannot vary the section or the materials.

10.9. BUTTONS



NOTE: IF LONG OVERHANGS ARE PRESENT, POST-TENSED STRANDS ARE DISABLED

The diagram shows a form layout with two main input areas. On the left, there is a 'Working title' field with a yellow background. On the right, there is a larger text area for 'NOTES OR COMMENTS FOR VARIOUS PROJECT. DISPLAYED WHEN USER PRINTS THE FORM.'.

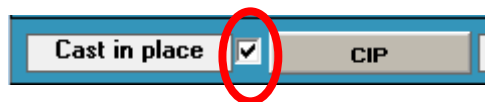
Working title

NOTES OR COMMENTS FOR VARIOUS PROJECT. DISPLAYED WHEN USER PRINTS THE FORM.

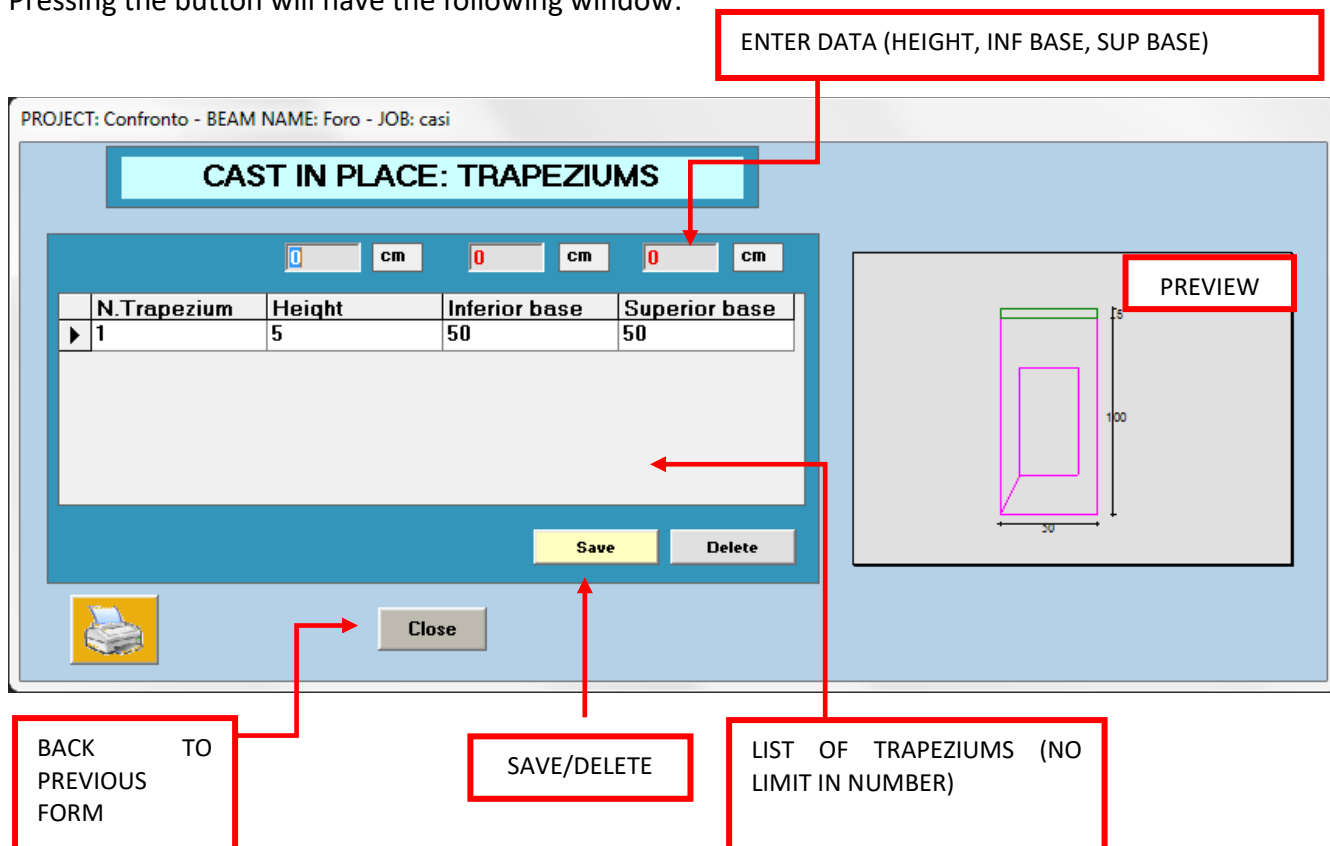
Then we have three check boxes: CAST IN PLACE, CONCENTRATED LOADS and PARTIAL LOADS, to check if you want to introduce, respectively, the casting and concentrated loads; **PARTIAL LOAD is not yet active.**

11. CAST IN PLACE

To enable the button "[CIP](#)" check "[Cast in place](#)" in the "BEAM DATA" window.



Pressing the button will have the following window:



ENTER DATA (HEIGHT, INF BASE, SUP BASE)

PROJECT: Confronto - BEAM NAME: Foro - JOB: casi

CAST IN PLACE: TRAPEZIUMS

0 cm 0 cm 0 cm

N.Trapezium	Height	Inferior base	Superior base
▶ 1	5	50	50

Save Delete

Close

PREVIEW

BACK TO PREVIOUS FORM

SAVE/DELETE

LIST OF TRAPEZIUMS (NO LIMIT IN NUMBER)

The screenshot shows a software window titled 'CAST IN PLACE: TRAPEZIUMS'. At the top, there are three input fields for 'Height', 'Inferior base', and 'Superior base', each with a unit 'cm' and a red arrow pointing to a label 'ENTER DATA (HEIGHT, INF BASE, SUP BASE)'. Below these is a table with columns 'N.Trapezium', 'Height', 'Inferior base', and 'Superior base'. The first row shows '▶ 1', '5', '50', and '50'. Below the table are 'Save' and 'Delete' buttons, with a red arrow pointing to a label 'SAVE/DELETE'. At the bottom left is a 'Close' button, with a red arrow pointing to a label 'BACK TO PREVIOUS FORM'. On the right is a 'PREVIEW' area showing a trapezoid with dimensions 5, 50, and 50. A red arrow points from the table to the preview area, with a label 'LIST OF TRAPEZIUMS (NO LIMIT IN NUMBER)'.

The cast in place is inserted by trapezoid upon the beam: begin from the beam top edge flap and proceeds upward.

Enter in order Height, Lower Base and Upper base in the text boxes at the top, then press "[SAVE](#)" to save the trapeze. The trapeze is added to the bottom of the list.

To delete a trapezoid select it from the list and click the button "[Delete.](#)"

12. POINT LOADS

To enable the button "**LOADS**" check "Concentrate loads" in the window "BEAM DATA".

Concentrate loads ☒ **LOADS**

PROJECT: Confronto - BEAM NAME: Foro - JOB: casi

CONCENTRATED LOADS

VIEW INSERTED LOADS

Distance from beam start	Spacing	G1 fully def. perm. loads on beam only (CIP+others)	G1 fully def. perm. loads on beam and CIP	G2 perm. loads NOT def. on beam + CIP	Qk1 var. loads on beam and CIP	Qk2 var. loads on beam and CIP
m	m	kg	kg	kg	DOMINANT kg	kg
0	0	0	0	0	0	0

ENTER DATA

Dist from L end

	Distance from beam start	G1 fully def. perm. loads on beam only (CIP+others)	G1 fully def. perm. loads on beam and CIP	G2 perm. loads NOT def. on beam + CIP	Qk1 var. loads on beam and CIP	Qk2 var. loads on beam and CIP
▶ 1.2	2000	0	0	2000	0	
8	2000	0	0	2000	0	
11	3000	0	2000	2000	0	

LIST OF LOADS

STAMPA DI QUESTA SCHERMATA

Save Edit Delete Delete ALL

SAVE/DELETE

Close

BACK TO PREVIOUS FORM

In this window enter: the distance of a single load from the left head of the beam, the permanent and variable load, by the same logic of the line loads. Click "**SAVE**" to save data.

To edit a combination of loads: select it from the list, edit the data in the boxes and then click the button "**EDIT**".

Select a combination of loads from the list and then press "**DELETE**" to delete it.

"**DELETE ALL**" deletes all present combinations of loads.

The loads have different colours from each other, but equal in the text and in the graphic representation for a quick control.

12.1. SPACING

You can set a combination of loads with a certain step and the program automatically inserts a *repetition of all loads* from the distance of the left beam end.

13. MATERIALS

PROJECT: Confronto - BEAM NAME: Foro - JOB: casi

MATERIALS

Beam concrete Rck at 28 days	C35/45	450	Kg/cm ²	Concrete Rckj at initial time	C28/35	350	Kg/cm ²
Strands Stressing	14000		Kg/cm ²	Strands Area for I [^] sec prc	1.39		cm ²
Strands fptk	1860		N/mm ²	Braids fptk	1900		N/mm ²
Concrete in situ Rck at 28 days	C20/25	250	Kg/cm ²	Fyk steel	B450C		N/mm ²
Loss on jack Coefficient	1.5		%	Shrinkage Coeff. (25=0.00025)	25		
Fluage Loss Coefficient 1.0	2		1.8 : 3	Loss at 1000 h in the strands	2.5		%
Short or long duration Variable	0.4		0.4 - 0.6	Maturity days at init. time	9		
Coeff Sic CLS γ_C	1.5		1.4 - 1.5	Storage days	15		
Cement class	N						
Cable Stressing	15000		Kg/cm ²				

☒ ROUGH surface beam and CIP c = 0.40 $\mu = 0.7$ EC2 6.2.5 (2)

☐ SMOOTH surface beam and CIP c = 0.20 $\mu = 0.6$ EC2 6.2.5 (2)

DEFAULT MATERIALS

for allowable stresses code

Concrete Ecm at initial time	336749	Kg/cm ²
Concrete Ecm at 28 days	381837	Kg/cm ²
Concrete in situ Ecm	284604	Kg/cm ²

for T.U. 2008 - EC2 calculation

Concrete Ecm at initial time	32.59	kN/mm ²
Concrete Ecm at 28 days	34.63	kN/mm ²
Concrete in situ Ecm	30.2	kN/mm ²

Close

Save

Yellow boxes are automatically calculated by the program but user can edit them.

Beam concrete Rck at 28 days and at initial time: As from Rules. You can enter any value in the box.

Strands Stressing and Strands Area for I[^] pres. Sec.: As from Rules. Strands area is used also to calculate then length of prestressing diffusion, to define the distance of the first prestressed section from the head.

Strands fptk (acciaio armonico) and **Fptk braids:** As from Rules.

Loss on jack Coefficient: At the time of stressing there is always a loss. We suggest you to set this value from 1.5 to 3 according to the tools of your factory.

Shrinkage Coeff: For Italian calculation Tensioni Ammissibili. We suggest you to set 25, as from experience in steam curing.

Fluage Loss Coefficient: For Italian calculation Tensioni Ammissibili. We suggest you to set 2, allowable in steam curing.

Short or long duration Variable: it can be 0.4 for long duration loads (recommended) or 0.6 for short duration loads; used to calculate crack width.

Loss at 1000 h in the strands: default is 2.5% for low relaxation strands, used to calculate losses for Relaxation of strands.

Conc. sic. coeff: can be equal to 1.4 for production under continuous monitoring, or 1.5 that is default, it is used in tests at the ultimate limit state. For Italian checks the program uses 1.6.

Maturity days at init. time: it proposes a number of equivalent days according to Rckj, this value is used to calculate the losses for shrinkage and relaxation to checks at demould and transportation.

Cement class (R/N/S): used to calculate the losses due to shrinkage and creep.

Storage days: by default it is set equal to 15 days; It is used to make to transport checks with the 50% of losses occurred.

Ec of concrete are generated automatically with the formula according to the Rules, and can then be changed by the user.

Ec concrete cast in place, at initial time and 28days: user can change the theoretical data with those derived from the tests.

<input checked="" type="radio"/>	ROUGH surface beam and CIP	c = 0.40	$\mu = 0.7$	EC2 6.2.5 (2)
<input type="radio"/>	SMOOTH surface beam and CIP	c = 0.20	$\mu = 0.6$	EC2 6.2.5 (2)

Surface between the beam and cast in place: there are two buttons for the calculation of the projecting stirrups from the beam which serve to connect it to the casting.

This calculation is performed according to EC2; you can choose between a ROUGH and SMOOTH surface. The choice of the latter involves an increase in the projecting stirrups area.

The "VERY SMOOTH" surface option was ignored.

13.1. DEFAULT MATERIALS

The data are as in Material window, but these do not refer to a specific project: are the materials that will be used by default for new projects. The program, every time you make a new project, will set these data instead of null values, and then the user can change them whenever he wants. The default materials are saved in the text file "MDES.TXT" in the program directory.

Eiseko computers

DEFAULT MATERIALS for new projects

Beam concrete Rck at 28 days	C40/50	500	Kg/cm ²	Concrete Rckj at initial time	C28/35	350	Kg/cm ²
Strands Stressing	14000		Kg/cm ²	Strands Area	0.93		cm ²
Strands fptk	1860		N/mm ²	Strands fptk	1900		N/mm ²
Concrete in situ Rck at 28 days	C20/25	250	Kg/cm ²	Fyk Steel	B450C		N/mm ²
Loss on jack Coefficient	1.5		%	Shrinkage Coeff. (25=0.00025)	25		
Fluage Loss Coefficient 1.0	2		1.8 - 3	Coeff. SLU var $\gamma_{Qk1} \gamma_{Qk2}$	1.5		
Cable Stressing	14000		Kg/cm ²	Coeff. SLU perm γ_{G1}	1.3		
Coeff. Sic CL5 γ_C	1.4		1.4 - 1.5	Coeff. SLU perm γ_{G2}	1.5		
Short or long duration Variable	0.4		0.6 - 0.4	Loss at 1000 h in the strands	2.5		%
% air vapour during maturity	60		%	Maturity days at initial time	5		
Exposure classes	XC3						
Cement class (R/N/S)	N			Storage days	15		

for allowable stresses code			for T.U. 2008 - EC2 calculation		
Concrete Ecm at initial time	336749	Kg/cm ²	Concrete Ecm at initial time	32.59	kN/mm ²
Concrete Ecm at 28 days	402492	Kg/cm ²	Concrete Ecm at 28 days	35.54	kN/mm ²
Concrete in situ Ecm	284604	Kg/cm ²	Concrete in situ Ecm	38.2	kN/mm ²

14. COORDINATES

It shows then geometric data of the beam: section area, volume, centre of gravity coordinates, weight, table of coordinates of points of the two sections, graphical view of the section with the points.

PROJECT: a1 - BEAM NAME: esempio - JOB: Prove

BEAM COORDINATES

MIDSPAN SECTION

N	X (cm)	Y (cm)
1	15	0
2	15	13
3	23	21
4	23	74
5	15	79
6	0	82
7	0	90
8	60	90
9	60	82
10	45	79
11	37	74
12	37	21
13	45	13
14	45	0
15	15	0

HEAD SECTION

N	X (cm)	Y (cm)
1	15	0
2	15	79
3	0	82
4	0	90
5	60	90
6	60	82
7	45	79
8	45	0
9	15	0

GEOMETRICAL DATA

Section Area	2,033.00	cm ²
Volume	0.2	m ³ /m
Weight	508	kg/m
X center of mass (G)	30.00	cm
Y center of mass (G)	49.83	cm

GRAPHIC DISPLAY OF SECTION WITH NUMBER OF POINTS, MAIN DIMENSIONS AND CAST IN PLACE

VIEW SECTION + coord. View coord. table Export sect. to DXF

Close

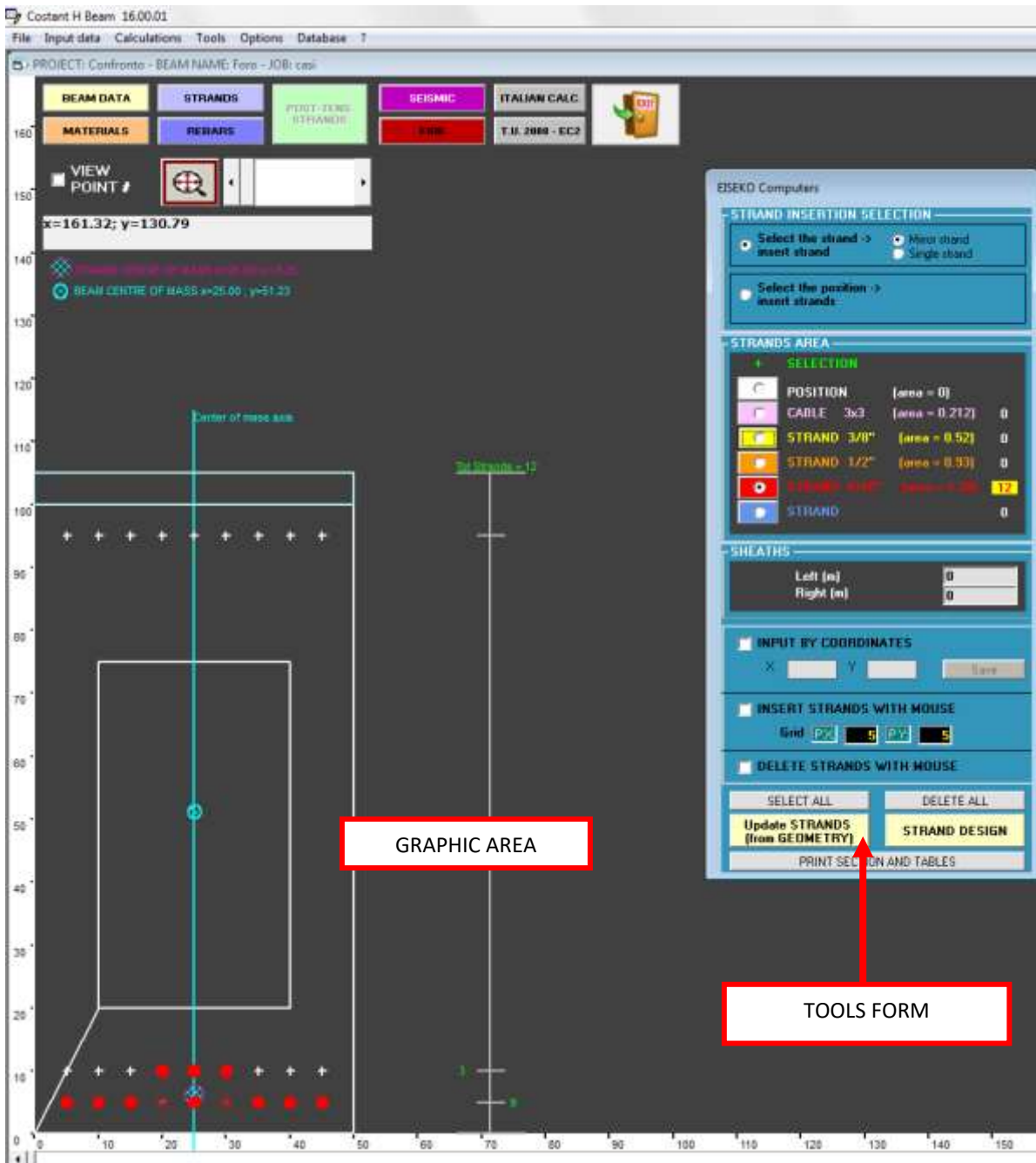
EXIT

VIEW SECTION + COORDINATES: open the printable reports with the display of the section with the points, the CIP, the strands and the rebars.

VIEW COORDINATES table: open the printable report with tables of coordinates of the points of the two sections.

Export sect. to DXF: Export the sections with and without the rebars/strands into a DXF file.

15. STRANDS



Pressing the "**STRAND**" button, you switch to the window to edit strands. This window consists of a graphics area and a working tool window. the graphic area shows the section with the strands: for new projects the program loads the strands mask saved with Geometry module. Then it can be changed for the current project.

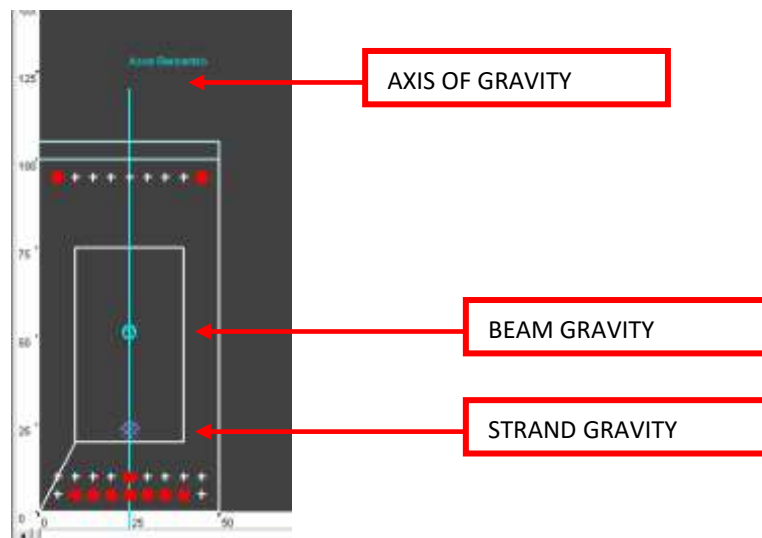
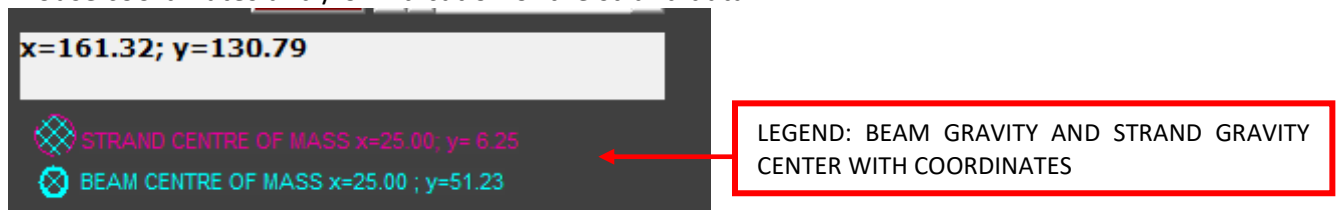
To put on / take off of the strands: select the area of the strand from the tool window (strands area) and click in the position in which you want to put the strand (the white "+"). If the strand is not there, it will be inserted, with the chosen area. If the strand is already there: if it has the same area, it is deleted, if it has a different area, the area is modified into the chosen one.

15.1. Graphic area

Buttons to access directly to the various windows:



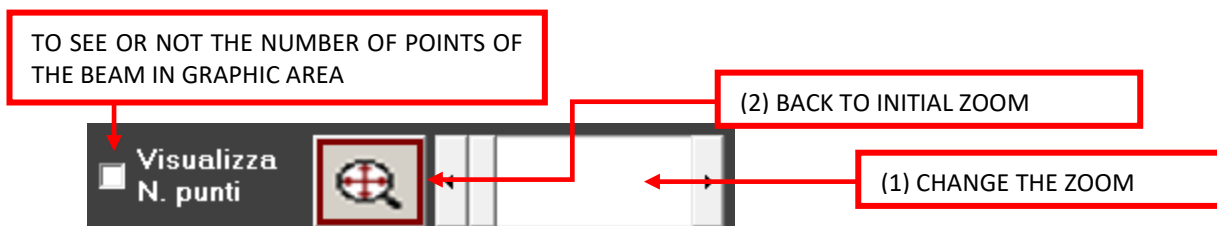
Mouse coordinates and / or indication of the strand data.



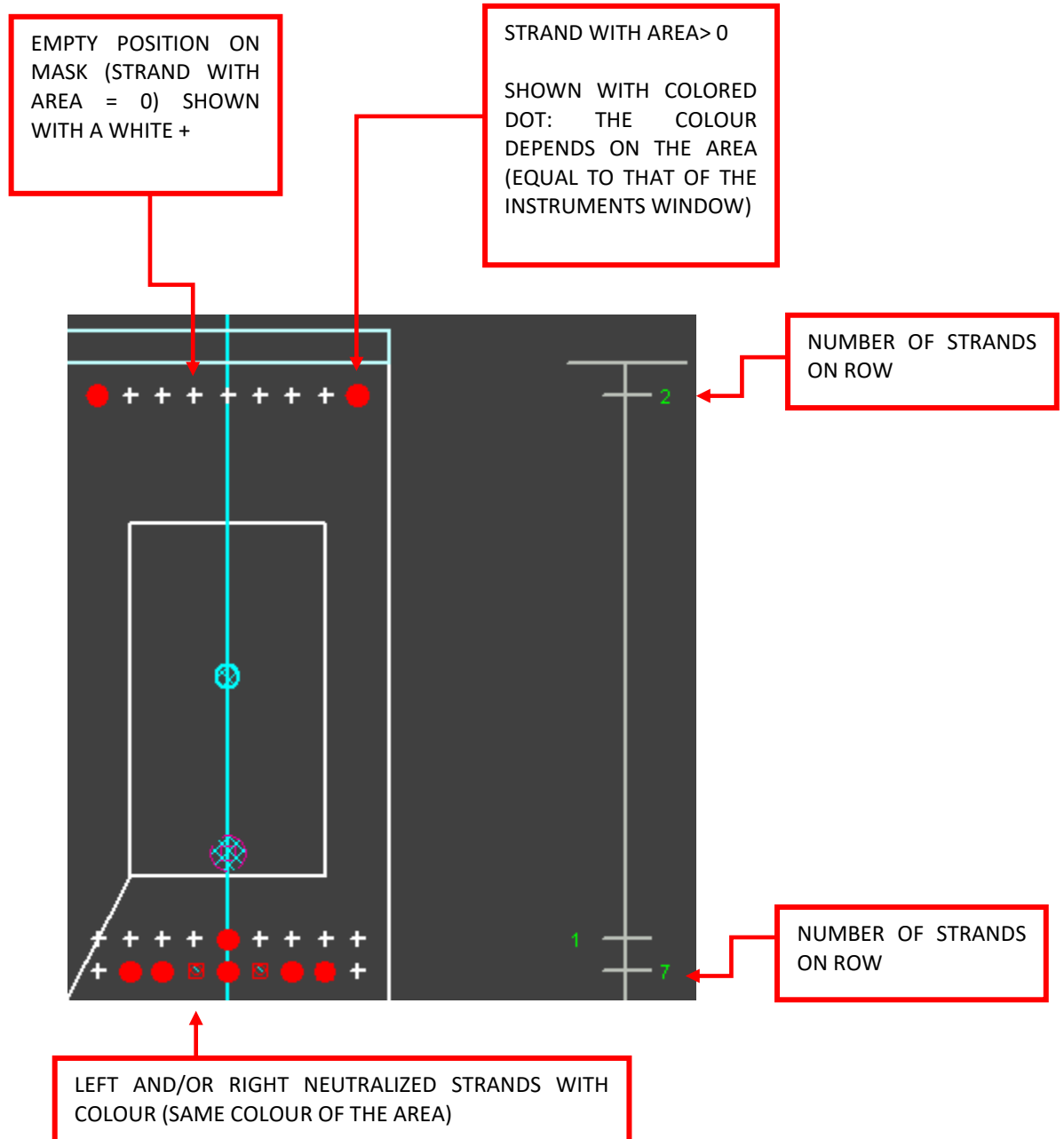
15.1.1. ZOOM

If the image is too small or too large, you can change the zoom:

1. Turn the mouse wheel
2. By moving the scroll bar (1)
3. By changing the scroll bars at the bottom and right of graphic area (the area is shifted in correspondence)
4. By pressing the button (2): returns to the initial zoom.



15.1.2. STRAND REPRESENTATION IN GRAPHIC AREA



15.2. TOOLS WINDOW

EISEKO Computers

☒ Select the strand ->
insert strand

☒ Mirror strand
☐ Single strand

☐ Select the position ->
insert strands

+

SELECTION

☐ POSITION (area = 0)

☐ CABLE 3x3 (area = 0.212) 0

☒ STRAND 3/8" (area = 0.52) 0

☐ STRAND 1/2" (area = 0.93) 0

☐ STRAND 6/10" (area = 1.39) 12

☐ STRAND 0

SHEATHS

Left (m) 0

Right (m) 0

☐ INPUT BY COORDINATES

X

Y

Save

☐ INSERT STRANDS WITH MOUSE

Grid

PX

5

PY

5

☐ DELETE STRANDS WITH MOUSE

SELECT ALL

DELETE ALL

Update STRANDS
(from GEOMETRY)

STRAND DESIGN

PRINT SECTION AND TABLES

“SELECT STRAND”: When *“Mirror strand”* is selected, the operation the user will select will be done directly on the selected strand and on the mirrored position respect to gravity (operations like insert, delete, change area, neutralize, etc.), if mirrored position is present; when *“Single strand”* is selected, operation will be done only on the selected position.

“ SELECT POSITION”: To select multiple strands just click a first strand, hold down the left mouse button and move the mouse to pass on the other strands that you want to select (will be highlighted in green). The selected operation will be performed directly on all selected strands.

“STRAND AREA”: Choose the area of the strand to insert. On the right the number of strands of that type inserted in the beam is shown.

“SHEATS”: To neutralize one strand, first insert the length of the neutralization in meters (starting from the corresponding end) in the box, then click in the graphic area on the strand to be neutralized.

To delete a sheat put lengths = 0 m and click on the strand.

The strand neutralized appears with a square of the colour of the corresponding area.

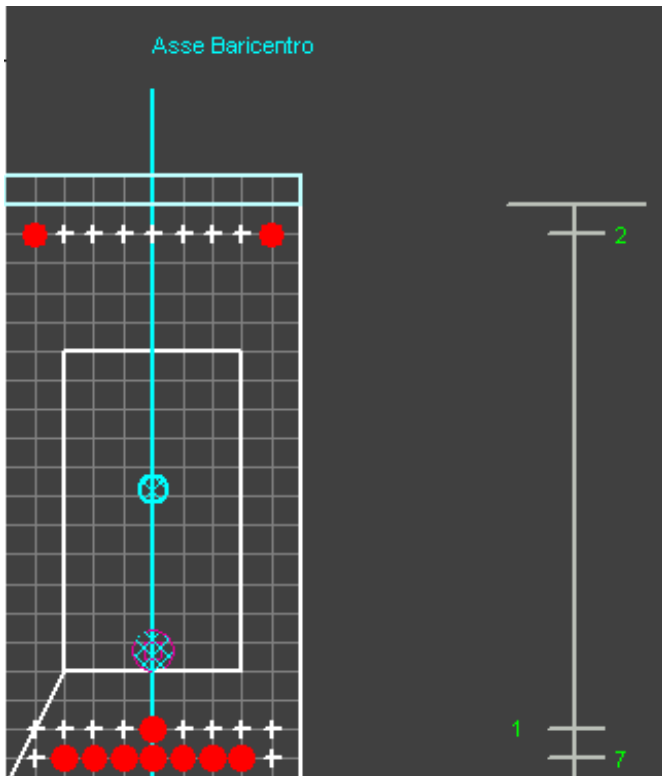
“INPUT BY COORDINATES”: To enter NEW POSITIONS of strands select this option, enter the new coordinates, select the correct area of

the strand, and press **“SAVE”**.

“INSERT NEW STRAND WITH MOUSE” To insert NEW positions of strands select this option and type the step in X and Y of the grid: the program will insert a grid as in the following figure.

Then select the correct area of the strand and click on the grid nodes to enter the new position. At the top it is possible to see the mouse coordinates.

The new positions will only be available in the current project and have no effect on the beam created with the program *“GEOMETRY”*.



“DELETE STRAND WITH THE MOUSE”: To delete the **positions** of strands (to remove the area just choose "POSITION" in the strands areas and click on it) select this option and click on the strand to delete the graphic area. If "Mirror Strand" at the top is selected, the mirrored strand is deleted too, if any.

“SELECT ALL”: To select all of the strands positions (switches automatically to the option "Select position"). Selecting an area of the strand, this will be assigned to all the positions.

“DELETE ALL”: To delete all AREAS of strands (positions remain).

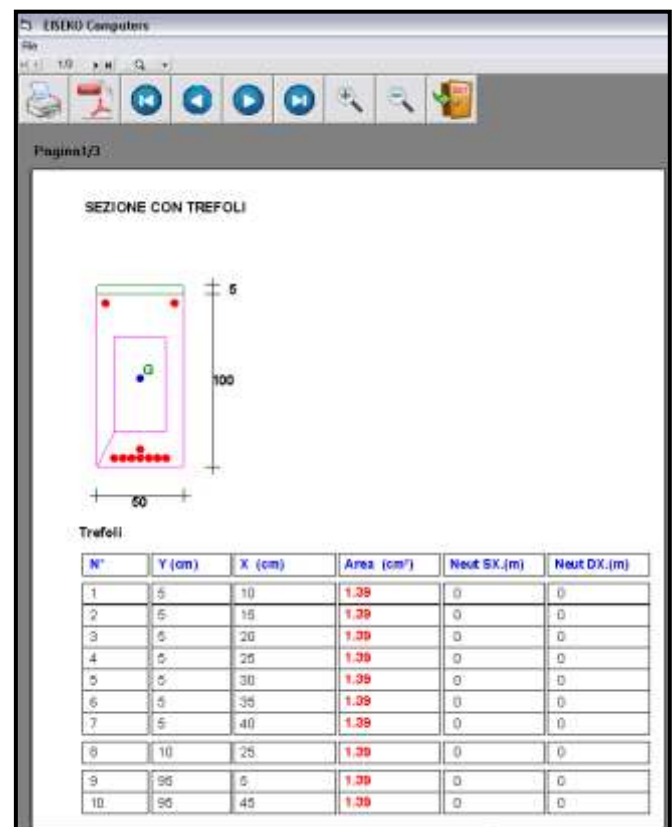
“UPDATE STRANDS (FROM GEOMETRY)” to reset the positions of strands: the strands will be restored from the beam as saved by the "GEOMETRY" program, both for the positions and areas.

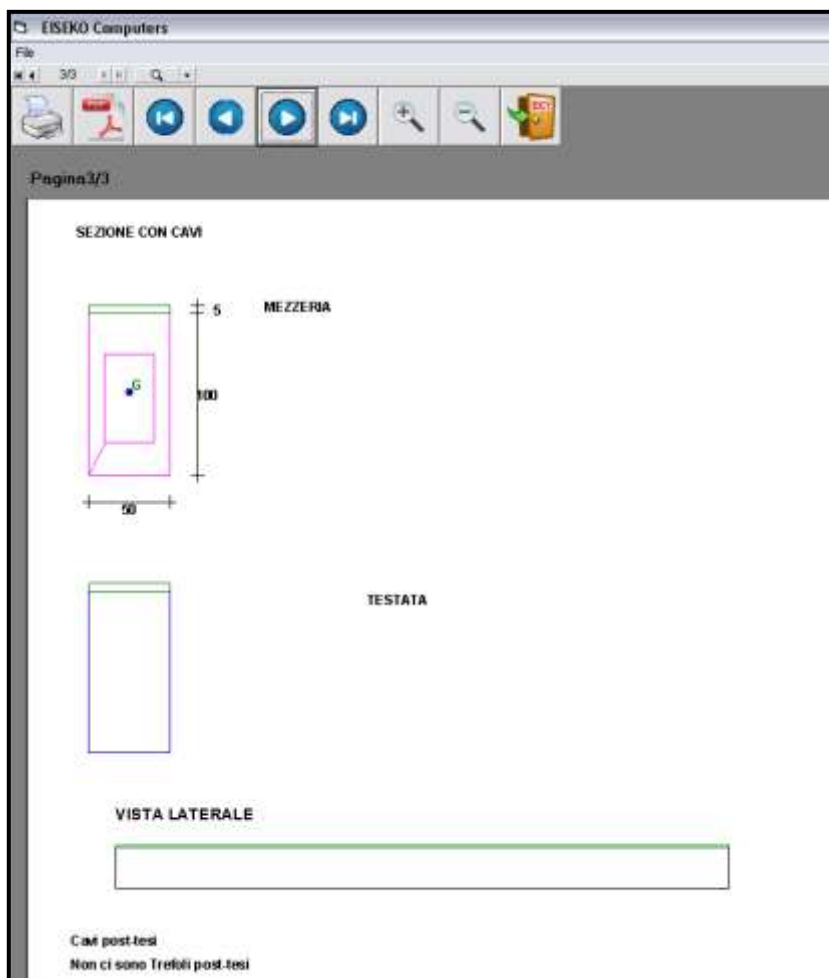
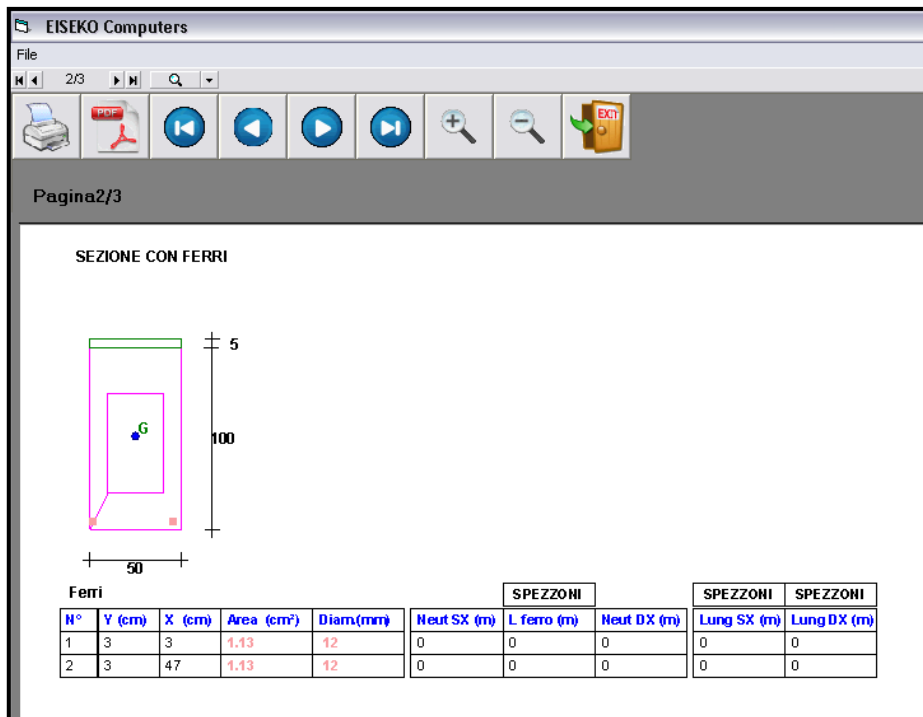
“DESIGNS STRAND” The program automatically designs the minimum required strands as required for break verification (this button is also found in "BEAM DATA").

If changes are made on the strands with the program "GEOMETRY", you must click the button "UPDATE BEAM" before loading the project (project manager in the mask) to see the changes here.

“PRINT SECTION AND STRAND TABLE”: to view a report with three pages, the first is the section with strands (pre-tensioned) and the strands table (pre-tensioned), in the second there is a section with rebars and table of rebars, in the third there is the section with post-tensioned cables and cable table (if any).

The sheats are always indicated, even if they are actually present only in the ends or only on the centre line.





The post-tensed cables will be treated in detail in a separate chapter.

16. REBARS


Costant H Beam 16.00.01

File Input data Calculations Tools Options Database ?

PROJECT: Confronto - BEAM NAME: Foro - JOB: casi

BEAM DATA STRANDS POST-TENS. STRANDS SEISMIC ITALIAN CALC. EXIT

MATERIALS REBARS FIRE T.U. 2008 - EC2

VIEW POINT # 

x=162.12; y=155.92

BEAM CENTRE OF MASS x=25.00 ; y=51.23

Center of mass axis

Tot Rebars = 6

2

GRAPHIC AREA

EISEKO Computers

REBAR INSERTION SELECTION

☒ Select the rebar -> insert rebar ☐ Mirror rebar

☐ Select the position -> insert rebars ☐ Single rebar

REBAR DIAMETER

ϕ	n	ϕ	n	ϕ	n
0	4	16	2	28	0
5	0	18	0	30	0
6	0	20	0	32	0
8	2	22	0	34	0
10	0	24	0	36	0
12	2	25	0	38	0
14	0	26	0	40	0

SHEATHS

☐ 0 0 0

☐ 0 0 0

INPUT BY COORDINATES

X Save Edit

Y

INSERT REBAR WITH MOUSE

Grid PX PY

DELETE REBAR WITH MOUSE

SELECT ALL DELETE ALL

Update REBARS (from GEOMETRY)

PRINT SECTION AND TABLES

TOOLS FORM

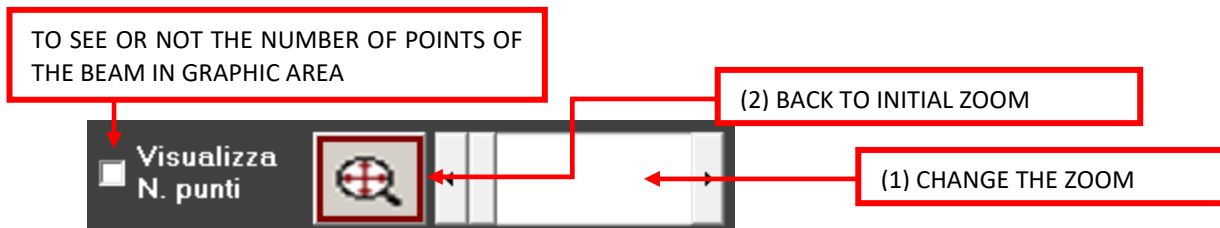
Buttons to access directly to the various windows:



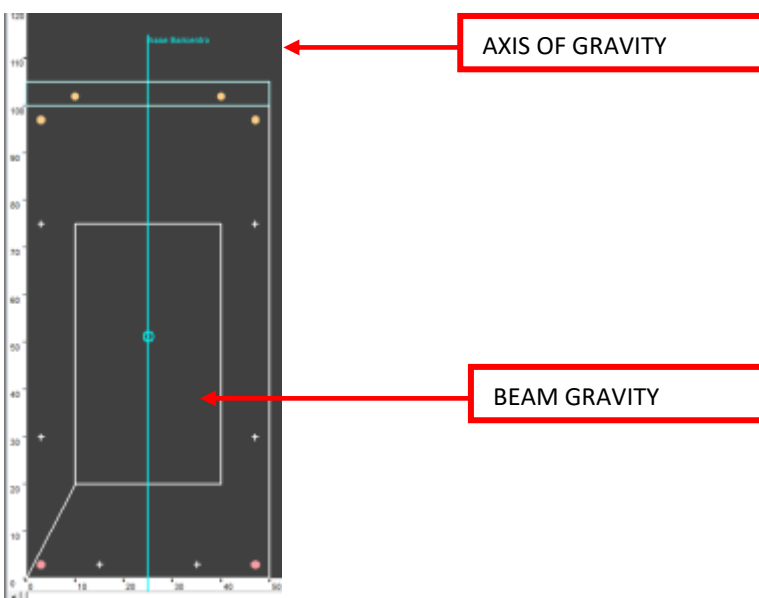
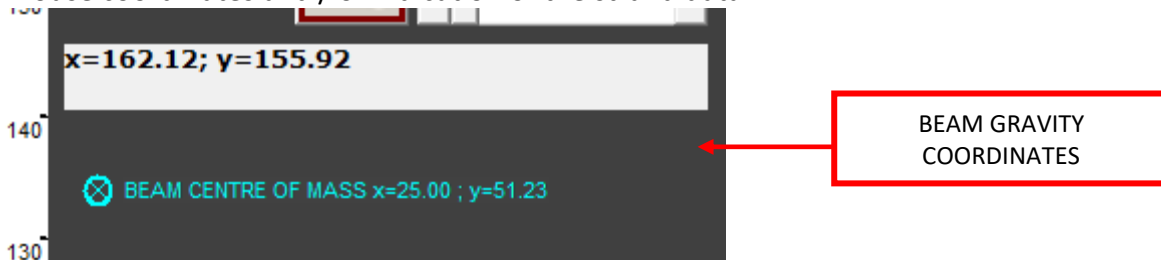
16.1. ZOOM AND GRAPHIC

If the image is too small or too large, you can change the zoom:

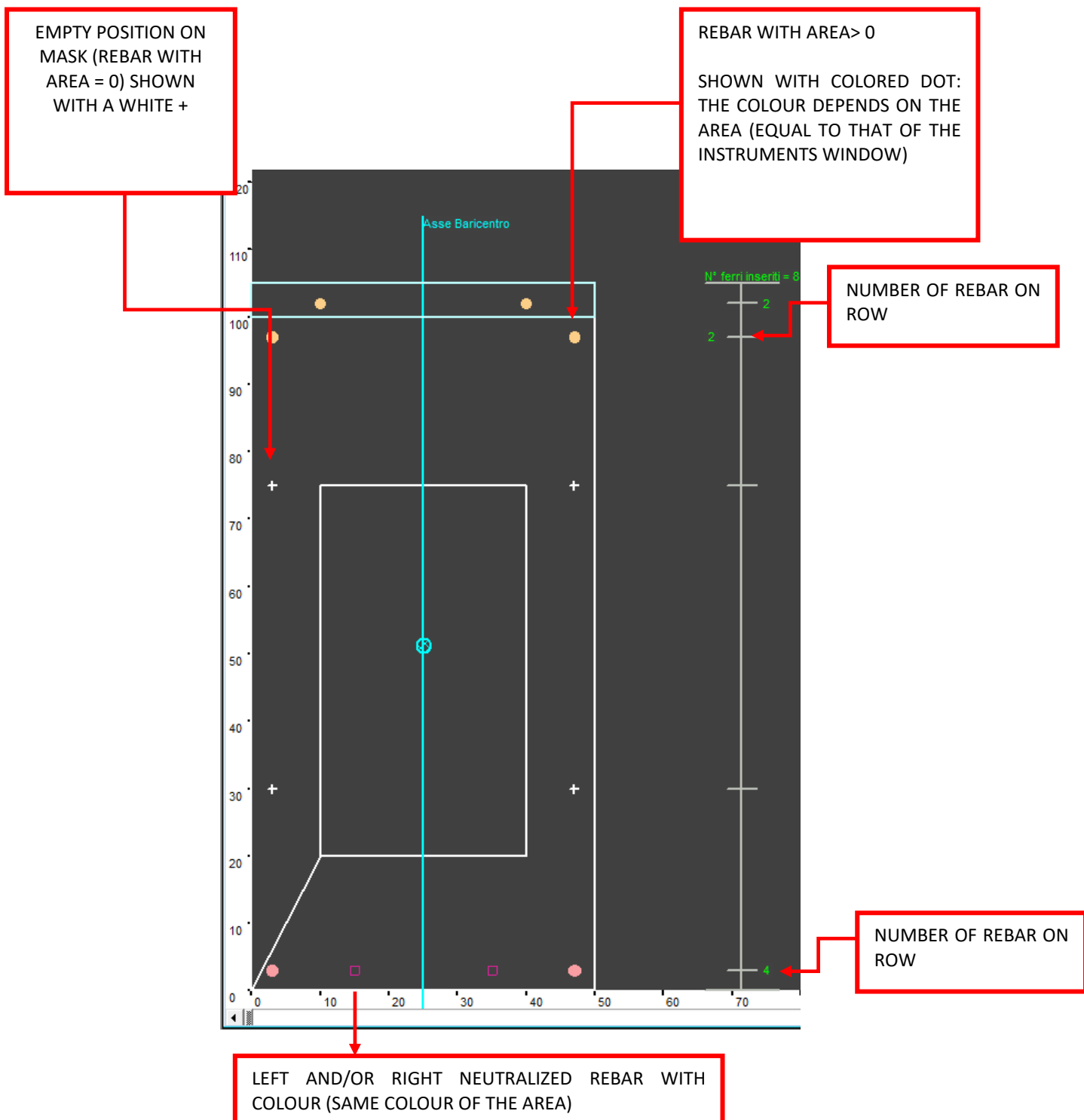
1. Turn the mouse wheel
2. By moving the scroll bar (1)
3. By changing the scroll bars at the bottom and right of graphic area (the area is shifted in correspondence)
4. By pressing the button (2): returns to the initial zoom.



Mouse coordinates and / or indication of the strand data



16.1. REBAR REPRESENTATION IN GRAPHIC AREA



16.1. TOOLS WINDOW

EISEKO Computers

☒ Select the strand -> insert strand

☒ Mirror strand
☐ Single strand

☐ Select the position -> insert strands

+

SELECTION

<input type="radio"/>	POSITION	[area = 0]	0
<input type="radio"/>	CABLE 3x3	[area = 0.212]	0
<input type="radio"/>	STRAND 3/8"	[area = 0.52]	0
<input type="radio"/>	STRAND 1/2"	[area = 0.93]	0
<input checked="" type="radio"/>	STRAND 5/16"	[area = 1.39]	2
<input type="radio"/>	STRAND		0

SHEATHS

Left (m)

0

Right (m)

0

☐ INPUT BY COORDINATES

X

Y

Save

☐ INSERT STRANDS WITH MOUSE

Grid

PX

5

PY

5

☐ DELETE STRANDS WITH MOUSE

SELECT ALL

DELETE ALL

Update STRANDS (from GEOMETRY)

STRAND DESIGN

PRINT SECTION AND TABLES

“SELECT REBAR”: When “Mirror rebar” is selected, the operation the user will select will be done directly on the selected rebar and on the mirrored position respect to gravity (operations like insert, delete, change area, neutralize, etc.), if mirrored position is present; when “Single rebar” is selected, operation will be done only on the selected position.

“SELECT POSITION”: To select multiple rebars just click a first rebar, hold down the *SHIFT* button and click on the other rebars that you want to select (will be highlighted in green). The selected operation will be performed directly on all selected rebars.

“REBAR DIAMETER”: Choose the area of the rebar to insert. On the right the number of rebars of that type inserted in the beam is shown.

“INPUT BY COORDINATES”: To enter NEW POSITIONS of rebars select this option, enter the new coordinates, select the correct area of the rebar, and press “SAVE”.

To edit an existing rebar, click on the rebar (its coordinates will be automatically inserted in boxes X, Y), change the coordinates and press “EDIT”.

“INSERT NEW REBARS WITH MOUSE” To insert NEW positions of rebars select this option and type the step in X and Y of the grid: the program will insert a grid as in the following figure.

Then select the correct diameter of the rebar and click on the grid nodes to enter the new position. At the top it is possible to see the mouse coordinates. The new positions will only be available in the current project and have no effect on the beam created with the program “GEOMETRY”.

NOTE: you can select a rebar and edit the data or select more than one to change them all at once: multiple selection is made by selecting the various rebars by holding down the “Shift” on your keyboard (allowed only if you click Select Position on top of Tools window).

“DELETE REBARS WITH THE MOUSE”: To delete BOTH the positions of rebars and the rebars (to remove only the area just choose “POSITION” button, while “0” in areas) select this option and click on the

rebar to delete in the graphic area. If "Mirror Rebar" at the top is selected, the mirrored rebar is deleted too, if any.

"SELECT ALL": To select all of the rebars positions (switches automatically to the option "Select position"). Selecting a diameter of the rebar, this will be assigned to all the positions.

"DELETE ALL": To delete all AREAS of rebars (positions remain).

"UPDATE REBARS (FROM GEOMETRY)" to reset the positions of rebars: the rebars will be restored from the beam as saved by the "GEOMETRY" program, both for the positions and areas.

If changes are made on the rebars with the program "GEOMETRY", you must click the button "UPDATE BEAM" before loading the project (project manager in the mask) to see the changes here.

"SHEATS": To enter just a piece of rebar (default has the same length of the beam, and considers automatically the concrete cover). Enter first the rebar, than change it into a piece:

Pieces can be on the ends of the beam or at midspan.

SHEATHS ON ENDS select upper option, enter the length of sheat, left and right (must enter both) and click on the desired rebar. The program considers these sheaths from the corresponding end.

SHEATHS ON MIDSPAN select lower option, enter the distance of the sheath from left end and right end respectively (must enter both and click on the desired rebar.

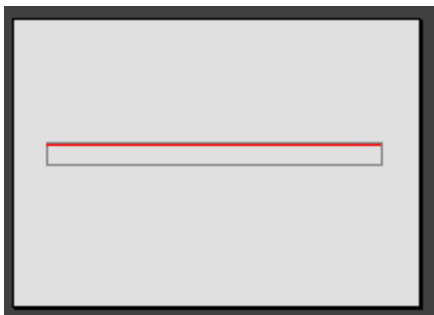
To insert only 1 sheath on 1 end, enter a midspan sheath near the desired end.

To create sheaths, select the option that best suits rebar to insert, write the lengths in the corresponding boxes and click on the rebar in the graphics area.

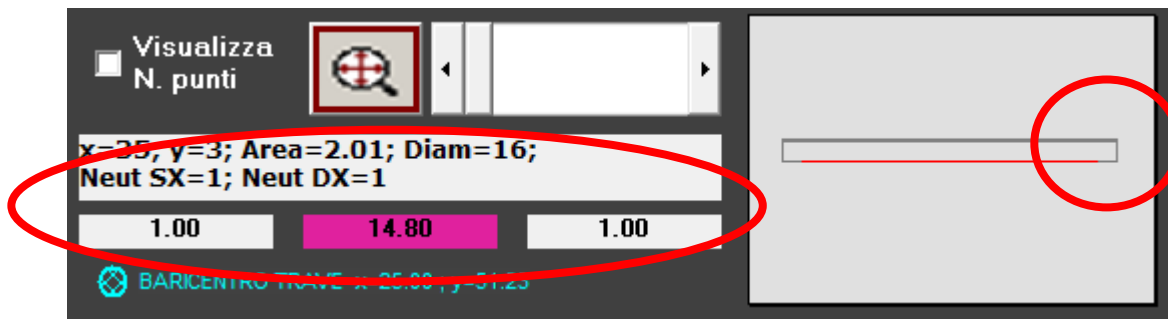
The length of the sheath to enter is the one used for the calculation, so you have to add bonded length to real length.

When you move your mouse on a rebar, this is plotted in red:

current rebar

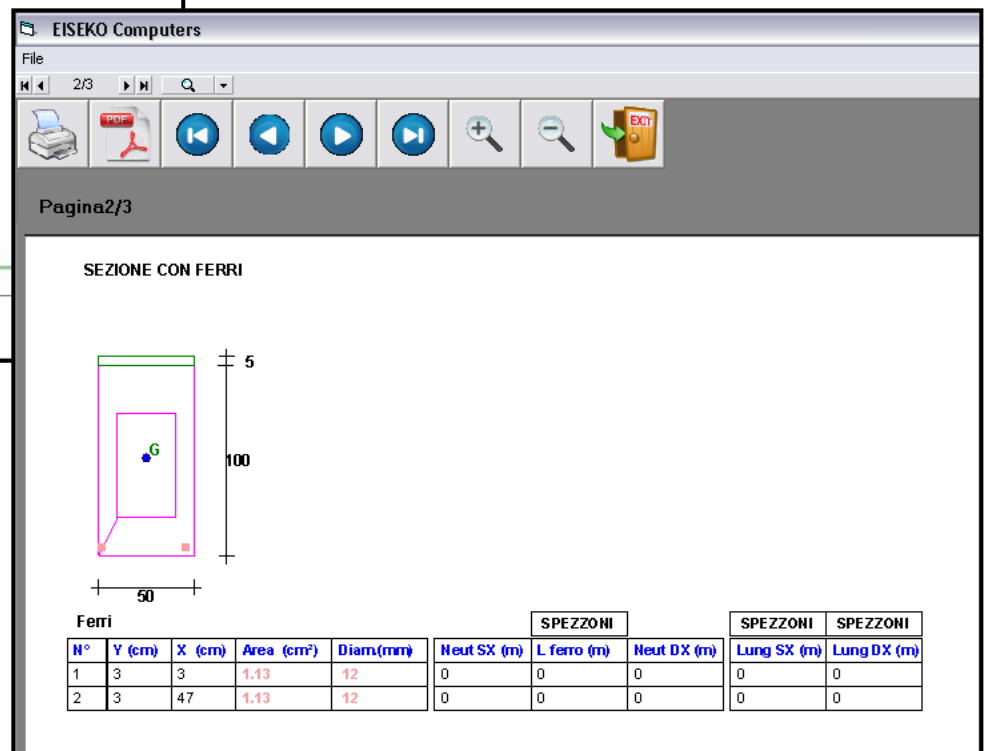
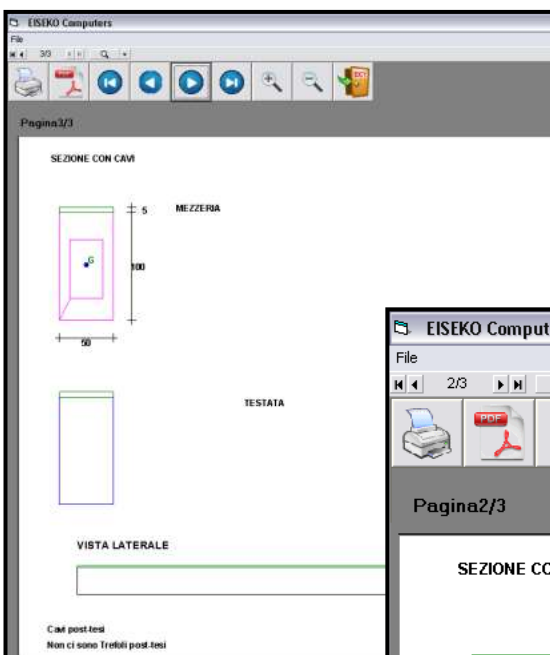
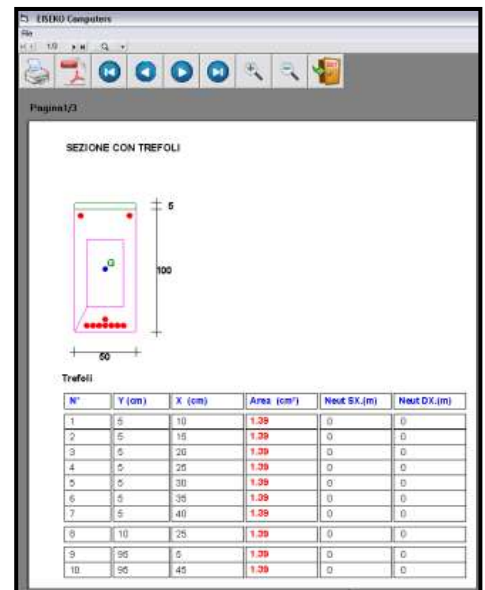


Rebar SHEATHS— in this case in addition to the coordinates of the rebar, diameter, area, are shown the measures of the segments too:



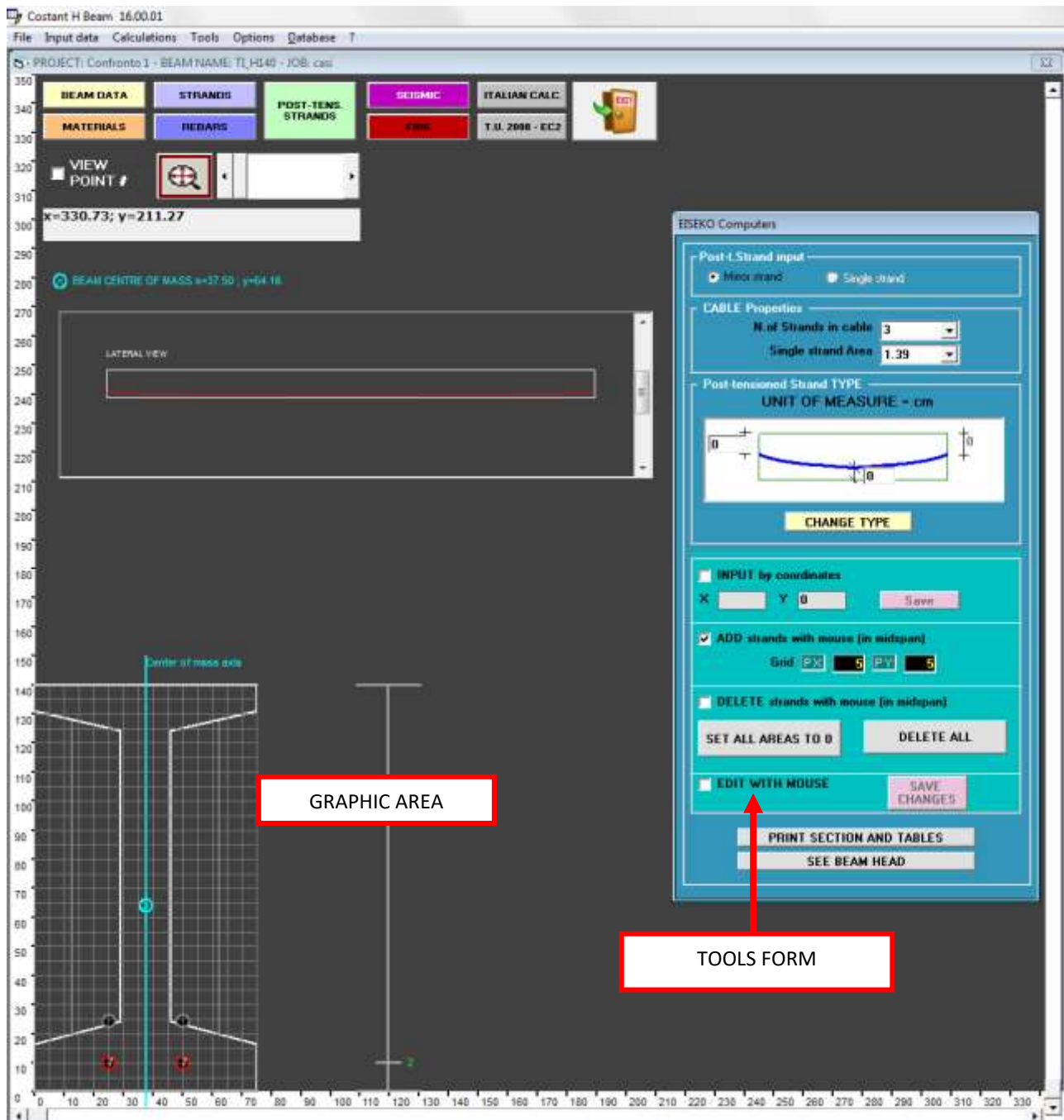
“ PRINT SECTION AND REBAR TABLE”: to view a report with three pages, the first is the section with strands (pre-tensioned) and the strands table (pre-tensioned), in the second there is a section with rebars and table of rebars, in the third there is the section with post-tensioned cables and cable table (if any).

The sheats are always indicated, even if they are actually present only in the ends or only on the centre line.



17. POST-TENSED STRANDS

NOTE: IF LONG OVERHANGS ARE PRESENT, IT IS POSSIBLE TO INSERT POST-TENSED STRANDS.



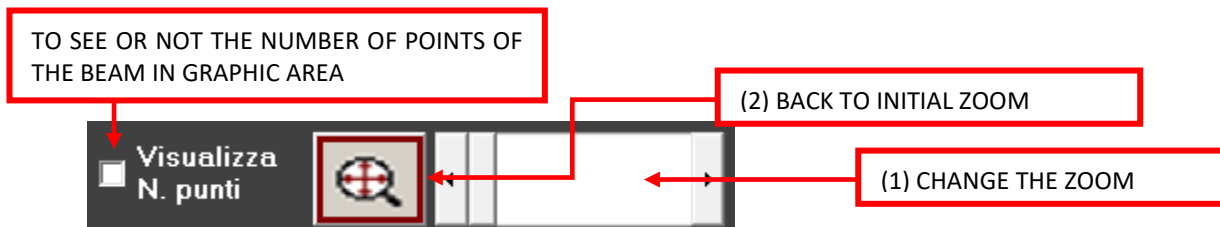
Buttons to access directly to the various windows:



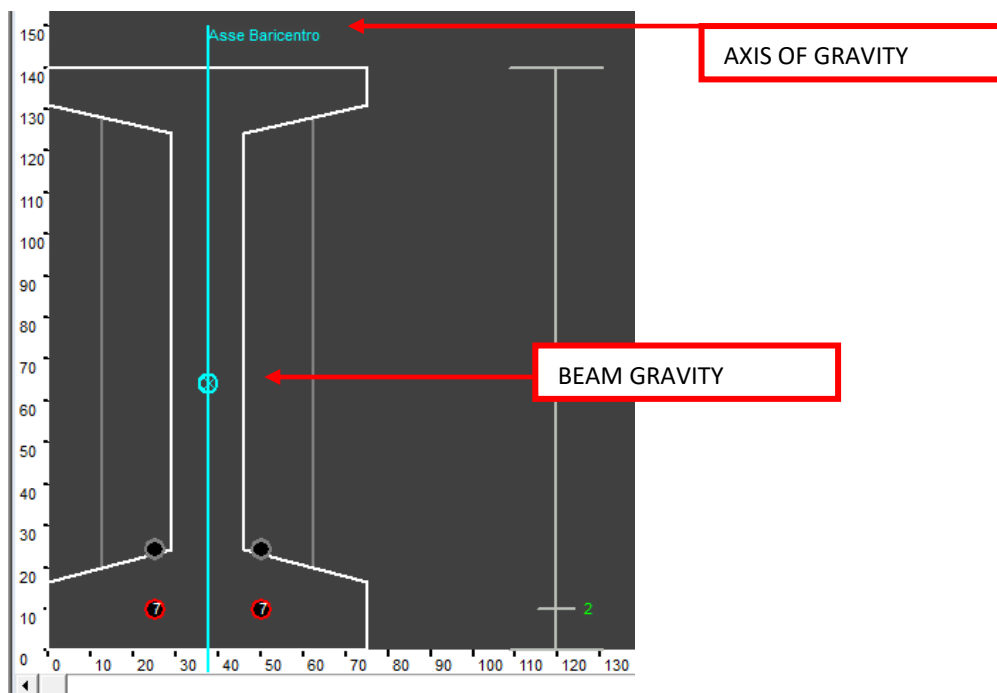
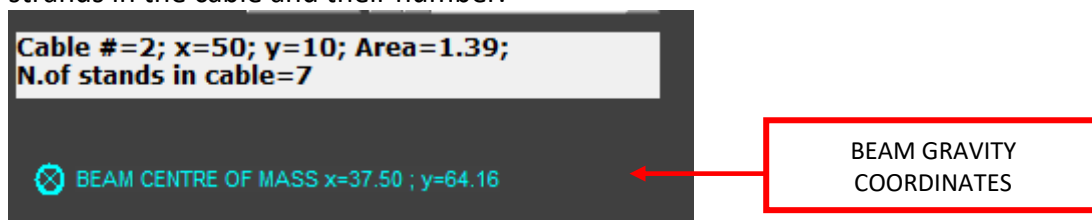
17.1. ZOOM AND GRAPHICS

If the image is too small or too large, you can change the zoom:

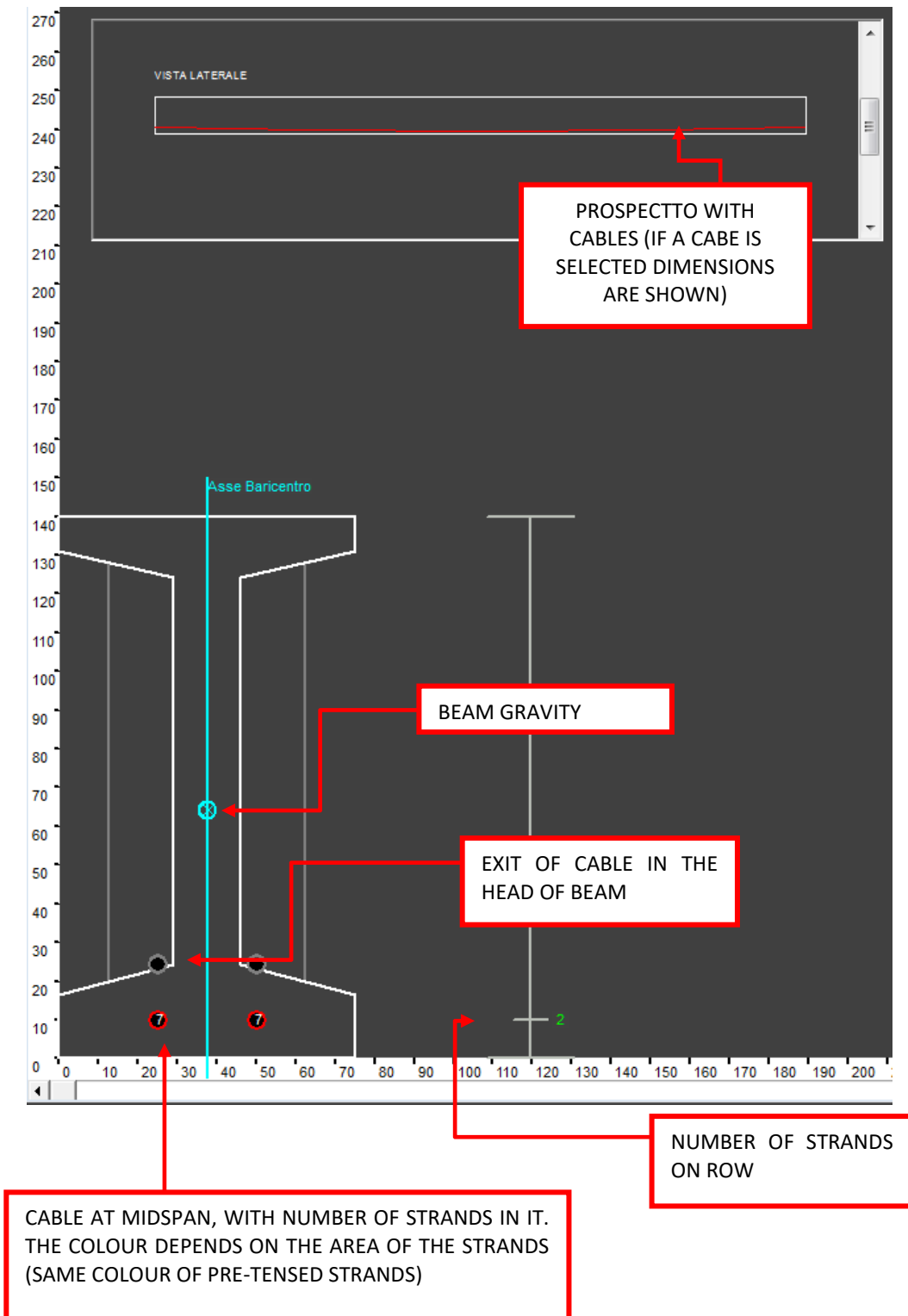
1. Turn the mouse wheel
2. By moving the scroll bar (1)
3. By changing the scroll bars at the bottom and right of graphic area (the area is shifted in correspondence)
4. By pressing the button (2): returns to the initial zoom.



Mouse coordinates and / or indication of the strand data: index, coordinates, area of single strands in the cable and their number.



17.2. GRAPHIC AREA



Before inserting a cable select from the tool window the **insertion type**: mirrored strands (the program will calculate and insert two identical cables, mirrored respect to centre of gravity axis) or single strand.

17.1. TOOLS WINDOW

The screenshot shows the 'Tools Window' interface for EISEKO Computers. The window is divided into several sections:

- Post-t. Strand input:** Contains two radio buttons: 'Mirror strand' (selected) and 'Single strand'.
- CABLE Properties:** Contains two dropdown menus: 'N. of Strands in cable' (set to 3) and 'Single strand Area' (set to 1.39).
- Post-tensioned Strand TYPE:** Contains a section titled 'UNIT OF MEASURE = cm' with a diagram of a cable profile. Below the diagram is a 'CHANGE TYPE' button.
- INPUT by coordinates:** Contains checkboxes for 'INPUT by coordinates' (unchecked), 'ADD strands with mouse (in midspan)' (checked), and 'DELETE strands with mouse (in midspan)' (unchecked). It also has input fields for 'X', 'Y' (set to 0), 'Grid PX' (set to 5), and 'PY' (set to 5), along with 'Save', 'SET ALL AREAS TO 0', and 'DELETE ALL' buttons.
- EDIT WITH MOUSE:** Contains a checkbox for 'EDIT WITH MOUSE' (unchecked) and a 'SAVE CHANGES' button.
- PRINT SECTION AND TABLES:** A button at the bottom.
- SEE BEAM HEAD:** A button at the bottom.

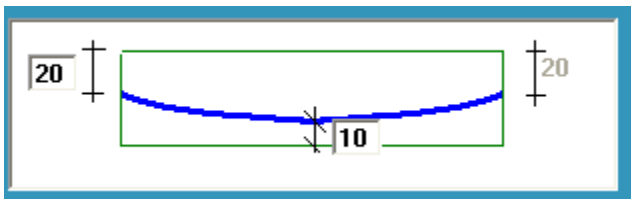
Annotations with red arrows point to specific features:

- INSERTION TYPE:** Points to the 'Mirror strand' radio button.
- PROPERTIES OF THE CABLE: NUMBER OF STRANDS AND AREA OF EACH STRAND IN THE CABLE TO INSERT:** Points to the 'N. of Strands in cable' dropdown.
- MEASURE IN CENTIMETERS TO POSITION THE CABLE - COORDINATE Y AT MIDSPAN IS NOT CONSIDERED IF USER INSERT THE CABLE WITH MOUSE:** Points to the 'UNIT OF MEASURE = cm' section.
- TO CHANGE TYPE OF CABLE TO INSERT:** Points to the 'CHANGE TYPE' button.

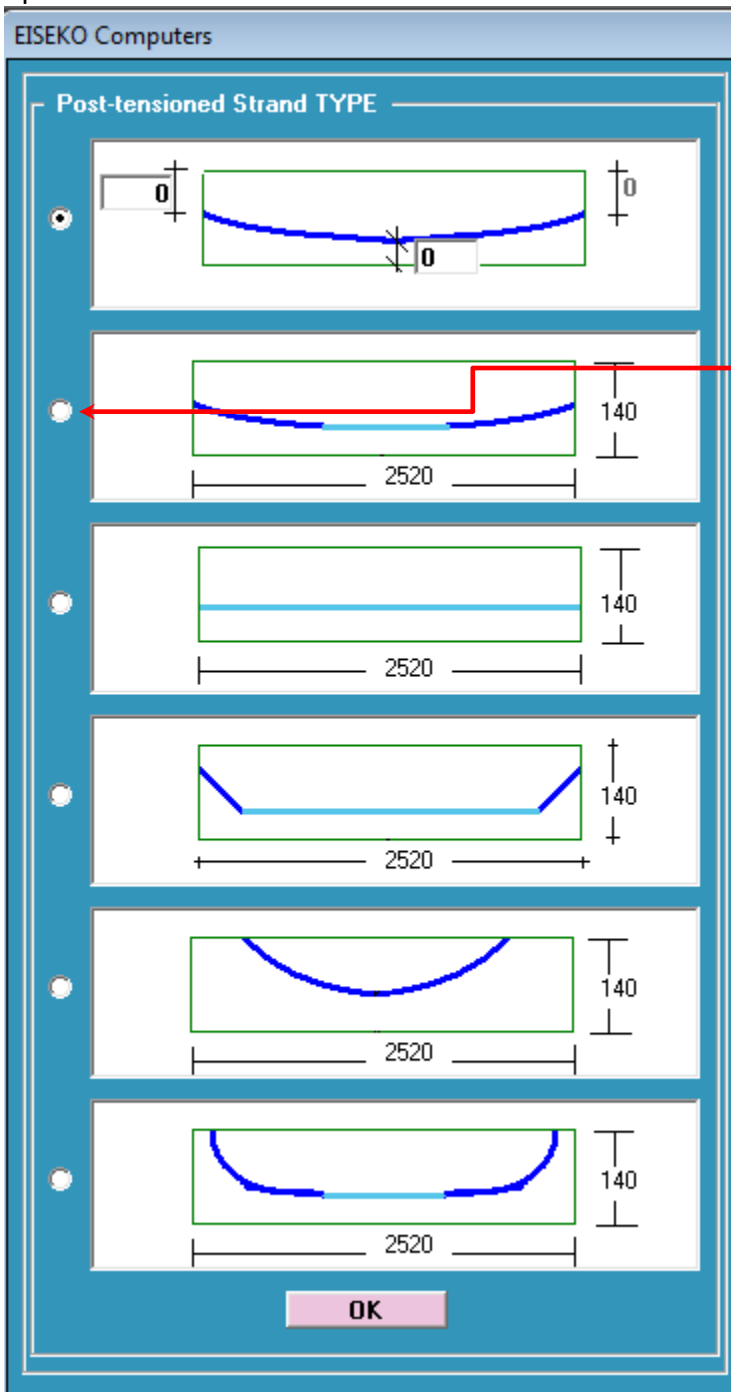
INSERTION TYPE: When “*Mirror strand*” is selected, the operation the user will select will be done directly on the selected strand and on the mirrored position respect to gravity (operations like insert, delete, change area, neutralize, etc.), if mirrored position is present; when “*Single strand*” is selected, operation will be done only on the selected position.

CABLE PROPERTIES: number of strands in the cable and area.

POST-TENSIONED strands Type: you can input various geometry of cables, straight, parabolic, with or without strait part in the middle and so on. As default we use the following:



To select another type of cable click the button “[CHANGE TYPE](#)” and the following window is opened:



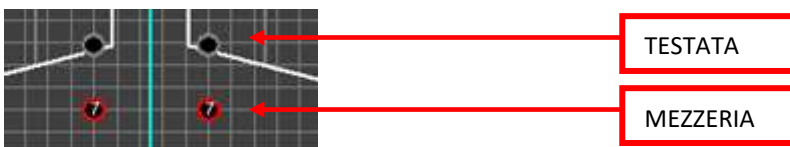
OPTION TO
SELECT THE
TYPE OF CABLE

To select the desired type click on the checkbox. They will enable all the dimensions needed to define the cable: it is possible to insert them directly or select "OK", return to the previous window and enter them there.

17.2. Insert/edit of cable:

As default the program lets you insert a cable with mouse click (option "[ADD STRANDS WITH MOUSE \(in midspan\)](#)" selected): just click in the graphical area the point for the strand position. **The cable dimension to be inserted is the one at midspan.** The grid helps to click in the correct position, to define the steps to change the values in the boxes PX, PY.

In the drawing area each cable is represented with two black dots: one represents the position in the head and the other the position at midspan. If coincident, you see only the midspan one.



"[INPUT by COORDINATES](#)" check this option, enter the X coordinate (the coordinate Y at midspan is already entered because taken from prospectus box) and click the button "[SAVE.](#)" Repeat for all cables.

"[DELETE strands WITH MOUSE \(in midspan\)](#)" check this option to delete individual existing cables: click with the mouse on the cable to be deleted (midspan one).

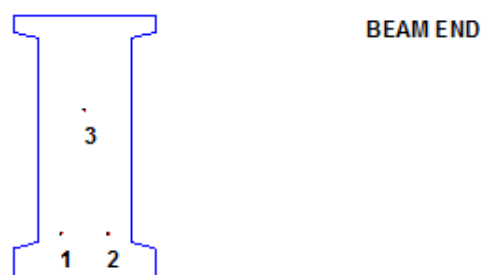
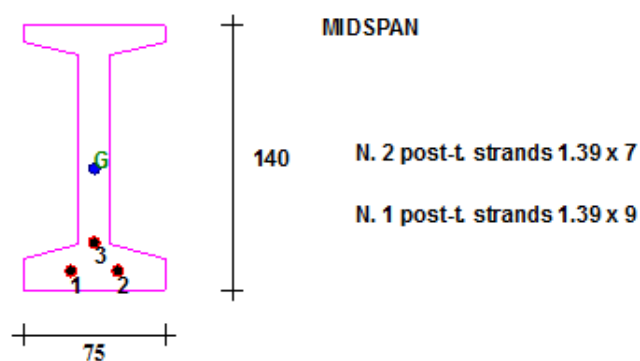
You can also cancel all the inserted cables (set their area = 0) by pressing the button "[SET ALL AREAS TO 0](#)", or delete all the inserted cables (and therefore eliminate positions too) by pressing the "[DELETE ALL](#)".

"[EDIT WITH MOUSE](#)" check this option to modify one or more existing cables: click with the mouse on the cable (or more than one to make the same change on all) to change the features to edit (number of strands in the cable and/or area of the single-strand and/or type of cable and/or dimension of each), then select "[SAVE CHANGES.](#)"

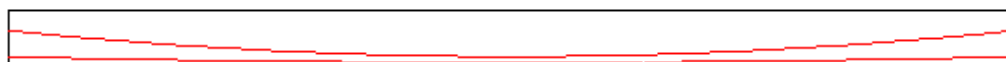
The following figure shows the insertion of the two lower parabolic cables with 9 strands each.

“PRINT SECTION AND TABLE”: to view a report with three pages, the first is the section with strands (pre-tensioned) and the strands table (pre-tensioned), in the second there is a section with rebars and table of rebars, in the third there is the section with post-tensioned cables and cable table.

SECTION WITH POST-TENSED CABLES



LATERAL VIEW



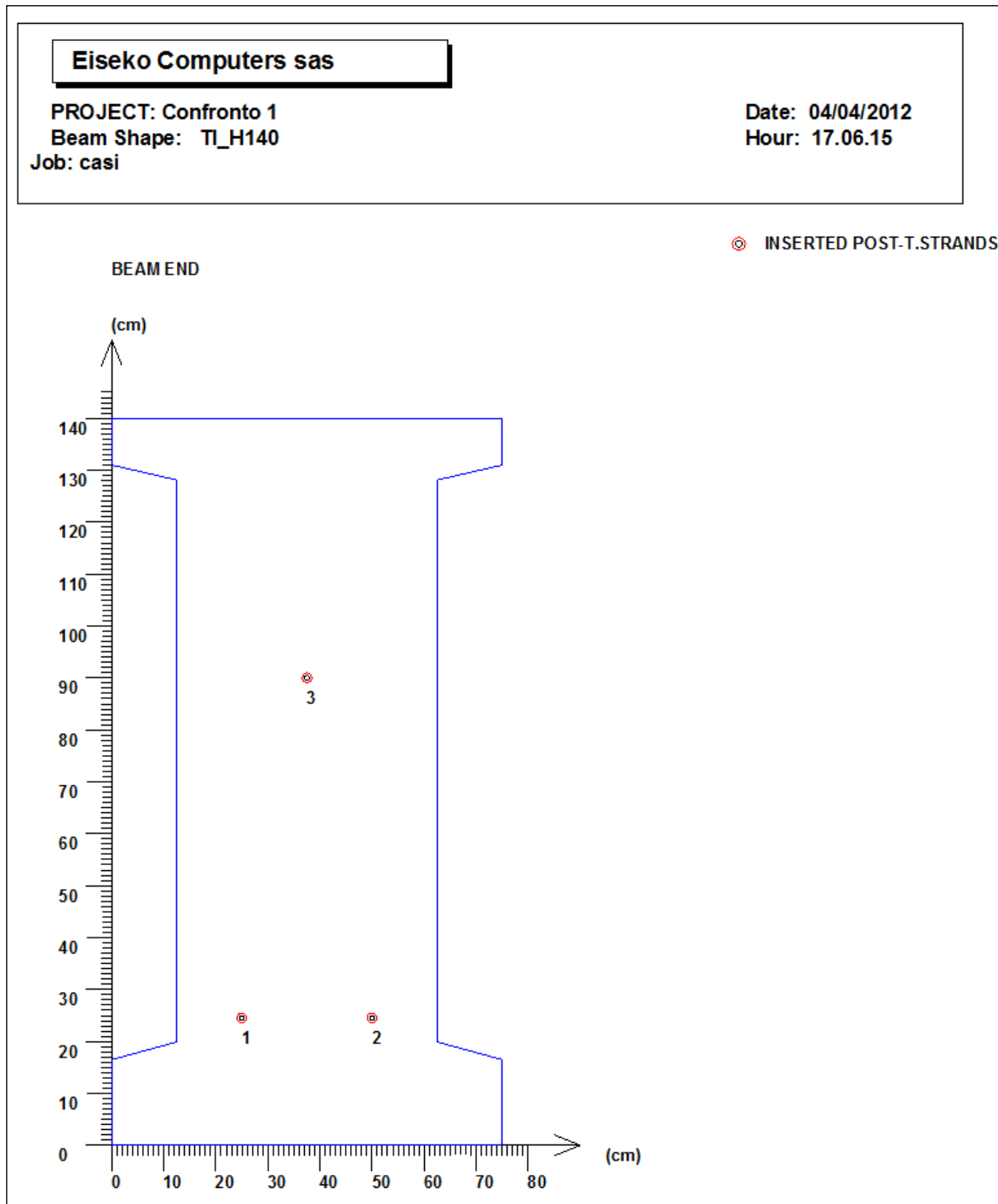
Post-tensed Strands

N	Y (cm)	X (cm)	Single Area(cm²)	Strand N.	Dist from head	Dist sup edge	Parab.Length
1	10.00	25.00	1.39	7	0	115.5	1260
2	10.00	50.00	1.39	7	0	115.5	1260
3	25.00	37.50	1.39	9	0	50	1260

Cable Heights

Sect.N.	Dist m	Cable H 1 cm	Cable H 2 cm	Cable H 3 cm
1	0.00	24.50	24.50	90.00
2	12.60	10.00	10.00	25.00

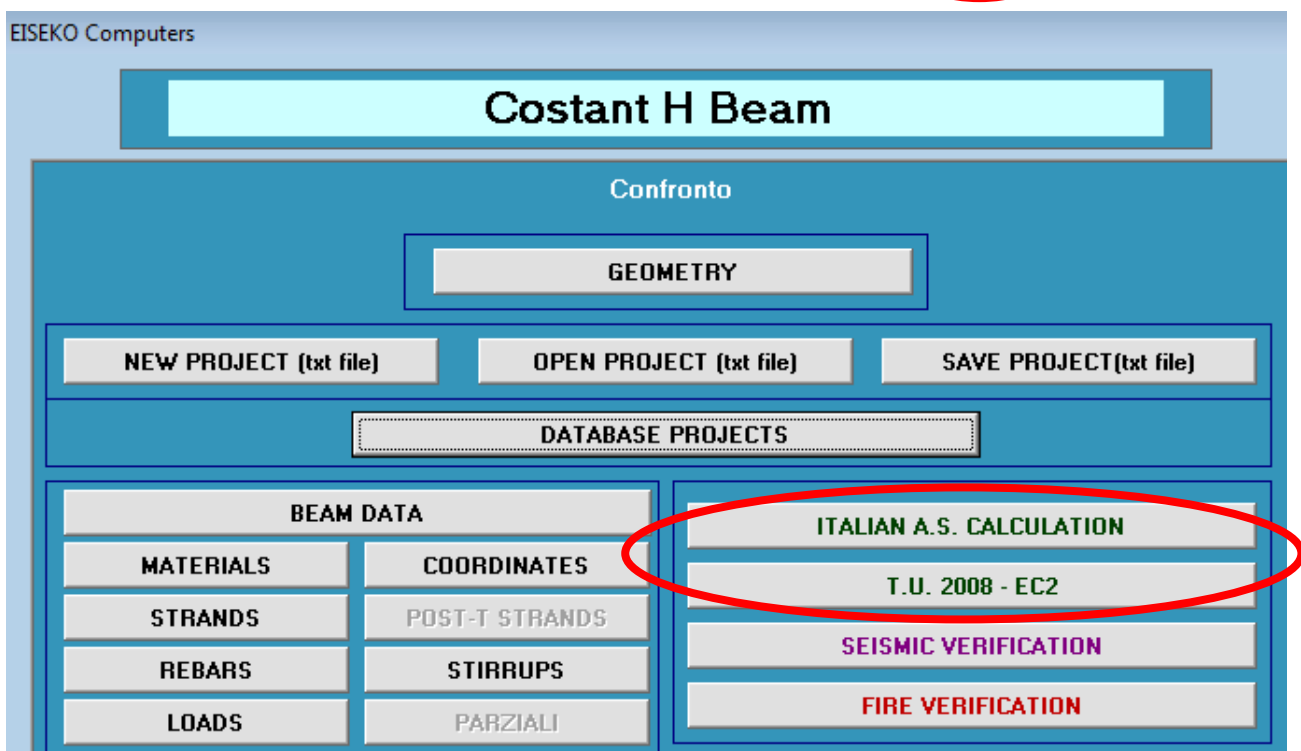
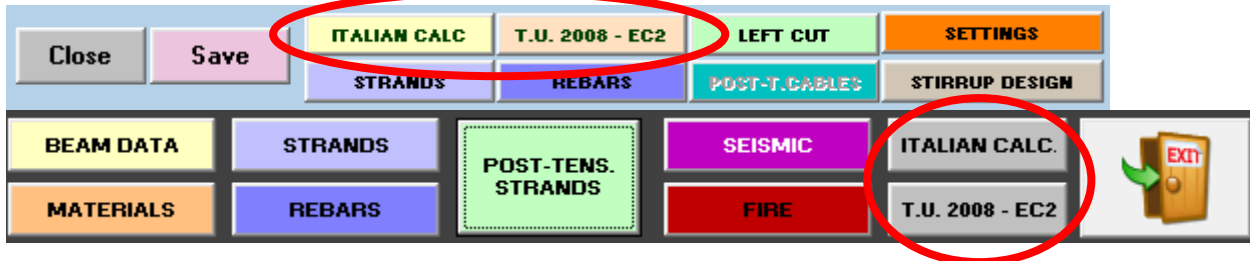
“[SEE BEAM HEAD](#)” Shows the beam end with the cable exits.



18. CALCULATIONS

Until you introduce all the data sufficient to perform the calculation of the beam, the buttons for calculations are disabled.

The verification of the beam, according to the method of Allowable Stress or EC2, can be made directly from windows "BEAM DATA", "STRAND", "REBARS", "POST-TENSED STRANDS" or the Main Window or from the menu



The result is on the screen in the "[Calculation report](#)" which is divided into three parts (see below): verification at midspan, verification at supports, verification at handling.

User can change the "[User Data](#)" and "[Company Data](#)" that appear in the headers of the reports, see chap. [Initial SET UP](#).

To choose the unit of measure used to calculate the Allowable Stresses, change the settings of the settings, see chapter 6.2.1.

The calculation is performed between the supports.

In the output report you find the table with the Serviceability/Ultimate limit state checks in various sections along the beam, the verification at the most stressed support and transport verification.

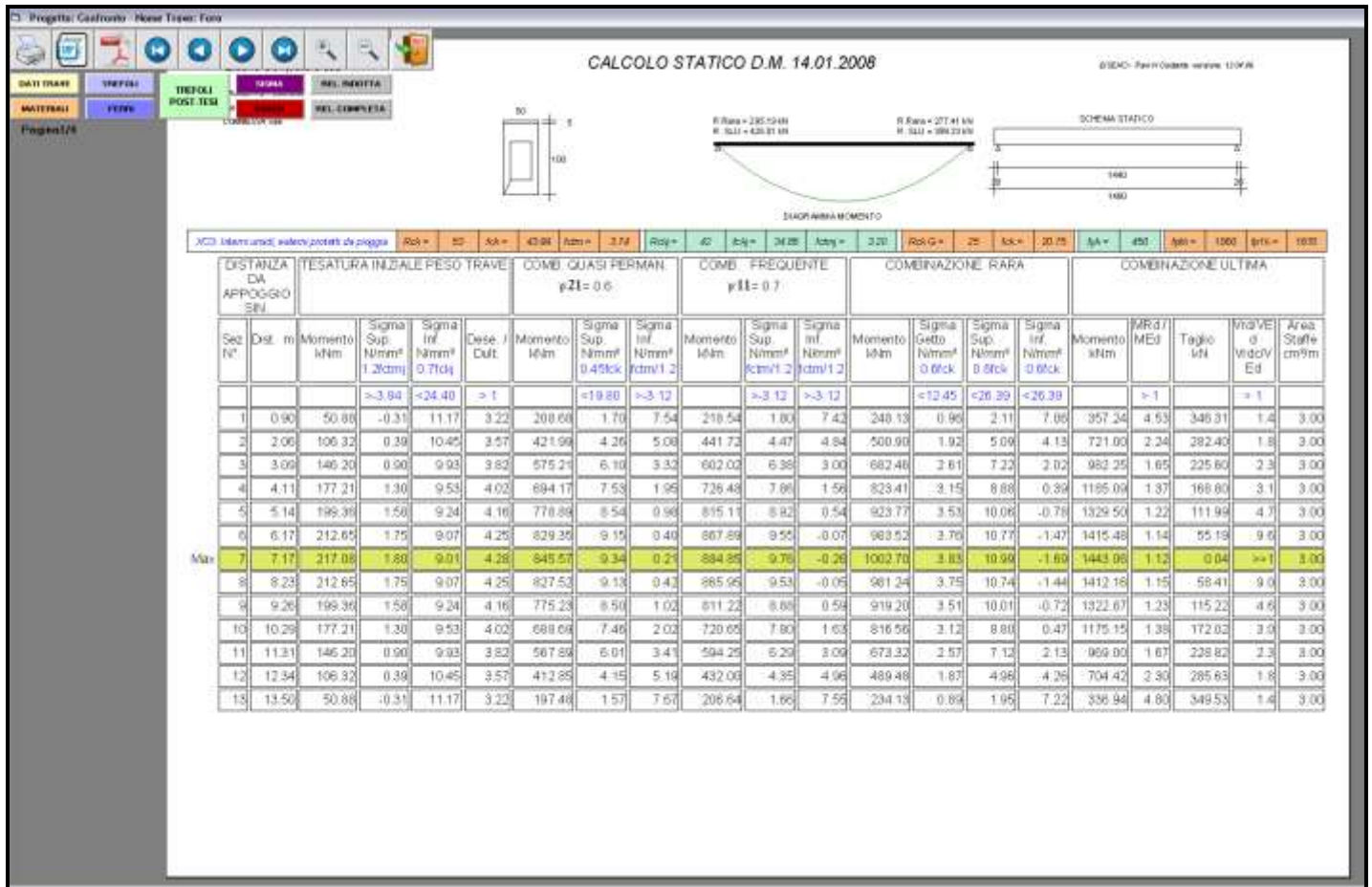
In the reports and charts:

Light blue values indicate the conditions that the calculation must respect (e.g. sigma min, max).

Red values indicate the checks not verified.

Violet with boundary values indicate the area of stirrups and of rebars to be included.

Press "T.U.: 2008 – EC2 to see the reports.

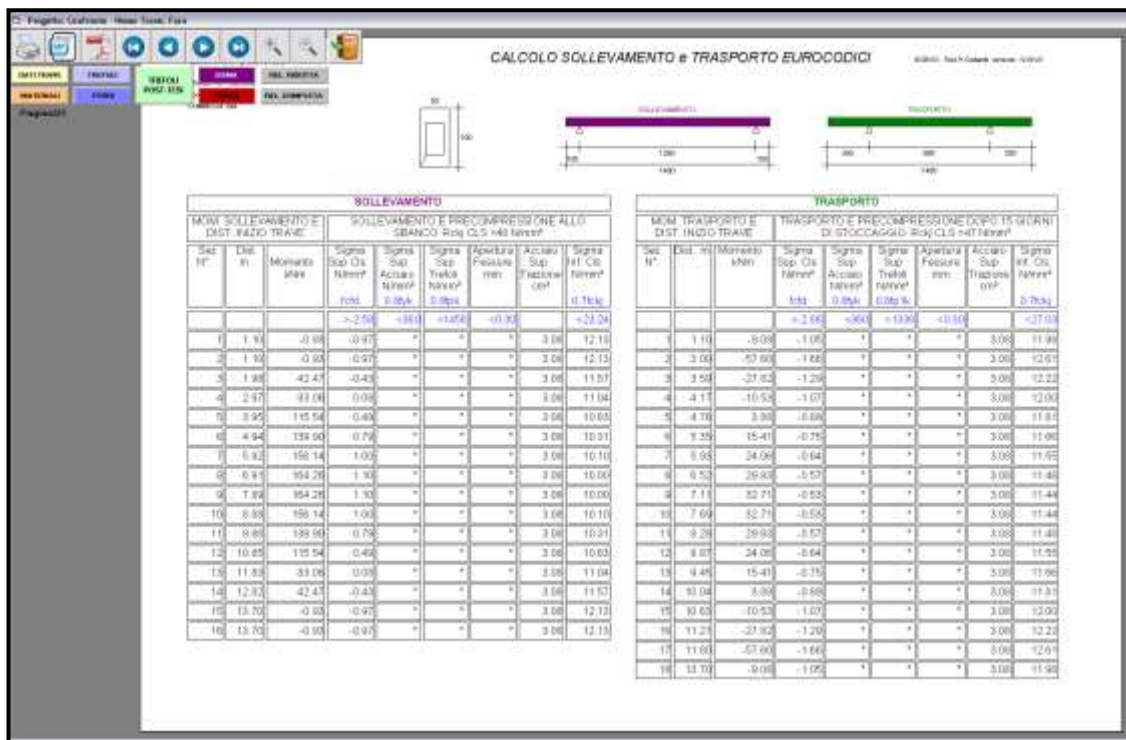


New: Display of Constraint reactions Rare and Ultimate left and right (also shown in the calculation report).

The verification is performed in a series of sections arranged at a constant step by the program, to which are added the first prestressed section, the most stressed section and the last prestressed section.

In the next page the shear verification in the most stressed support is shown.

The checks are performed according to EC2 as permitted by the DM 14/01/08.



Lifting and transport checks are performed in a series of prestressed sections starting from the left end.

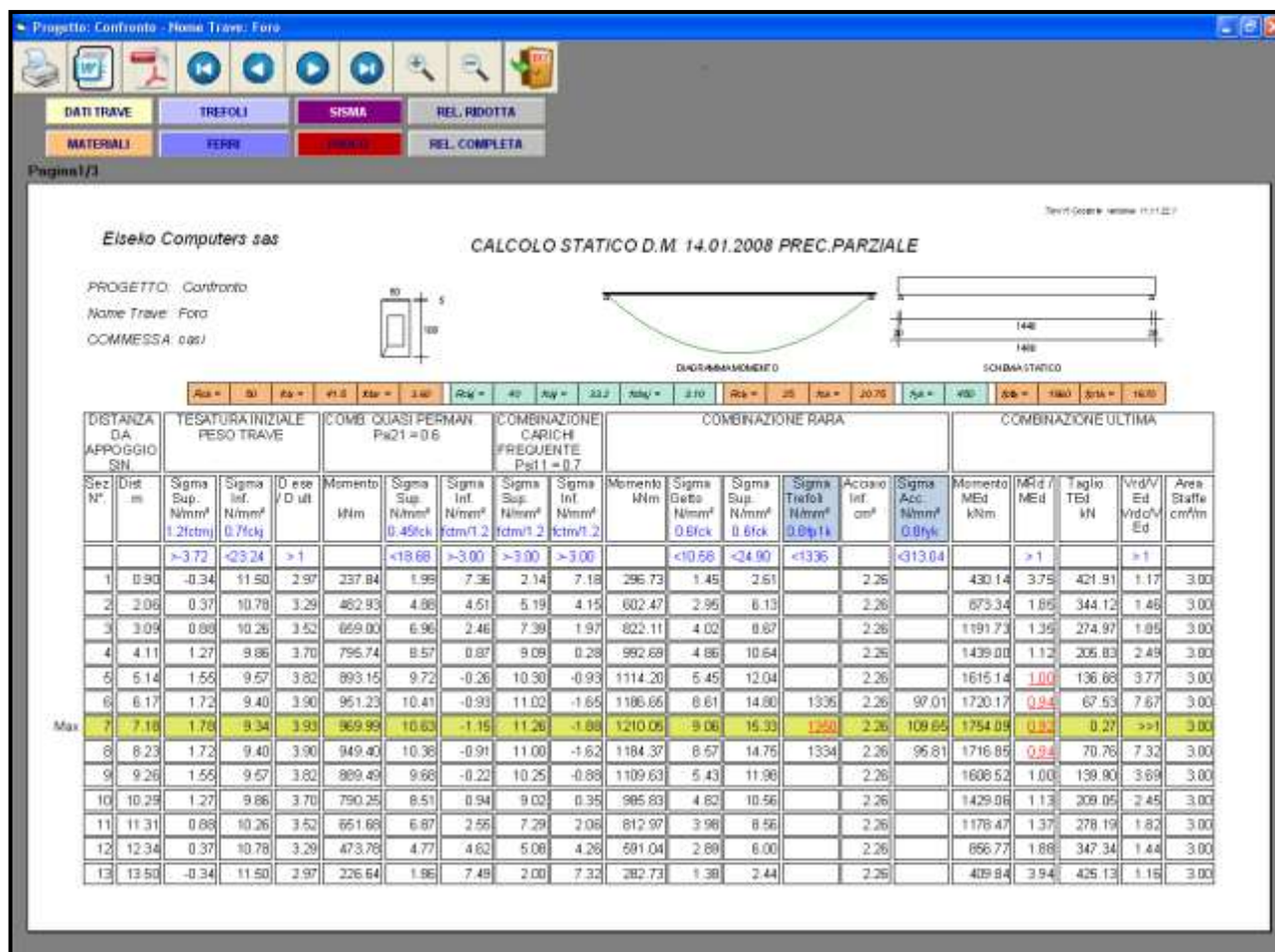
18.1. PARTIAL PRESTRESS VERIFICATION

The program switches to calculate using partial prestressing if during verification of the stresses at section intrados exceed the tensile limit valid for the calculation with entirely reacting section ($f_{ctm} / 1.20$).

The concrete is no longer considered reacting for traction as well as you do in the calculation of non-prestressed concrete beams, the beam is stressed with the serviceability moment acting simultaneously with the effort due to prestressing.

When the tensile maximum sigma of section exceeds the sigma limit in traction of the concrete, there is an immediate transfer of all the traction section from the tensile zone in concrete to reinforcement, with the consequent opening of crack. To limit the size of the cracks and the increase of stress on the strands, the program adds rebars (the user is alerted with a message on the screen) by placing a rebar $\varnothing 12$ for each of the possible positions which are in the lowest row of rebars in mask: in the example the program has two positions in the lower row and so inserts two rebars $\varnothing 12$ for a total area of 26.2 cm².

This is not a reinforcement "by rules", but only a proposed solution to the problem starting from a minimum of bottom reinforcement.

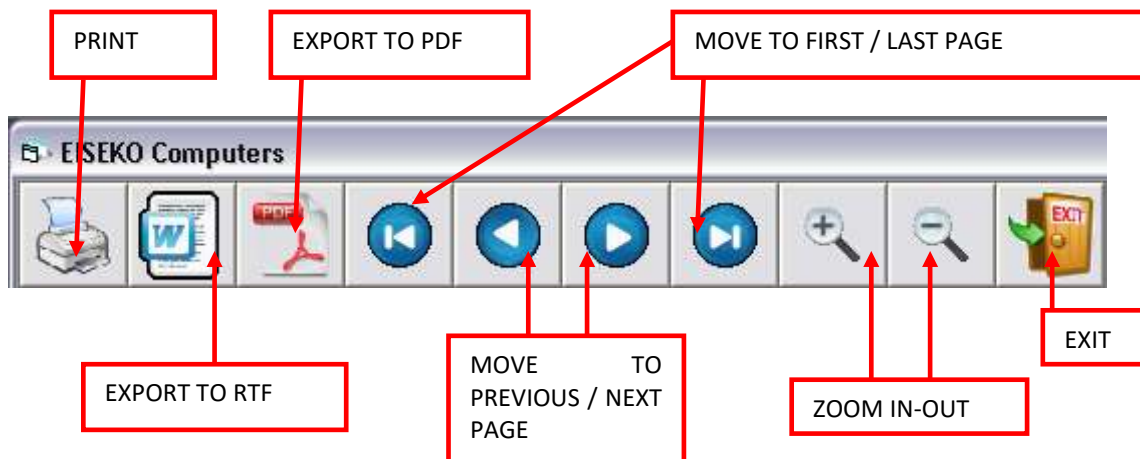


In the calculation report of the example, sections 6, 7 and 8 are calculated in a partial prestressing, sigma in the concrete on the top edge and the traction sigma in the strands and in the reinforcement are calculated by comparing them with the limits according to Rules. In the example, in section 7 report shows that the strands are too stressed and in the sections 5, 6, 7 and 8 report shows that moment capacity lower than ultimate moment, therefore, we must proceed to integrate the prestressing reinforcement with other strands, or to change the rebars with other with bigger diameter.

Of course the rebars $\varnothing 12$ inserted by the program are not required, the user can change them in the "rebars" window.

The program behaves in the same way for the LIFTING or TRANSPORT verifications in which it enters rebars $\varnothing 12$ for each of the possible positions which are in the highest row of rebars mask, in case both exceeded the traction limit in these phases (i.e. $f_{ctm} / 1.20$).

18.2. BUTTONS FOR REPORTS



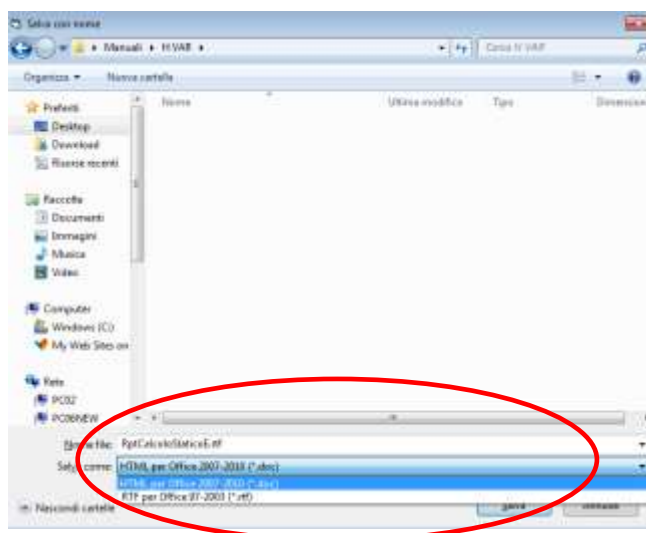
You can also zoom in with a simple double click on the report (double-click the right button decreases the zoom). NOTE: The initial zoom and the possible presence of grey parts of the screen depend on the screen resolution.

18.3. EXPORTATION

From menu:

EXPORT TO PDF: it's possible to save the report in a PDF file.

EXPORT TO RTF: it's possible to save the report in a file editable with any text editor (i.e.: Microsoft Word®, Notepad...). Files are saved in RTF or HTML: If you have Microsoft Word ® 2007 or later choose HTML, if you have Microsoft Word ® 2003 or earlier choose RTF. For other editors see which format is best with a test.



The resulting file may have the incorrect orientation of the page, depending on the printer, and the text editor used: just change it from the "File" menu, "Page Setup" editor.

PROGETTO: prime prove
Nome Trave: esempio
COMMESSA: a

MOVE ICON

Rck =	55	fck =	45.65	fctm =	
DIST. DA APPOGGIO SIN.				TE	TR
Sez N°	Dist m	Altezza Sez. cm		Sig Su N/r	
				1.2	
				>3	
1	0.67	70.20		-0.3	
2	2.03	83.82		0.8	

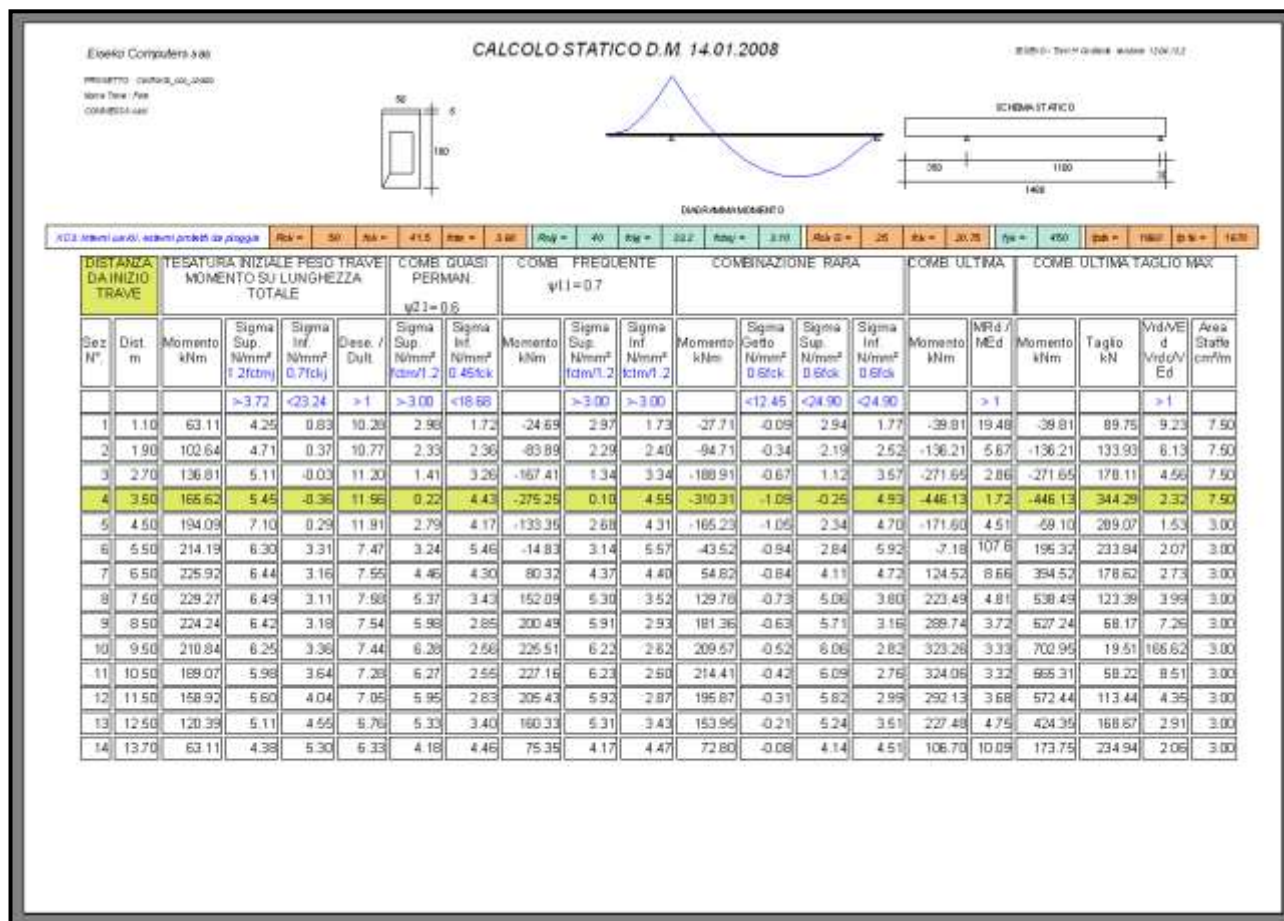
PROGETTO: prime prove
Nome Trave: esempio
COMMESSA: a

DIST. DA APPOGGIO SIN.		TE		TR	
Sez N°	Dist m	Altezza Sez. cm		Sig Su N/r	
				1.2	
				>3	
1	0.67	70.20		-0.3	
2	2.03	83.82		0.8	

65

19. VERIFICATION WITH LONG OVERHANGS (SUPPORTS)

If the beam has very long support distance from the ends, i.e. with dimensions exceeding the length of diffusion of prestressing, which is typically about one meter (see also section "Calculation System"), the program adds a verification with combinations of loads that give the maximum stresses on the supports and minimum between the supports.



The main difference compared to the table in paragraph "Calculations", which is calculated using the minimum loads on supports and maximum loads on midspan, is that in this case report shows also the checks on some prestressed sections on supports.

NOTE: IN PRESENCE OF LONG SUPPORT IT'S NOT POSSIBLE TO INSERT POST-TENSED STRANDS.

20. STIRRUP DESIGN

DESIGN OF THE SHEAR + TORSION STIRRUPS AND REBARS AT THE SUPPORT

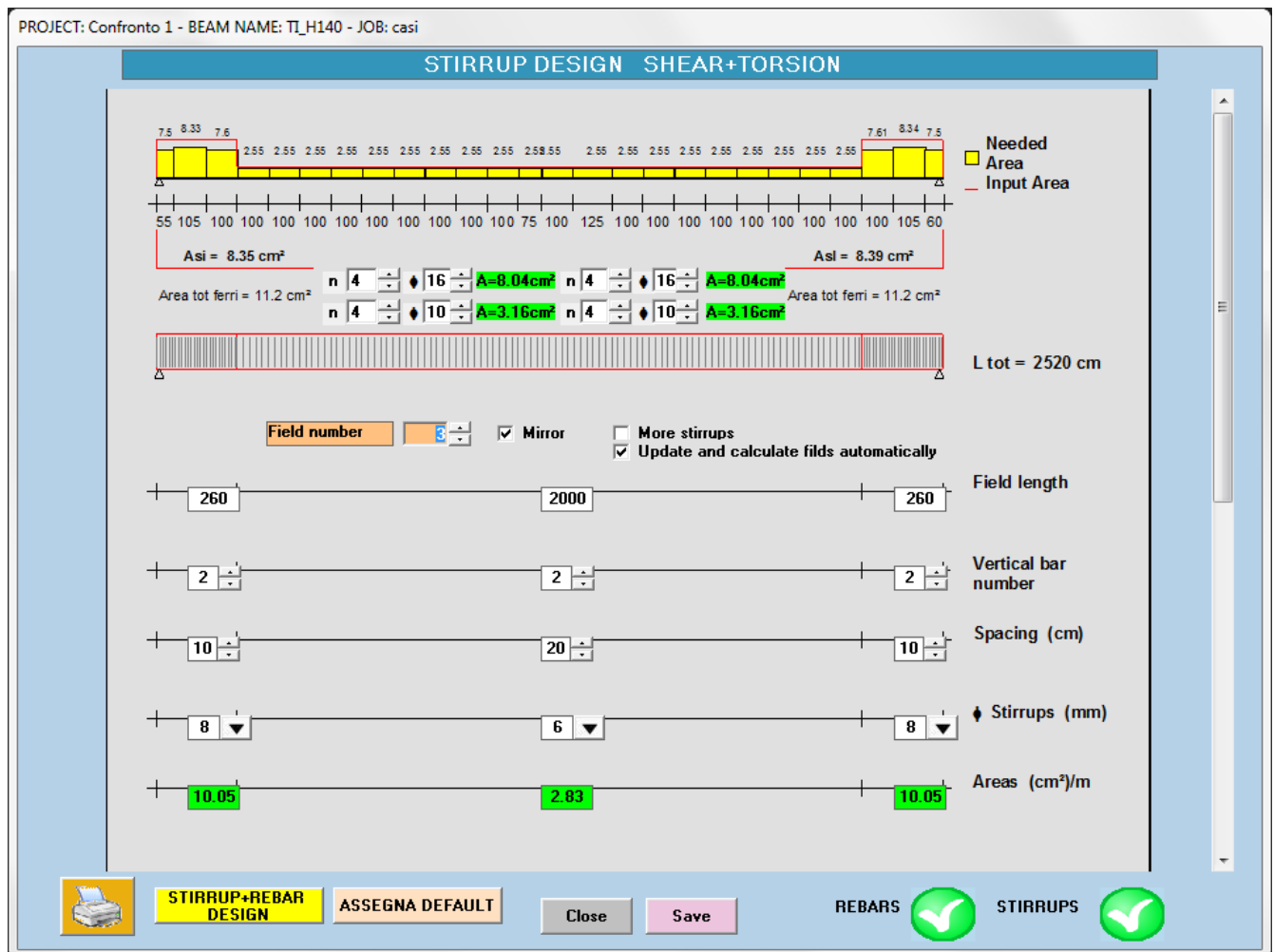


The program can design automatically the needed stirrups along the beam.

In the report and in the calculation table the necessary stirrups along the beam are shown, using a yellow graph representing the minimum stirrup area required calculated by the program, at the various step along the whole beam (the last column of the calculation tables).

Then the program can design a proposed set of stirrups according to this calculation results.

The user edit the proposed stirrups, dynamically modifying them with this convenient graphical environment, until they are like desired, checking dynamically if they are still according to calculation. Stirrups are subdivided in fields, grouped according to similar characteristic.



When user run this window, calculation is performed too.

20.1. The process of design

The first time you open the design of stirrups window in the current project, the program performs the calculation to update necessary areas, and sets the proposed stirrups. If the design of stirrups has already been performed and the data is saved, the saved data are displayed, even if user changed the data. NOTE: in this case user have to check if the stirrups are still verified.

20.1.1.SHEAR REBARS

Asi Calculated SHEAR REBARS NECESSARY AREA

Tools to set SHEAR REBARS in l. and r. head

Asi = 6.83 cm²

Area tot ferri = 2.26 cm²

n 2 d 12 A=2.26cm²

n 0 d 0 A=0cm²

total INPUT rebar area

Asl = 6.83 cm²

Area tot ferri = 2.26 cm²

A red rebar is represented with L-shape (or straight) with the area required from shear (in this case Asi = 6.83 in both the left and right head). Below the introduced area is shown: total area rebars = 2.26 in this case, calculated by setting the default rebars, in this case 2 ϕ 12 + 0.

Modify the number of rebars and the diameters using the up-down arrows or by typing the data in the boxes. It's possible to insert two different groups of rebars (ie two diameters: each group has a number of rebars of the same diameter).

For each change the data "Tot Rebars Area " is dynamically updated and it is possible to verify whether it is sufficient in a very simple and immediate way: when it is ok it's green:

Asi = 6.83 cm²

Area tot ferri = 7.6 cm²

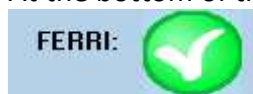
n 2 d 22 A=7.6cm²

n 0 d 0 A=0cm²

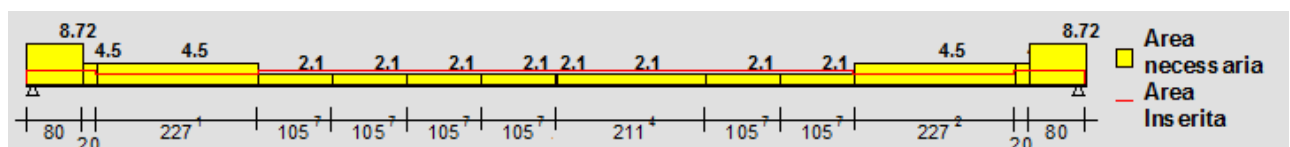
Asl = 6.83 cm²

Area tot ferri = 7.6 cm²

At the bottom of the window the green check appears:



20.1.2.STIRRUPS



Stirrup graph along the whole beam is printed on top of the window, with inserted stirrup area, red, overlapping to see immediately if it is verified or not. Red line should be higher than yellow box, to be verified.

GRAPHIC PRESENTATION OF RESULTS ALONG THE BEAM: FOR EACH STIRRUP FIELD, STIRRUP CHARACTERISTIC ARE SHOWN

Ltot TOTAL BEAM LENGTH
L tot = 1500 cm

NUMBER OF STIRRUPS FIELDS
Numero campi: 5 ☒ Specchia ☐ Più tipologie (staffe doppie)

SYMMETRIC / DOUBLE STIRRUPS

Lunghezze Campi (cm)	Numero braccia	Passo (cm)	Staffe (mm)	Aree (cm ² /m)
100	2	20	6	2.83
230	2	20	5	1.96
840	2	20	6	2.83
230	2	20	5	1.96
100	2	20	6	2.83

STIRRUP AREA IN THE FIELD. IF > OR = TO THE ONE CALCULATED, IT'S GREEN, OTHERWISE IT'S RED

LENGTH OF STIRRUPS FIELD + STIRRUPS DATA: n. branches, step, diameter

First you set the number of stirrups fields to use.

Then define the **lengths of the fields**. The top graph shows the ranges in which the sections in the calculation are calculated.

Mirror: Check to mirror all of the input data (except the centre field, if the number of fields is odd).

More types of stirrups (double stirrups): To have two types of stirrups available for each field. The mask will change to:

Numero campi: 5 ☒ Specchia ☒ Più tipologie (staffe doppie)

Lunghezze Campi (cm)	Numero braccia	Passo (cm)	Staffe (mm)	Aree (cm ² /m)
100	0 2	0 20	0 6	2.83
230	0 2	0 20	0 5	1.96
840	0 2	0 20	0 6	2.83
230	0 2	0 20	0 5	1.96
100	0 2	0 20	0 6	2.83

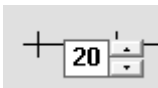
Where the input data added to existing ones represent any additional stirrups of separate fields.

To add a stirrup in the field to enter all its data.

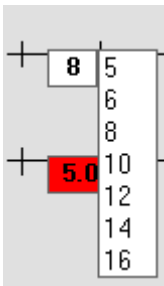
To leave just one stirrup in the field, set null data.

Each field must have at least one stirrup.

The diameters of the brackets must be between the minimum and maximum values given in complex settings, see section [STIRRUP DESIGN tab](#).



Press the arrow up or down to change data faster. The results are updated for each change. For the spacings, the arrows change the entered value by step of ± 5 cm; for the branches, ± 1 .



To change the diameter click on the arrow and select the desired diameter of the list. The diameters of the stirrup must be between the minimum and maximum values given in complex settings, see section [STIRRUP DESIGN tab](#).

For each change, areas in the last row of dimensions are updated. If displayed in red, they are lower than the calculated minimum required, whether in green they are equal to or greater and they are verified:



21. SERVICEABILITY VERIFICATION IN ANY SECTION

It is possible to run the calculation in a single section chosen by the user: it can be any section of the beam, but it must be prestressed.

You get the output of report for only the chosen section instead of the whole beam.

PROJECT: Confronto - BEAM NAME: Foro - JOB: casi

SECTION SELECTION

From Left edge m

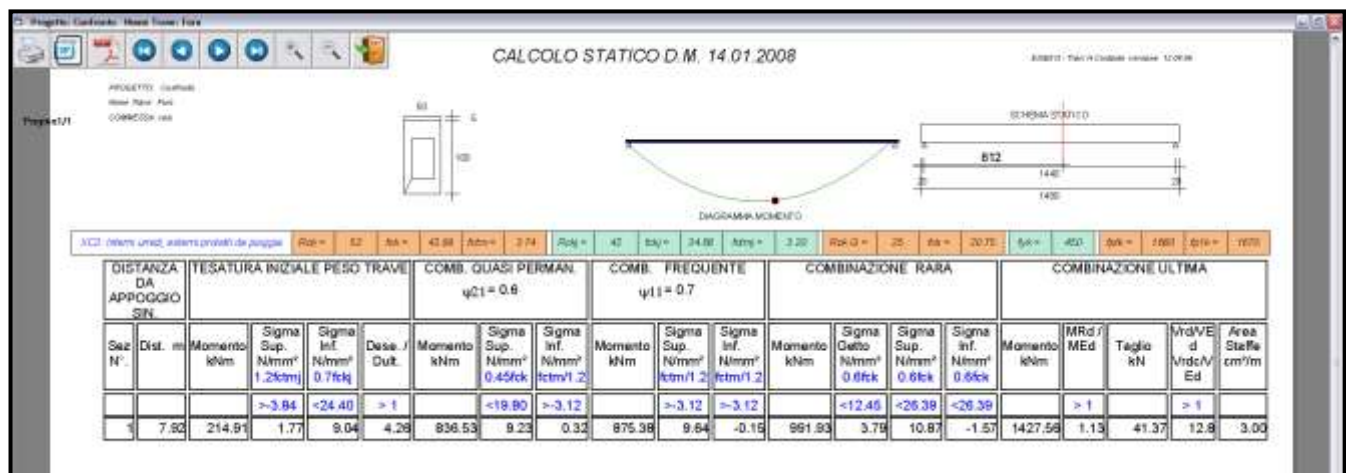
TYPE OF CALCULATION

☒ Italian Calculation ☒ T.U. 2008 - EC2

UNIT OF MEASURE

☒ Kg / cm² ☒ N / mm²

Set the value of the distance of the section to be calculated from the left (the default value is the most stressed section found in the calculation, if it has already been calculated), choose the type of calculation (TU default) and click the button "SECTION CALCULATION".



For Italian calculation:

Eiseko Computers sas

PROGETTO : Confronto

Nome Trave: Foro

Commessa: casi

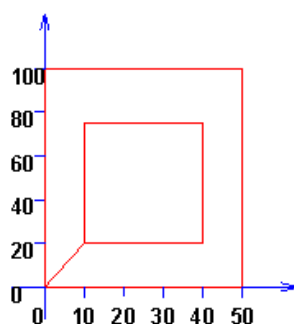
Data : 27/05/04

Ora : 18.53.40

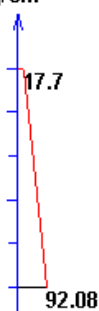
Sezione Singola

Distanza Sezione 8.12 m

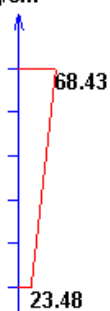
SEZIONE TRAVE



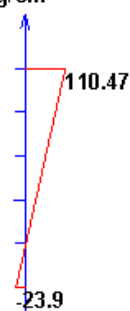
SIGMA SBANCO
Kg/cm²



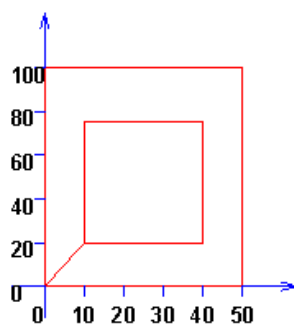
SIGMA PERMANENTE
Kg/cm²



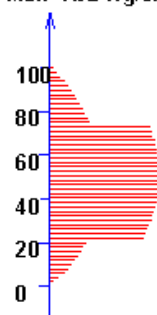
SIGMA FINALI
Kg/cm²



SEZIONE TRAVE



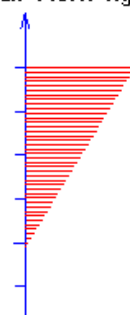
TAU TOTALE
Max 1.92 Kg/cm²



SIGMA PRINC. TRAZ.
Max -23.90 Kg/cm²



SIGMA PRINC. COMP.
Max 110.47 Kg/cm²



22. Research Spans and Capacities

HAVING REINFORCEMENT AND STRANDS, IT DESIGNS THE DIAGRAM EFFECTIVE SPAN / CAPACITY.

PROJECT: Confronto - BEAM NAME: Foro - JOB: casi

RESEARCH SPANS AND CAPACITIES


TYPE OF CALCULATION

☒ Italian calculation
 ☐ D.M. 14/01/2008

Max. Initial Effective Span	<input type="text"/>	m	Min. Final Effective Span	<input type="text"/>	m
Min. Initial Capacity	<input type="text"/>	Kg / m	Max. Final Capacity	<input type="text"/>	Kg / m
Research Spans Range	<input type="text"/>	m	Loads Range	<input type="text"/>	Kg / m
Breaking Point Coefficient	1.5	(1 - 2)	Traction Limit Coefficient	0.06	(0,06 x Rck)
Permanent Loads	<input type="text"/>	%			

Effective Span Leff m
 Beam Capacity Kg / m
 ☒ Stop to print

I	Dist	Moment	Shear	Sig S I	Sig I I	Sig Cast.F	Sig S F



The texts have the following meaning:

Max. Initial Effective Span max effective span to start the research: the program will mate it with min load at the beginning, it's recommended to set it more than the maximum possible.

Min. Final Effective Span min effective span, under this value the research is not interesting.

Min. Initial Capacity Kg/m capacity to start the research: the program will find, starting from the max span the first possible span with this capacity.

Max. Final Capacity Kg/m capacity to end research.

Research Spans Steps Step for the next calculation: whenever the span is not valid, the program decreases it of this value and performs another calculation.

Research Loads Steps Step for the next calculation: whenever the load is not valid, the program increases it of this value and performs another calculation. For example, if a 20 m beam supports 1000 Kg/m, the program increases the loads of this value and finds a new span.

Permanent Loads % Used for the calculation of the stress losses: it's the presumed percentage of permanent load of the total used load.

Breaking Point Coefficient The coefficient which, if not reached, triggers the exclusion of span by decreasing it.

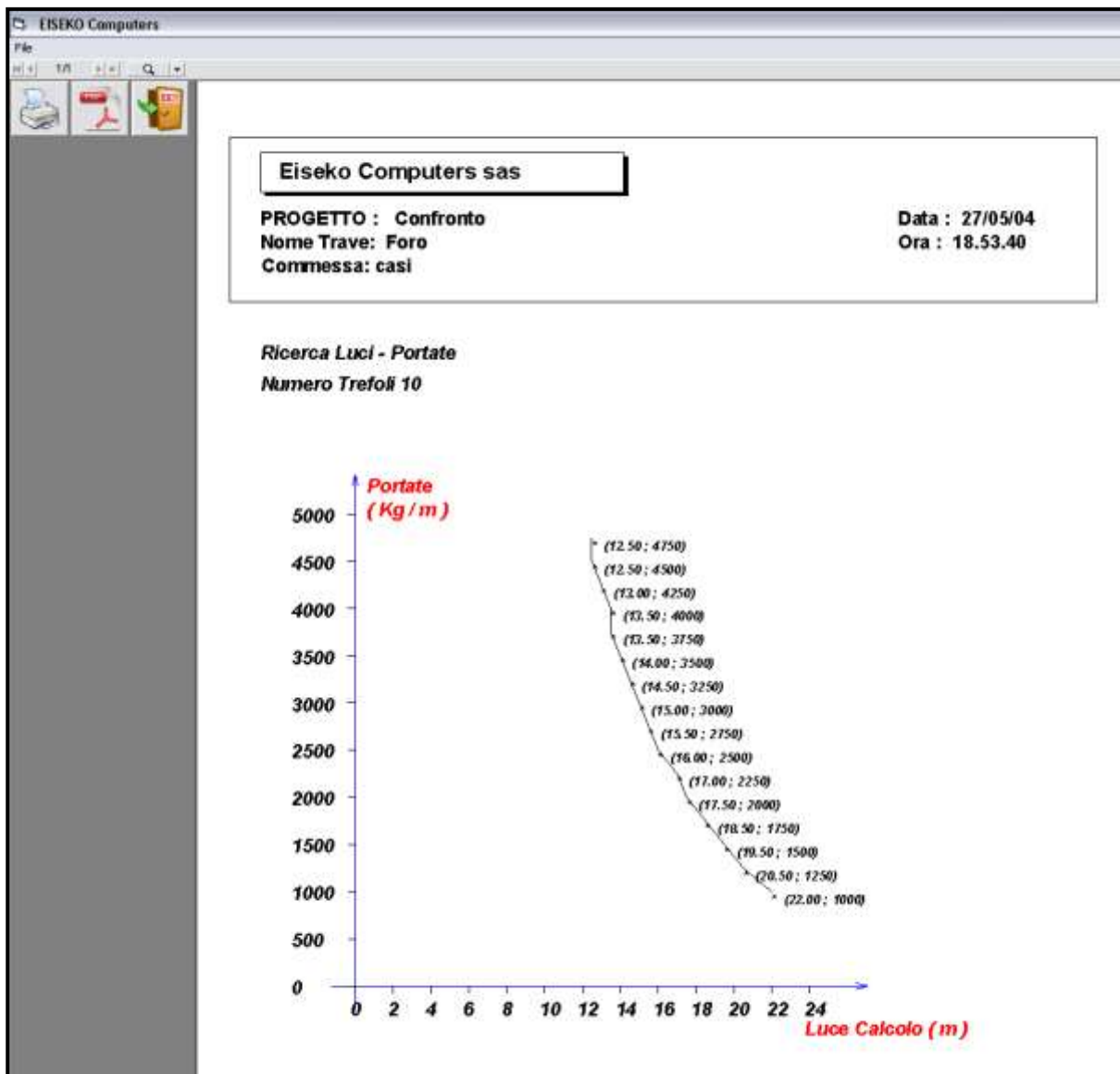
Traction Limit Coefficient The same for inferior traction.

Once you have entered all the data, and choose the Rules, you start the calculation by clicking on the "[SEARCH](#)" button at the bottom left of the window.

The program stops at the first possible span at 1000 kg / m, the first capacity of the research, and asks if you want to make a print to control the output data.

We can also decide to continue ignoring the print, or to cancel the entire calculation by pressing the Cancel button.

Button "[View Graph](#)" shows the following diagram (printable and exportable):



23. SEISMIC VERIFICATION

PROJECT: Confronto - BEAM NAME: Foro - JOB: casi

SEISMIC VERIFICATION

USE CLASS

<input type="radio"/> I	Buildings of lesser importance to public safety	Vn = 50	Cu= 0.7
<input type="radio"/> II	Ordinary buildings	Vn = 50	Cu= 1
<input checked="" type="radio"/> III	Important buildings in relation to the consequences of possible collapse	Vn = 50	Cu= 1.5
<input type="radio"/> IV	Buildings whose fuction has fundamental importance for civil protection	Vn = 100	Cu= 2

DELETE LOCATION
CHOOSE LOCATION
UDINE (UD)

Location Latitude	46.063	Location Longitude	13.236
Beam Nominal life (10-50-100)	50 years	Usage coef.	1.5
Period of reference	75 years		
Ultimate limit state (LLS)	0.1	Serviceability Limit state (DLS)	0.63
Structure factor	1.5	Hor. Structure factor	1.5
Topographic Category	T1	Subsoil Category	A

Pver	Tr	Ag	Fo	T*c	
81	45	0.0695	2.47	0.26	SLO
63	75	0.0901	2.46	0.27	SLD
10	712	0.2414	2.47	0.34	SLV
5	1462	0.3198	2.47	0.36	SLC

ONLY VERTICAL SEISM
☒
DLS - LLS CALCULATION
LLS CALCULATION
REPORT

Close
SET SEISMIC PARAMETERS

USE CLASS: four possible options are proposed with the corresponding values of Cu; default values for Design Working Life Vn are proposed, which may be changed in a following mask;

CHOOSE LOCALITION: pressing this button the window "seismic hazard assessment" is opened. Input the location of the building (in the "Location" textbox), and press "Find". If there are multiple locations with the same name but in different provinces, the program shows all of them: click on the one that interests you. The program finds the latitude, longitude and seismic parameters of the site. If the user wishes to change the coordinates he can do it directly in the boxes Longitude and Latitude, and then pressing "Apply".

Probability "PVER" corresponding to the various checks are those of the Regulation, these can be changed too, and then remember to press "Calculate".

Hor. Structure factor: for the horizontal verification, the default value is the minimum 1.5, but generally must be assigned by the user.

Topographic Category and Subsoil Category: these are list boxes with the possibility of Rules.

Valutazione della pericolosità sismica

ISTITUTO NAZIONALE DI GEOFISICA E VULCANOLOGIA

Legend: < 0.025, 0.025-0.050, 0.050-0.075, 0.075-0.100, 0.100-0.125, 0.125-0.150, 0.150-0.175, 0.175-0.200, 0.200-0.225, 0.225-0.250, 0.250-0.275, 0.275-0.300, 0.300-0.350, 0.350-0.400, 0.400-0.450, 0.450-0.500, 0.500-0.600, 0.600-0.700

p.e. 10% in 50 anni

Nota: per il calcolo dei parametri sismici
1) inserire le coordinate geografiche 2) introdurre Vn e Cu

Per le isole è possibile utilizzare come località: gruppo isole N [con N = 1,2,3,4,5]

Id nodo	Longitudine	Latitudine	Distanza [km]
12731	11.071	45.411	2.186
12732	11.142	45.412	3.687
12510	11.140	45.462	5.777
12509	11.069	45.461	5.006

Coordinate geografiche

Località: Trova

Longitudine: Latitudine: Applica

Parametri per le forme spettrali

Pver	Tr [anni]	ag [g]	Fo	T*c [sec]
81	30	0.040	2.500	0.240
63	50	0.055	2.490	0.250
10	475	0.154	2.430	0.280
5	975	0.200	2.470	0.280

Periodo di riferimento per l'azione sismica

Vita Vn [anni] Coeff. uso Cu Periodo Vr [anni]

Calcola

Annulla OK

In tab **Time reference point for the seismic action** you set the values of the Design Working Life and use coefficient, and press "Calculate".

If the parameters for the spectral shapes are fine, press "OK" and return Check seismic window, which gets all the seismic values provided by the window of Seismic hazard evaluation.

PVR-ULTIMATE LIMIT STATE LLS: is the probability of exceeding the corresponding limit state LLS, which by default is 0.1 (10%), but can be changed from the user and the seismic parameters automatically update in the LLS row.

STRUCTURE VERTICAL FACTOR: for vertical checks, default is equal to 1.5

STRUCTURE HORIZONTAL FACTOR: for horizontal checks, the default is equal to the minimum value of 1.5, but generally it should be assigned from the user.

TOPOGRAPHICAL CATEGORY and CATEGORY OF THE UNDERGROUND: they are list boxes with the possibility according to Rules.

At this point you can proceed to the calculation of the beam.

If you want to test only the vertical earthquake, you select the check box "ONLY VERTICAL SEISM" and then press "CALCULATION SLD-SLV" with which we get the summary calculation table for DLS and LLS limit states. If you want to test the vertical earthquake combined with the horizontal one, you

deselect the "ONLY VERTICAL EARTHQUAKE" check box, then you can get the summary calculation table for DLS and LLS limit states pressing respectively "DLS CALCULATION" and "LLS CALCULATION". You can the open the calculation report (button "REPORT").

23.1. ASSOCIATE SEISMIC PARAMETERS TO JOBS

From the main screen, press "SETTINGS" and then "SET SEISMIC PARAMETERS". This will open the following screen, where it is possible to associate all the seismic data relative to the location where you will find the selected job (in this case "casi").


EISEKO Computers

SEISMIC PARAMETERS

Job Order: **casi**

DELETE LOCATION **CHOOSE LOCATION**

Location Latitude	<input type="text"/>	Location Longitude	<input type="text"/>
Use Class - Usage coef. - Cu	II - 1		
Structure factor	1.5	Hor. Structure factor	1.5
Topographic Category	T1	Subsoil Category	B

 **Close** **Save**

Every time you make a new project associated with the job "casi", the program will propose automatically the data of seismic parameters (they can of course be changed). The existing projects, created before the definition of these parameters, will continue to use the previously saved data instead.

24. FIRE VERIFICATION

Pressing the button "[FIRE VERIFICATION](#)" or "[FIRE](#)" the following window is opened:

PROJECT: Confronto - BEAM NAME: Foro - JOB: casi

FIRE VERIFICATION

SECTION SELECTION

From Left edge	9.80	m
Section for Moment calculation	9.80	m
Section for Shear calculation	2.2	m

Per. Load Rid. Coef.
G1 + G2

1

Var. Load Rid. Coef.
Qk1

ψ_{21} 0.6

Var. Load Rid. Coef.
Qk2

ψ_{22} 0.6

Fire Combination

$G1 + G2 + P + Ad + \psi_{21} * Qk1 + \psi_{22} * Qk2$ T.U. 2.5.6

SETTINGS

REI	120	min	Stirrups °C	0	°C
MESH	4	cm			


☒ UNI 9502 may 2001
☐ UNI EN 1992-1-2:2005 simplified
☐ UNI EN 1992-1-2:2005 advanced

CONCRETE

☒ Calcareous ☐ Siliceous

FIRE EXPOSITION

<input checked="" type="checkbox"/> All left edges	<input type="checkbox"/> All top edges
<input checked="" type="checkbox"/> All right edges	<input checked="" type="checkbox"/> All bottom edges



Exit

FIRE CALCULATION

The program proposes automatically the sections with maximum moment and shear, editable from the user.

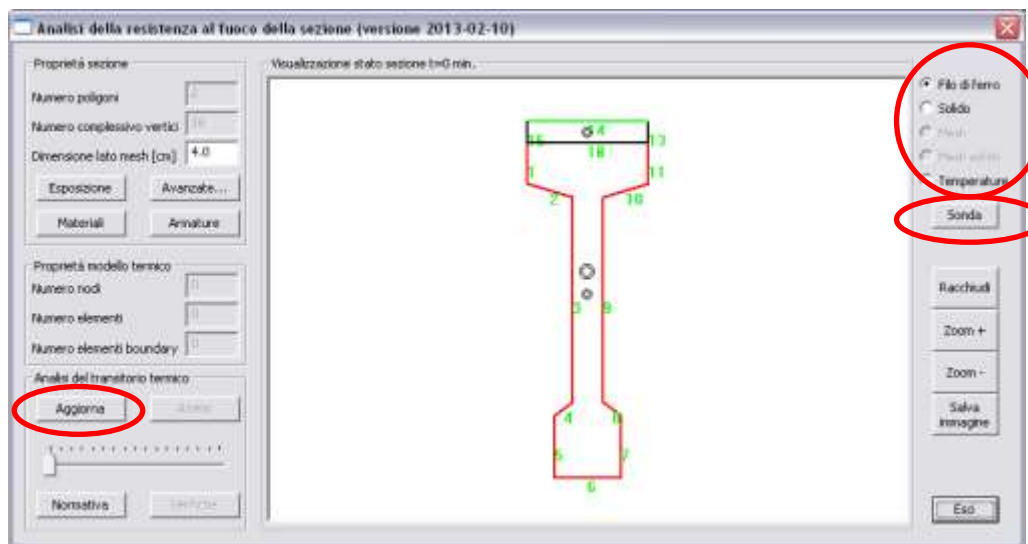
Furthermore, it proposes automatically fire exposure for the sides left / right / top / bottom, they can be modified from the user too: just select or deselect the check and the program automatically set the exposure to heat or to air, respectively, for all segments of the beam section that are on the chosen side. To choose other types of exposure you have to enter the next window.

The exposure for the cast and the holes are also provided automatically, and always editable (in next window).

The settings (rules, REI, mesh, stirrups temperature and aggregate type) are read by the default settings and assigned here. The user can change the values and the change will be saved along with all other project data, so that if you return later on the project, you will find again the saved data.

See [dedicated chapter](#).

Pressing "[Fire Calculation](#)" opens the main analysis of the fire analysis:



In the graphics area the section, the existing reinforcement and the geometric centre of gravity of the beam are shown, in case cast in place is present, centre of gravity of cast in place and the entire section are shown too.

In the top left corner it is possible to choose the preferred type of representation for the beam. "Filo di ferro" is shown in the figure: the segments of the section are numbered, the reinforcement has blue color, the prestressed strands have light blue color.

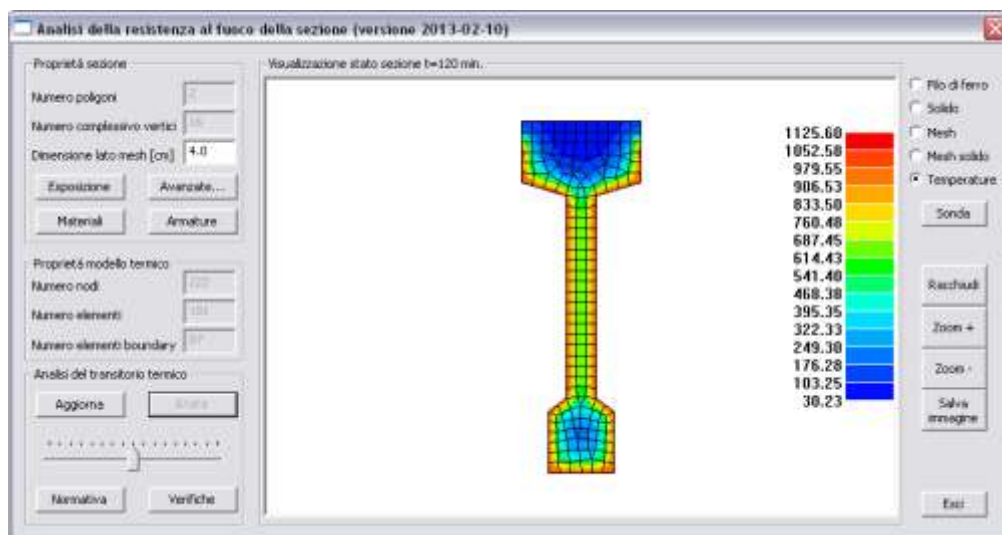
Pressing the "Aggiorna" button the mesh is calculated automatically. We strongly recommend you to choose a mesh size of about 4-5 cm, if you chose a lower mesh size calculation times are getting longer. We strongly recommend minimum size of 2 cm. Press the option "Mesh" at the top right to display it.

NOTE: the window for fire check allows you to modify some characteristics of the beam such as the number, the position and the type of the rebars of reinforcement, the strength class of the beam and the cast in place. All these changes are internal to this verification window, they are not brought back to the cold calculation, and if you want to keep the changes you should input them manually in the input data.

Please read the specific manual of the fire verification for the detailed explanation: it is possible to download it from our site www.eiseko.com/download.

Pressing "Analysis" you run the verification. When finished, the window displays the temperatures required for REI. By moving the cursor under the button "Analysis" it is also possible to view minor REI too.

By means of the "Sonda" button on the right, it is possible to see the exact value of the temperature inside the section: click on the desired point and a window with the temperature values appears.



Pressing “Verifiche” the following window appears.

Analisi della resistenza al fuoco della sezione

Caratteristiche limite M-N

Coefficienti di sicurezza

- calcestruzzo: 1.20
- acciaio: 1.00

Sezione tesa: -1476.7 kN

Sezione compressa: 10271.7 kN

Tese fibre inferiori: 1162.2 kN m

Tese fibre superiori: -268.0 kN m

Tese fibre a sinistra: 238.8 kN m

Tese fibre a destra: -238.0 kN m

Aggiorna

Caratteristiche limite V

Coefficienti di sicurezza

- calcestruzzo: 1.20
- acciaio: 1.00

Asw/s: 9.93 cm²/m

fyw: 450.0 N/mm²

Tmed: 20.0 C

Delta: 1.00

Bw ini: 0.00 m

Bw fin: 0.50 m

d: 1.06 m

V (fcd): 3389.5 kN

V: 240.4 kN

Vcd: 323.1 kN

Rd / Ed: 3.12

Vwd: 426.4 kN

V lim: 749.6 kN

Aggiorna

Caratteristiche limite M-N proporzionali

N: 0.0 kN

N lim: 0.0 kN

N positivo: compressione

M i-s: 881.1 kN m

M i-s lim: 1162.2 kN m

M i-s positivo: tende le fibre inferiori

M s-d: 0.0 kN m

M s-d lim: -0.0 kN m

M s-d positivo: tende le fibre a sinistra

Rd / Ed: 1.32

Aggiorna

Salva verifica

Esci

25. LEFT CUT

Verification of Gerber Support in the ends

The geometry data are imported directly from "[BEAM DATA](#)". The default cut is 20 cm in height and 20 cm in length, editable, while the higher value of Shear between the 2 support is set. The values entered in the ORANGE and YELLOW fields can be edited from the user to make immediate assessments with different values. The "a" model provided by EC2 is adopted for checks, with the exception of the calculation of T3 where the model "b" is adopted in favour of safety. The program reports automatically if the cut is not verified and where.

Progetto: prime prove - Nome Trave: esempio

Verifica SELLA Sinistra

VERIFICA CON GETTO

Rck 55 N/mm²
 fck 45.7 N/mm²
 fcd 25.9 N/mm²
 fyk 450 N/mm²
 fyd 391 N/mm²

R 22426 Kg ok verificato <VRd
 b 30 cm larghezza trave
 α 45.0 gradi considerato per T2
 α 69.0 gradi d 42.0 cm
 β 65.2 gradi pl 0.005
 v 0.5 6.6N EC2 k 1.69

Tirante T1 = C1 sin α + C2 sin β 37764 Kg As1 = 9.66 cm²
 Tirante T2 = R / tan α 22426 Kg As2 = 5.74 cm²
 Tirante T3 = R 22426 Kg As3 = 5.74 cm²

Puntone C1 = R / sin α 24021 Kg ok verificato <VRdmax
 Puntone C2 = T2 / (sin β + cos β) 16897 Kg ok verificato <VRdmax
 Puntone C3 = C2 sin β / sin 45° 21692 Kg ok verificato <VRdmax

VRd,c = 4.1.14 T.U. 6550 Kg
 VRdmax = 6.5 EC2 81585 Kg
 VRd = 6.9 EC2 73427 Kg

As1 = T1/fyd 9.66 cm²
 As2 = T2/fyd 5.74 cm²
 As3 = T3/fyd 5.74 cm²
 As4 = R/(0.9dlyd)* 20 3.03 cm²

Salva dati per disegno:

φ mm	n	Area cm ²	
12	12	13.57	OK
20	2	6.28	OK
20	2	6.28	OK
5	16	3.14	OK

Progetto: prime prove
 Commessa: n
 Nome Trave: esempio

OK

Button "[PRINT](#)" prints the windows as you see it.

26. SNOW - WIND LOADS

The button “[SNOW - WIND LOADS](#)” opens this simple window.

SNOW - WIND LOADS for all areas of Italy in the various load situations.

27. UTILITY

The button “[UTILITY](#)” opens this simple window.



27.1. CONVERSION OF UNIT OF MEASURES to convert lengths.



27.2. STRANDS AREAS TABLE

TABELLA TREFOLI				
TIPO	Diametro mm	Diametro Equivalente mm	Sezione cm²	Peso Kg/m
TRECCIA 3X3	6.5	3.19	0.212	0.167
TREFOLO 3/8 "	9.30	8.14	0.52	0.408
TREFOLO 1/2 "	12.50	10.88	0.93	0.73
TREFOLO 6/10 "	15.2	13.30	1.39	1.09

27.3. REBARS AREAS TABLE

TABELLA AREE FERRI													
Diametro mm	Peso Kg/m	Sezione cm²											
		Numero Barro											
		1	2	3	4	5	6	7	8	9	10	11	12
5	0.154	0.2	0.4	0.6	0.8	1	1.2	1.4	1.6	1.8	2	2.2	2.4
6	0.222	0.28	0.56	0.84	1.12	1.4	1.68	1.96	2.24	2.52	2.8	3.08	3.36
8	0.395	0.5	1	1.5	2	2.5	3	3.5	4	4.5	5	5.5	6
10	0.617	0.78	1.56	2.37	3.16	3.95	4.74	5.53	6.32	7.11	7.9	8.6908	9.48
12	0.888	1.13	2.26	3.39	4.52	5.65	6.78	7.91	9.04	10.17	11.3	12.43	13.56
14	1.209	1.54	3.08	4.62	6.16	7.7	9.24	10.78	12.32	13.86	15.4	16.94	18.48
16	1.578	2.01	4.02	6.03	8.04	10.05	12.06	14.07	16.08	18.09	20.1	22.11	24.12
18	1.998	2.54	5.08	7.62	10.16	12.7	15.24	17.78	20.32	22.86	25.4	27.94	30.48
20	2.466	3.14	6.28	9.42	12.56	15.7	18.84	21.98	25.12	28.26	31.4	34.54	37.68
22	2.984	3.8	7.6	11.4	15.2	19	22.8	26.6	30.4	34.2	38	41.8	45.6
25	3.854	4.91	9.82	14.73	19.64	24.55	29.46	34.37	39.28	44.19	49.1	54.01	58.92
26	4.168	5.31	10.62	15.93	21.24	26.55	31.86	37.17	42.48	47.79	53.1	58.41	63.72
28	4.834	6.16	12.32	18.48	24.64	30.8	36.96	43.12	49.28	55.44	61.6	67.758	73.92
30	5.545	7.07	14.14	21.21	28.28	35.35	42.42	49.49	56.56	63.63	70.7	77.77	84.84
32	6.314	8.04	16.08	24.12	32.16	40.2	48.24	56.28	64.32	72.36	80.4	88.44	96.48
34	7.128	9.08	18.16	27.24	36.32	45.4	54.48	63.56	72.64	81.72	90.8	99.88	108.96

27.4. CALCULATION OF REBARS AREAS

To calculate the area of one or more rebar, to sum them and to calculate the area of rebars / m.

CALCOLO FERRI - AREE FERRI	
Area ferro N. <input type="text" value="1"/> ϕ <input type="text" value="10"/> = <input type="text" value="0.79"/> cm²	Peso L (cm) <input type="text" value="0"/> → <input type="text" value="0"/> kg
Somma ferri (N. <input type="text" value="1"/> ϕ <input type="text" value="10"/>) + (N. <input type="text" value="1"/> ϕ <input type="text" value="10"/>) = <input type="text" value="1.58"/> cm²	Peso L (cm) <input type="text" value="0"/> → <input type="text" value="0"/> kg
Area ferri / m N. <input type="text" value="1"/> ϕ <input type="text" value="10"/> / <input type="text" value="20"/> = <input type="text" value="3.93"/> cm²/m	Peso L (cm) <input type="text" value="0"/> → <input type="text" value="0"/> kg/m
Somma area ferri / m (N. <input type="text" value="1"/> ϕ <input type="text" value="10"/> / <input type="text" value="20"/>) + (N. <input type="text" value="1"/> ϕ <input type="text" value="10"/> / <input type="text" value="20"/>) = <input type="text" value="7.85"/> cm²/m	Peso L (cm) <input type="text" value="0"/> → <input type="text" value="0"/> kg/m
<input type="button" value="TABELLA AREE FERRI"/>	

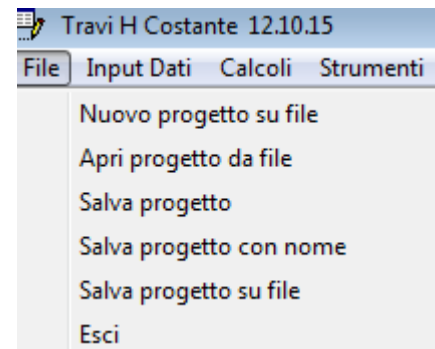
28. MENU BAR

The menu bar gives you direct access to the various windows or program features.

28.1. File

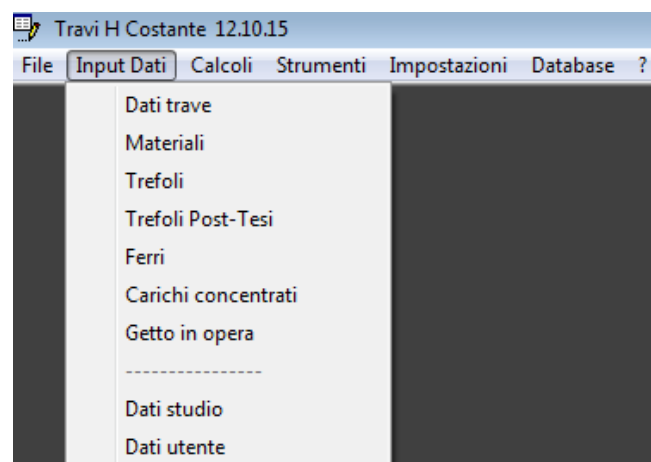
The "**File**" menu allows you to do operations on individual projects saved in text format, which are then easily transferable to other PCs.

"**Save project on file**" allows you to save on text file a project created on database.



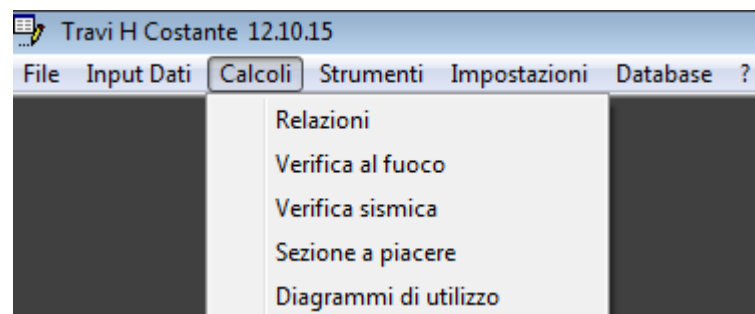
28.2. Input Data

The "Input Data" menu allows access to the various window to change the input data.



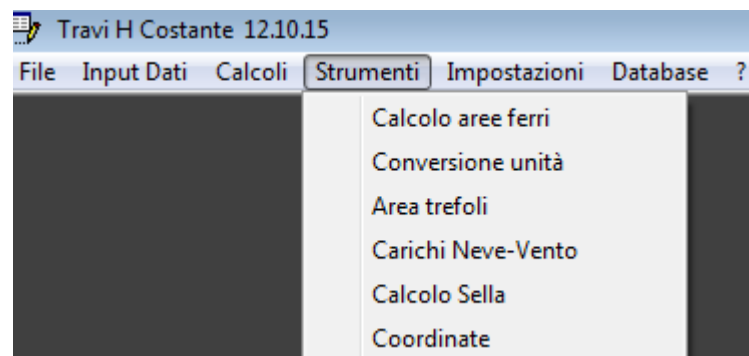
28.3. Calculations

The "Calculations" menu allows you to perform various types of calculations.



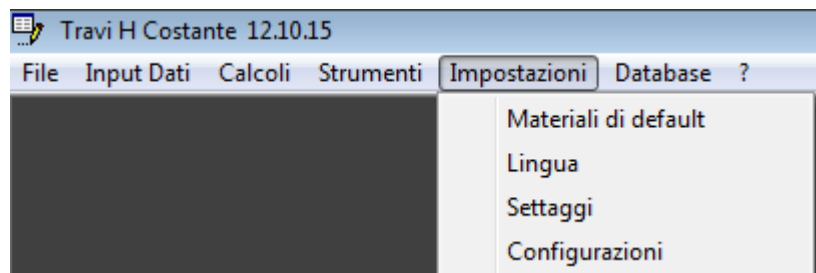
28.4. Tools

The "**Tools**" menu shows the same commands of the "Utility", the window for the calculation of the Cut, the Snow - Wind Loads and Coordinate window.

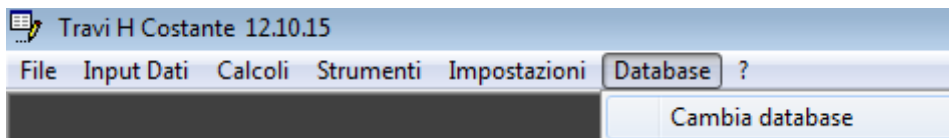


28.5. Settings

The "**Settings**" menu allows you to change the default data (materials and settings), and program settings, such as language (Italian, English and Spanish available) and configurations (whether to automatically or manually search for updates).



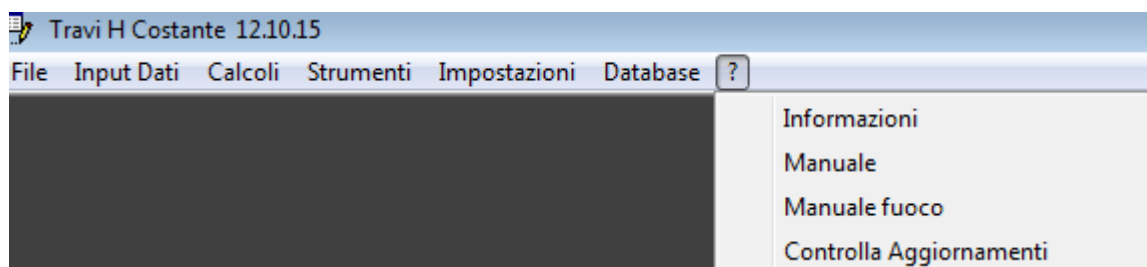
28.6. Cambia database



The "**Change database**" menu allows you to change the database.

28.7. ?

The "?" Menu allows you to see the information (program version and database you're using), view manuals and check for updates manually.



29. DATA MANAGEMENT

You can work on text files or databases. It should be used the same method for the calculation program and the editor of the section geometry " H constant Geometries " associated.

You starts by saving your own sections with the program of the Geometries. After you create a section you can save it in the database (default function) or you can save it to a text file.

If you save on database you have all the beams (created with the program Geometry) and projects (created with the calculation program) stored in the same file, which is managed automatically by the program and you don't have to worry about where to save it (you can see name and path of the file in the "Info" menu). The list could get very long. Anyway you can do all the operations such as save multiple databases, use network database, change database and so on.

If you save to text file you can save the file where you want to, in the local PC or network (for example in the job directory). When you create a new project file with the calculation program, you'll have to manually select the text file of the section (created with the Geometry module, you have to browse to the directory where you saved).

If you saved the projects on database, you can open them by selecting "**PROJECTS ON DATABASE**" (you will see a window with the list of projects and a preview of it when you select it), if you saved them on file will select "**OPEN FROM FILE**" and then browse to select them in the correct directory (the program opens the classic Windows form to open files).

To delete a project in the database, select it from the list and then press "**Delete project**", in the "Project Manager" window; to delete a project saved on file, simply delete the file.

29.1. SAVE - OPEN – DELETE projects

29.1.1. Save

FILE (text file .TXT) you proceed as for many other commonly used programs (MS Word®, MS Excel® ...): you can save the files wherever you want, on your local computer or network, and when you open a saved file you will need to manually select it from the directory where it is located.

In this case, from the main window, you create new projects or open existing projects with the respective buttons:



When you load the project the button **SAVE** will be enabled.

DATABASE: you have all projects and all sections stored in one database, which is managed automatically by the program, and you should not worry about where you save the files (you can see the name and path of the database in the menu "?" -"Information"). You can do all the operations of multiple databases, use of network database, database change. To work on the

database from the main window you choose "[PROJECTS ON DATABASE](#)" and automatically manage the projects from the Projects Manager form (extensively described previously 9.3).



29.1.1.OPEN

FILE Click "**Open project from file**" and select the project TXT file. The geometry is saved with the project, so it is not necessary to select the geometry files. If you want to change the geometry of the beam associated with new or update existing data modified with geometries form, open the "BEAM DATA" window and select another beam as explained in Chapter [BEAM DATA](#).

DATABASE Click " [DATABASE PROJECTS](#)" and double-click the project in the list.

29.1.1.DELETE

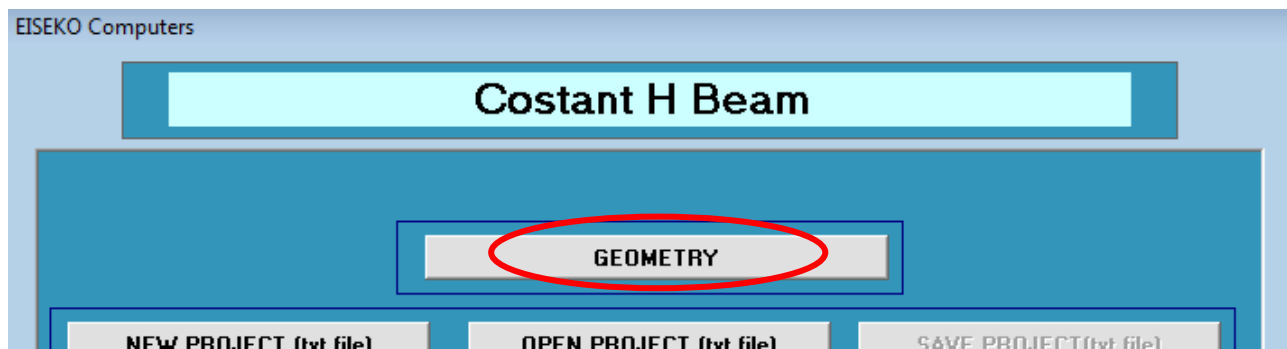
FILE delete the file (Windows Explorer).

DATABASE Click "[DATABASE PROJECTS](#)" and select the project to be deleted from the list, then press the button "[DELETE PROJECT](#)".

To delete a beam from database use the editor "Geometries": from the "File" menu, select "Delete Beam".

**IN BOTH CASES PLEASE NOTE THAT IT IS GOOD
HABIT TO ENSURE A REGULAR BACKUP DATA.**

30. "GEOMETRY" EDITOR



30.1. What is it for:

The program allows you to introduce the geometry of the beams that will be calculated with the associated program "Calculation Beams constant H".

You must first insert a midspan section, which defines the shape of the beam. Then you can insert a head section, if necessary, for example if the beam is thickened at the ends.

NOTES: no need for any cut such as the Gerber support, which are treated separately.

Then insert the strand positions (all the positions of the mould), and if you want you can add some real strands, inserting them with an area >0, and then insert some typical rebars for that beam (necessary).

You first create the beam with the "Geometries" editor, then you do the calculation by creating a new project from the calculation program, selecting the beam that you created.

30.2. FILE MANAGEMENT

Even in this Module, as in the Calculation one, it is possible to manage the files of the beam sections with text file or database (automatic control). **NOTES: you must use the same method for both programs.**

If you work on the database, you have all the sections and projects created with the program of calculation and geometry within the same file, which is managed automatically by the program and not have to worry about where to save it (you can see the name and path of the file in the "[Information](#)" menu).

If you change a section from the editor, and there are already existing projects using this section, you must click the button "UPDATE BEAM" in the Projects Manager window to update the section with the changes. Otherwise, the beam of the project will remain with the previous section data (like it was at the time of the creation of the project , not updated to the latest changes). The project will remain intact with its values (Materials, BEAMS DATE etc.), the program will only update the geometry of the section (points, location of rebars / strands areas).

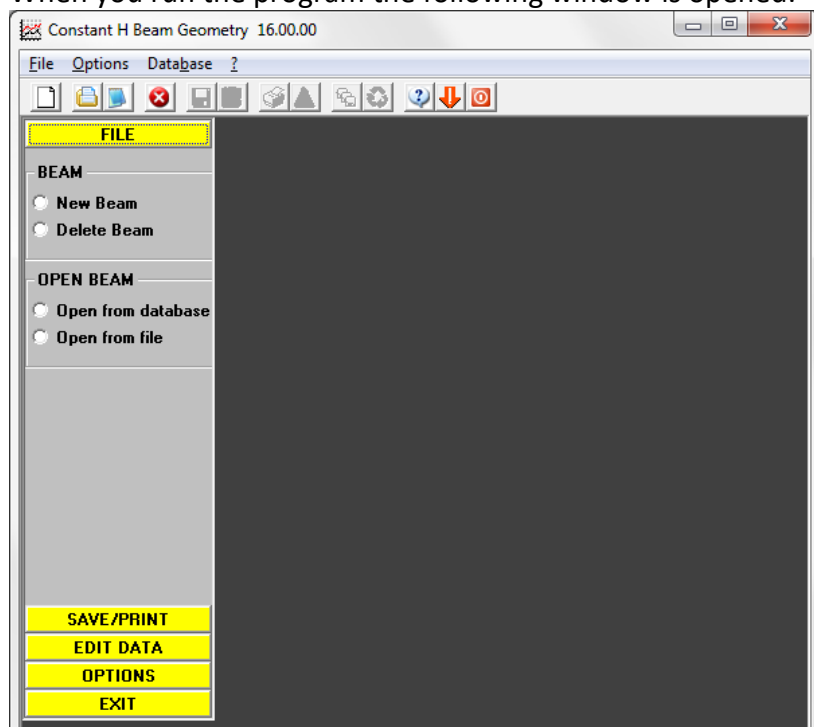
If you work on files with the program of calculation you have to work on files even with this Geometry editor, with which you will create the sections of text files. Then, when you create a project, you will have to manually select the files to be associated with the beam from the directory where you saved it.

In the Menu of Geometries Editor: if you saved the beams in the database, you can reopen them by selecting "Open From List" (you will see a form with the list of beams and a preview of it when I click on it), if you saved you will select the file "Open from file" and you're going to select the correct location (opens the classic Windows® form to open files).

IF YOU SAVE A BEAM WHICH DOES NOT HAVE ALL THE NECESSARY DATA, LIKE REBARS OR STRANDS, THE BEAM CAN BE OPENED BY THE GEOMETRY MODULE TO FINISH DATA INPUT, BUT IT WILL NOT BE AVAILABLE BY THE CALCULATION MODULE TO CREATE PROJECTS.

30.3. GEOMETRY EDITOR FEATURES

When you run the program the following window is opened:



At first, you should insert a midspan section, which defines the shape of the beam. then you can insert a head section, if necessary, for example if the beam is thickened at the ends. The length of this portion will be set in the calculation program, because it usually depend on the particular beam you're designing.

NOTES: no need for any cut, which are treated separately in calculation module, see chap. 25.

Then, input the mask of the possible positions of strands (as from your mould) and of the most important rebars (as from experience).

The commands are in the menu, or in the vertical bar or in the top bar, see dedicated sections.

In the menu "**File**" we can start with "New section", or open an existing geometry ("Open From List" or "Open from file" depending on whether you work on databases or files), delete a section "Delete section" , or save the one on which you are working with the same name or a different name.

When you create a new section, a wizard will guide you step by step through all the process. We see this process in detail.

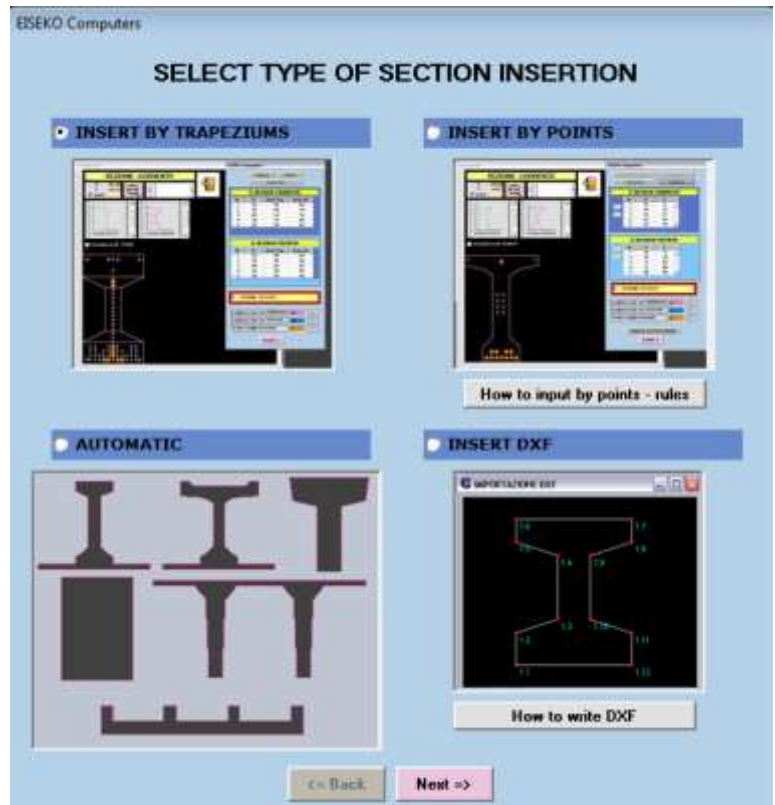
30.4. INPUT OF NEW SECTIONS

Input of the section

At first, you should insert the midspan section choosing between the following methods (we will analyse each method in detail):

- Section by trapeziums
- Section by points
- Parametric sections
- Sections by DXF files

The two buttons "How to input by points - rules" and " How to write DXF" are two screens that explain all the rules you have to follow to enter the data.



30.4.1. SECTION BY TRAPEZIUMS

The following main screen opens with a graphic black area and a tools window on the right.



Add the data of trapeziums in the table on the right, starting from the HIGHER trapezium and proceeding downward. to facilitate the data input, the program will take the last written trapezium lower base and put it as upper base of the next trapezium.

To move from one textbox to the other, press Enter.

The graphic area on the left is dynamically updated.

To change a value, just click in the box, enter the new data and **press Enter.**

If the last line contains empty data is automatically deleted by the program.

Button "**Add**": Allows you to add a trapezium among the others (add a line). Select the **previous** trapezium to the trapezium to be added: it will be inserted after the selected one. **NOTE:** To add a trapezium at the bottom of the table it is not necessary to press the "Add" button, just press Enter on the last row, and the program will create the new row and the cursor will directly be on the first column.

Button "**Delete**": delete a trapezium (a table row). First select the trapezium to be deleted and then click the button.

Button "**set data to ZERO**" is used to cancel all the data of all the trapeziums (set 0 to all the columns of all the rows in the table), without erasing the lines. **It's not possible to undo this command.**

EISEKO Computers

Add Delete

Set data to zero

1) SECTION C = MIDSPAN

N.	H	Sup. side:	Inf. side
1	20	54	54
2	10	5	
3	40	1	
4	10	1	
5	20	5	

3) SECTION A = HEAD

N.	H	Sup. side:	Inf. side
1	20	54	54
2	10	54	54
3	40	54	54
4	10	54	54
5	20	54	54

☒ HEAD SECTION

Midspan sect. min width 14 cm

Head sec. min width 54 cm

Min H head sect.for shear 100 cm

☐ Use section by points for the Fire Verification POINTS

Next =>

Midspan sect. min width	14	cm
Head sec. min width	54	cm

Enter the minimum midspan section WIDTH. This value is used to calculate the minimum stirrups area. If there is even the head section, it is also required

for the latter. In the calculation program this data will be taken as default value for new projects created using this section, and will then can be edited for single projects.

Min H head sect.for shear	100	cm
---------------------------	-----	----

Minimum height of the section to calculate the Shear at support. The program automatically

calculates the sum of all the trapeziums, but it is possible to enter a lower value.

<input type="checkbox"/> VIEW TRAP. #

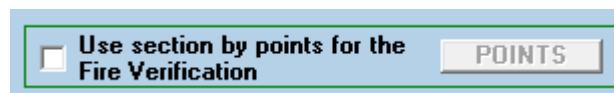
Check "VIEW TRAP. #" in the graphic area to see the number of EACH trapeziums of the section in the graphic area.

<input type="checkbox"/> HEAD SECTION

Once the midspan section is complete you can move on to the head section: Check "**HEAD SECTION**" and a second table with the same data as the first one

appears, for easy editing.

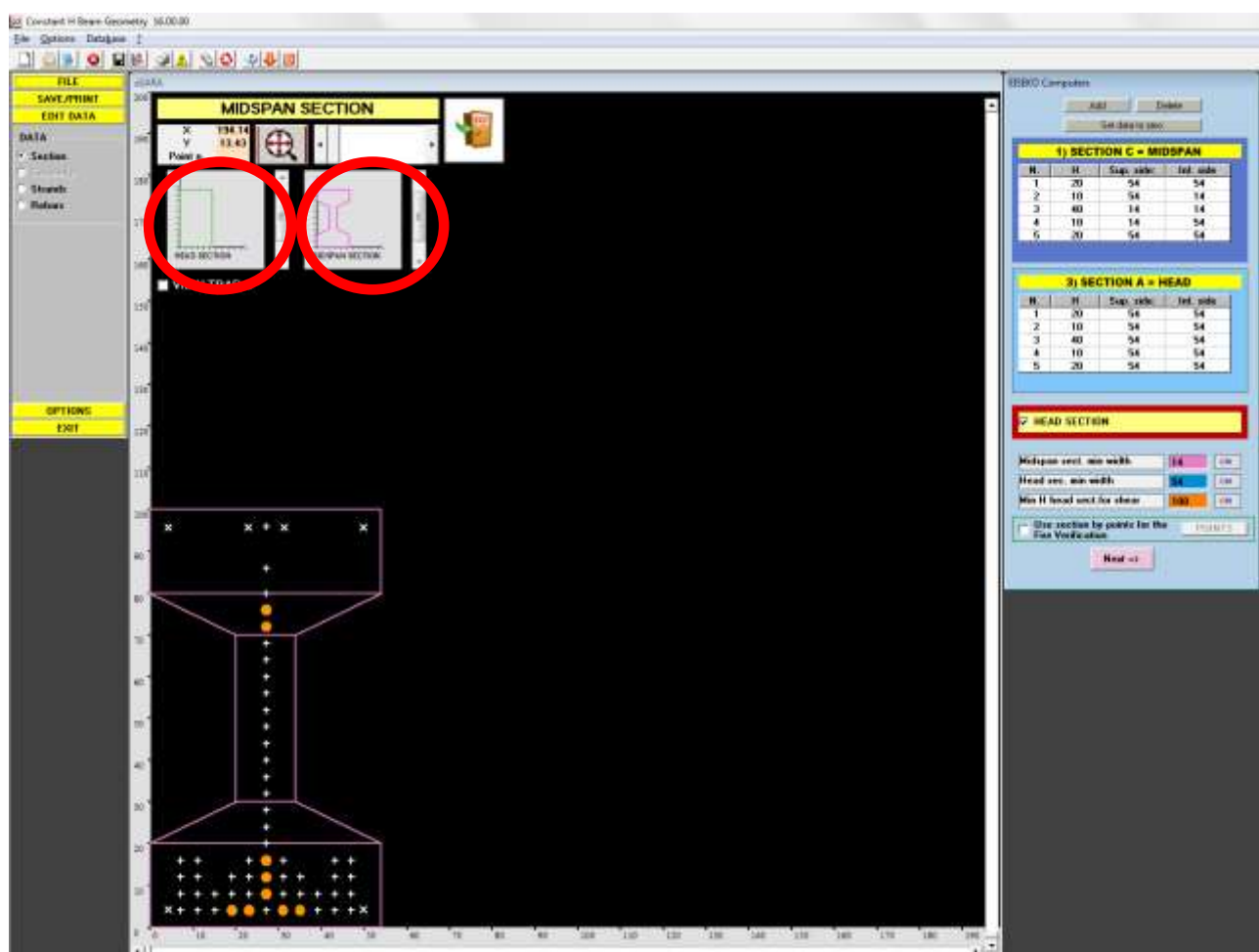
To switch from one section to the another just click on the corresponding table: the displayed section is highlighted in blue.



If you realize sections that entered by trapeziums are not equal to the real ones (for example a TT), it is possible to introduce a section by points, which will be displayed in the graphics of the calculation module and used for fire verification. For other calculations it will always be used for the section introduced by trapeziums. The section should be inserted with the same rules and methods of sections for points, see chapter 30.4.2.

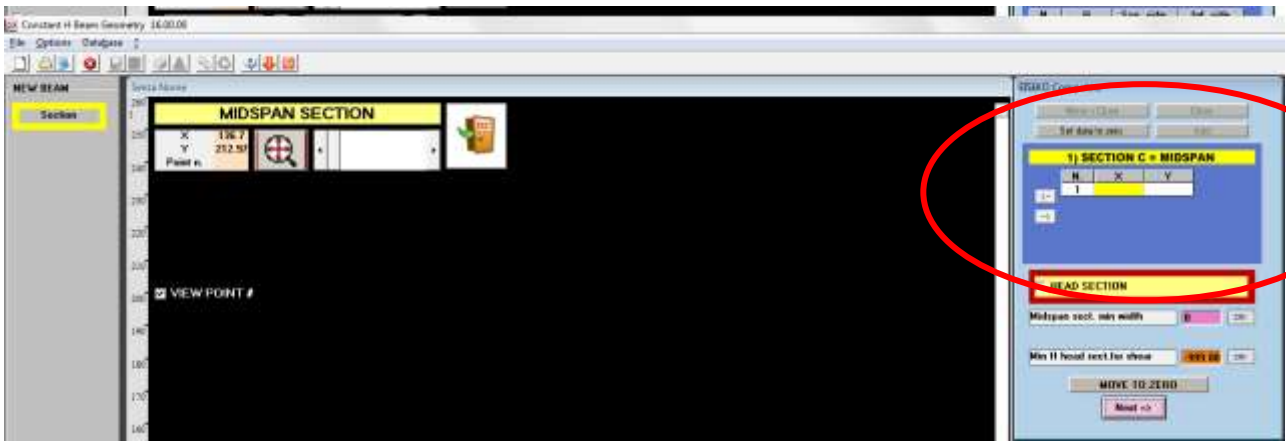
The drawing in the graphic area is updated automatically: midspan section is pink, head section is green.

To switch from one section to the another you can also click on the corresponding drawing top in the graphic area.



30.4.2. SECTION BY POINTS

The following main screen opens with a graphic black area and a tools window on the right.



Enter the coordinates of the points of the section in the table on the right.

POINTS SHOULD BE ENTERED BY PROCEEDING CLOCKWISE.

The section should be closed: the starting and ending points must be coincident.

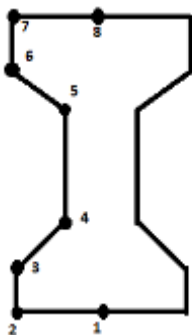
To move from one textbox to the other, press Enter.

The graphic area on the left is dynamically updated.

To change a value, just click in the box, enter the new data and **press Enter.**

You can enter all the points and then click the button "**Close**" (the program will add the last point coincident with the first one) or enter directly all the points and also the end, coincident with the first one, and the program understands it's finished.

The points can also be entered by clicking with the mouse in the graphics area (top left the mouse coordinates) are indicated.



We recommend you to start from the point at the bottom centre as in the figure on the left, and get to the point in the top centre, and then click the button "Mirror + Close": the program automatically will place all points of the right part and close the beam.

Once the midspan section is complete you can move on to the head section: check "**HEAD SECTION**" and a second table appears, with the same data as midspan section for easy editing. Modify the data that have been inserted automatically.

EISEKO Computers

Specchia + Chiudi Chiudi

Azzera Dati Aggiungi

1) SEZIONE CORRENTE

N.	X	Y
1	0	0
2	0	100
3	50	100
4	50	0
5	0	0

<- -->

☐ SEZIONE TESTATA

Larghezza min sez. CORRENTE cm

H min a taglio in testata cm

TRASLA TUTTO A ZERO

Avanti =>



EISEKO Computers

Specchia + Chiudi Chiudi

Azzera Dati Aggiungi

1) SEZIONE CORRENTE

N.	X	Y
1	0	0
2	0	100
3	50	100
4	50	0
5	0	0

<- -->

2) SEZIONE TESTATA

N.	X	Y
1	0	0
2	0	100
3	50	100
4	50	0
5	0	0

<- -->

☒ SEZIONE TESTATA

Larghezza min sez. CORRENTE cm

Larghezza min sez. TESTATA cm

H min a taglio in testata cm

TRASLA TUTTO A ZERO

Avanti =>

To switch from one section to the another just click on the corresponding table: the displayed section is highlighted in blue.

The drawing in the graphic area is updated automatically: midspan section is **pink**, head section is **green**.

To switch from one section to the another you can also click on the corresponding drawing top in the graphic area.



The **arrows** next to the coordinates of the table allow you to delete or add points to the table: back arrow = take off the last point; forward arrow = re-enter the last point taken off.

Button "**Add**": Allows you to add a point between others (add a line). Select the previous point to the point to be added: the point will be inserted after the selected point. **Note** to add a point at the bottom of the table it is not necessary to press the "Add" button, just press Enter on the last row, and the program will create the new row and the cursor will directly be on the first column. **It's not possible to undo this command.**

Button "**set data to ZERO**" is used to cancel all the data of all the points (set 0 to all the columns of all the rows in the table), without erasing the lines. **It's not possible to undo this command.**

Button “**MOVE TO ZERO**”: It allows you to move the section so that the point more on the left has X coordinates=0. **It's not possible to undo this command.**

Midspan sect. min width	14	cm
Head sec. min width	54	cm

Enter the minimum midspan section WIDTH. This value is used to calculate the minimum stirrups area. If there is even the head section, it is also required

for the latter. In the calculation program this data will be taken as default value for new projects created using this section, and will then can be edited for single projects.

Min H head sect.for shear	100	cm
---------------------------	-----	----

Minimum height of the section to calculate the Shear at support. The program automatically

calculates the sum of all the trapeziums, but it is possible to enter a lower value.

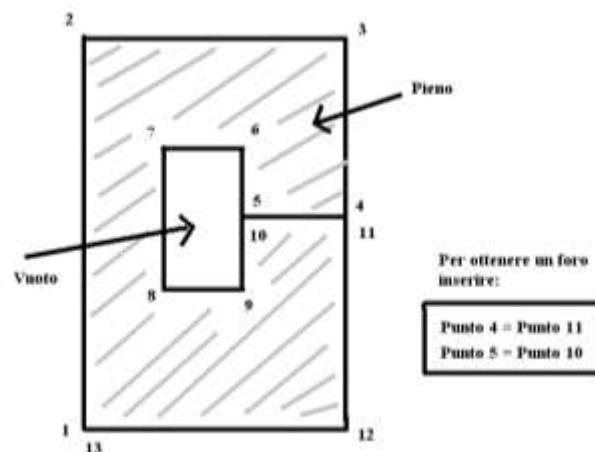
If you have similar sections, we recommend to start from the widest and then change the section editing the coordinates and then using this button. It is not possible to start from the thinner and then enlarge it, because you cannot enter negative numbers and then move it.

HOLES

You can also get internal holes entering the profile of the holes proceeding counterclockwise (section will be negative, and then removed).

The holes should be joined to the outer profile by two coincident lines.

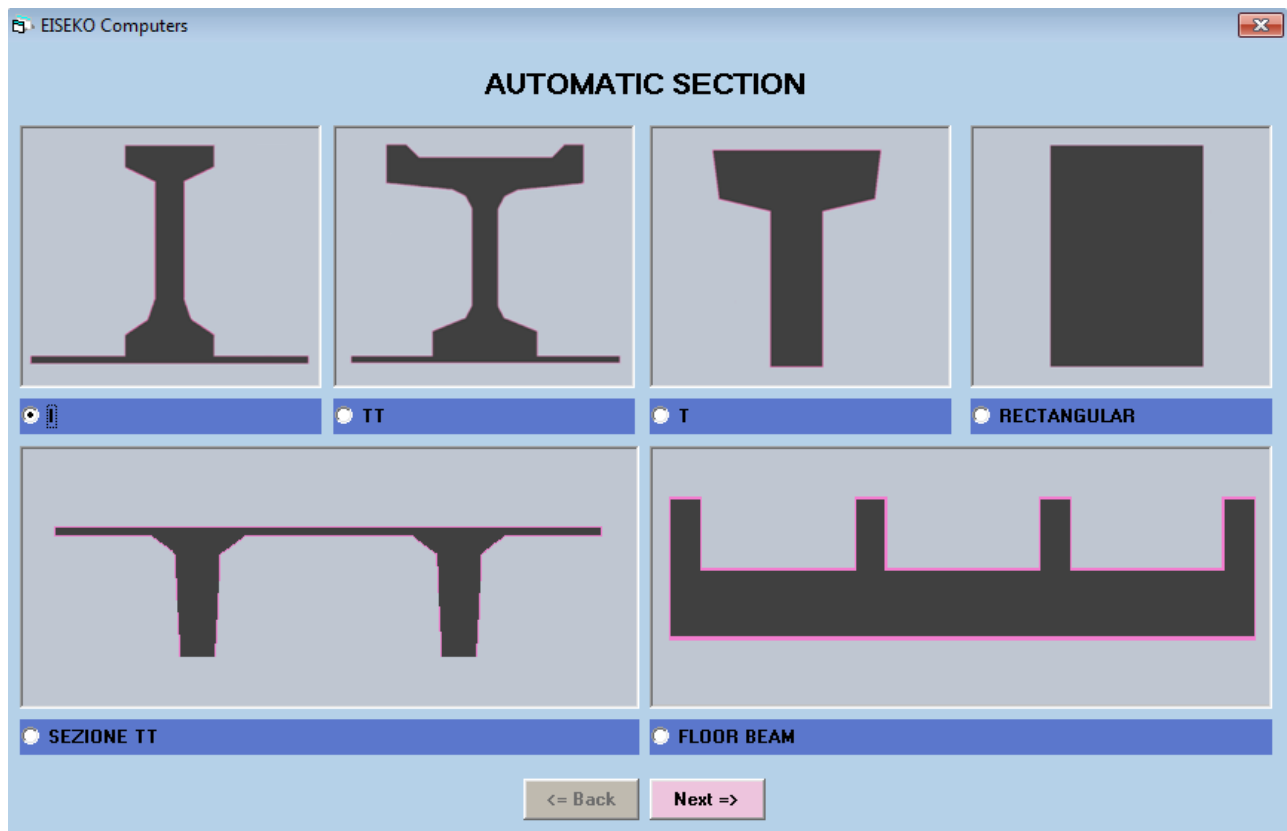
NOTE: to enter sections with holes you can use only one of the methods “section by points” or “section by DXF” (which follows the same rules).



30.4.3. PARAMETRIC SECTION

You can enter some common sections entering the measures as dimensions, rather than enter any points or trapeziums. The available parametric sections are:

- I (with eventual inferior slab)
- Complex I : with eventual inferior slab and more tapers.
- T
- Rectangular
- TT
- Floor Section

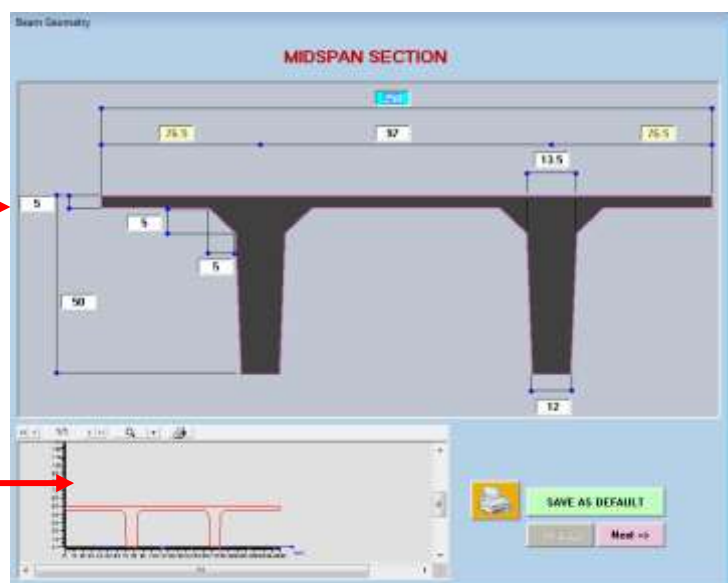


We choose as an example the TT (you can also double click on the image to select):

Enter the dimensions of your section and press "Next->".

ENTER DIMENSIONS

GRAPHIC DISPLAY OF DYNAMIC RESULT BASED ON DIMENSIONS



Button "**SAVE AS DEFAULT**": allows you to save the section data as defaults. When you create a new section, the program will propose automatically the default data for the selected section, then you can edit them.

Once the midspan section is complete, you switch to the window below:



Now you can switch to the head section: check "**HEAD SECTION**" and click the button "**EDIT SELECTED SECTION(head)**" [For example in TT slabs will not be necessary]. It goes back to previous window, where it is possible to give the dimensions of the head section (starting from the values entered for midspan for simplicity).
NOTE: Check that you have selected the correct section (the one selected is blue).

NOTE: Check that you have selected the correct section (the one selected is blue).

CLICK ON THE LIGHT BLUE BOX TO SELECT THE SECTION.

THE SELECTED SECTION IS HIGHLIGHTED BLUE.

EDIT SELECTED SECTION (MIDSPAN) -(HEAD): OPENS THE WINDOW TO EDIT THE DIMENSIONS

In case the head section is not specified, it will be considered equal to the midspan section.

30.4.4. SECTION BY DXF

The DXF files should be created with these features:

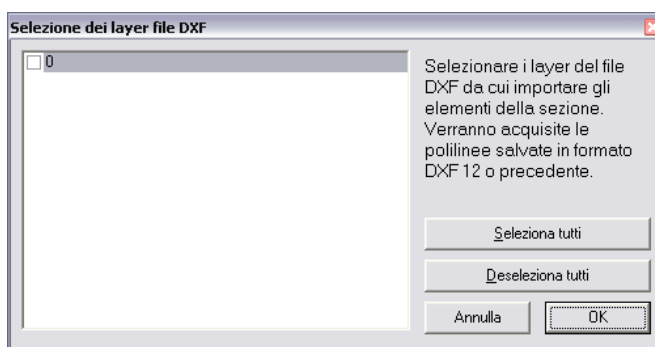
- Midspan section should be drawn with one single polyline, with the points generated in a clockwise direction (as for the insertion of the section by points: counterclockwise only for any holes, always connected to the external profile by two coincident lines)
- The section in DXF should be in cm and with a scale of 1: 1.
- Polylines should be formed by lines, not arcs: it is necessary to discretize the section.

We recommend you to open the file containing the section, delete everything else (dimensions, texts etc...), create a new polyline tracing the profile (discretizing any arc), up to close it ending with a point coinciding with the first. DO NOT use the CAD close command: If necessary you can create only half section, and then use the command Mirror within the program. In this case the first and the last point must be on the axis of mirroring. Copy the new polyline only, create a new dxf and paste the polyline. Also clean the DXF with the command "PURGE" or "EN" if necessary. Save the new DXF and use it to insert the section.

If the section is not closed, the program asks if you close it automatically or not when reading the DXF.

Once you have chosen the insertion of section via DXF, the program asks you to select the DXF file to read the section. Select the file on your computer or network, and press "Open".

In the next dialog it is possible to select the desired layer, or all. Check the layer (in the list on the left), then press "OK".



Now the preview of the selected file appears, with the section(s) within the file:



To use another DXF click the button "[BROWSE DXF](#)", otherwise press "[OK](#)". In the preview, the sections are numbered indicating vertices with a.b = (number of section). (Number of the point): If there are more sections the program will ask you which one you want to save. You can only import one section at a time.

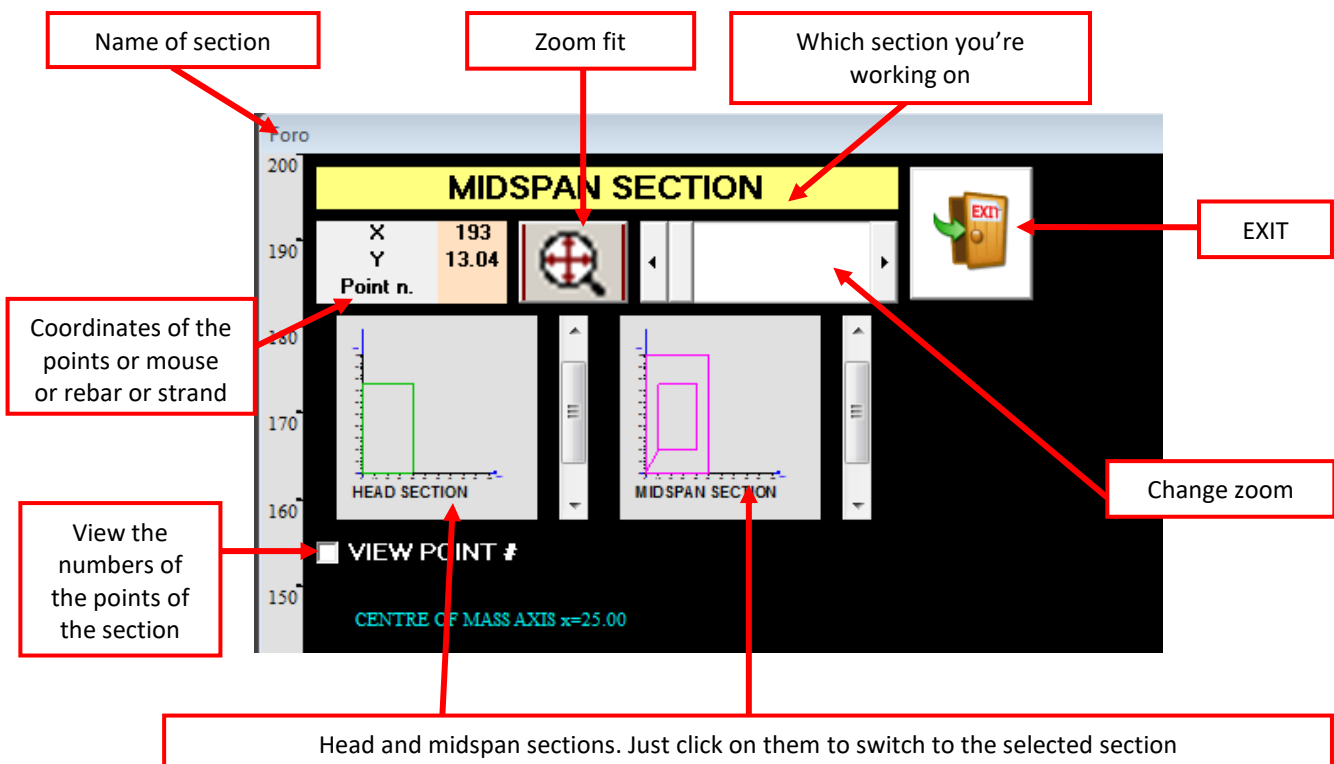
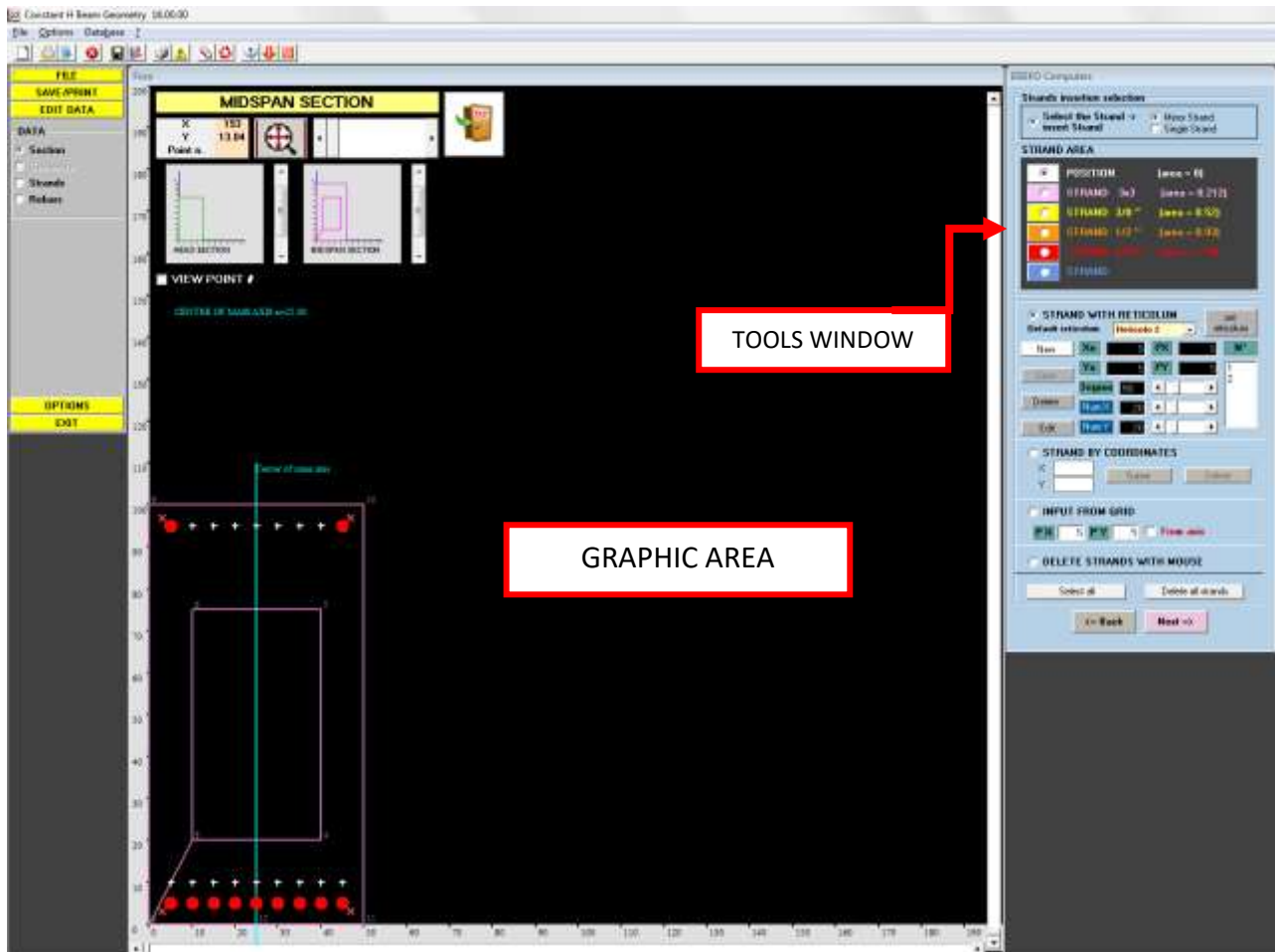
Pressing "[OK](#)" you return to the same window for the introduction of the section by points, and the beam will be treated as those entered by points, *there won't be any link to the DXF file.*

The head section is automatically set equal to this section, then you can edit the points coordinates: click the box of the coordinates of the point to be changed and enter the new values, then press enter.

You cannot enter the head section by DXF.

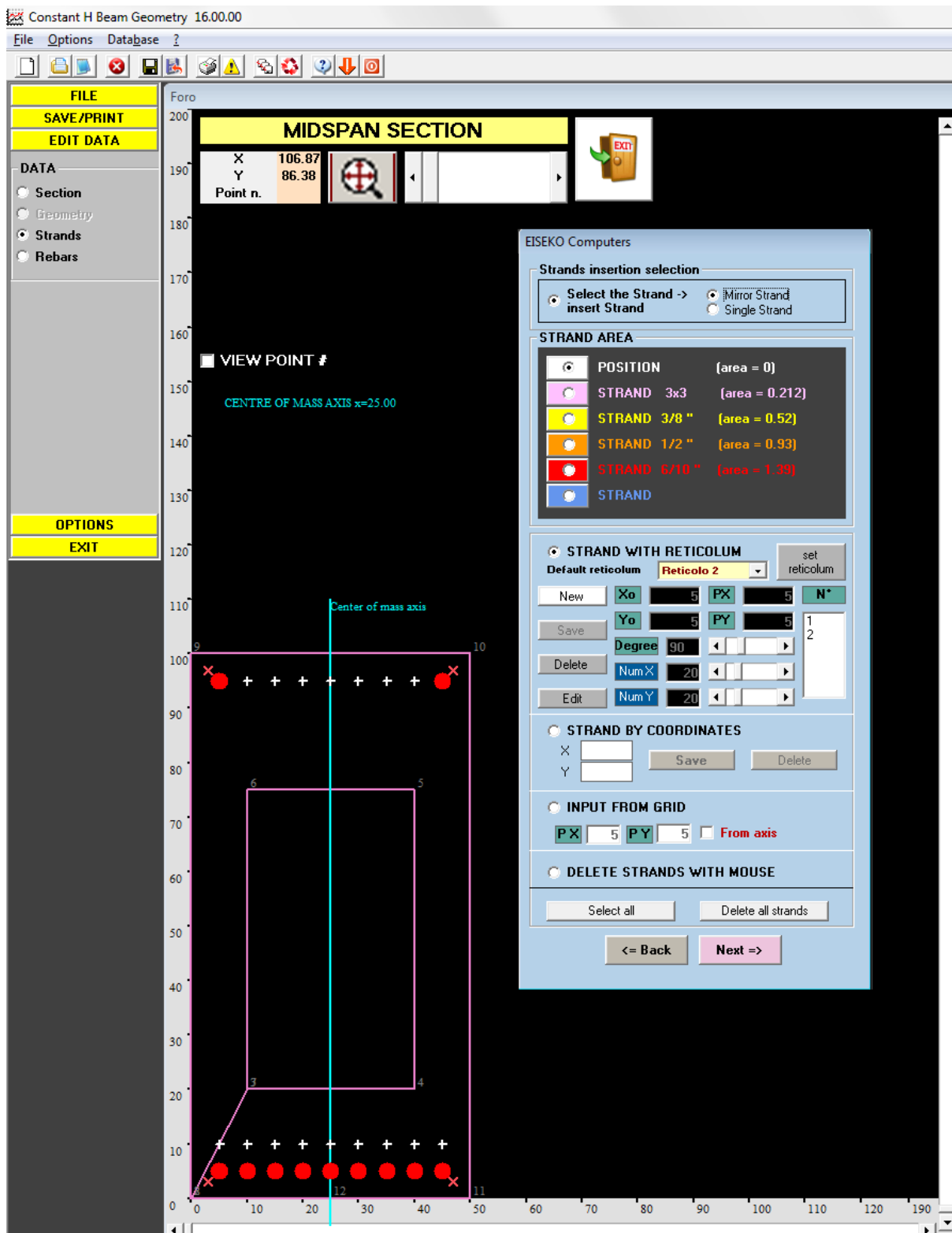
We recommend to create the DXF of the ***left*** half of the section (as in the picture above), and then mirror it with the appropriate command: in this case you are sure to get a symmetrical beam (also make sure with the CAD that the beam is symmetrical).

30.5. GRAPHIC AREA



30.6. STRANDS

Once the introduction of the section is complete, you switch to the window to enter the mask of strands:



You can insert the strands by coordinate, by reticulum or by grid. You can enter only the positions of the strands (area = 0) or the real strands (area > 0). You should always enter the full schema of strands of your mould, with all the possible positions and, if you want, the most used strands with their area. In this way, when you make a new project with this section, you will have the most used strands already inserted (then you can add or remove unnecessary strands directly from the calculation program). The strands here entered will be available for all projects realized with this section, while the strands inserted in the calculation program will be available, however, for the sole project on which you inserted them.

EISEKO Computers

Strands insertion selection

☒ **Select the Strand -> insert Strand** ☒ **Mirror Strand** ☐ **Single Strand**

STRAND AREA

<input checked="" type="radio"/>	POSITION	(area = 0)
<input type="radio"/>	STRAND 3x3	(area = 0.212)
<input type="radio"/>	STRAND 3/8 "	(area = 0.52)
<input type="radio"/>	STRAND 1/2 "	(area = 0.93)
<input type="radio"/>	STRAND 6/10 "	(area = 1.39)
<input type="radio"/>	STRAND	

STRAND WITH RETICULUM ☒ **Default reticulum** **Reticolo 2**

 Xo **PX** **N°**

 Yo **PY**

 Degree

 NumX

 NumY

☐ **STRAND BY COORDINATES**

X

Y

☐ **INPUT FROM GRID**

PX **PY** ☐ **From axis**

☐ **DELETE STRANDS WITH MOUSE**

Callout boxes:

- Enter a single strand of a couple, the second is the mirrored one, respect to gravity axis
- Select the area of the strand to enter (=0 means set only a position)
- Strand with non-standard area (from input): select this option to enter the input area
- Enter strands using a reticulum
- Enter strands by coordinates
- Enter strands by grid. Default grid starts from 0, if you want a specular grid from gravity axis check "From axis"
- Back to section input or forward to rebars input

1. Select if you want to enter each single strand (Single Strand option) or if you want to enter the mirrored strand all at once (Mirror strand option). With the latter, you just enter half of the mask and the program dynamically mirror it.
2. Select the area of the strand you want.
3. Select the type of insertion method (reticulum, coordinates or grid).

We analyse the three cases in detail.

INSERTING STRAND BY RETICULUM

Check the "**STRAND WITH RETICULUM**" option, select the default reticulum to use (see Chapter 30.8): the program automatically proposes data from the default reticulum selected from the list, and the user can modify this data.

Otherwise, you can press "**New**" to create a new reticulum:

X_0 , Y_0 are the coordinates of the starting point of the reticulum;

Degrees is the angle of inclination from the horizontal (press the arrows to change);

Num X / Y are the number of steps in X and Y of the grid (press the arrows to change);

PX, *PY* are the spacings in X and Y.

The grid is updated in the graphics as you change the data.

If you click in the graphics area, the origin of the grid will be the mouse clicked point.

Press "**Save**" to save the reticulum: the number of reticulum is inserted in the list of reticulums on the right (under "N").

You can store up to 17 reticulums.

The button "**Set reticulum**" allows you to return to the window to create / modify the default reticulum (SETTINGS).

Once the reticulum is saved, select it from the list and insert the strands with your mouse in the graphics: if you click on a node, the strand is added there (if you have only selected the location, there will be a cross, or a dot with colour corresponding to the area). **If you click on an existing strand, the strand will be removed.**

The button "**Delete**" deletes the selected reticulum in the list, with all the strands in it.

NOTES: The grid is used only to define the positions of the strands without worrying about the corresponding coordinates. It will be the program to set for the calculation module the X, Y.

NOTES: When you enter multiple reticulums, you should first enter the reticulum, then the strands in it, and THEN go to the next reticulum. You cannot insert all the reticulums and then all the strands.

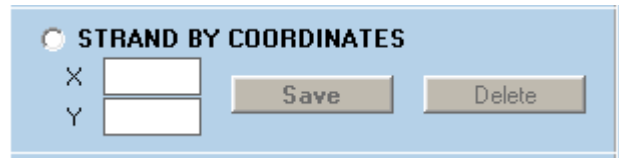
You can **change** an already inserted reticulum:

- Select the reticulum from the list
- Change data
- Click on "[Edit](#)"

All the strands placed in that reticulum will be automatically moved according to the new data.

INSERTING STRAND BY COORDINATES

Select "[STRAND BY COORDINATES](#)", enter the coordinates X and Y and press "[SAVE](#)" to save or "[Delete](#)" to delete the strand. Area should be already selected.



INSERTING STRAND BY GRID



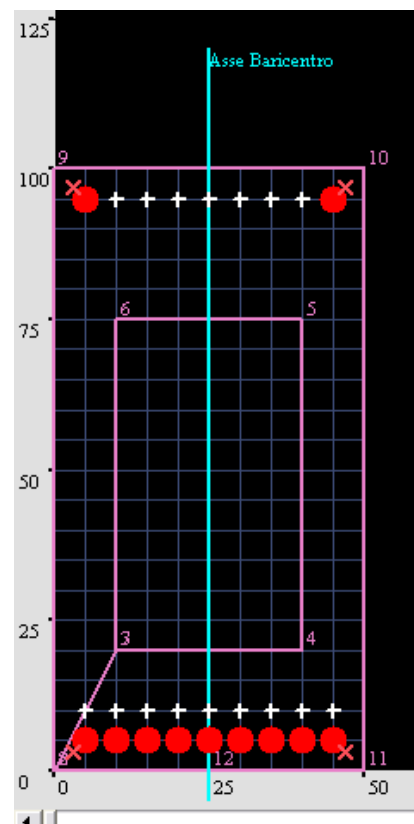
Select the option "[Input by grid](#)", the program proposes automatically data from the default settings, user can edit them. In the graphic area a grid will appear.

Check "[From axis](#)" to start the grid from the centre of gravity of the beam (symmetrically). To insert the strands just click in the grid nodes. To delete an already inserted strand just click on it.

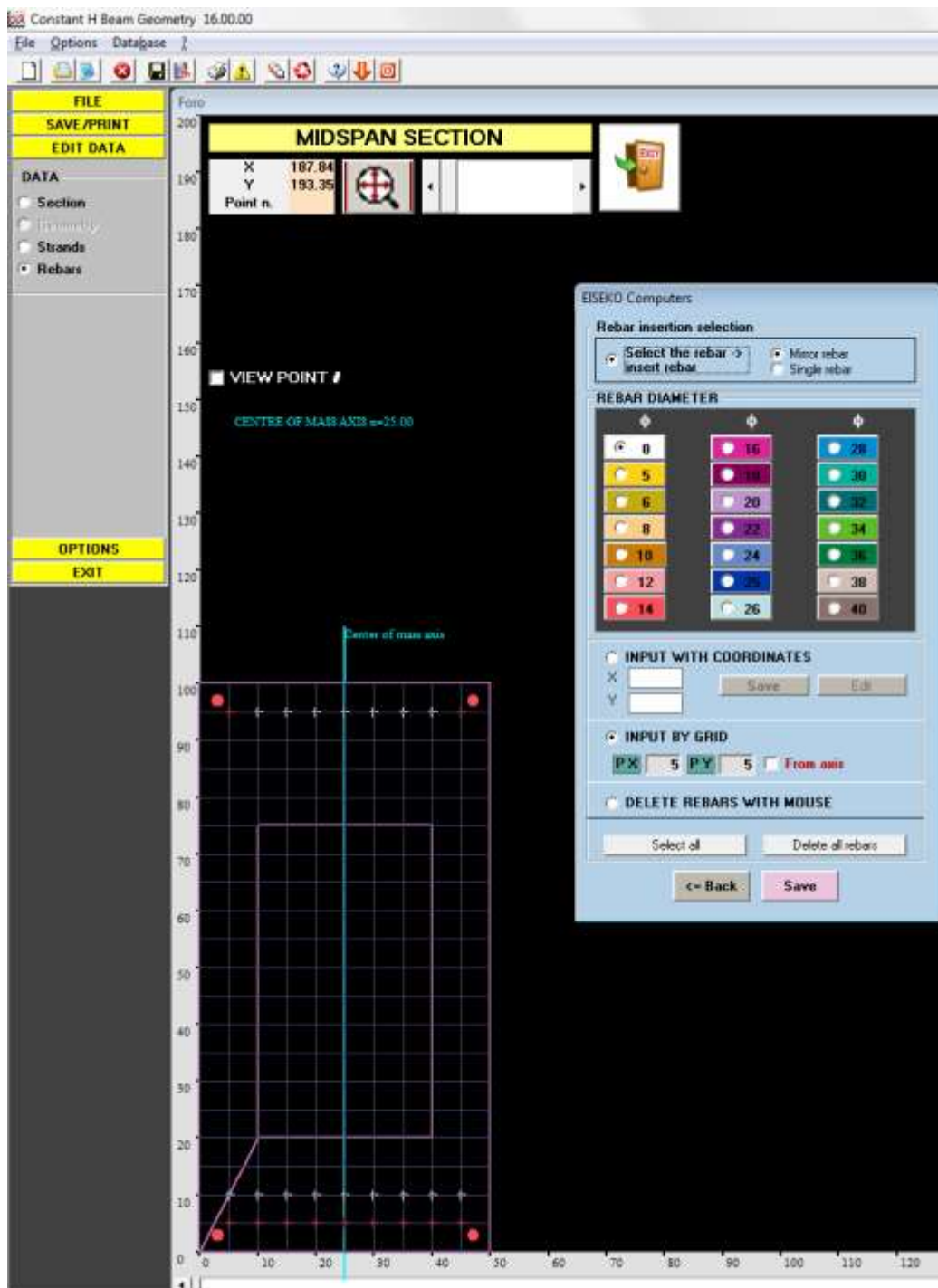
Button "[Select All](#)" allows you to select all the strands and, for example, change of all area by pressing the button of the desired area.

Button "[Delete all](#)" strands eliminates all the inserted strands (removes not only the area but also the positions). It is not possible to undo the command.

See chap. 15 for other details.



30.7. REBARS



You can enter the rebars by coordinates or by a grid. You can enter real rebars (area > 0) or only the positions (area = 0). We recommend to insert the more locations possible and the most used rebars with their area, so when you make a new project of this beam, you will have more rebars used already entered and then you can add or remove unnecessary directly from the calculation program. It's faster.

Then in the calculation program you can also enter new positions, available only for the project you are working on.

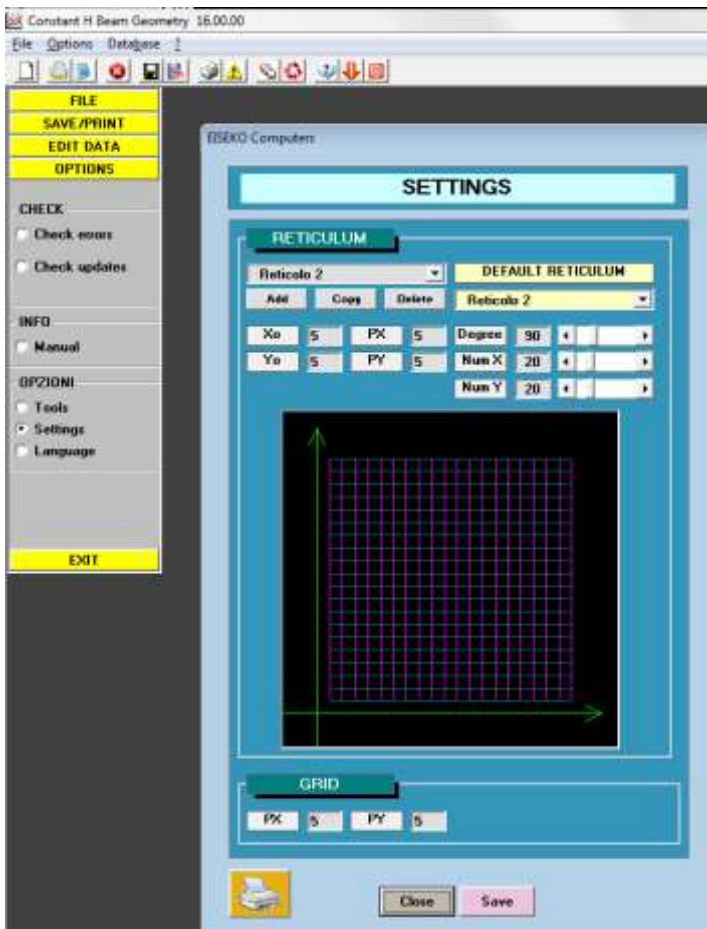
1. **Inserting by coordinates** Select this option, enter the X and Y coordinates of the rebar and press "**Save**": the rebar will be displayed in the graphics area with a colour dot corresponding to the choice, or with a white cross if he has an area = 0.
2. **Inserting by grid** Select this option and change if necessary the steps of the grid in X (PX) and Y (PY): the graphic area will be updated dynamically. Then select the grid nodes where to place the rebars. The rebar will be displayed with a colour dot corresponding to the selected area. The grid begins from origin 0,0 and covers the whole beam, even if the section has holes. *User should not enter rebars in positions external to the beam or in the holes. The only error messages that appear is whether the x coordinate of the rebar exceeds the maximum width of the beam, or if the y coordinate exceeds the maximum height.*
Select "**From axis**" to start the grid from the centre of gravity of the beam (symmetrically).
3. **Delete rebars with mouse** To delete a rebar just select this option and click on the rebar to remove in the graphic area.
4. **"Select all rebars"** To select all entered rebars (then if you select an area, this will be assigned to all rebars).
5. **"Delete all rebars"** To delete all existing rebars. It is not possible to undo this command.

To assign the area to a rebar select the option of the area before placing the rebar in the graphic area, otherwise select the rebar (by clicking on it with the mouse) and then click the option with the area to be assigned.

Now, the beam has all the data and the Wizard let you save the beam (if you save before it is complete, you can edit the section with the geometry module but you can't use it in the calculation module:



30.8. SETTAGGI



In SETTINGS it is possible to set the default reticulum to use for the new sections [you have to load a section to enable SETTINGS].

Modify the existing reticulum with your own preferences, or create new ones with the button "ADD". The button "COPY" allows you to save a grid with another name and the button "DELETE" deletes unwanted reticulums.

X_0, Y_0 are the coordinates of the starting point of the reticulum;

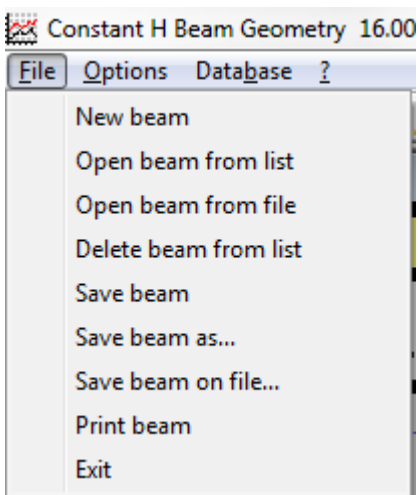
Degrees is the angle of inclination from the horizontal (press the arrows to change);

Num X / Y are the number of steps in X and Y of the grid (press the arrows to change);

PX, PY are the spacings in X and Y.

The graphic area displays the preview.

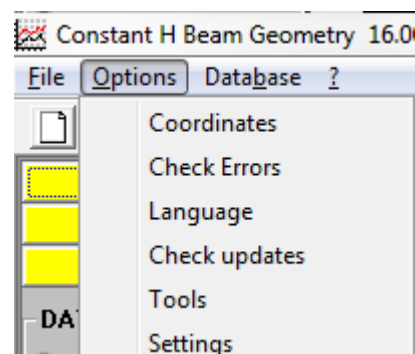
30.9. MENU'



The "File" menu allows you to create a new geometry or open an existing one, delete a section from the list of geometries, and savings.

The "Options menu:

- Coordinates: displays the tables of the coordinates of the section (midspan + head, if available).



EISEKO Computers

SECTION COORDINATES

MIDSPAN SECTION

N	Coord. X (cm)	Coord. Y (cm)
1	25	0
2	0	0
3	0	75
4	40	75
5	40	0
6	25	0

HEAD SECTION

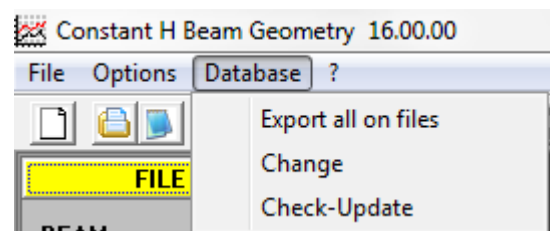
N	Coord. X (cm)	Coord. Y (cm)
1	25	0
2	0	0
3	10	20
4	40	20
5	40	75
6	10	75
7	10	20
8	0	0
9	0	100
10	50	100
11	50	0
12	25	0

OK

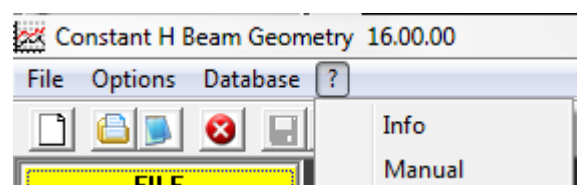
- Check errors: Check if section points actually follow the clockwise direction.
- Language: Choice of Language (Italian, English and Spanish available)
- Check Updates: to check if a program update is available (internet connection should be available).
- Configurations (whether to automatically or manually search for updates.)
- Settings: opens the window to set the default reticulum

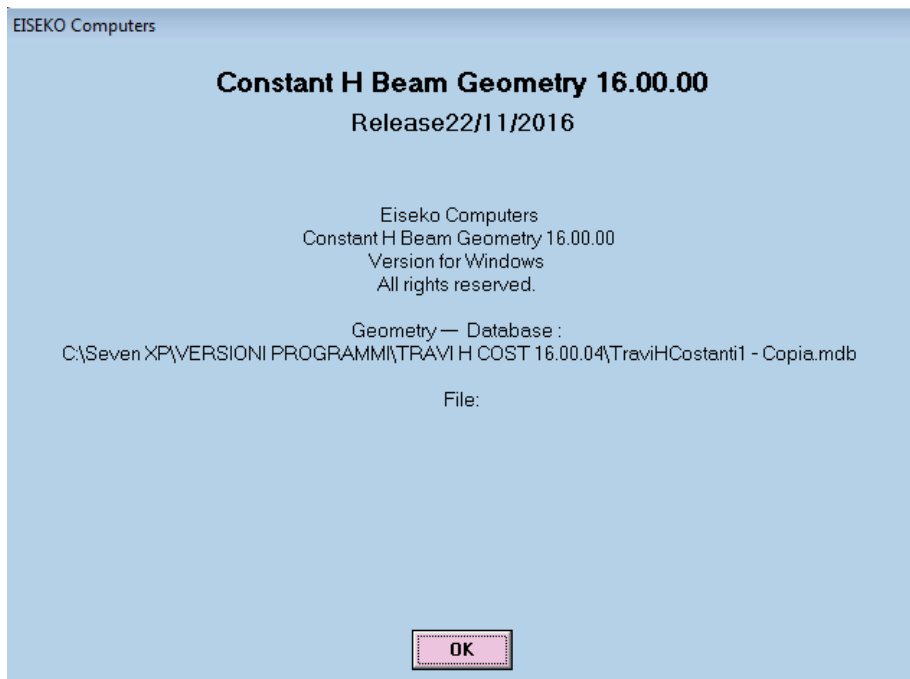
The “Database” menu:

- **Export all on files**: exports all the geometries of the existing sections in the database to a text file (one for each section, where the name of the file = name of section). It asks for a directory where to save all the files.
- **Change**: lets you change the database you’re working on. It must be an EISEKO database (it may be a copy of the original one, for example)
- **Check-Update**: check if database needs to be updated, and in case, it updates it.



The “?” menu shows the information (version of program and connected database) and opens the manual (Italian manual, to open the English one browse it to the program directory).





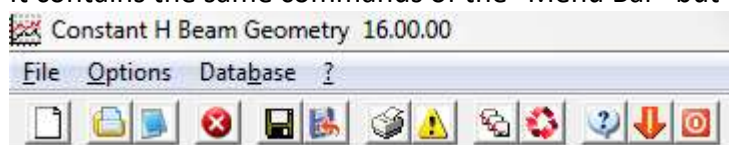
Name of program and
release number
Release date














Database file with path

Exportation File (if any)

30.10. TOOLBAR

It contains the same commands of the "Menu Bar" but like icons.



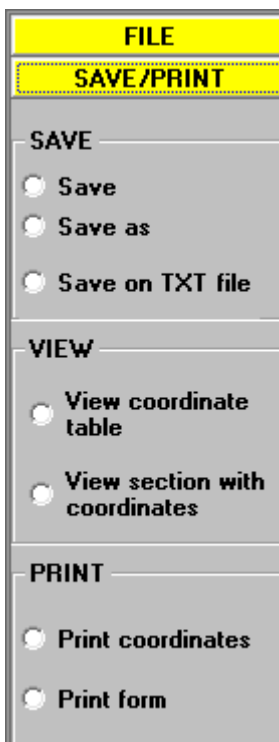
	New section		Open the section from list (database)
	Open the section from file		Delete section from list (database)
	Save		Save in file (txt)
	Print section		Check errors
	Export all on file		Change database
	Information		Check updates
	Exit		

30.11. VERTICAL TOOLBAR

ON THE LEFT IT 'S ALWAYS VISIBLE A VERTICAL TOOLBAR THAT ALLOWS YOU TO REACH DIRECTLY ALL THE WINDOWS AND OPTION AVAILABLE IN THE PROGRAM.



In the "FILE" TAB it is possible to create a new beam, open or delete an existing beam.



In the "SAVE / PRINT" TAB it's possible to save the beam, save it as, save it in a txt file.

-View coordinates table: displays the coordinates of the sections in a table

- View Section with coordinates: opens a report with the representation of the section, overlapped and the rebars.

"VIEW COORD" = Show coordinates in the vertices.

" VIEW POINTS" = Display numbers of points at vertices.

-Print coordinates: allows you to view the coordinates of the table in a printable version.

-Print form: allows you to print the displayed form (from any active form).

BDD Computer

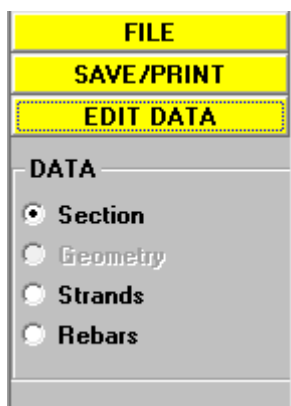
SECTION COORDINATES

MIDSPAN SECTION

HEAD SECTION

pt	Coord. X (mm)	Coord. Y (mm)	pt	Coord. X (mm)	Coord. Y (mm)
1	0	0	1	0	0
2	10	10	2	10	10
3	10	20	3	10	20
4	0	20	4	0	20
5	0	10	5	0	10
6	10	0	6	10	0
7	20	0	7	20	0
8	20	10	8	20	10
9	20	20	9	20	20
10	10	20	10	10	20
11	0	20	11	0	20
12	0	10	12	0	10
13	10	0	13	10	0
14	20	0	14	20	0
15	20	10	15	20	10

OK



The tab "EDIT DATA" lets you open all the windows to change the data: sections, geometry (enabled only for parametric sections) strands, rebars.

FILE
SAVE/PRINT
EDIT DATA
OPTIONS
CHECK
<input type="radio"/> Check errors
<input type="radio"/> Check updates
INFO
<input type="radio"/> Manual
OPZIONI
<input type="radio"/> Tools
<input type="radio"/> Settings
<input type="radio"/> Language

The tab "OPTIONS":

CHECK

Error Check: Check if there are any errors in the current section.

Check Updates: accesses automatically to www.eiseko.com to check if there are newer versions of the program

INFORMATION

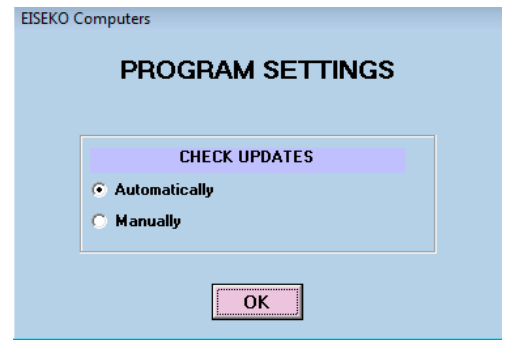
Manual: Launches Program Manual

OPTIONS

Tools: allows you to choose whether to search for updates automatically (when you start the program, it attempts to connect to the internet to compare the version of the program with that present in the site) or manual (the user will have to visit the site and check).

Settings: opens the settings window

Language: opens the window to change the language



FILE
SAVE/PRINT
EDIT DATA
OPTIONS
EXIT
<input type="radio"/> End

With the tab "EXIT" you can end the program.

You will be prompted to save any unsaved changes.