WinAqua TUTORIAL

SOFiSTiK AKTIENGESELLSCHAFT

WinAqua TUTORIAL

Table of Contents

1. Polygonal column

WinAqua can be used for creating a polygonal column as shown on the Fig.1.1.

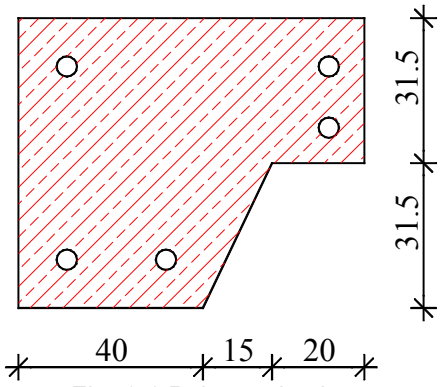


Fig. 1-1 Polygonal column

The column is made of B25 concrete and the same is reinforced with standard reinforcement BST 420. The position of the reinforcement is shown on the Fig.1.2.

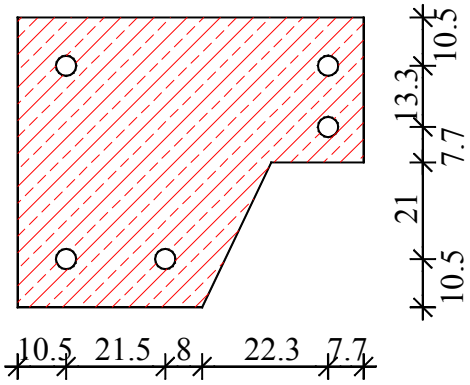


Fig. 1-2 Reinforcement placement

Some previous preparations are necessary for creating column geometry in WinAqua. For example user should define the position of the coordinate beginning (0,0), the units in which the cross section will be entered, the coordinate of the points which define column contour, the points where the reinforcement is and the sections in which user would like to control the share stresses.

1.1. Previous preparations:

Defining of the coordinate beginning and characteristic points of the cross section:

Polygon column geometry will be described with six characteristic points (1,2,3,4,5 and 6) with which the contour of the column will be defined. User should define another five points with which the position of the single reinforcement will be defined. These are points 101,102,103,104 and 105 shown on the Fig.1.4. The coordinates will be given in unit's meters [m] from the SI system. The coordinates of the points from the cross section will be defined in relation with previously determined coordinate system. For this example the coordinate beginning will be taken as it is shown on Fig.1.3. The points with which we will define the cross section are shown on the Fig.1.4.

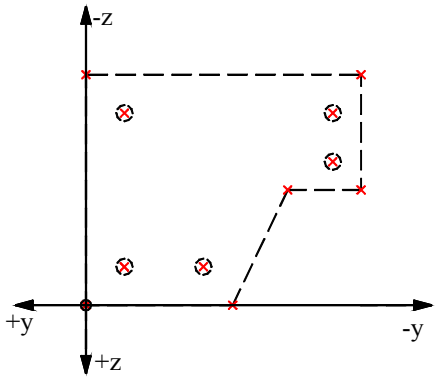


Fig. 1-3

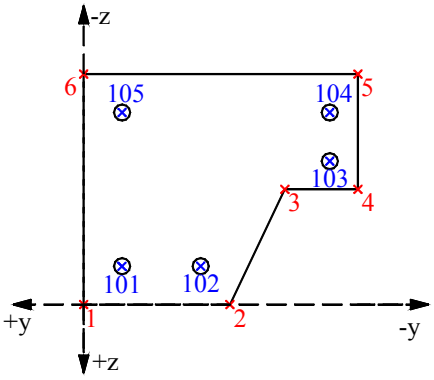


Fig. 1-4

The coordinates of the points with which the geometry of the column is defining are given in the Table 1.1, and the coordinates of the points with which the single reinforcement is defining are given in the Table 1.2

Table 1-1

Point	Coordinate	
	Y	Z
1	0.000	0.000
2	-0.400	0.000
3	-0.550	-0.315
4	-0.750	-0.315
5	-0.750	-0.630
6	0.000	-0.630

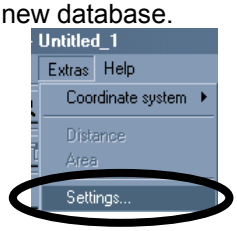
Table 1-2

Point	Coordinate		As
	Y	Z	cm^2
101	-0.105	-0.105	55.44
102	-0.320	-0.077	36.96
103	-0.673	-0.392	24.64
104	-0.673	-0.525	36.96
105	-0.105	-0.525	36.96

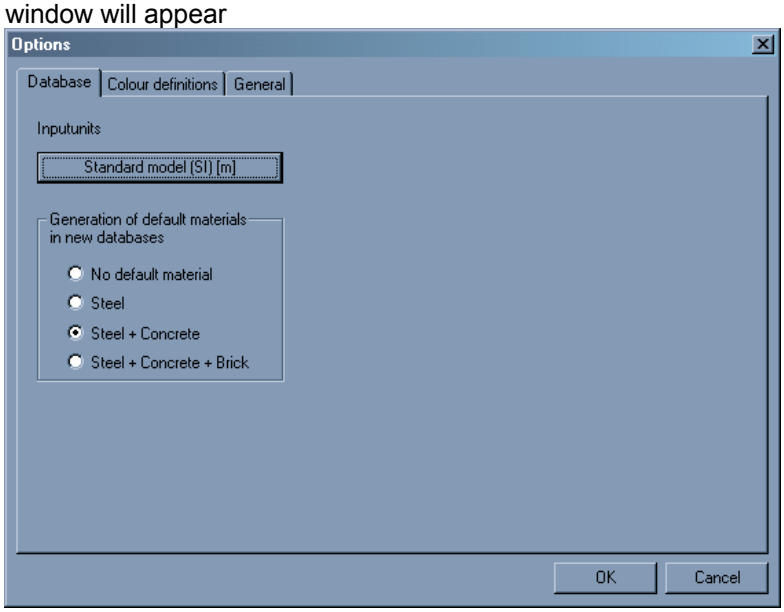
1.2. Solving the problem with WinAqua

1.2.1. Defining default units and materials in database file

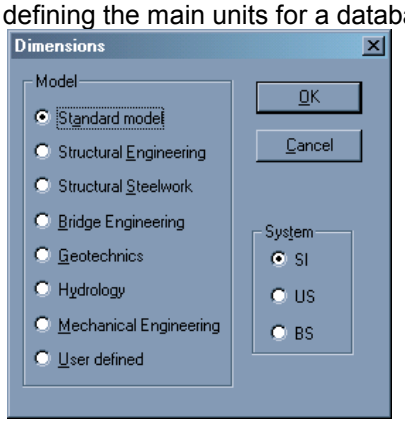
Form “Extras” menu select command “Settings” to define the main input parameters of WinAqua for a new database.



Option window will appear



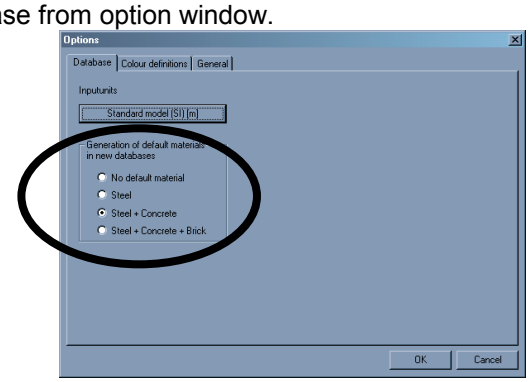
With “input units” user can define the units in which database file will be drawn. If user selects a button located under “input units” a new window will appear which will allow defining the main units for a database.



User can define:
- Standard units of SI-System
- General structural engineering (e.g. cm instead of m)
- Steel construction, SI-System (e.g. mm instead of m)
- Concrete bridge construction, SI-System (e.g. MN instead of kN)
- Geo-techniques, SI-System (e.g. MN instead of kN)
- Hydrology
- Standard units of US-System
- General structural engineering in US-System
- Steel construction in US-System
- Concrete bridge construction in US-System
- Geo-techniques in US-System

For this example user should select System SI, Standard mode so he can use standard units like m and kN for defining new database.

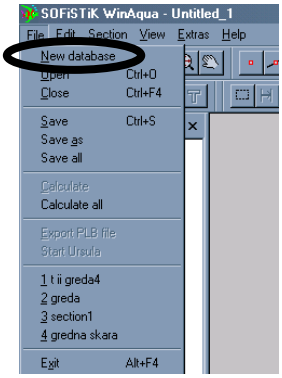
User can define default materials, which will be included in the new database. This can be done with “Generation of default materials” options in a new database from option window.

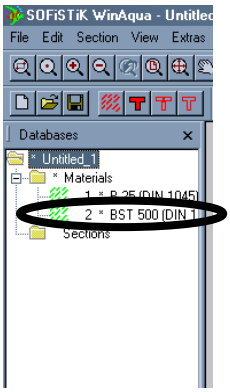


If the user selects “No default material” option then there will be no material input in the new database.
If “Steel” option button is selected then WinAqua will automatically add only default steel material Structural steel 235 (DIN 18800).
If “Steel + Concrete” option button is selected then WinAqua will automatically add default Standard reinforcement steel BST500 (DIN 1045) and standard concrete material B 25 (DIN 1045).
If “Steel + Concrete + Brick” option button is selected then WinAqua will automatically add default Standard reinforcement steel BST500 (DIN 1045) , standard concrete material B 25 (DIN 1045) , standard structural steel 235 (DIN 18800) and brick MZ4 I (DIN 1053-1).
For Example 1 user should select “Steel + Concrete”
From the tab “color definition” user can define visual parameters such as type and color of the cross section lines, reinforcement lines, shear cut lines, text height etc. More detail information about tab “color definition” are given in Example 2 step 1

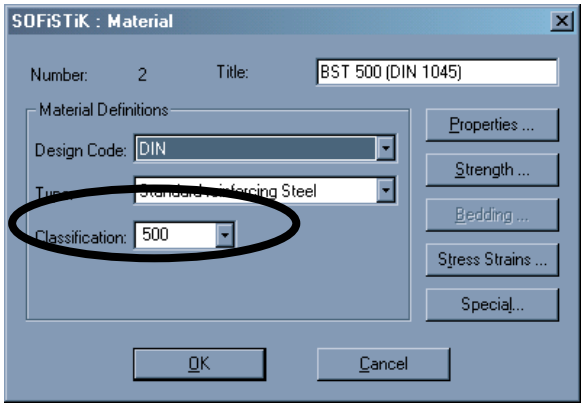
1.2.2. Creating a new database file, changing default materials

From “File” menu select “New database”.

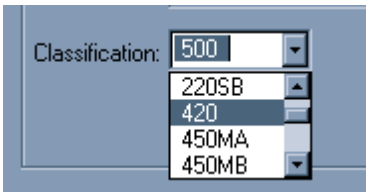




WinAqua will open a new database file Untitled_1.cdb that will contain default materials previously defined in step 1. User can explore which materials are inputted in a new database file in a window database. By double clicking on a material in database window, user can change the default material and the designed code if is necessary. In this example it is not necessary to change concrete materials because B25 concrete shall be use for the polygon column. User should change only reinforcement steel because WinAqua added Standard reinforcement BST 500(default material). To change standard reinforcement user has to double click on a “2 * BST 500” from database window. A new window will appear called “SOFiSTiK: Materials”.



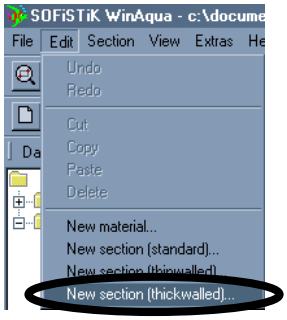
This window will allow user to change standard reinforcement steel classification from 500 to 420.

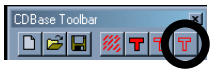


By pressing on the button OK WinAqua will change standard reinforcement steel from BST500 to BST420.

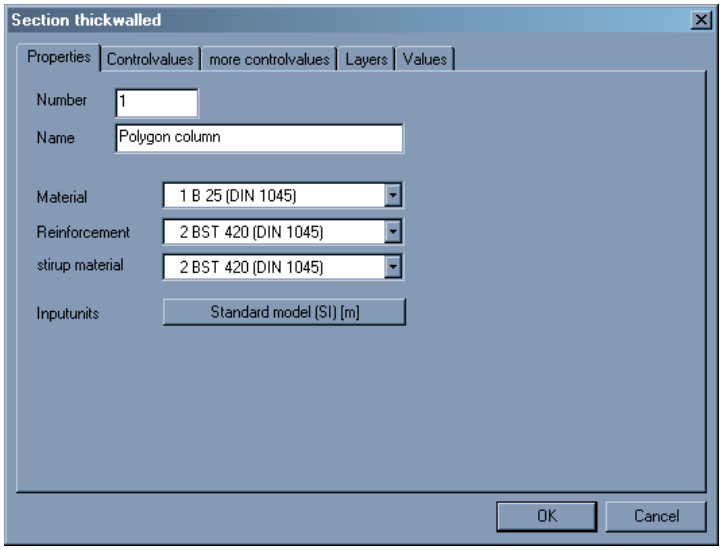
1.2.3. Creating Section

To create new section from the menu “Edit” select “New section [thick walled]…” or select icon from “CDBase Toolbar”.





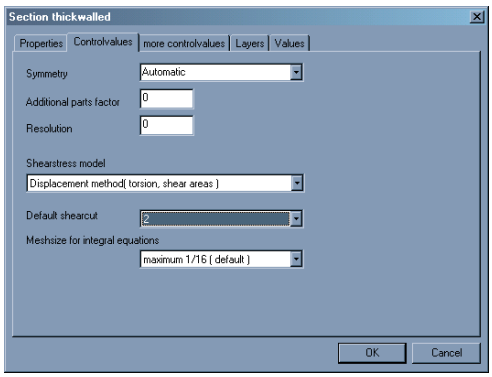
A new window will appear allowing user to define some global parameters for new cross section.



Here user can define number of the cross section by which the cross section will be stored in a database, the name of a cross section (example:

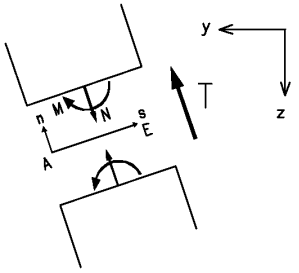
Polygon column) and the material for cross section, reinforcement and stirrup. User can only select materials previously defined in step 2. Here the user can define a unit if that is not previously done in step 1.
In a “Control values” tab the user can define some parameters such as:

Shear stress model
Default shear cut
Mesh size for integral equations

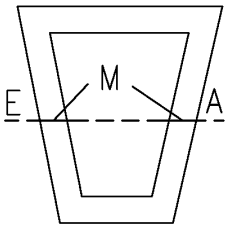


“Shear stress model” controls the computation of shear stresses in solid sections:
1- force method
2- displacement method only for It and location of shear center (default for concrete)
3- displacement method for torsion,
4- displacement method for torsion and shear, shear deformation areas are determined (default for steel, timber).
Options 2 and 3 should be used for composite sections, but with caution. For more details see aqua manual 2.7. Shear Stresses in Solid Sections.

“Default shear cut” allows the user to control how many of these standard cuts will be generated (0/1/2). The cut can hit the cross-section several times creating partial cuts. Each partial cut has a direction s and three defined points of interest: beginning (A), middle (M) and end (E):



Sign convention



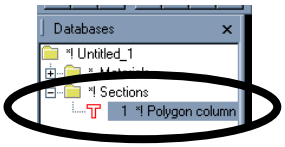
Thin wall equivalent cross section

The internal forces perpendicular to the cut M and N act in such way that positive axial forces cause tensile stresses, and positive moments cause tensile stresses at the End-Point.
The shear stressing is described primary by the resistance module of the shear stresses at the three points. Additional values are calculated for the proportioning of reinforced concrete structural elements:
A mean torsion shear stress which, after being multiplied by the width of the partial section, must be covered by reinforcement. This corresponds to a section modulus for the shear flow.

- The total cut width, by which the shear stresses due to the shear force must be multiplied in order to obtain the shear flow from shear force. These distinctions are very significant in the definition of equivalent hollow cross-sections.

“Mesh size for integral equations” controlled the fineness of the subdivision for the integral equation method. This indicates how large an element may be compared to the largest dimension of the cross-section.

0 No subdivision
1 maximum 1/2
2 maximum 1/4
3 maximum 1/8
4 maximum 1/16 (default)
By pressing the button “OK” a new item will appear in database window in the folder “Section”.

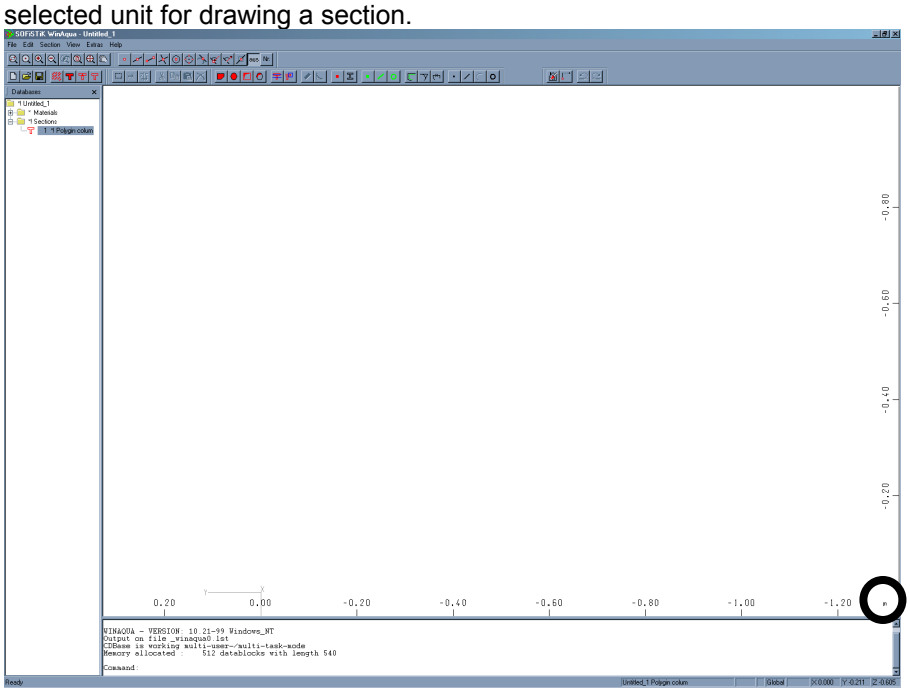


Description:

Icon  shows the user that he used “thick walled” section

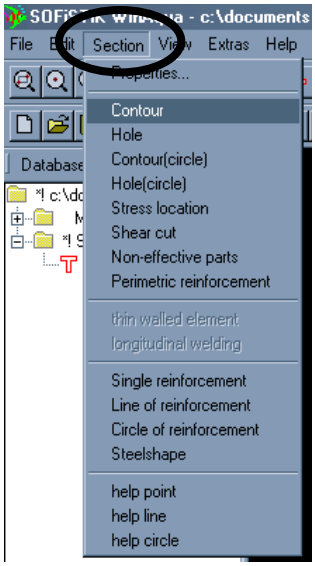
1 tells user that he selected No 1 for a cross section to be stored in a database
* tells user that the changes in a data base are still not saved
! tells user that there is no calculated result for a cross section
Pol.. tells user the name that he selected for a cross section No 1

On the right side a new drawing area will appear where user can draw section. In the right bottom of the drawing window WinAqua will show selected unit for drawing a section.



1.2.4. Drawing section

With WinAqua a cross section can be drawn using the menu “Section” and all available options in this menu, or by using “Edit section” toolbar.



Tools for drawing section can be divided in four groups:
Groups for defining contour of the cross section,

Icon	Description
	Contour
	Circle contour
	Polygonal hole
	Circle hole

Groups for defining reinforcement of the cross section,

Icon	Description
	Single reinforcement
	Line reinforcement
	Circle reinforcement
	Contour reinforcement
	Steel shape

Groups for defining characteristic shear cut, non effective parts and stress point on a cross section,

Icon	Description
	Shear cut
	Non –effective part
	Stress point

Groups for defining help line for easier creating of cross section using on-snap.

Icon	Description
	Help Point
	Help Line
	Help Circle

During the use of these commands it is necessary for user to pay attention on the command window in the bottom of the WinAqua window. Command window will be guide for correct use of all these commands.

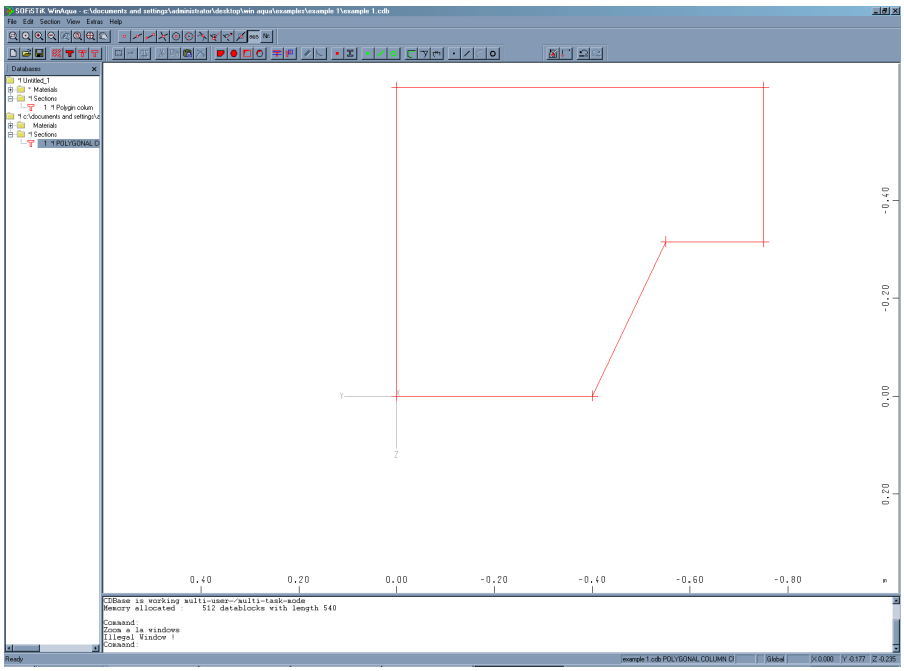
1.2.5. Description of the section drawing command (necessary for defining cross the section in Example 1)

 **Contour**

After activating the command “Contour”, WinAqua will expect the user to define first and the last point of a contour of the cross section. Points can be added in the drawing by clicking with mouse on the drawing area or by typing y,z coordinates in the command window. The command will be active until closed polygon is created. By pressing escape, the command will be canceled and WinAqua will automatically create closed polygon by connecting the first input point with the last input point.
For Example 1 user can activate command contour and can define cross section by inputting coordinates of six characteristic points of a cross section defined in Table 1.1 in the command bar

Command:

New Polygon, first point [m] 0,0
New Polygon, next point [m] -0.4,0
New Polygon, next point [m] -0.55,-0.315
New Polygon, next point [m] -0.75,-0.315
New Polygon, next point [m] -0.75,-0.63
New Polygon, next point [m] -0.,-0.63
[Press ESC to close the couture or press right mouse button and select Close command]



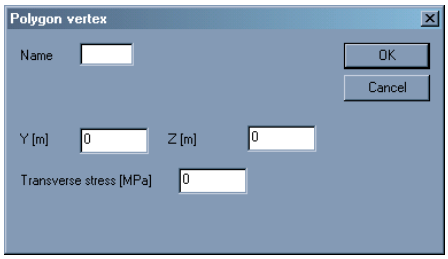
Another way to draw a section is by using local coordinate system instead of global. Local coordinate system is activated when in front of the y,z coordinates user puts sign “@”, then WinAqua will place the local coordinate system in the last input point in the drawing.

Command:

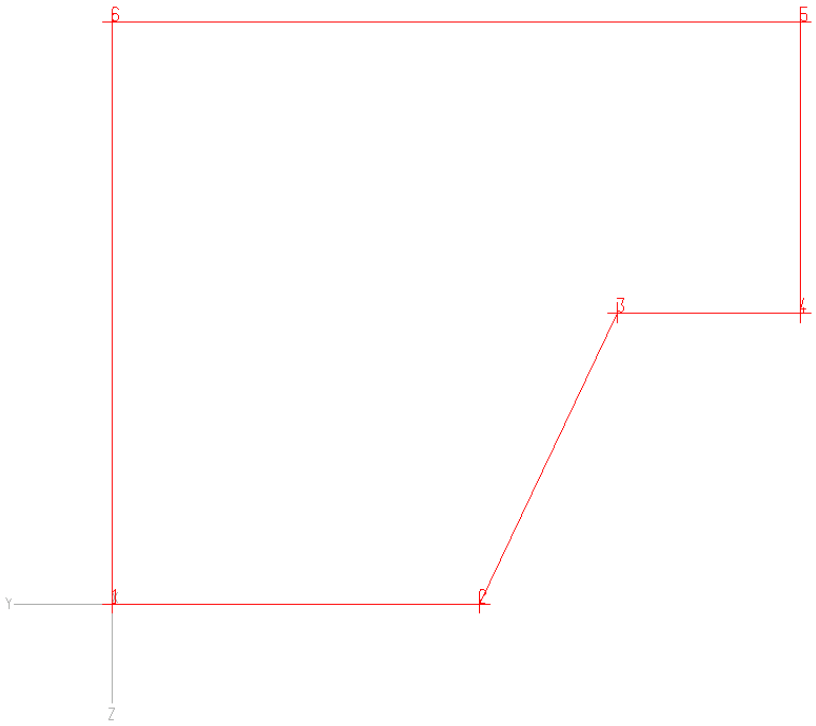
New Polygon, first point [m] 0,0
New Polygon, next point [m] @-0.4,0
New Polygon, next point [m] @-0.15,-0.315
New Polygon, next point [m] @-0.20,0
New Polygon, next point [m] @-0,-0.315
New Polygon, next point [m] @0.75,0

[Press ESC to close the couture or press right mouse button and select Close command]

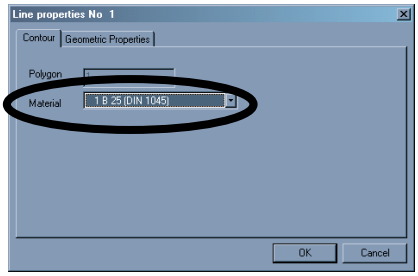
To define point name entered in a drawing, simply double click on a point that was drown. The first click with the mouse will select a point. The second click on the selected point will show “Polygon vertex” window where user can define name of the point, redefine it’s coordinates if necessary or user can give transverse stress in a selected point.

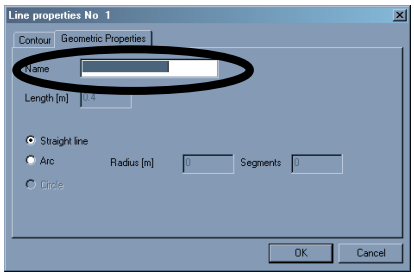


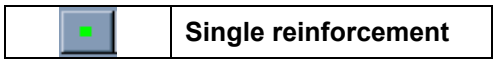
Naming of a point will allow easier further control of stresses in a cross section. After naming all points in a drawing, numbers will appear on every point.



By double clicking on a line, WinAqua will select a line and “line property” window will appear. Here user can define material and name of the selected line.

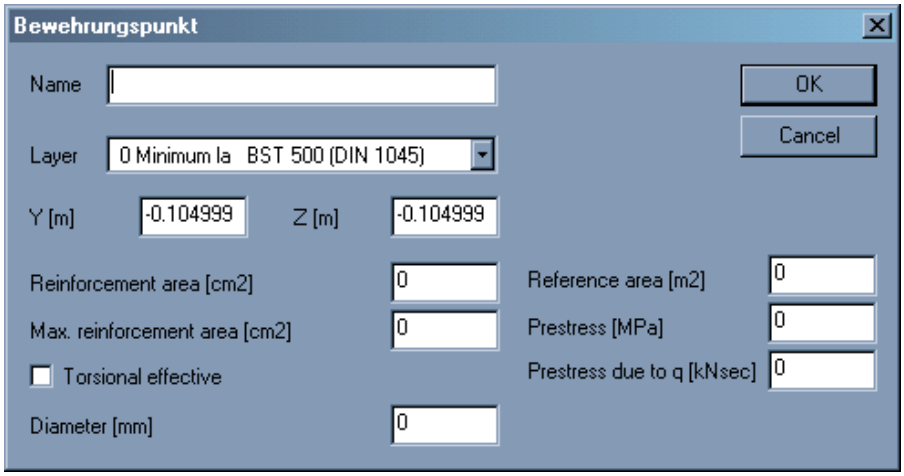




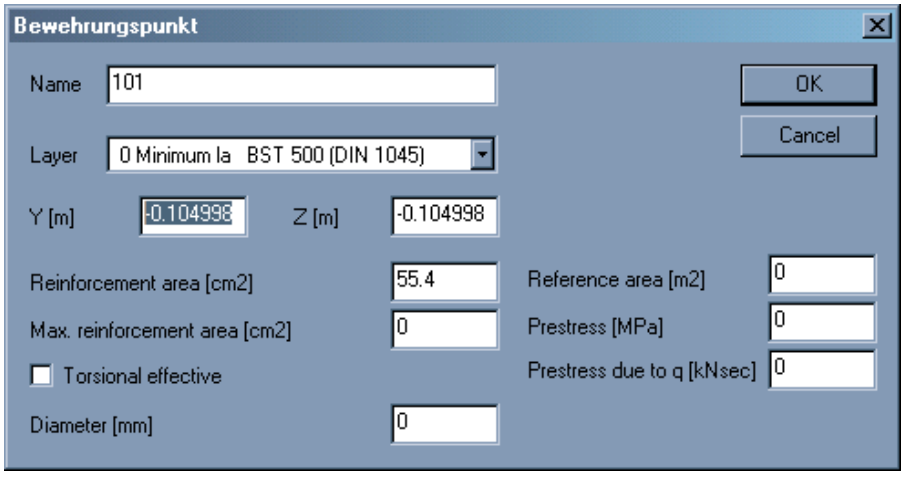


By activating the command for “Single reinforcement”, WinAqua will ask the user to input coordinate for single reinforcement. Coordinate can be define by clicking in the drawing area or by typing global y,z coordinates in a command window. After inputting the coordinate “Bewehrungspunkt”,

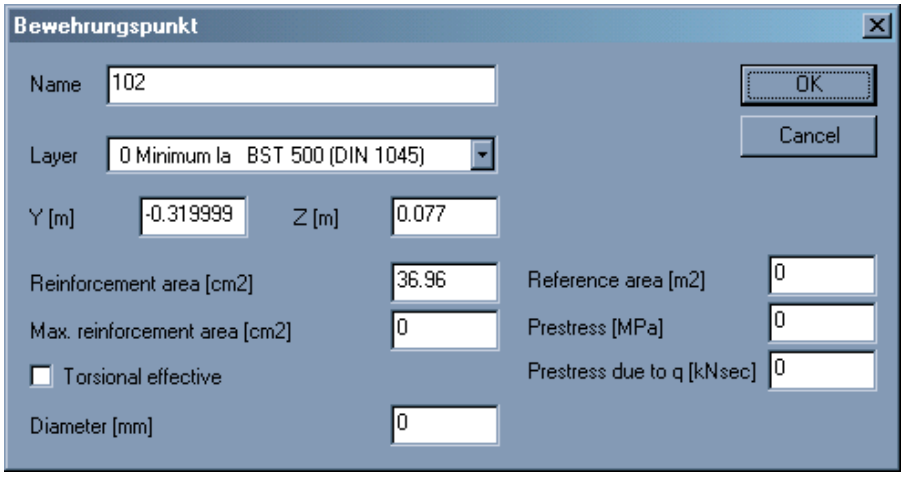
window will appear in which the user can define the name of reinforcement, material for reinforcement, layer, coordinate etc.

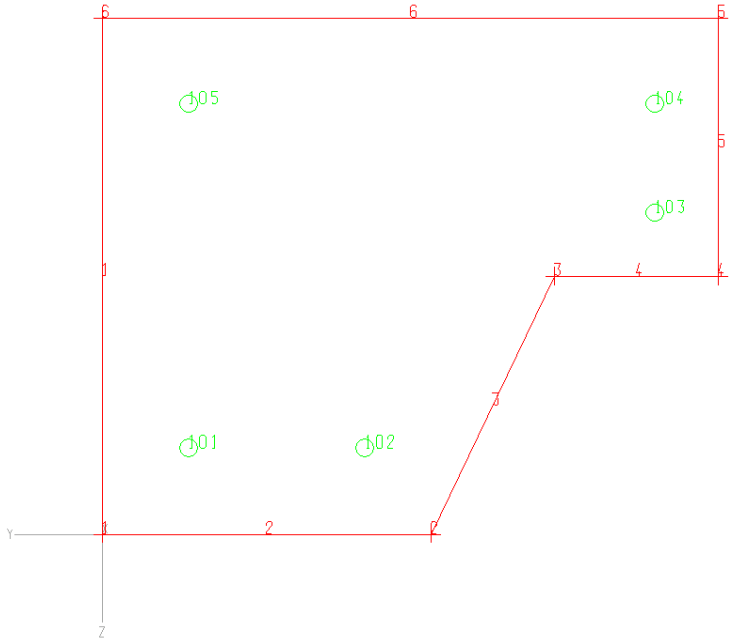


For more detailed description for other parameters (reinforcement area, max. reinforcement area, reference area, pre-stress, torsion effect, diameter etc.) see aqua manual paragraph 3.37. Reinforcement.
For Example 1 user can activate the command “Single reinforcement” and can define single reinforcement with the appropriate parameters given in Table 1.2.
Command:
New Single reinforcement [m] -0.105, -0.105



New Single reinforcement [m] -0.320,0.077

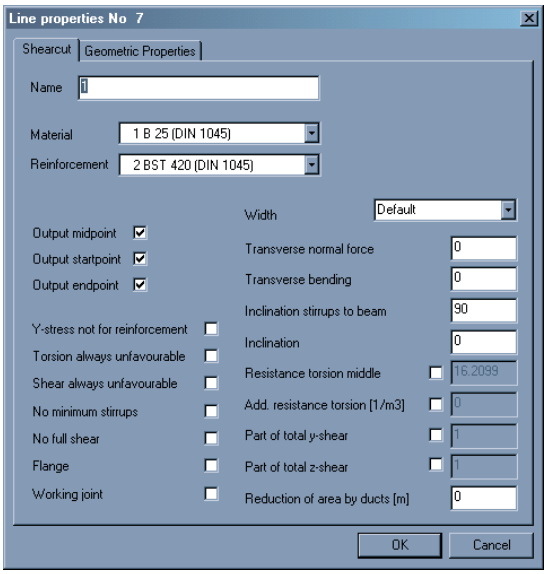




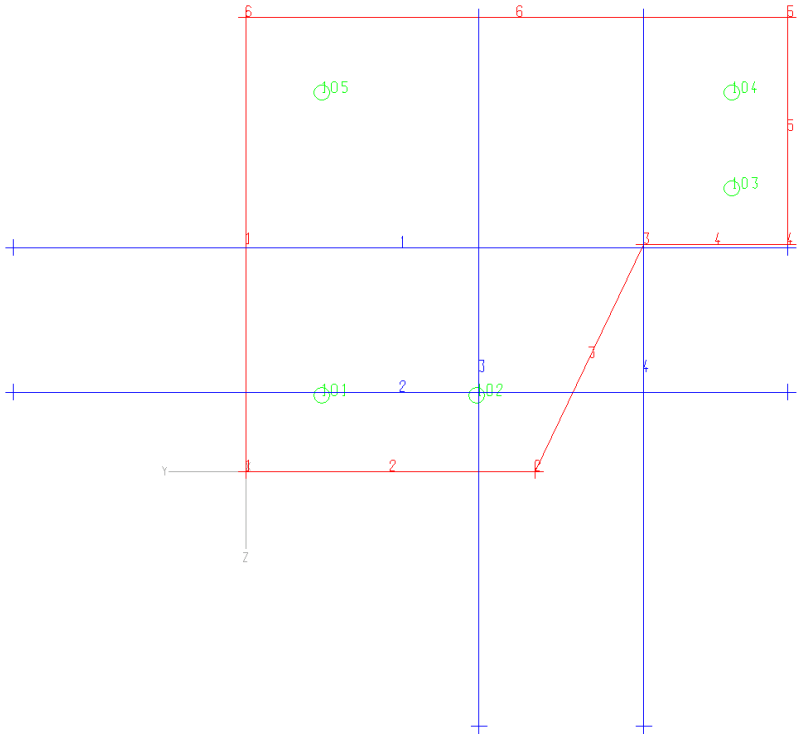
Shear cut

By activating the command for “Shear cut”, WinAqua will ask the user to input two coordinates, which will define a line (section) in where shear stresses will be analyzed.

In Example 1 user should examine four shear cuts. The first two shear cuts will be parallel with y axe (with z coordinate z=-0.311 and z=-0.11). The next two shear cuts will be parallel with z axe (with y coordinate y= -0.322448 and y=-0.55).
Start the command “Shear cut”, WinAqua will ask the user for the first and last point of the shear cut line.
Command:
New shear cut, first point [m] 0.32,-0.31
New shear cut, next point [m] -0.75,-0.31
[Press ESC to end the command or press right mouse button and select Abort command]
Double click drown line, line properties window will appear.

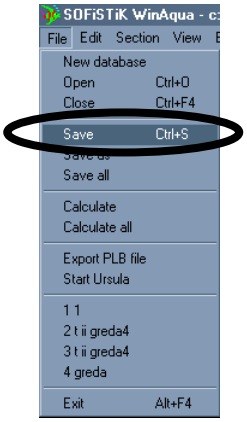


Here user can define some control parameters for shear cut calculation. More details for these parameters can be found in aqua manual in paragraph 3.28. CUT - Shear Sections
The same way as mentioned above the rest of the shear cut will be inserted in WinAqua drawing.



1.2.6. Saving database

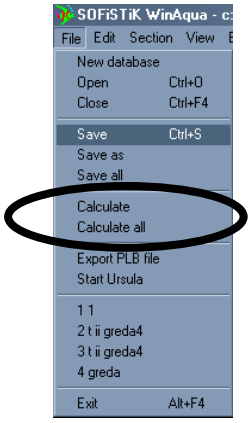
From the “File” menu select options in “Save”.



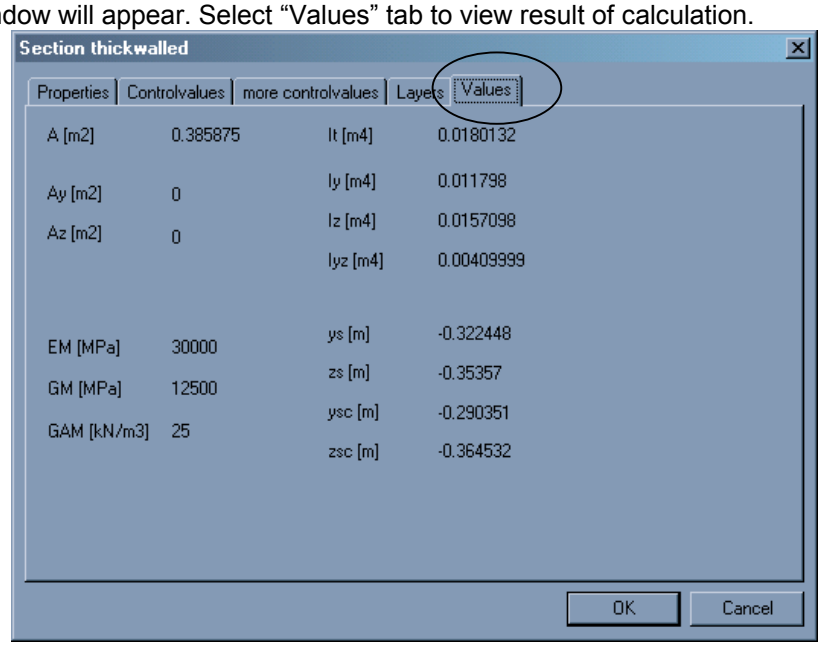
Standard save-as window will appear. Give the name and define the path in which database should be saved. WinAqua will create database file name.cdb in the selected path.

1.2.7. Calculating cross section

From the “File” menu select options in “Calculate”. If you have defined more then one cross section in the same database then you can select option “Calculate all”.



WinAqua will do the calculation for the section and the result will be stored in the name.cdb database. If you want to check the result double-click on the section in a database window. As it was described, “Section tick walled” window will appear. Select “Values” tab to view result of calculation.

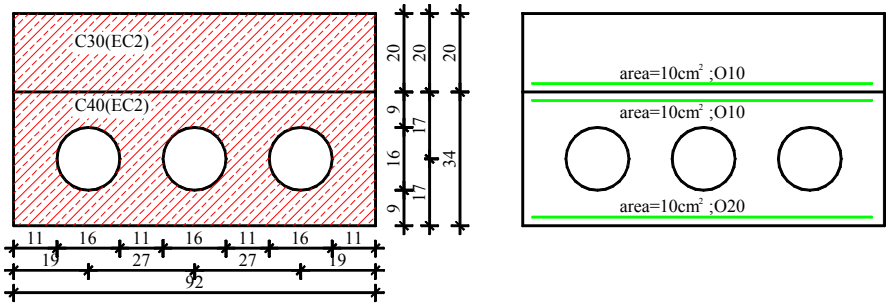


Now file name.cdb is ready for further use in SOFiSTiK static calculation.

2. Freely defined solid with circle holes

A freely defined solid cross-section consists of any number of outer and inner perimeters in the form of circles or polygons, as well as of reinforcement elements. Structural steel shapes can be integrated. The Example 2 is consisted of two outer (plate) and three inner (circles) perimeters.
In this example a composite cross section made of prefabricated plate with three holes and in site cast plate should be created.

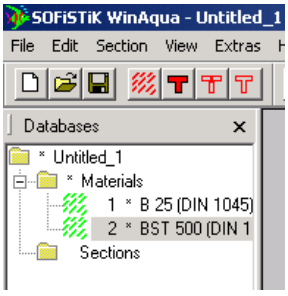
2.1. Concrete Reinforcement



2.2. Solving the problem with WinAqua

2.2.1. Default property setting

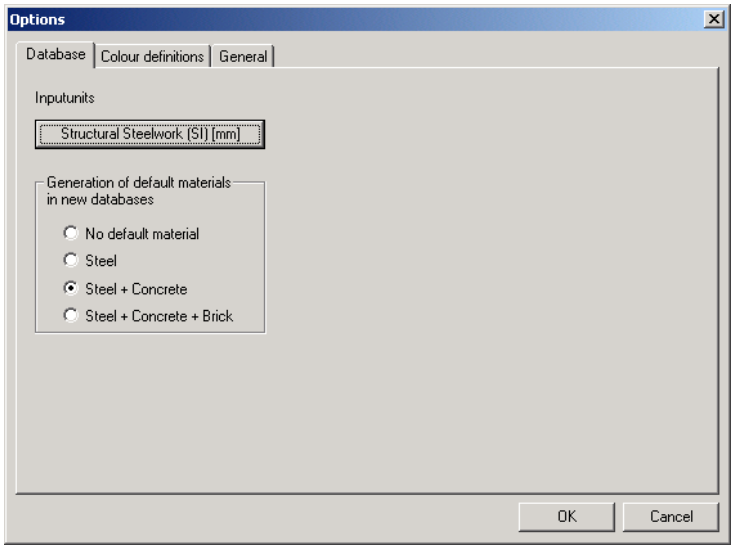
When the new project is stated, WinAqua created by default a dtabase named “Untitled_1” with two predefined materials: B 25 (DIN 1045) and BST 500 (DIN 1045).



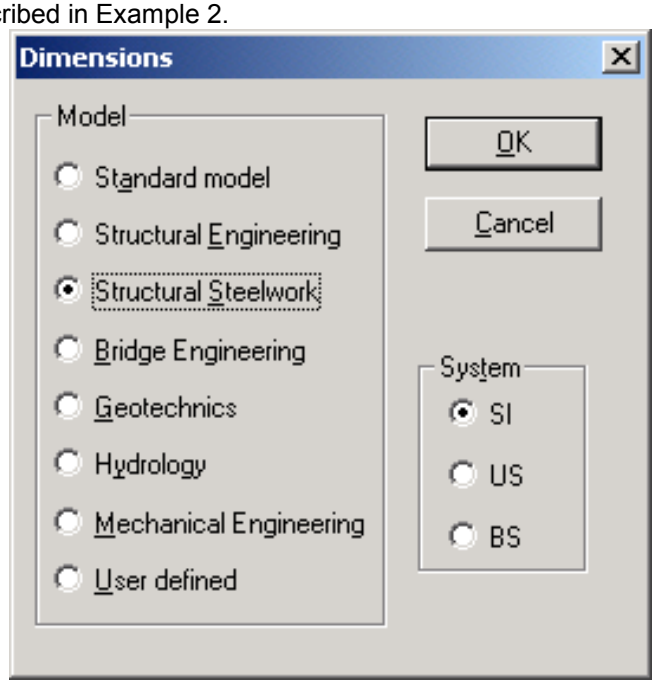
This option can be changed with the command Settings

Menu: Extras→Settings

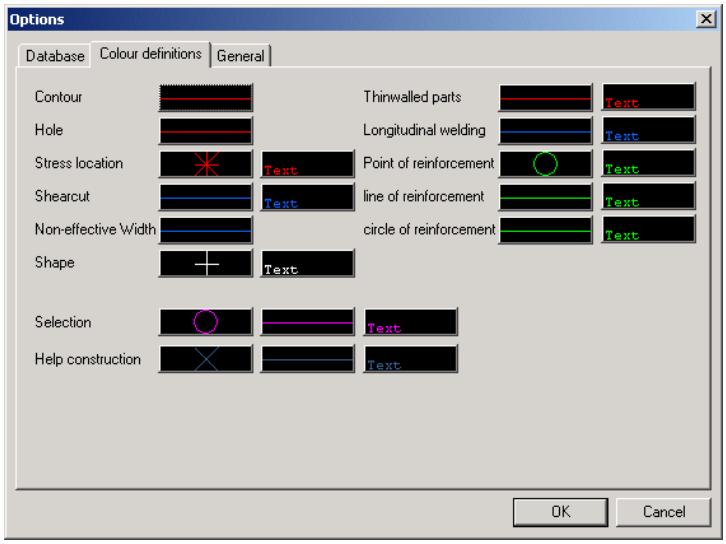
The “Options” window appears on the screen and the user can choose one of the four options to generate the default materials in new database.



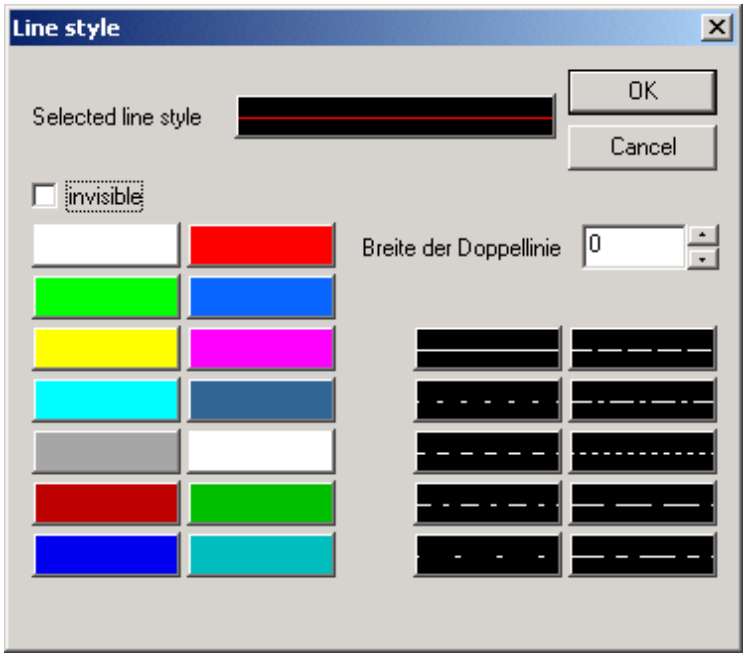
With the help of the “Options” windows the user can change several default properties also. Clicking on the command button Structural Steelwork (SI)[mm] input units can be changed. The new window Dimensions appears on the screen and the user can choose between several System and Model units described in Example 2.



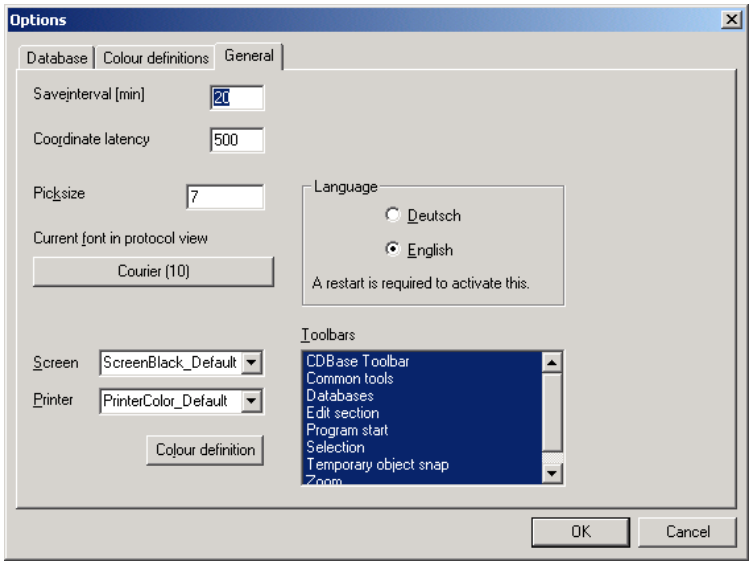
The tab “Color Definitions” of the “Options” window, gives the opportunity some property of the graphical entities to be changed.



For example, clicking on the picture Couture, the Line Style and the color of the contour can be changed.

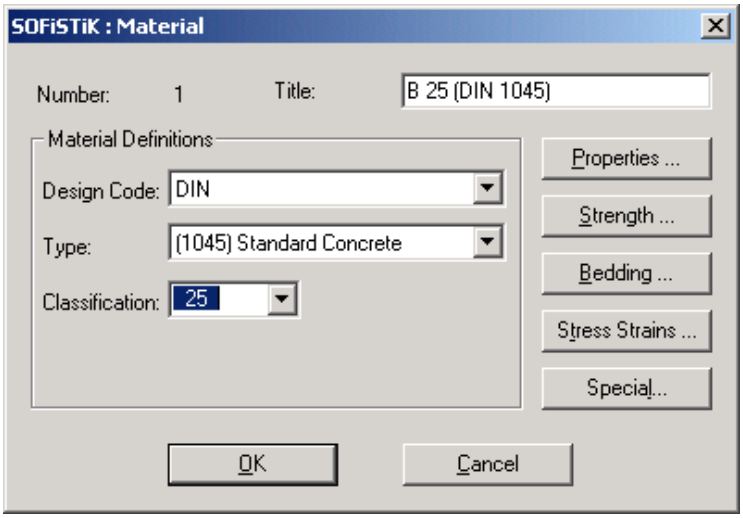


The Tab page General is used for setting the language, toolbars that will be showed on the screen, saving interval, coordinate latency, picking size, current font in protocol view, screen and printer defaults.

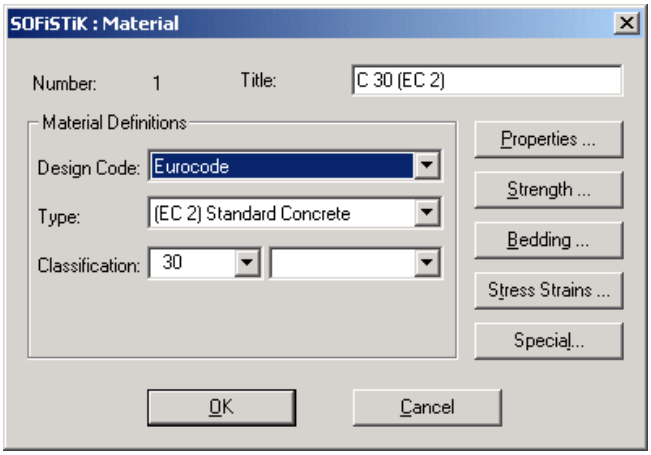


2.2.2. Material definition

The process of crating a new section begins with definition of the materials. The section is made of three materials: (EC2) Standard Concrete Classification 30 and 40, and (EC2) Standard reinforcement Steel Classification 500. At the beginning the property of the material 1 will be change. Double click on the name of the material 1 opens the SOFiSTiK "**Material**" window.



In the combo boxes “**Design Code**”, the “**Euro code**” standard should be selected. The type is set to “**(EC2) Standard Concrete**” and Classification is set to 30.



In a similar way the property of the material 2 will be changed to “**(EC2) Standard**” and the reinforcement to “**Steel Classification 500**”.
New material in the database can be added using command

Menu: Edit→New material… or
Icon: New material

The (EC2) Standard Concrete, Classification 40 will be selected to the material 3

2.2.3. Section creation

The new section can be added to database with the command

Menu: Edit→New section

There are several options to create the new section: standard, “**thinwalled**” and “**thick walled**” type. In this example the “**thick walled**” type should be selected. Firstly the plate with three holes will be created.
The contour of the cross section should be drawn with command

Menu: Section→Contour
Icon: Contour

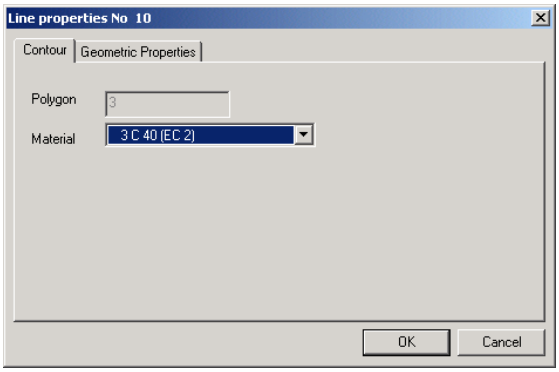
The contour’s point can be input by mouse or by typing in the command window.

Command:

New polygon, first point [cm]: **0,0**
New polygon, next point [cm]: **@0,-34**
New polygon, next point [cm]: **@92,0**
New polygon, next point [cm]: **@0,34**

[Press ESC to close the couture or press right mouse button and select Close command]

The material property for this contour can be set by double mouse click over contour, or by right mouse click and choose the item Properties… from the floating menu.



In the next steps several help line should be created. At first, two help lines that connect the mid points of the opposite sides of the plate should be created. The command

Menu: Section→Help line
Icon: Help line

draws the construction line between two points. The temporary "**object snap**" toolbar can be used for precisely definition of the mid points on the line.
The Move command is used to change the position of some object or to make a copy of the same.

With the Move icon m toolbar "**Selection**" vertical help line should be offset on the distance 27 cm. At first the vertical line should be selected by pointing with the mouse on it. After that the Move command is started

Command:

Form point:

[Select intersection between vertical and horizontal mid line]

*To point:**@27,0***

*Delete old element <1>yes <2> no (2):**2*** [not delete the original]

The prefix "@" is used to set coordinate relatively to the last drawn point.

Mirroring the previously created line, the last help line should be created. The mirroring process begins with selection of the line that should be mirrored and activating the command Mirror from toolbar Selection.

Command:
First point of axis or axis [cm]:
[select the first point of the mirror axis "that is bottom point of the first drawn help line"]

End point of the axis [cm]:

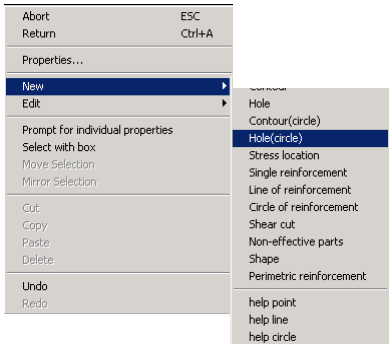
[select the second point of the mirror axis "that is upper point of the first drawn help line"]

*Delete old element <1>yes <2> no (2):**2*** [not delete the original]

There are several ways to create hole in the section. One is to use

Menu: Section→Hole (circle)
Icon: Hole (circle)

Another way is by clicking on the screen with the right mouse button. The floating menu appears



Selecting

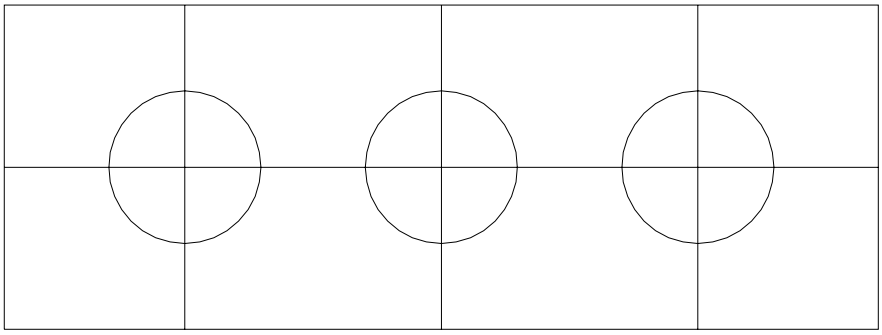
New→Hole(Circle)

the process of creating the new hole started. In the our example the three holes are created as a follow

Command:
*Center Point for circle [m]: **19,-17*** [center of hole]
*Point on radius [m]: **19,-25*** [radius 8]
Segments <10>: [Press Entry to accept this]

Center Point for circle [m]: ***46,-17***
Point on radius [m]: ***46-,25***
Segments <10>:
Center Point for circle [m]: ***73,17***
Point on radius [m]: ***73,25***
Segments <10>:

The plate cross section with three circle holes was created.



2.2.4. Reinforcement setting

The top and bottom reinforcements are presented with as distributed line of reinforcement. They can be insert in the section by command

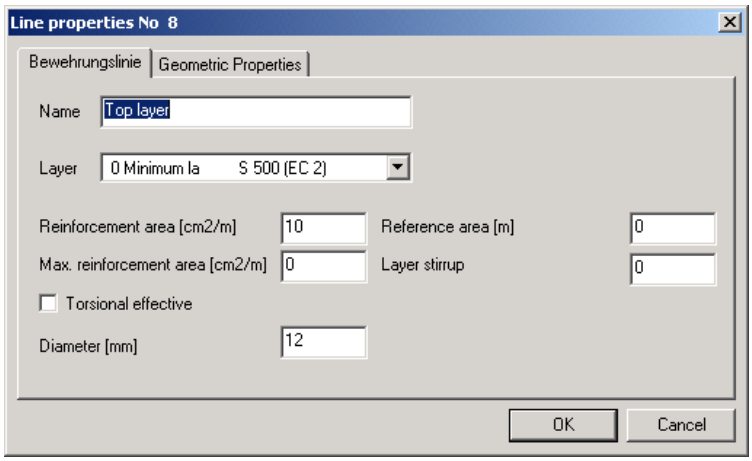
Menu: Section→Line of reinforcement
Icon: e of reinforcement

In our example two line of reinforcement should be set.
Command:
New line of reinforcement, start point [m]: 4,-4

New line of reinforcement, end point [m]: ***@84,0***
New line of reinforcement, end point [m]:
[Press Enter to not start next segment]

Chose reference point:
[Press Entry to not start next line by ref.]

When the line of reinforcement is drawn, a Line property window appears on the screen

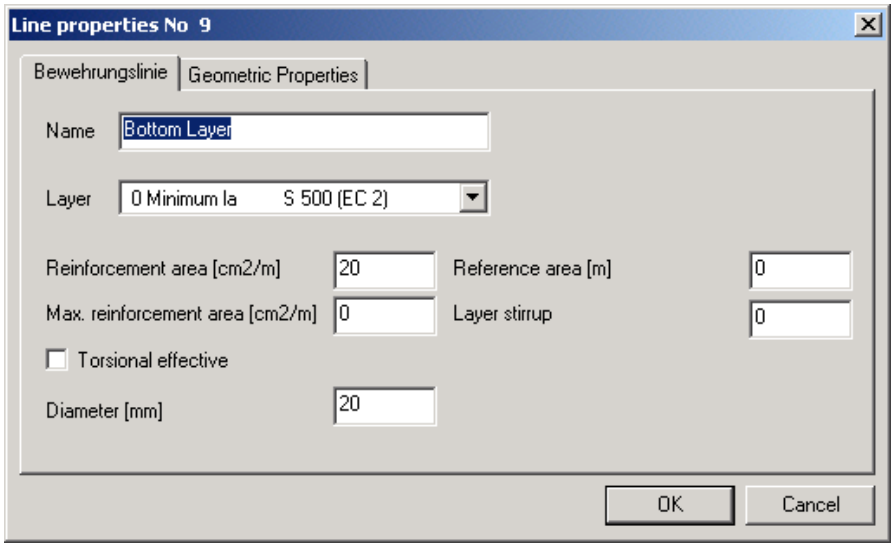


The reinforcement's name, area and diameter should be inserted in the appropriate text boxes.
In a similar way the second line of the reinforcement is inserted in the section.

Command:

New line of reinforcement, start point [m]: ***30,4***
New line of reinforcement, end point [m]: ***@88,0***
New line of reinforcement, end point [m]: [Press Entry to not start next segment]

Chose reference point: [Press Entry to not start next line by ref.]



2.2.5. Upper part of the cross section

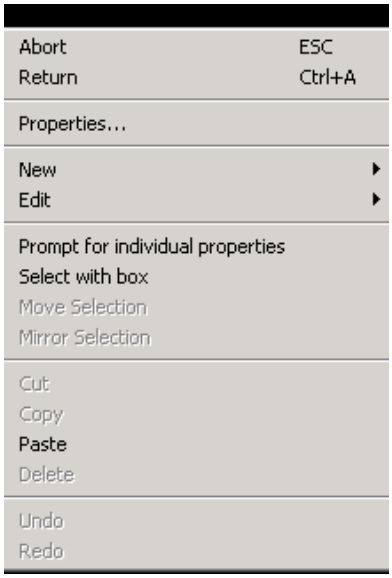
To create the upper pert of the cross section a new contour should be drawn.

Menu: Section→Contour
Icon: Contour

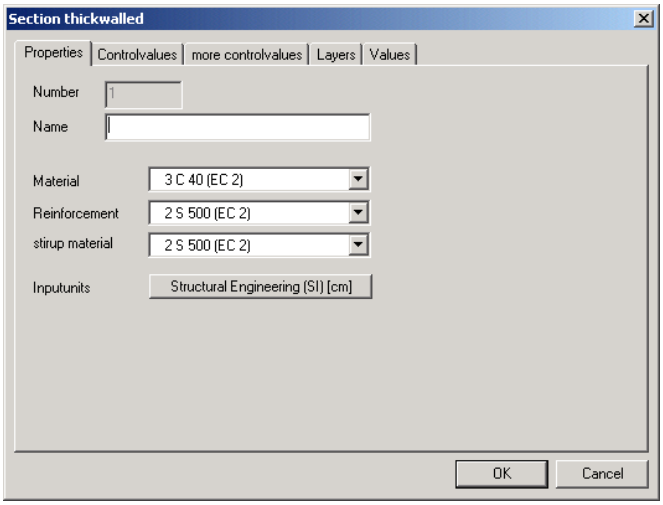
A four-point contour is drawn.
Command:
*New polygon, first point [cm]:**0,0***
*New polygon, next point [cm]:**@0,-20***
*New polygon, next point [cm]:**@92,0***
*New polygon, next point [cm]:**@0,20***

[Press ESC to close the couture or press right mouse button and select Close command]

The new contour inherits the default property of the cross section. In this case the material number is 3 C40 [EC 2]. The material property can be change with right click over one of the contour segments, and from floating menu the item "**Property…**" should be selected

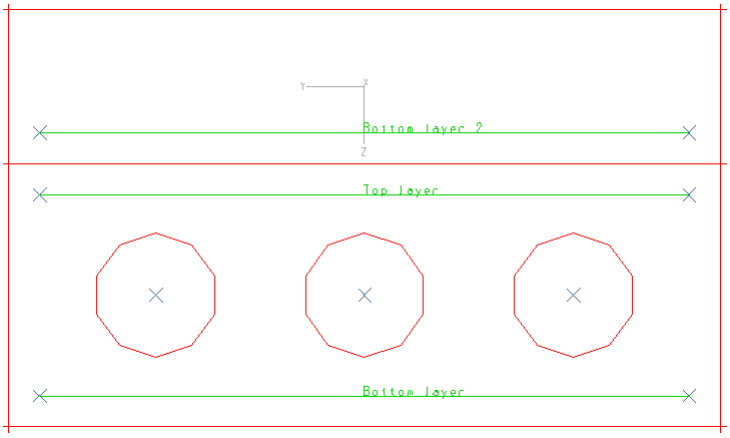


The Section "**thick walled**" window appears on the screen



In the combo box Material the appropriate material 1 C 30 (EC 2) should be selected.

At the end the line of reinforcement, named Bottom layer 2, with area 10 and diameter 10 is drawn.



2.2.6. Control points

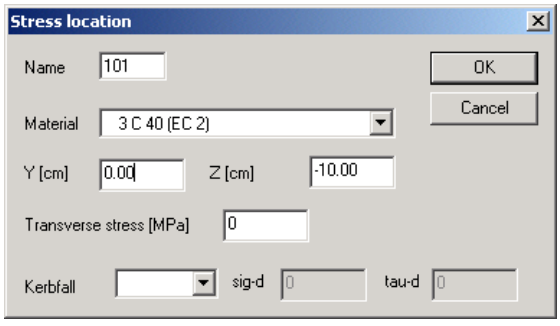
Stress location points
Resistance module for all stresses is only available at distinct points of the cross-section. Two points for stress calculation will be defined: point 101 placed at the middle of the upper cross section's edge and point 102 placed at the middle of the down edge.

Menu: **Section→Stress location**
Icon: **Stress location**

Command:

*New stress location point [cm]:**0,-10*** [use the temporary object snap toolbar to point middle of the upper and down edges or input the coordinates of the points]

The Stress location window appears on the screen.



The Name should be set to 101. Entering the coordinates of the next points should continue the process.

Command:
New stress location point [cm]: ***0,-44***
New stress location point [cm]:
[Press Entry two times to stop entering]
All the points entered by one Stress location command should have the same name, in our example 101. The number, the material and the location of the points can be changed with double click on the point. To change the number of the down point, double click on the point. The Stress location window will appear and the number and the material should be change.

2.2.7. Shear Cuts

The user uses the command CUT to define a so-called cut through the sectional geometry where a check of the shear stresses should take place. Each cut is assigned identification with three characters. The cut can be defined as parallel to an axis or as a freeform polygon line. Every segment has its own material number and it will cut only through cross-section elements with the same material number. Gaps between the segments will be closed by means of virtual connections. A width of the substitute torsional cross-section is available as a special option for the description of equivalent hollow cross-sections of reinforced and pre-stressed concrete. Two partial cuts are generated for each section in this case.
If the user does not supply any input, one or two axis-parallel cuts will be created through the gravity center. This generally is not sufficient even for a simple T-Beam, or for composite sections, where the reference material number of the section is not necessarily represented at that location. The user will see a warning for general sections therefore.

In the Example 2, one shear cut section will be defined. It will split the cross section through the centers of the holes.

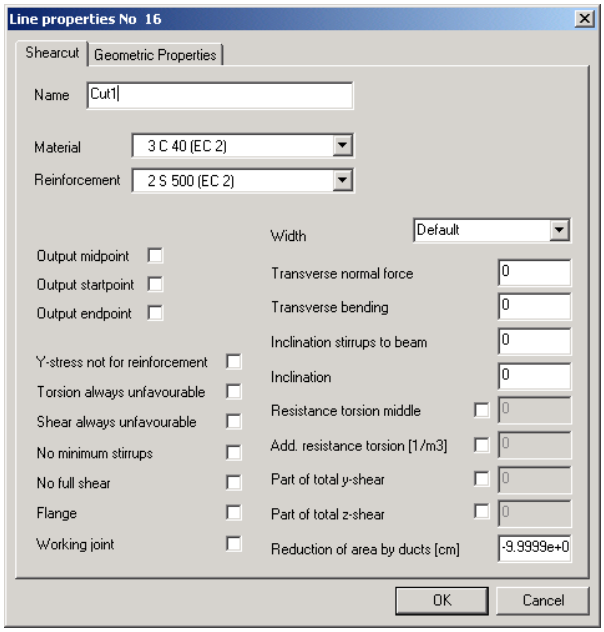
Menu: **Section→Shear cut**
Icon:  **Shear cut**

Command:

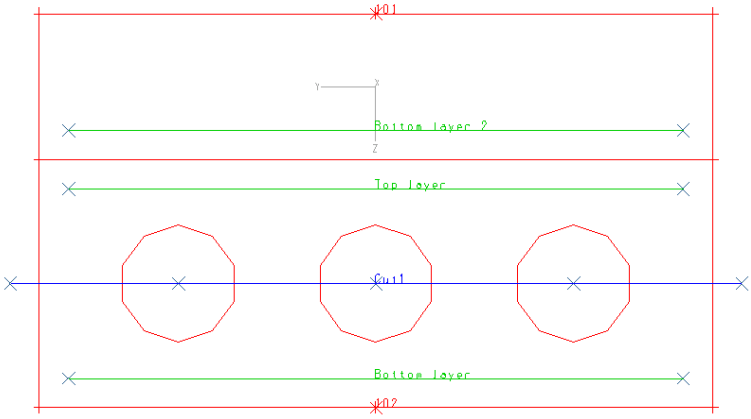
New shear cut, first point [cm]: ***-50,-27***
New shear cut, first point [cm]: ***50,-27***
New shear cut, first point [cm]:

[Press Esc to stop entering]

The shear cut section should be named. Double click on the shear cut section opens the Line property window that allows the user to enter or change the name and some other properties.



The effects of the last two commands are shown on the figure that follows.



2.2.8. Calculation – cross section property

After section creation, the cross section properties should be calculated. There are to commands that calculate cross section properties.

- Calculate - only the property of the actual cross section will be calculated

- Calculate all - all sections in the database will be calculated

In this example the command Calculate is activate and the property of the cross section are calculated. The results of the calculation are shown at the following

```
CROSS-SECTION NO     1

Static properties of cross section
 NNo       A|m2| Ay/Ax/Ayx Iy/Ix/Iyx   ys/xs  y/x-sc   modules      gam
 NNs      It|m4|       |m2|       |m4|    |cm|    |cm|      |MPa|   |kN/m|
   3 4.2270E-01           1.090E-02    0.00    0.00      34525   10.91
   2  9.571E-03           3.104E-02   16.13   16.46      14386

Additional static properties of cross section
  Alfa-I      ymin      xmin    hymin        AK   NE      Tau-I     Tau-Vy
              ymax      xmax    hxmin        AE           Tau-E     Tau-Vx
              |cm|      |cm|     |cm|       |m2|         |1/m3|     |1/m2|
 1.0E-05    -46.00    -26.13          2.59E-01   2  1.77E+01  1.89E-07
             46.00     27.87          4.36E-01                5.46E+00

Partial cross sections
 NNo       A|m2| Ay/Ax/Ayx Iy/Ix/Iyx   ys/xs   modules      gam
 NNs      It|m4|       |m2|       |m4|    |cm|      |MPa|   |kN/m|
   3 2.5248E-01           2.917E-03    0.00      34525    6.31
   2  9.571E-03           1.903E-02   27.00      14386

   1 1.8400E-01 3.367E-01 6.133E-04    0.00      31939    4.60
   2  0.000E+00 3.200E-01 1.298E-02    0.00      13308

Design values of cross section
 NNo       A|m2| Ay/Ax/Ayx Iy/Ix/Iyx   ys/xs   modules      gam
 NNs      It|m4|       |m2|       |m4|    |cm|      |MPa|   |kN/m|
   3 4.2270E-01 3.114E-01 1.090E-02    0.00      23017   10.91
     9.571E-03 2.960E-01 3.104E-02   16.13       9590

Additional Design Data
  circum-O  circum-I   t-min   t-max am.parts    thet-p   thet-y   thet-x  thet-yx
    |m2/m|     |m2/m|    |cm|    |cm|     |o/o|   |tm2/m|   |tm2/m|   |tm2/m|   |tm2/m|
     4.760     1.508                      0.0     1.083    0.282    0.800    0.000

Reinforcement global values
Layer mS  mR      area  lower-A  upper-A      yL      xL     L-tocs      M-pc     M-pc
                 |cm2|     |cm2|     |cm2|    |cm|     |cm|      |cm|      |kN|    |kNm|
   0   3   2     33.6     33.6              0.00   25.00
```

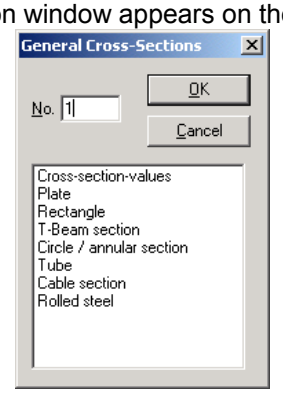
2.2.9. Using already defined cross section

Already defined cross section can be used in the process of creating the new section. Using Cut and Paste command one can simplify creation of the new section that has similar form with already defined sections. In this example the same cross section will be created using previously defined cross sections.

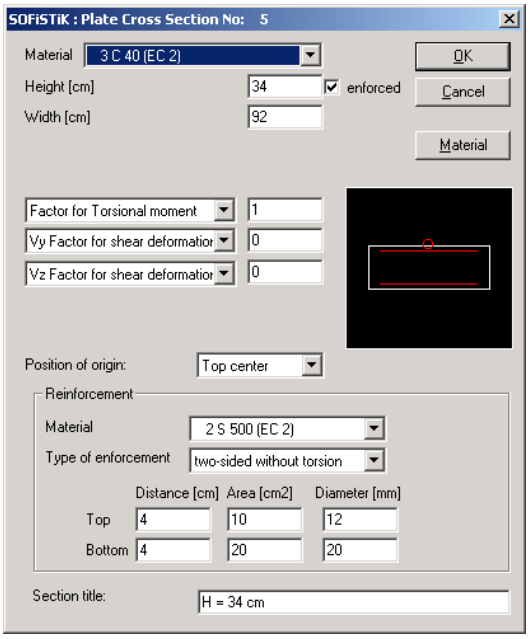
Step 1 - The down part of the cross section (plate with holes)
The standard plate cross section will be use to start creation of this part.

Menu: Edit→New section (standard)…
Icon: New section (standard)

The General Cross-Section window appears on the screen.



The number of selection is set to 1. After selections of the plate type the window SOFiSTiK: Plate Section will be ready to accept the property of the section



Some of the properties should be changed:

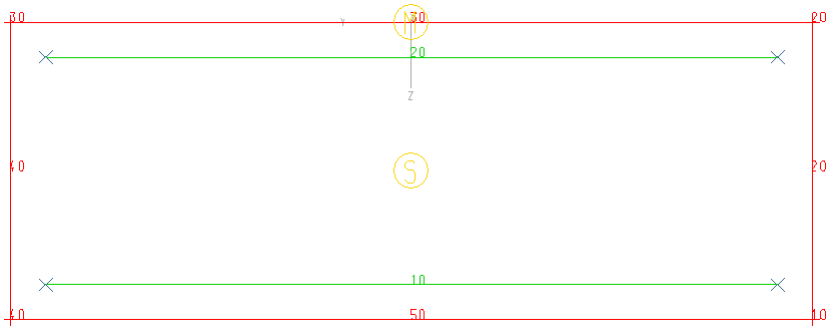
Material: **[3 C 40 (EC2)]**
Height: **34**
Width: **92**

Position of origin: Top center
Reinforcement

Distance	Area	Diameter	
Top	**4**	**10**	**12**
Bottom	**4**	**20**	**20**

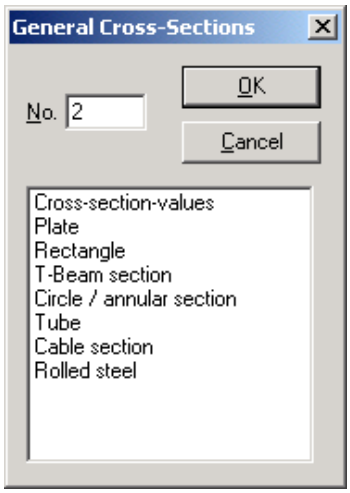
The other parameters should remain unchanged.

The section appears in the graphical area of the WinAqua environment.

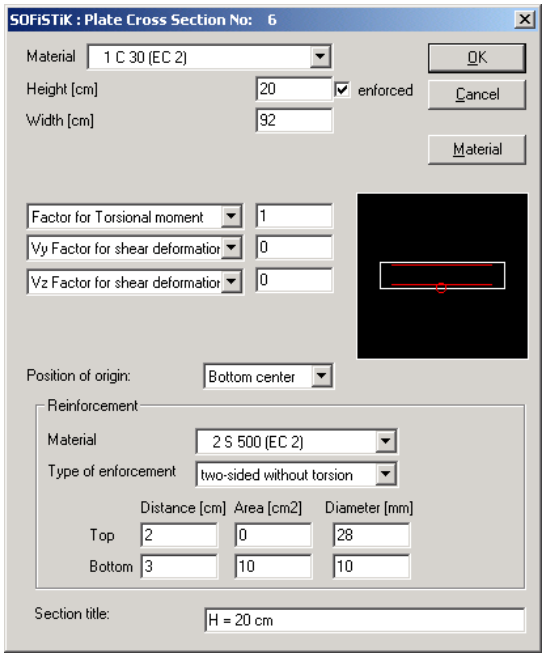


In the similar way the upper part of the section is created

Menu: Edit→New section (standard)…
Icon: New section (standard)



The general cross section Plate should be selected.



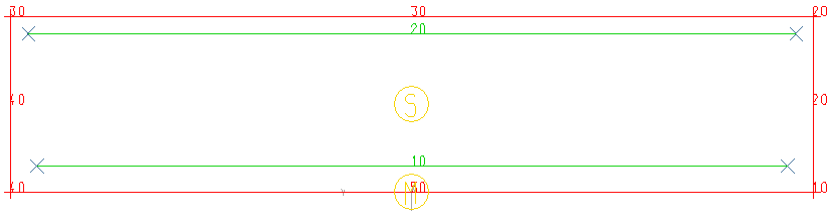
Same of the property should be changed:

Material: **[1 C 30 (EC2)]**
Height: **20**
Width: **92**
Position of origin: **Bottom center**

Reinforcement

	Distance	Area	Diameter
Top	**2**	**0**	**28**
Bottom	**3**	**10**	**12**

The upper part of the section is created.

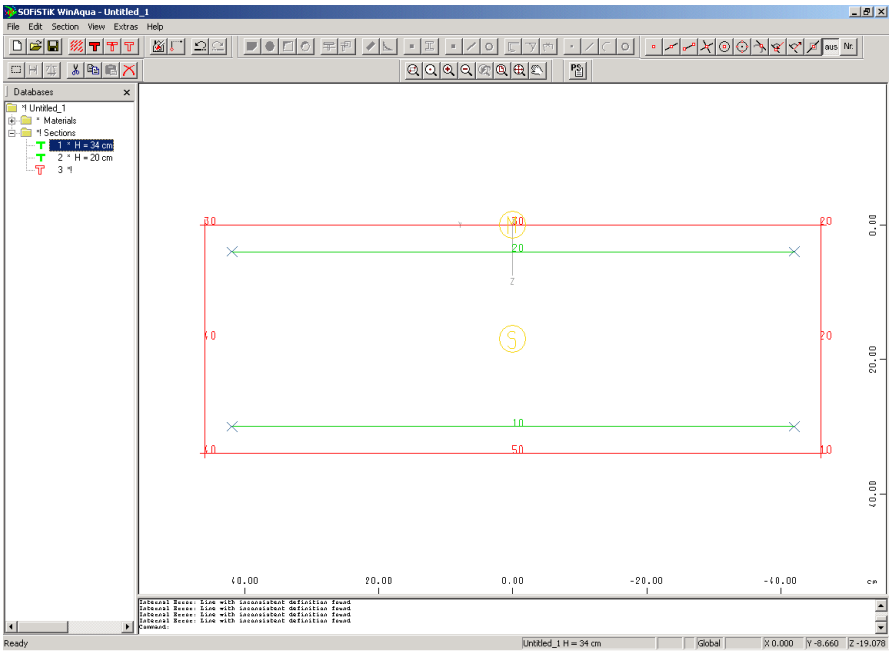


In the next step the standard cross sections, previously created, will be used to create the composite section of the Example 2.

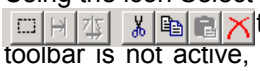
New section type “**thick walled**” should be started.

Menu: Edit→New section (thick walled)…
Icon: New section (thick walled)

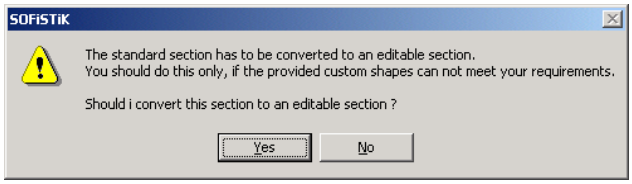
Using “**Copy**” and “**Paste**”, the standard sections will be copied into the new thick walled section. In the Database, section 1 should be selected.



Using the icon Select with box , from the toolbar Selection



the cross section will be selected. If the Selection toolbar is not active, one of the graphical objects should be select (simply mouse click). That will made the Selection toolbar active. Pressing the icon copy will put the selected section into the clipboard. In this moment a window dialog box will appears on the screen

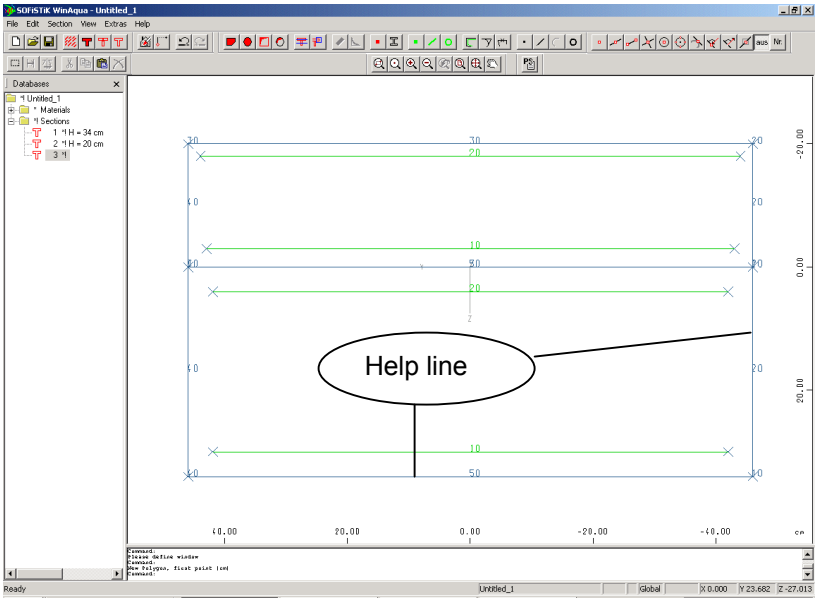


The “**Yes**” command button should be pressed to convert the standard section into an editable section. After that the content of the clipboard should be pasted into the section 3.

Select the section 3 in the Database

Paste the clipboard into the graphical editor.

During the “**Copy**” and “**Paste**” process some properties of the standard cross section should be missed. The contour of the cross section should be turned in the help line.

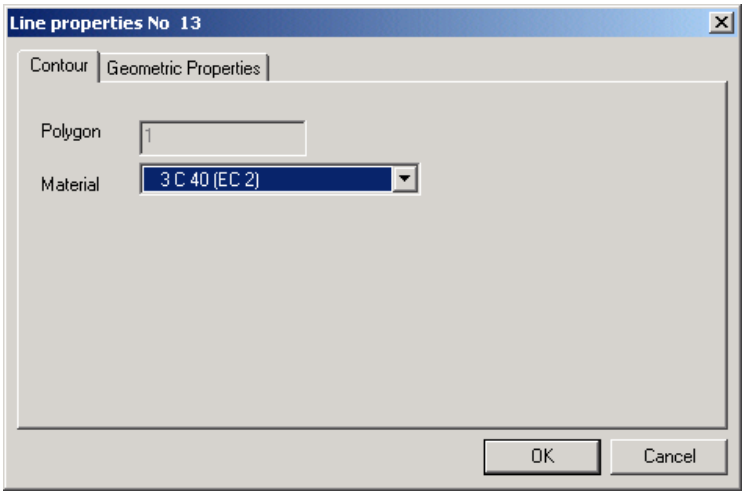


The user must once again to draw the contour of the plate and set their property.

Menu: Section→Contour
Icon: Contour

The contour of the down plate should be redrawn over the help line.

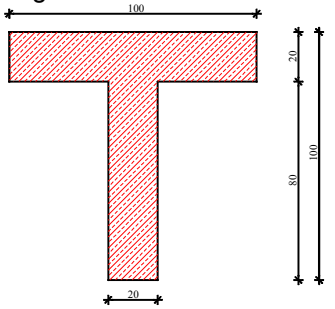
In the next steps the material for that contour should be set. Double click on one of the contour's segment will activate the Line properties windows. Sometimes WinAqua selects the help line instead of contour. In that case after drawing the contour one help line should be erased and the contour's segment that pass over this line should be selected.



In the Material combo appropriate material is selected.
The process repeats with upper part of the section
At the end the three holes should be inserted into the section using the standard procedure.

3. T-beam

The program WinAqua should create two T-beam cross sections. Geometrical characteristics of the first T-beam section are given on the Fig.3.1. Second T-beam section is given on the Fig.3.2 and this section has the same geometrical characteristics as the first section but here we will have non-effective part on the flange. Both sections are made of B55 concrete and standard reinforcement BST 500. The position of the reinforcement is shown on the Fig.3.3 for the first T-beam section and on the Fig.3.4 for a second cross section.



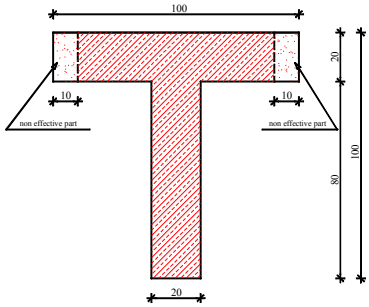
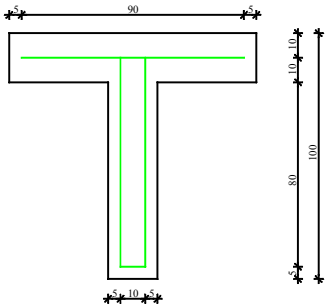


Fig. 3-1T-Beam section Fig. 3-2 T-Beam section with non-effective part



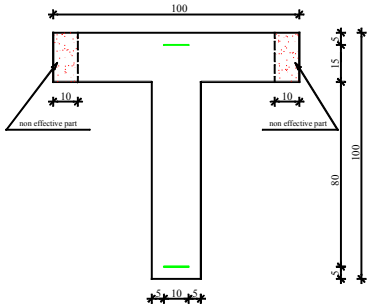
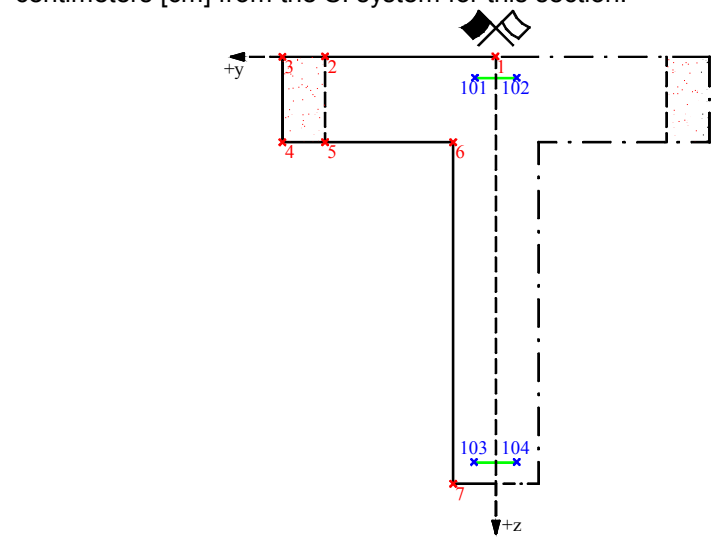


Fig. 3-3 Reinforcement *Fig. 3-4 Reinforcement*

Section shown on the Fig.3.1 will be drawn using "**standard cross sections**" from WinAqua. Because of that user do not need to do some previous preparations for drawing units. By default WinAqua will set units to meters [m] from the SI system for this section.

To draw section shown on the Fig.3.2 coordinate beginning will be taken as it is shown on Fig.3.5. The Cross section is symmetric so we will use mirror option to draw symmetrical part of a section. We will select and use units' centimeters [cm] from the SI system for this section.



x Section point
x Reinforcement point
Contour of a section
Contour which will be drawn with mirror command
Axes

Fig. 3-5 Position of the drawing

The coordinates of the points with which the geometry of the T-beam section is defined are given in the Table 2.1, and the coordinates of the points with which the line reinforcement is defined are given in the Table 2.2

Table 3-1

Point	Coordinate	
	Y	Z
1	0	0
2	40	0
3	50	0
4	50	20
5	40	20
6	10	20
7	10	100

Table 3-2

Point	Coordinate		As
	Y	Z	cm^2
101	8	5	125
102	-8	5	
103	8	95	62.5
104	-8	95	

3.1. Solving the problem with WinAqua

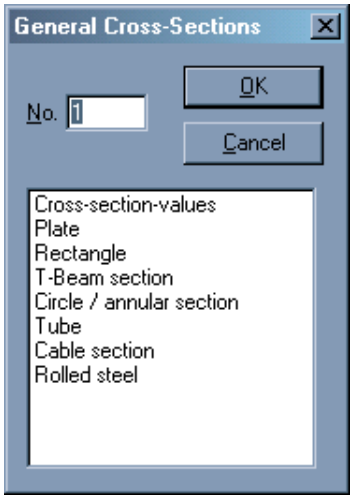
3.1.1. Defining default units and materials in database file (and creating a new database file)

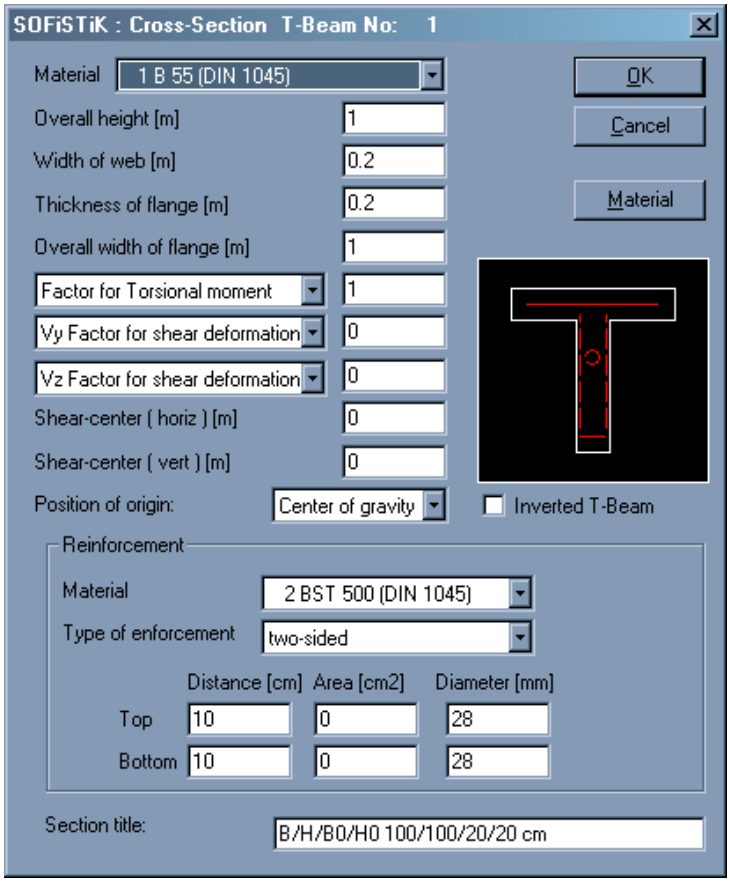
This step was described in details in Example 1. For section shown on Fig2.1 we will use standard WinAqua section and for this option there is no need to define the units because the option uses default units meters [m]. Do not forget to change default materials form B 25 to B 55 as it was described in step 2 from Example 1.

3.1.2. Drawing section 1 (shown on Fig2.1) using standard sections

From menu "**Edit**" select "**New section [standard]…"** or select icon from toolbar "**CDBase Toolbar**".

WinAqua will open a new "**General Cross-sections**" window. User can here define number "**No**" of the cross section, with which section will be stored in a database. User selects the type of a section. Depending on a selected type, different window will appear after pressing button "**OK**". For Example 2 user should select "**T-Beam section**". A new window will appear which will help user to define geometry of the T-beam section.





To define geometry of a section see Fig2.6.

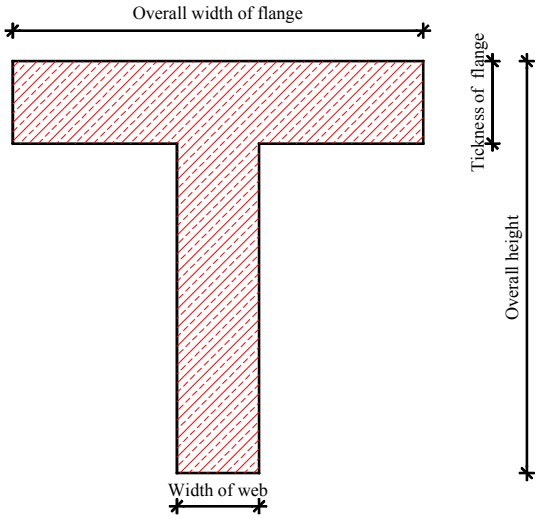
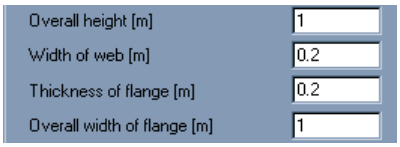


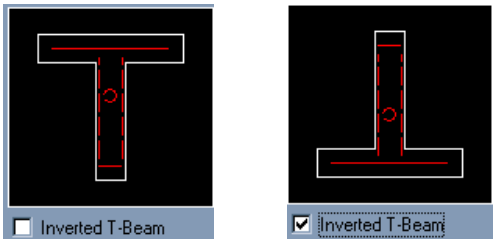
Fig. 3-6 Parameters for defining T-Beam section

From this option user can select the type of material for the section. In this list box only materials previously defined in step 1 will be stored.



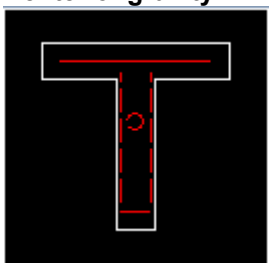
Parameters for defining T-beam geometry

Using Invert T-beam

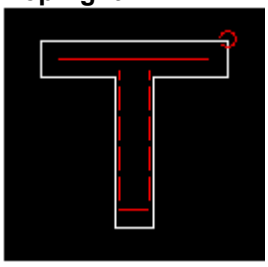


Using position of origin

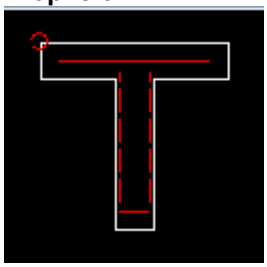
Center of gravity



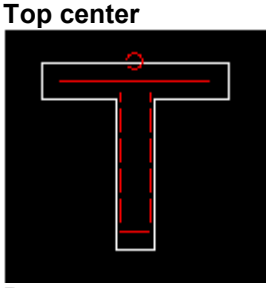
Top right



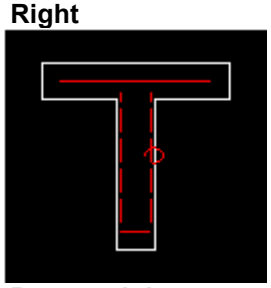
Top left



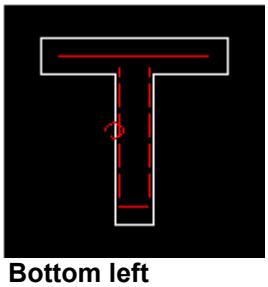
Top center



Right



Left



Bottom center

Bottom right

Bottom left

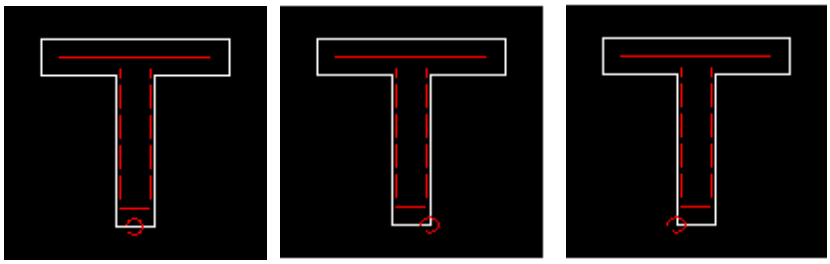
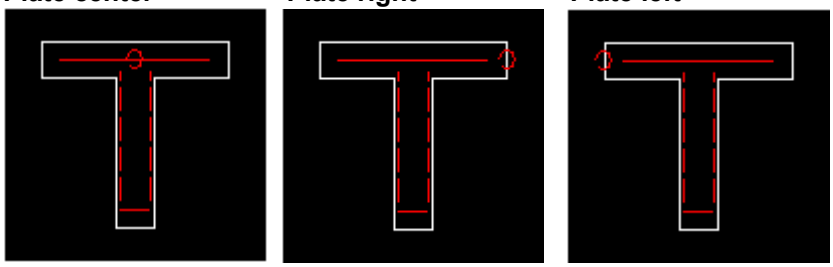
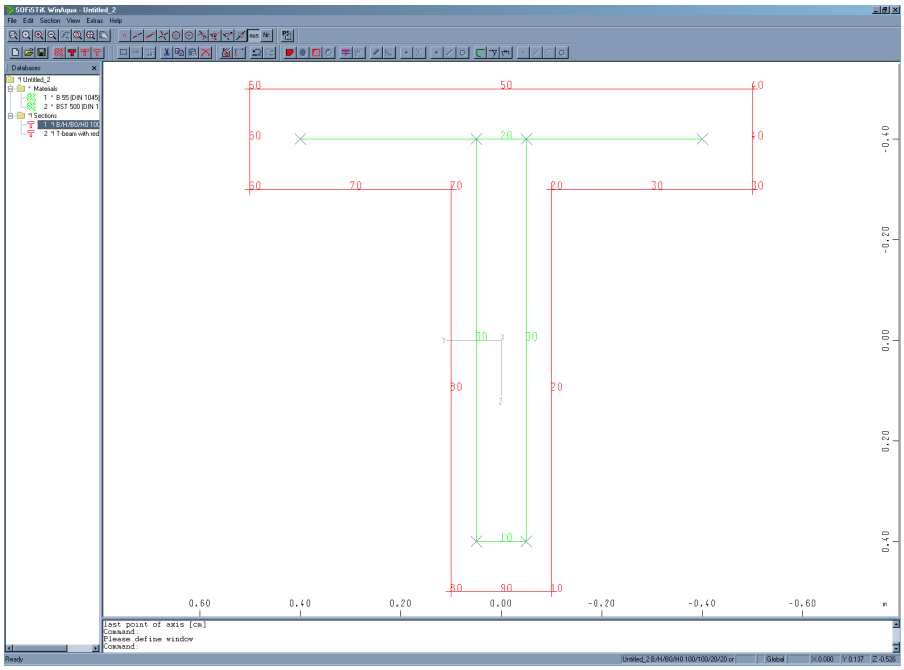


Plate center **Plate right** **Plate left**



For more details about other options in this window see Aqua manual paragraph **3.17. SREC - Rectangle, T-beam, Plate.**



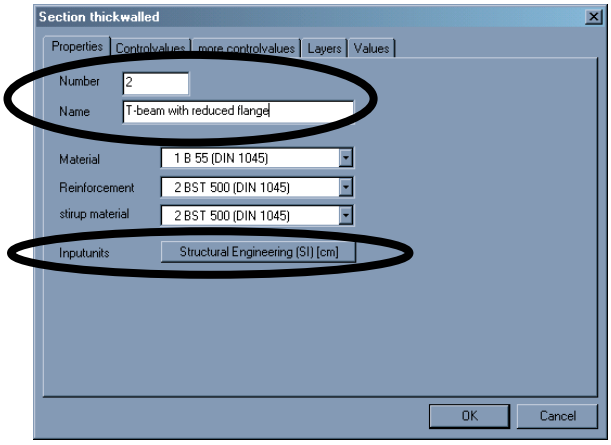
Linear reinforcement is automatically defined with the parameters, which were given in the **Cross section T-beam** window.

Note: "**Edit section toolbar**" is unavailable if you create a section using "**Standard cross section**".

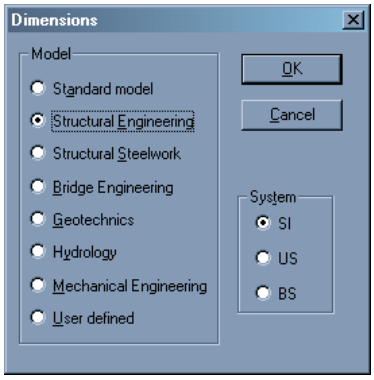
3.1.3. Drawing section 2

(shown on Fig3.2) using thick walled section

From **Edit** menu select "**New section [thick walled]…**". Set text box "**Number**" to ***2*** and set section name to "***T-beam with reduced flange***".



Click on the button by the "**Inputs"** and select units [cm] as it is shown on the next figure.



A new drawing area will be opened ready for inputting a new thick wall section.

Start the command "**Couture**" from the menu "**Selection**" or by selecting icon form '  **it section**" toolbar. Enter the coordinates for section 2 given in Table 2.1.

Command:

New Polygon, first point [cm] ***0,0***
New Polygon, next point [cm] ***40,0***
New Polygon, next point [cm] ***50,0***
New Polygon, next point [cm] ***50,20***
New Polygon, next point [cm] ***40,20***
New Polygon, next point [cm] ***10,20***
New Polygon, next point [cm] ***10,100***

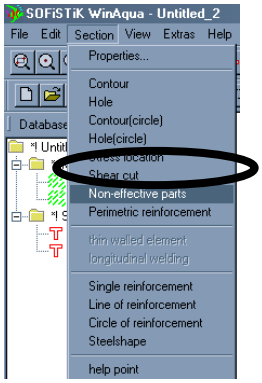
[Press right mouse button and select Mirror all points]

WinAqua will ask you to define the mirror line

First point of axes or axes [cm] ***0,0***
Last point of axes or axes [cm] ***0,100***
Command:

3.1.4. Defining non-effective part of a section

To defining non-effective part of a section select the menu “**Section**” and select “**Non-effective parts**” command or select icon from **Edit section toolbar.**



WinAqua will ask from the user to define rectangular area where the non-effective area is placed. Non-effective rectangular area is defined with two diagonal points (defining diagonal of a rectangular) see Fig 3.7.

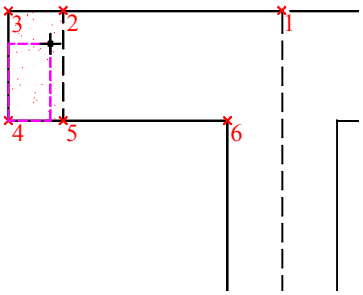


Fig. 3-7

There are two ways to define non-effective area. User can define non-effective area by giving the coordinates with clicking the mouse in a drawing area, or by typing the coordinates in the command window.

User will define the first non-effective area (on the left side of flange) in this example by using the mouse and for the second non-effective area (on the right side of flange) user will define it using the coordinates typed on the command window.

3.1.4.1. - First non-effective area on the right side of flange

Start the command “**Non-effective parts**” as it was described above. Activate “**end point**” snap from “**temporary object snap**” toolbar or by pressing **F3** key. Approach with the mouse towards the point 4 in a WinAqua drawing. Press the left mouse button and WinAqua will

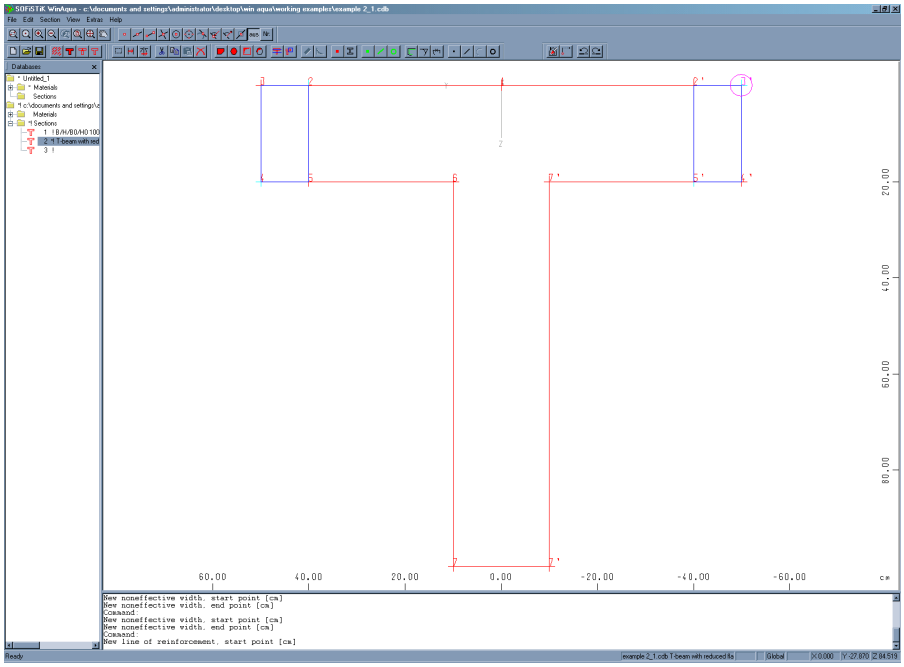
automatically place the first corner of non-effective area in the point 4. Activate “**end point**” snap again and approach towards point 2. Press the right mouse button again and WinAqua will define the last point of non-effective area.

3.1.4.2. - Second non-effective area on the left side of flange

Start the command “**Non-effective parts**” as it was described above. In command prompt type the coordinates of the point 5’ and after that coordinates of the point 3’.

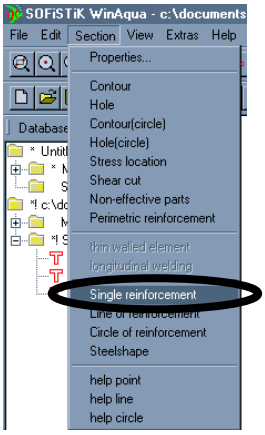
Command:
New noneffective width, start point [cm] –40,20
New noneffective width, end point [cm] –50,0

This is the second way to define non-effective parts of a cross section.



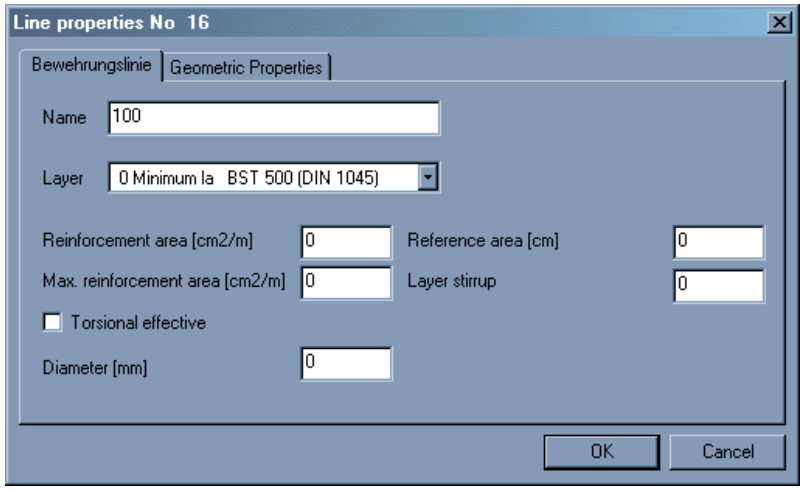
3.1.5. Defining linear reinforcement

To define linear reinforcement use the command “**Single reinforcement**” from menu “**Section**” or click on the icon from the “**Edit section**” toolbar.



Linear reinforcement is defined with coordinates of the two points (at the beginning and in the end of linear reinforcement line). User can enter these coordinates by typing them in the command line or by clicking with the mouse in the drawing area.
Start the command “**Linear reinforcement**” as it was described above.
Enter the coordinate given in the Table 2.1.
Command:
New line of reinforcement, start point [cm] 8,5
New line of reinforcement, start point [cm] -8,5

A window will appear that will ask the user to define the name of the reinforcement line (which will be used in further stress and reinforcement analyses), Reinforcement area [cm^2/m], Diameter etc.



For more detailed description of parameters in this window see aqua manual paragraph **3.37. Reinforcement.**

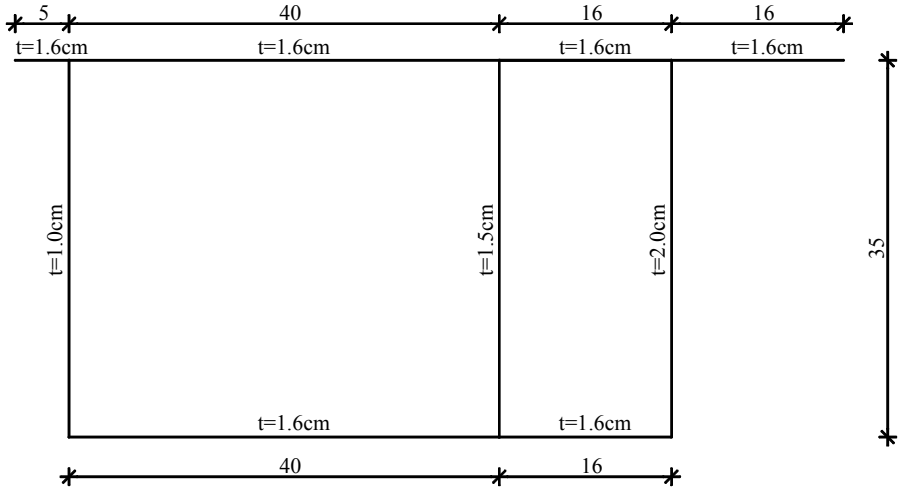
In the same way linear reinforcement will be defined at the bottom section.

3.1.6. Saving and calculating section database (This step was described in step 5 and 6 in Example 1.)

Note: For calculation use the command "**calculate all**". This way the section 1 and section 2 will be calculated. If user uses the command "**calculate**", then only the active section will be calculated.

4. Thin-walled steel box

A twin walled non-symmetric steel box with different plate thickness is examined. Geometrical characteristics of the thin walled steel box are given on the Fig.4.1. The section is made of structural steel S37 (DIN 17100).



(t- thick of a thin wall element) Fig. 4-1Cross section

For this section user should define coordinate beginning and all characteristic points that define thin-walled steel box. For this example we will select and use units' meters [m] from the SI system for this section. In WinAqua thin-walled steel box will be defined as it is shown on the Fig4.2. For easier stress analysis we recommend you to define name of a thin wall element as is shown on the Fig4.2.

The coordinates of the points with which the geometry of the T-beam section is defined are given in the Table 4.1

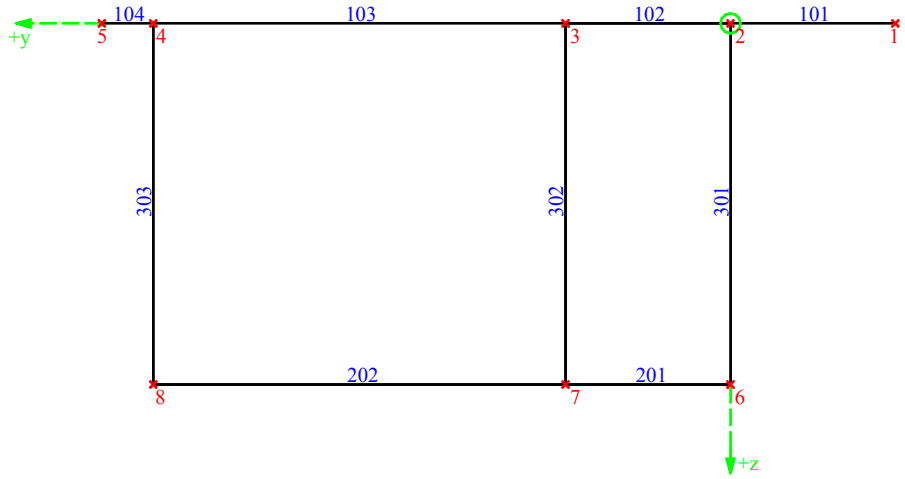


Fig. 4-2

Table 4-1

Element	Point	Coordinate	
		Y	Z
101	1	-0.16	0.00
	2	0.00	0. 00
102	2	0.00	0. 00
	3	0.16	0. 00
103	3	0.16	0. 00
	4	0.56	0. 00
104	4	0.56	0. 00
	5	0.61	0. 00
201	6	0.00	0.35
	7	0.16	0.35
202	7	0.16	0.35
	8	0.46	0.35
301	2	0.00	0. 00
	6	0.00	0.35
302	3	0.16	0. 00
	7	0.16	0.35
303	4	0.56	0. 00
	8	10	100

4.1. Solving the problem with WinAqua

4.1.1. Defining default units and materials in database file (and creating a new data base file)

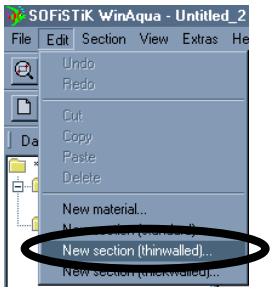
This step was described in details in Example 1.

Do not forget to change default materials from structural steel S 235 (DIN 18800) to structural steel S 37 (DIN 17100) as it was described in step 2 from Example 1.

4.1.2. Drawing section using thinwalled sections

From menu Edit select New section [thinwalled]… or select

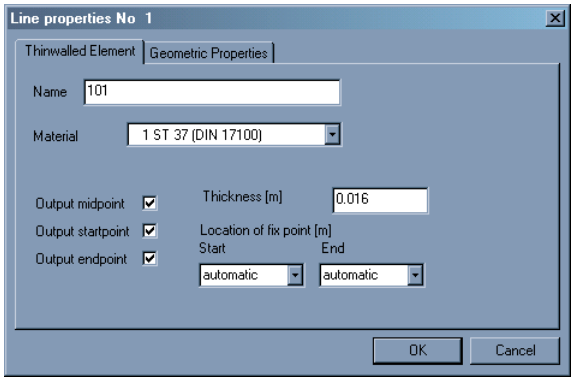
icon from toolbar **CDBase Toolbar**.



WinAqua will ask the user to define two points (start point and end point of a thin walled element). Enter coordinate given in the Table 4.1.

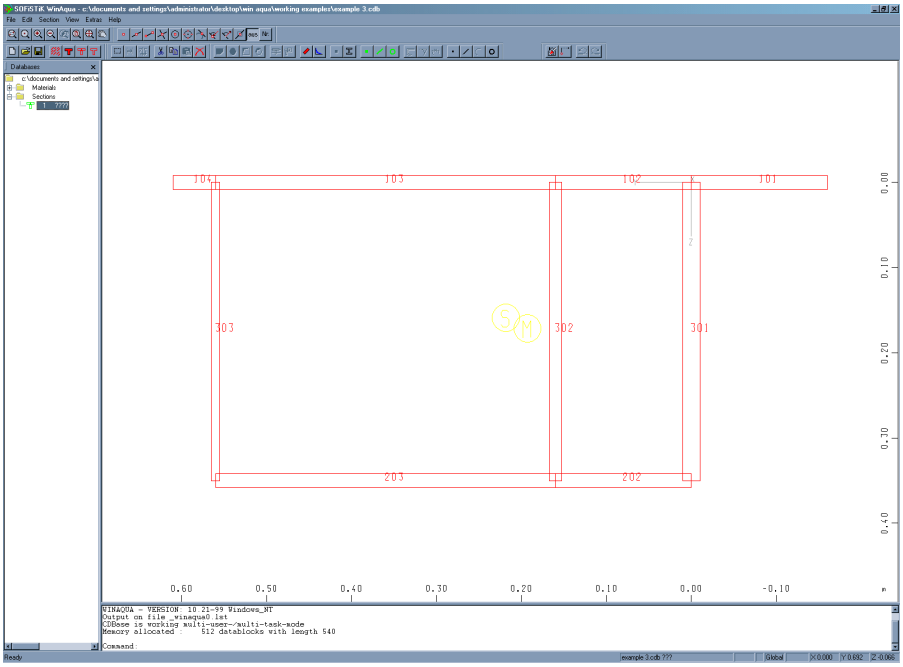
Command:
New thin walled element, start point [m] ***–0.16,0***
New thin walled element, end point [m] ***0,0***

A new window will appear asking the user to input name of a thin-walled element, thickens of a element.



Enter the name of the first thin walled element, as described in the Fig4.2 (***101***) and input the thickens of a element as is was shown in the Fig4.1 (***0.016***).

Repeat this procedure for all element defined in table 4.1

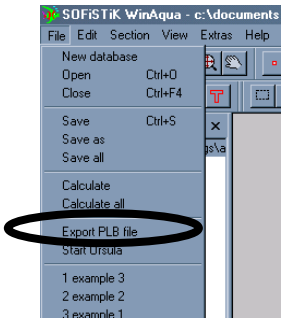


4.1.3. Saving and calculating section database

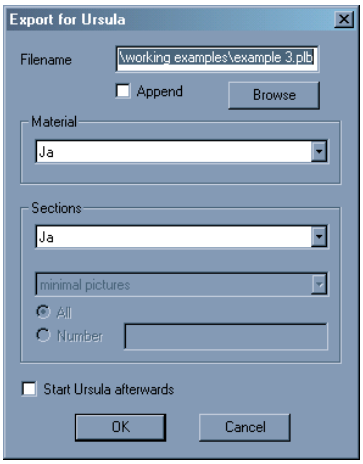
This procedure was described in step 5 and 6 in Example 1.

4.1.4. Exporting result in to plb file and using program Ursula

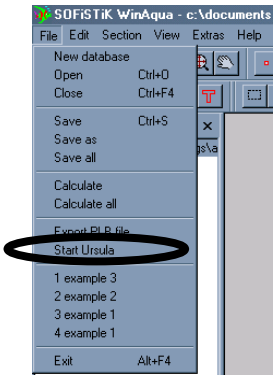
For more detailed result reports user should use program "**Ursula**". To use "**Ursula**" first you have to create "***.plb**" file. With command "**Export PLB file**" WinAqua will create "**name.plb**" file by exporting already created "**name.cdb**" file. From "**File**" menu select command "**Export PLB file**".



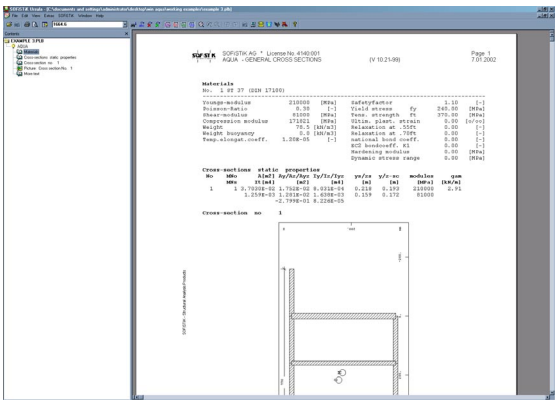
After activating command “**Export PLB file**” a new window “**Export for Ursula**” will appear. Here user can select export parameters.



After creating file “**name.plb**” user can view the result in Ursula. From “**File**” menu select command “**Start Ursula**”.



Program Ursula will start.



This is a detail report from all calculation made in WinAqua.

Materials

No. 1 ST 37 (DIN 17100)

Youngs-modulus	210000	[MPa]	Safetyfactor		1.10	[-]
Poisson-Ratio	0.30	[-]	Yield stress	fy	240.00	[MPa]
Shear-modulus	81000	[MPa]	Tens. strength	ft	370.00	[MPa]
Compression modulus	171821	[MPa]	Ultim. plast. strain		0.00	[o/oo]
Weight	78.5	[kN/m3]	Relaxation at .55ft		0.00	[-]
Weight buoyancy	0.0	[kN/m3]	Relaxation at .70ft		0.00	[-]
Temp.elongat.coeff.	1.20E-05	[-]	national bond coeff.		0.00	[-]
			EC2 bondcoeff. K1		0.00	[-]
			Hardening modulus		0.00	[MPa]
			Dynamic stress range		0.00	[MPa]

Static properties of cross section

MNo	A[m2]	Ay/Az/Ayz	Iy/Iz/Iyz	ys/zs	y/z-sc	modules	gam
MNs	It[m4]	[m2]	[m4]	[mm]	[mm]	[MPa]	[kN/m]
1	3.7030E-02	1.752E-02	8.031E-04	218.2	192.6	210000	2.91
	1.259E-03	1.281E-02	1.638E-03	159.1	171.5	81000	
		-2.799E-01	8.226E-05				

Main axis of inertia rotated at -84.42 [grad]

Main moments of inertia 1.6456E-03 7.9511E-04 [m4]

Additional static properties of cross section

Alfa-T	ymin	zmin	hymin	AK	MB	Tau-T	Tau-Vy
	ymax	zmax	hzmin	AB		Tau-B	Tau-Vz
	[mm]	[mm]	[mm]	[m2]		[1/m3]	[1/m2]
1.2E-05	-378.2	-167.1		1.90E-01		2.66E+02	6.91E+01
	391.8	198.9				1.46E+03	8.80E+01

Section values for warping

Wmin[m2]	Wmax[m2]	CM[m6]	CMS[m4]	ASwyy[m6]	ASwzz[m6]	ry[mm]	rz[mm]
-8.024E-03	1.942E-02	8.004E-07	5.488E-05	2.073E-06	2.285E-07		

Design values of cross section

MNo	A[m2]	Ay/Az/Ayz	Iy/Iz/Iyz	ys/zs	modules	gam
MNs	It[m4]	[m2]	[m4]	[mm]	[MPa]	[kN/m]
1	3.7030E-02	1.752E-02	8.031E-04	218.2	190909	2.91
	1.259E-03	1.281E-02	1.638E-03	159.1	73636	
			8.226E-05			

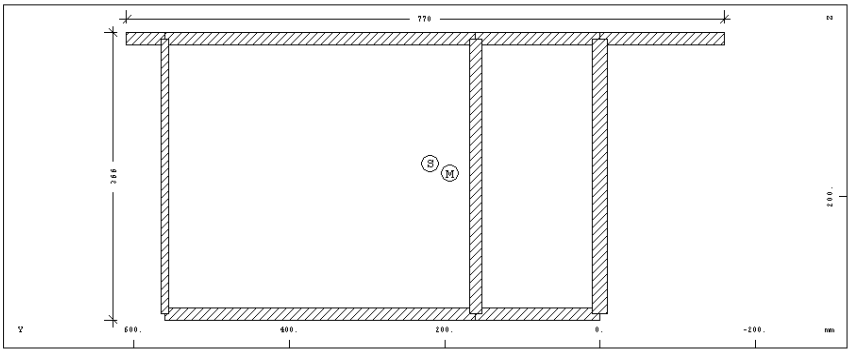
Full plastic force and moments

(K/D = characteristic/design)

	N[kN]	Vy[kN]	Vz[kN]	Mt[kNm]	My[kNm]	Mz[kNm]	y[mm]	z[mm]	BUCK
K	8887.2	2948.64	2182.38	536.82	1209.46	1548.79	163.2	137.7	B B
K	-463.4				1214.43	0.00	0.0	159.1	COMB
K	1573.5				0.00	1612.14	218.2	0.0	COMB
D	8079.3	2680.58	1983.99	488.02	1099.51	1407.99	163.2	137.7	
D	-421.3				1104.03	0.00	0.0	159.1	COMB
D	1430.5				0.00	1465.58	218.2	0.0	COMB

Additional Design Data

circum-O	circum-I	t-min	t-max	sm.parts	thet-p	thet-y	thet-z	thet-yz
[m2/m]	[m2/m]	[mm]	[mm]	[o/o]	[tm2/m]	[tm2/m]	[tm2/m]	[tm2/m]
4.760		10.0	20.0	0.0	0.192	0.063	0.129	0.006



5. Polygonal box section

The program WinAqua can be used to create polygonal box section as it is shown on the Fig.5.1

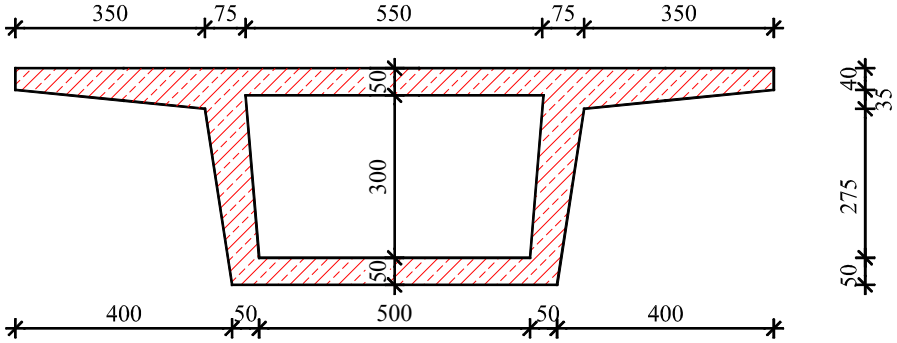


Fig. 5-1Cross section

The polygonal box section is made of B25 concrete. User should analyze shear stress in four characteristic cuts as shown on the Fig.5.2.

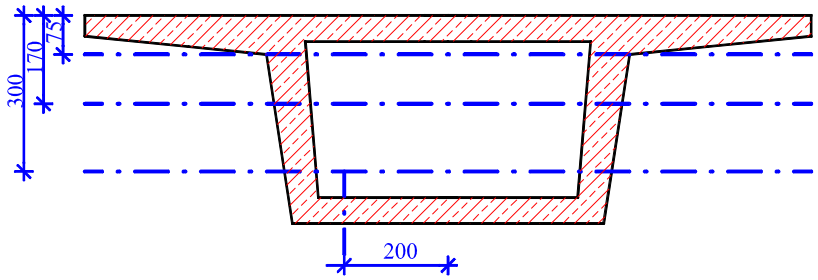


Fig. 5-2 Position of shear cut

The section will be oriented as shown on the Fig5.3

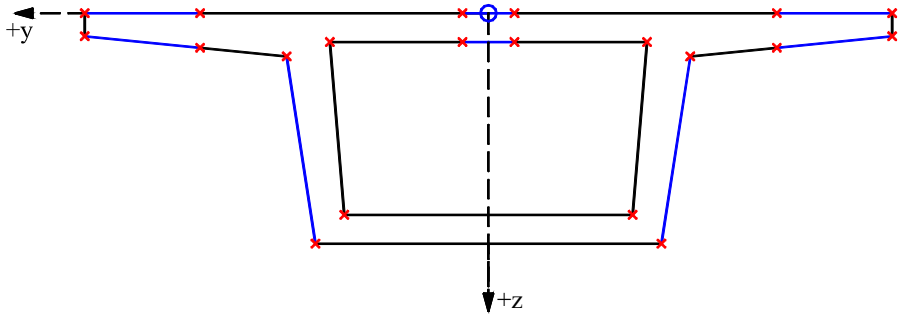


Fig. 5-3 Orientation of the cross section

For this example user should use units meters [m] from the SI system. To define geometry in WinAqua, program AutoCAD2000 will be used. User should draw section in AutoCAD in y-z plane as shown on the Fig5.4. All units must be in meters [m].

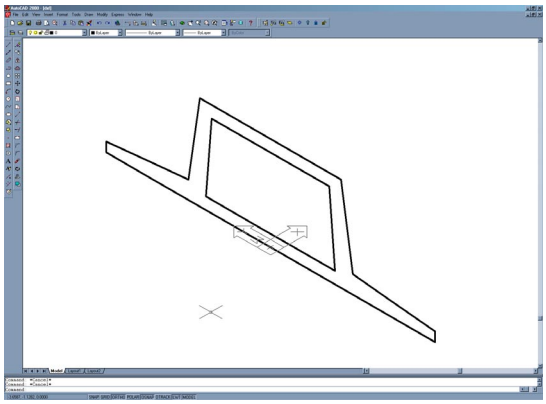
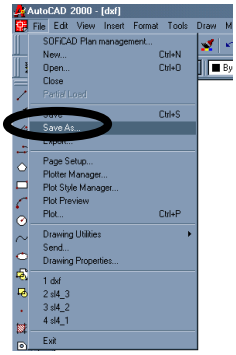


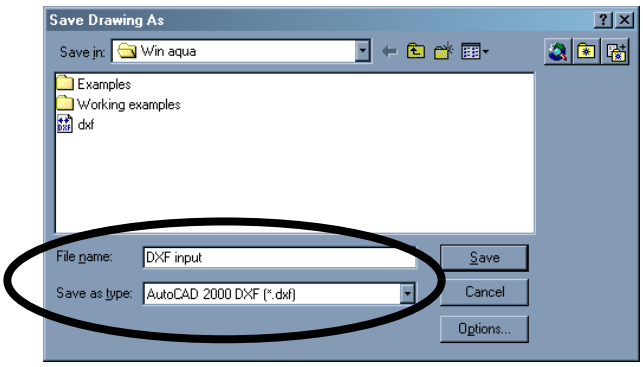
Fig. 5-4 Cross section in AutoCAD

This drawing should be exported in ***dxf*** file. Procedure for exporting AutoCAD drawing in dxf file is:

- From "**File**" menu select "**Save As**".



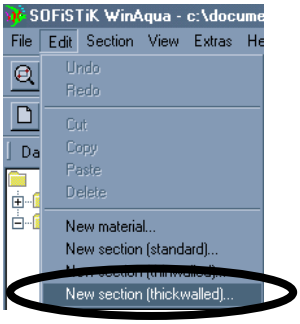
- Select a name for exported drawing (for example **DXF input**) and select "**AutoCAD 2000 DXF (*.dxf)**"save as type.



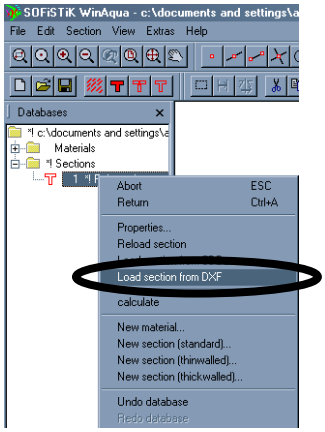
5.1. Solving the problem with WinAqua

5.1.1. Drawing polygonal box section in WinAqua

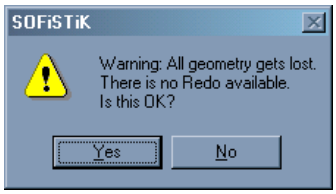
After creating database file from the "**Edit**" menu select "**New section[thick walled]…**" or select icon from CDBase Toolbar



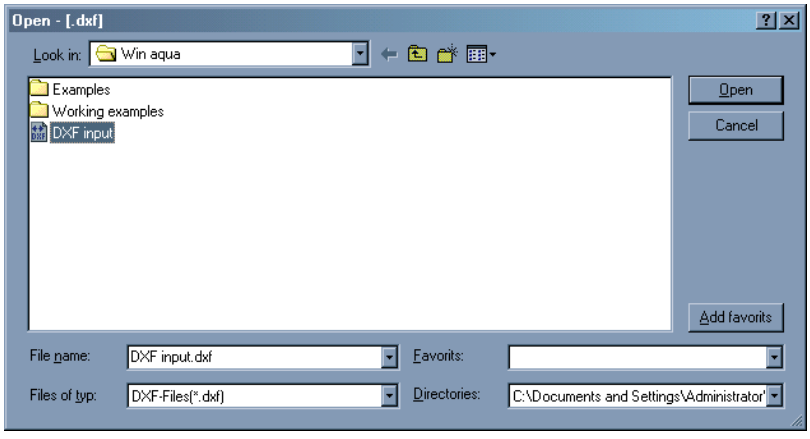
A new item will appear in database window under "**Section**" folder. Point with the mouse on a section name in database window and click on the right mouse button. A new floating menu will appear. Select command "**Load section from DXF**"



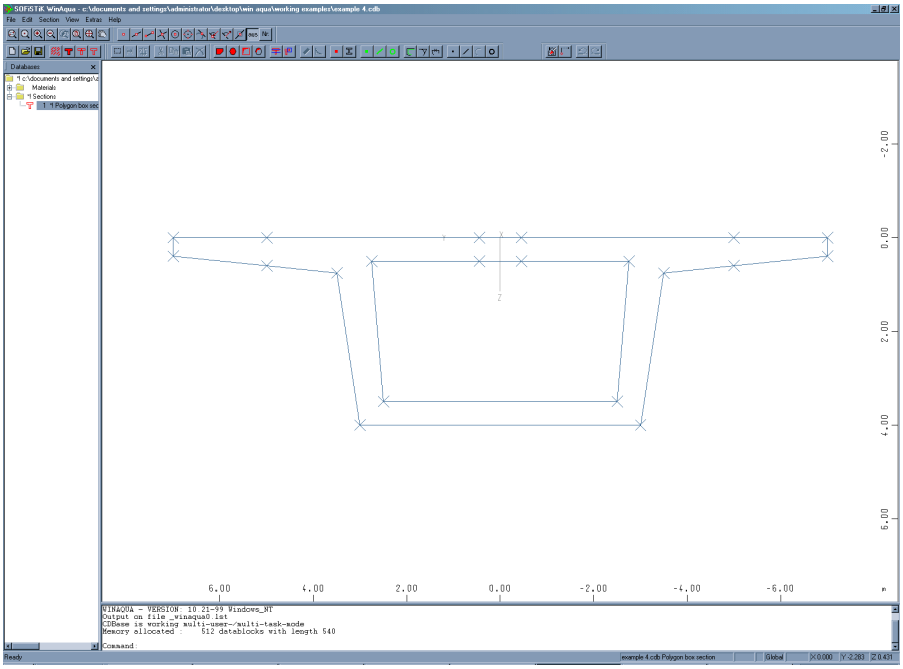
Warning message will appear. WinAqua warns user that all previously drawn line in selected section from database will be deleted.



Import command will be canceled if user selects "**No**".If button "**Yes**"is selected WinAqua will ask for path and name of dxf file. Find and select previously created "**DXF input.dxf**" file.

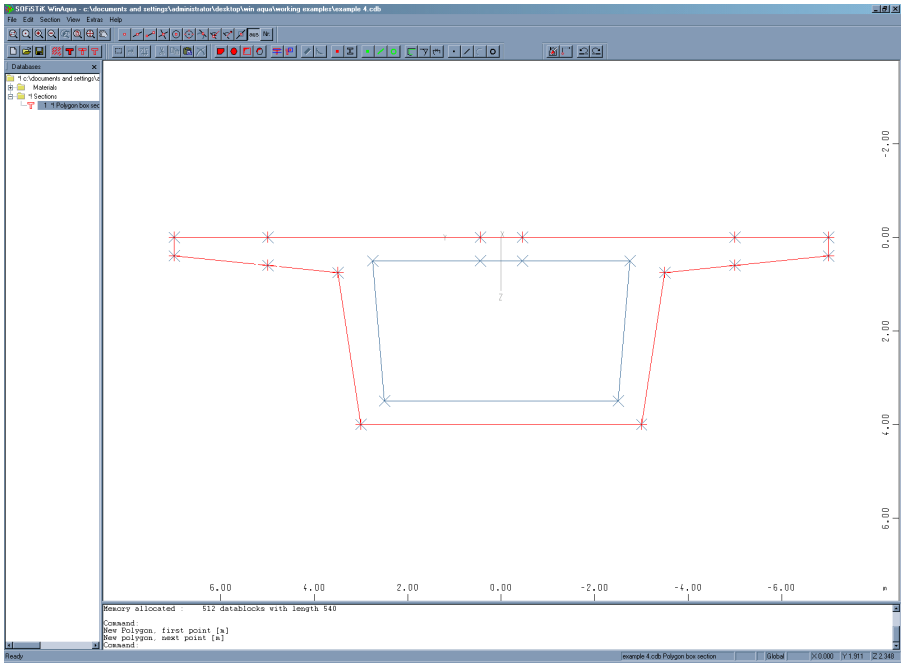


WinAqua will draw section in the drawing window using help lines.
Note: Command “Load section from DXF” is available only for “thick-walled” sections!



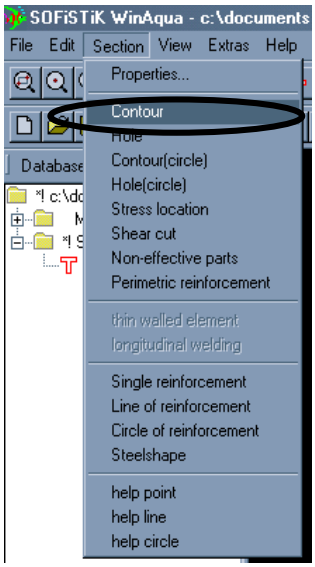
Position and orientation of a section in WinAqua will be the same as it was defined in AutoCAD. The section will be drawn whit help lines.

To define section suitable for calculation user should define section by using “**contour**” command and appropriate snap. Help line will help user for easier defining outer contour of the section.

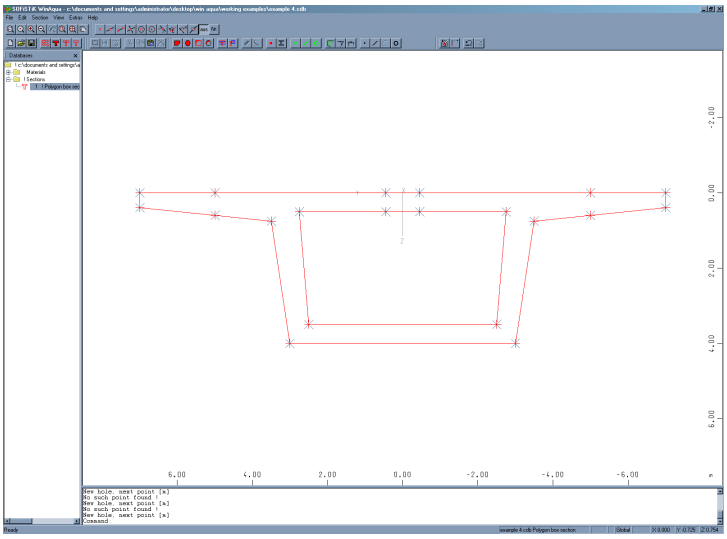


5.1.2. Defining inner contour of a polygon box section.

Using "***Hole***" command user can define the inner contour of the section. From the "***Section***" menu select command "***Hole***" or select icon from "***Edit section***" toolbar.



This command works exactly like command "***Contour***" as previously described in Example 1 in step 4. Use snap and command "***Hole***" to define inner contour of a section.



5.1.3. Defining shear cut

This procedure was described in step 2 paragraph “**Shear cut**” in Example 1.

5.1.4. Saving and calculating section database

This procedure was described in step 5 and 6 in Example 1.

5.1.5. Exporting result in to plb file and using program Ursula

This procedure was described in step 4 in Example 4.

Materials

No. 1 B 25 (DIN 1045)

Youngs-modulus	30000	[MPa]	Safetyfactor		1.00	[-]
Poisson-Ratio	0.20	[-]	Strength	fc	17.50	[MPa]
Shear-modulus	12500	[MPa]	Nomin. strength	fcn	25.00	[MPa]
Compression modulus	16667	[MPa]	Tens. strength	fctm	2.56	[MPa]
Weight	25.0	[kN/m3]	Tens. strength	fctk	2.14	[MPa]
Weight buoyancy	0.0	[kN/m3]	Tens. strength	fctk	3.08	[MPa]
Temp.elongat.coeff.	1.00E-05	[-]	Compr.failure energy		20.00	[kN/m]
			Tens.failure energy		0.05	[kN/m]
			Friction in crack		0.20	[-]

No. 2 BST 500 (DIN 1045)

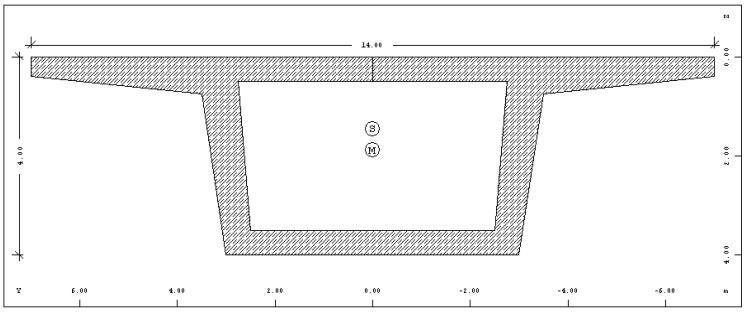
Youngs-modulus	210000	[MPa]	Safetyfactor		1.00	[-]
Poisson-Ratio	0.30	[-]	Yield stress	fy	500.00	[MPa]
Shear-modulus	80769	[MPa]	Tens. strength	ft	550.00	[MPa]
Compression modulus	175000	[MPa]	Ultim. plast. strain		0.00	[o/oo]
Weight	78.5	[kN/m3]	Relaxation at .55ft		0.00	[-]
Weight buoyancy	0.0	[kN/m3]	Relaxation at .70ft		0.00	[-]
Temp.elongat.coeff.	1.20E-05	[-]	national bond coeff.		200.00	[-]
			EC2 bondcoeff. K1		0.80	[-]
			Hardening modulus		0.00	[MPa]
			Dynamic stress range		240.00	[MPa]

Static properties of cross section

MNo	A[m2]	Ay/Az/Ayz	Iy/Iz/Iyz	ys/zs	y/z-sc	modules	gam
MNs	It[m4]	[m2]	[m4]	[m]	[m]	[MPa]	[MN/m]
1	1.4650E+01		3.068E+01	0.000	0.002	30000	0.366
2	5.495E+01		1.675E+02	1.456	1.886	12500	

Additional static properties of cross section

Alfa-T	ymin	zmin	hymin	AK	MB	Tau-T	Tau-Vy
	ymax	zmax	hzmin	AB		Tau-B	Tau-Vz
	[m]	[m]	[m]	[m2]		[1/m3]	[1/m2]
1.0E-05	-7.000	-1.456		1.60E+01	2	6.26E-02	2.00E-02
	7.000	2.544		1.46E+01			2.22E-01



6. Composite Section

The program WinAqua can be used for creating composite section. With WinAqua a square steel shape with concrete core should be made. Geometry of this composite section is as shown on the Fig.6.1.

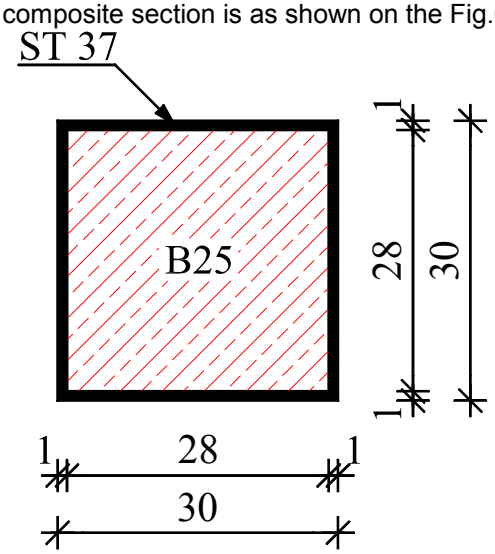


Fig. 6-1 Cross section

Material from which this section is made is given on the Fig.6.1. In WinAqua the section will be oriented as is shown on the Fig.6.2.

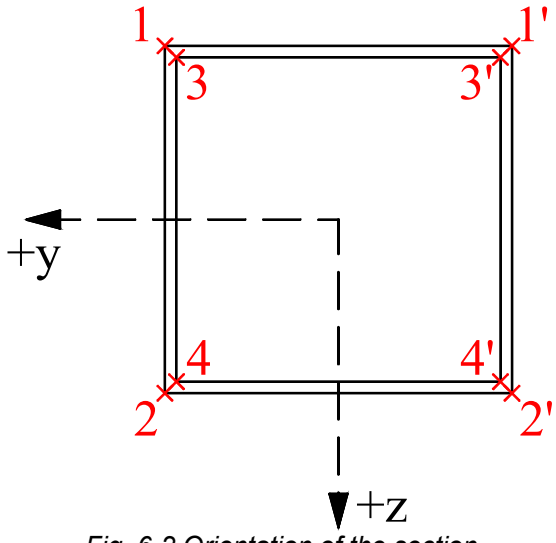


Fig. 6-2 Orientation of the section

The main geometry of this section is defined with coordinates are given in Table 6.1.

Table 6-1

Point	Coordinate	
	Y	Z
1	0.15	-0.15
1’	-0.15	-0.15
2	0.15	0.15
2’	-0.15	0.15
3	0.14	-0.14
3’	-0.14	-0.14
4	0.14	0.14
4’	-0.14	0.14

Note: The sectional values can refer to steel or concrete material. For example, the area can be calculated:

- If section values refer to steel material

$$E_{Steel} = 210000MPa \quad E_{concrete} = 30000MPa$$

$$reffered\ material\ is\ steel\ with\ E_{Steel} = 210000MPa$$

$$A_i = A_{Steel} + \frac{E_{concrete}}{E_{Steel}} \cdot A_{concrete} = 0.0228 + \frac{30000}{210000} \cdot 0.0784 = 0.0228m^2$$

- If section values refer to concrete material

$$E_{Steel} = 210000MPa \quad E_{concrete} = 30000MPa$$

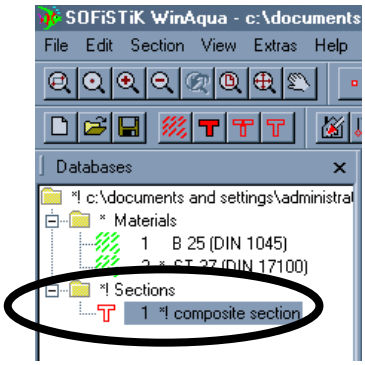
$$reffered\ material\ is\ concrete\ with\ E_{concrete} = 30000MPa$$

$$A_i = \frac{E_{Steel}}{E_{concrete}} \cdot A_{Steel} + A_{concrete} = \frac{210000}{30000} \cdot 0.0228 + 0.0784 = 0.1596m^2$$

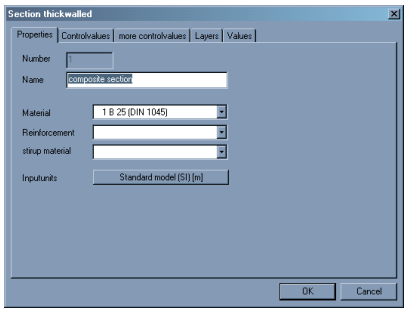
6.1. Solving the problem with WinAqua

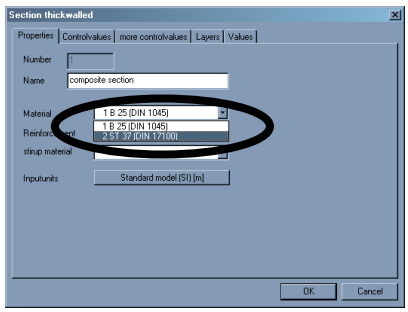
6.1.1. Drawing composite section in WinAqua

Create new database and import two materials (B25 and ST 37) for the composite section. From the "**Edit**" menu select "**New section[thick walled]…**" . Before starting to draw the section user should define current material for the section. Double click on the section name in the "**Section"** folder from "**Database**" window.



"Section Thick walled" window will upper.





Set current material to "**2 ST 37(DIN 17100)**". Note list box for reinforcement and stirrup is empty because there is no reinforcement in this section. After defining the default material user should draw outer (steel) contour of the section using command "**Contour**". To complete steel part of the section user should activate command "**Hole**" and draw inner contour of the section.

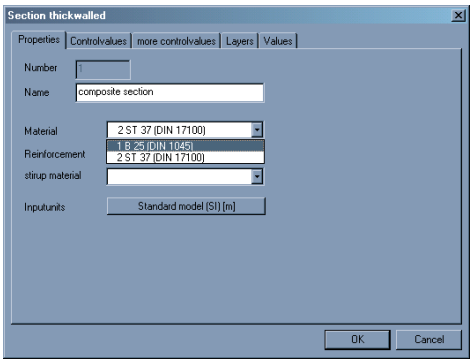
Start command “Contour”

Command:
New Polygon, first point [m] ***0.15,0.15***
New Polygon, next point [m] ***0.15,-0.15***
[Press right mouse button and select command Mirror all points]
first point of axis or axis [m] ***0,0***
last point of axis or axis [m] ***0,0.15***
Outer (steel) couture of the composite section will be drawn.

Start command “Hole”

Command:
New hole, first point [m] ***0.14,0.14***
New hole, next point [m] ***0.14,-0.14***
[Press right mouse button and select command Mirror all points]
first point of axis or axis [m] ***0,0***
last point of axis or axis [m] ***0,0.15***

As it was previously described prepare drawing area for drawing concrete part of a composite section by setting current material to “**1 B 25(DIN 1045)**”.



To complete all composite section user should activate command “**Contour**” and draw contour of the concrete part of the section.

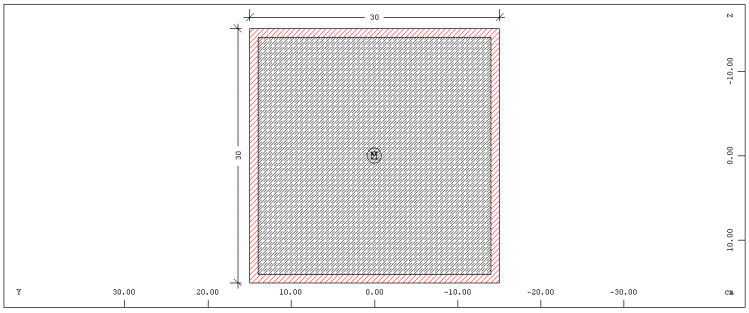
Start command “Contour”

Command:
New Polygon, first point [m] ***0.14,0.14***
New Polygon, next point [m] ***0.14,-0.14***
[Press right mouse button and select command Mirror all points]
first point of axis or axis [m] ***0,0***
last point of axis or axis [m] ***0,0.15***

Save database and the section is prepared for calculation.

6.1.2. Calculating composite section

For calculating section values user should set to which material the section value will refer to. This operation can be done using current material.
If current material is "**1 B 25(DIN 1045)**" concrete sectional values will refer to the concrete.



Materials

No. 1 B 25 (DIN 1045)

Youngs-modulus	30000	[MPa]	Safetyfactor		1.00	[-]
Poisson-Ratio	0.20	[-]	Strength	fc	17.50	[MPa]
Shear-modulus	12500	[MPa]	Nomin. strength	fcn	25.00	[MPa]
Compression modulus	16667	[MPa]	Tens. strength	fctm	2.56	[MPa]
Weight	25.0	[kN/m3]	Tens. strength	fctk	2.14	[MPa]
Weight buoyancy	0.0	[kN/m3]	Tens. strength	fctk	3.08	[MPa]
Temp.elongat.coeff.	1.00E-05	[-]	Compr.failure energy		20.00	[kN/m]
			Tens.failure energy		0.05	[kN/m]
			Friction in crack		0.20	[-]

No. 2 ST 37 (DIN 17100)

Youngs-modulus	210000	[MPa]	Safetyfactor		1.10	[-]
Poisson-Ratio	0.30	[-]	Yield stress	fy	240.00	[MPa]
Shear-modulus	81000	[MPa]	Tens. strength	ft	370.00	[MPa]
Compression modulus	171821	[MPa]	Ultim. plast. strain		0.00	[o/oo]
Weight	78.5	[kN/m3]	Relaxation at .55ft		0.00	[-]
Weight buoyancy	0.0	[kN/m3]	Relaxation at .70ft		0.00	[-]
Temp.elongat.coeff.	1.20E-05	[-]	national bond coeff.		0.00	[-]
			EC2 bondcoeff. K1		0.00	[-]
			Hardening modulus		0.00	[MPa]
			Dynamic stress range		0.00	[MPa]

Static properties of cross section

MNo	A[m2]	Ay/Az/Ayz	Iy/Iz/Iyz	ys/zs	y/z-sc	modules	gam
MNs	It[m4]	[m2]	[m4]	[cm]	[cm]	[MPa]	[kN/m]
1	1.5960E-01		1.652E-03	0.00	0.00	30000	2.87
	1.106E-03		1.652E-03	0.00	0.00	12500	

Additional static properties of cross section

Alfa-T	ymin	zmin	hymin	AK	MB	Tau-T	Tau-Vy
	ymax	zmax	hzmin	AB		Tau-B	Tau-Vz
	[cm]	[cm]	[cm]	[m2]		[1/m3]	[1/m2]
1.0E-05	-15.00	-15.00		8.95E-02		5.59E-10	
	15.00	15.00		7.84E-02			

Partial cross sections

MNo	A[m2]	Ay/Az/Ayz	Iy/Iz/Iyz	ys/zs	modules	gam
MNs	It[m4]	[m2]	[m4]	[cm]	[MPa]	[kN/m]
2	1.1600E-02		1.628E-04	0.00	210000	0.91
	0.000E+00		1.628E-04	0.00	81000	
1	7.8400E-02		5.122E-04	0.00	30000	1.96
	1.106E-03		5.122E-04	0.00	12500	

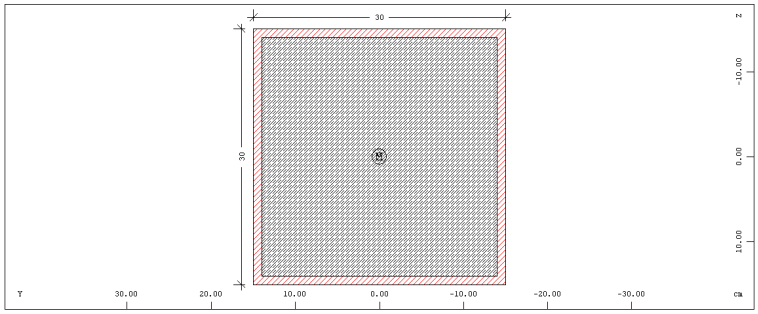
Design values of cross section

MNo	A[m2]	Ay/Az/Ayz	Iy/Iz/Iyz	ys/zs	modules	gam
MNs	It[m4]	[m2]	[m4]	[cm]	[MPa]	[kN/m]
1	1.5222E-01	1.476E+00	1.548E-03	0.00	30000	3.81
	1.106E-03	1.476E+00	1.548E-03	0.00	12500	

Full plastic force and moments
(K/D = characteristic/design)

	N[kN]	Vy[kN]	Vz[kN]	Mt[kNm]	My[kNm]	Mz[kNm]	y[cm]	z[cm]	BUCK
K	-4156.0	31851.00	31851.00	460.63	334.67	334.67	4.73	-4.73	B B
K	-686.0		30768.68		350.90	0.00	0.00	0.00	COMB
K	-686.0	30768.68			0.00	350.90	0.00	0.00	COMB
K	2784.0	29686.36	29686.36	349.74	-334.67	-334.67	-4.73	4.73	
K	-686.0		30768.68		-350.90	0.00	0.00	0.00	COMB
K	-686.0	30768.68			0.00	-350.90	0.00	0.00	COMB
D	-3902.9	29152.25	29152.25	428.84	306.10	306.10	5.03	-5.03	
D	-686.0		28069.92		323.37	0.00	0.00	0.00	COMB
D	-686.0	28069.92			0.00	323.37	0.00	0.00	COMB
D	2530.9	26987.59	26987.59	317.94	-306.10	-306.10	-5.03	5.03	
D	-686.0		28069.92		-323.37	0.00	0.00	0.00	COMB
D	-686.0	28069.92			0.00	-323.37	0.00	0.00	COMB

If current material is “**2 ST 37(DIN 17100)**” steel sectional values will refer to the steel.



Static properties of cross section

MNo	A[m2]	Ay/Az/Ayz	Iy/Iz/Iyz	ys/zs	y/z-sc	modules	gam
MNs	It[m4]	[m2]	[m4]	[mm]	[mm]	[MPa]	[kN/m]
2	2.2800E-02		2.360E-04	0.0	0.0	210000	2.87
	1.106E-03		2.360E-04	0.0	0.0	81000	

Additional static properties of cross section

Alfa-T	ymin	zmin	hymin	AK	MB	Tau-T	Tau-Vy
	ymax	zmax	hzmin	AB		Tau-B	Tau-Vz
	[mm]	[mm]	[mm]	[m2]		[1/m3]	[1/m2]
1.2E-05	-150.0	-150.0		8.95E-02		5.59E-10	
	150.0	150.0		7.84E-02			

Partial cross sections

MNo	A[m2]	Ay/Az/Ayz	Iy/Iz/Iyz	ys/zs	modules	gam
MNs	It[m4]	[m2]	[m4]	[mm]	[MPa]	[kN/m]
2	1.1600E-02		1.628E-04	0.0	210000	0.91
	1.106E-03		1.628E-04	0.0	81000	
1	7.8400E-02	2.142E-01	5.122E-04	0.0	30000	1.96
	0.000E+00	2.142E-01	5.122E-04	0.0	12500	

Design values of cross section

MNo	A[m2]	Ay/Az/Ayz	Iy/Iz/Iyz	ys/zs	modules	gam
MNs	It[m4]	[m2]	[m4]	[mm]	[MPa]	[kN/m]
2	2.3920E-02	1.083E-02	2.433E-04	0.0	190909	1.88
	1.106E-03	1.083E-02	2.433E-04	0.0	73636	

Full plastic force and moments
(K/D = characteristic/design)

	N[kN]	Vy[kN]	Vz[kN]	Mt[kNm]	My[kNm]	Mz[kNm]	y[mm]	z[mm]	BUCK
K	-4156.0	1376.37	1376.37	460.63	334.67	334.67	47.3	-47.3	B B
K	-686.0		1329.60		350.90	0.00	0.0	0.0	COMB
K	-686.0	1329.60			0.00	350.90	0.0	0.0	COMB
K	2784.0	1282.83	1282.83	349.74	-334.67	-334.67	-47.3	47.3	
K	-686.0		1329.60		-350.90	0.00	0.0	0.0	COMB
K	-686.0	1329.60			0.00	-350.90	0.0	0.0	COMB
D	-3902.9	1259.75	1259.75	428.84	306.10	306.10	50.3	-50.3	
D	-686.0		1212.98		323.37	0.00	0.0	0.0	COMB
D	-686.0	1212.98			0.00	323.37	0.0	0.0	COMB
D	2530.9	1166.21	1166.21	317.94	-306.10	-306.10	-50.3	50.3	
D	-686.0		1212.98		-323.37	0.00	0.0	0.0	COMB
D	-686.0	1212.98			0.00	-323.37	0.0	0.0	COMB

6.2. Composite Section (steel profile + concrete plate)

The program WinAqua can be used to create composite section that is made of steel profiles and concrete sections as shown on the Fig.6.3.

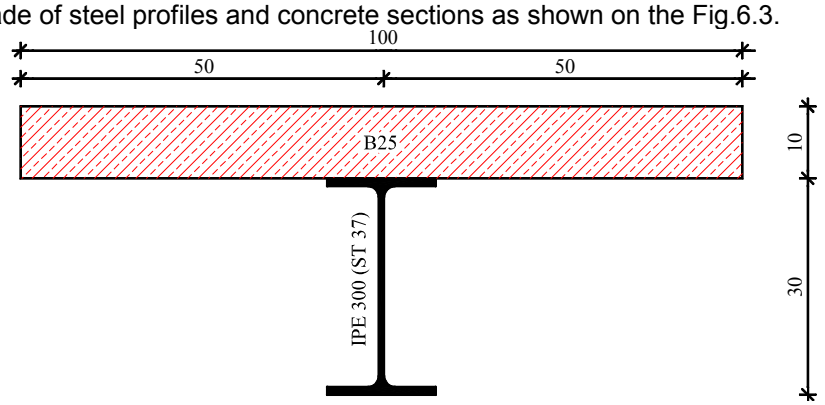


Fig. 6-3 Cross section

Materials from which this sections are made are given on the Fig.6.3. In WinAqua the section will be oriented as shown on the Fig.6.4

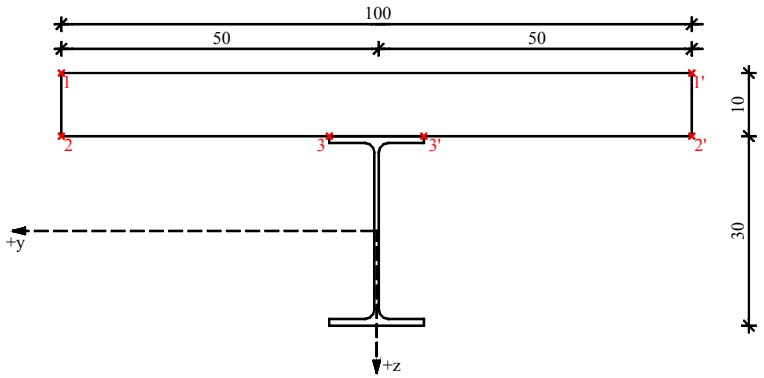


Fig. 6-4 Orientation of the section

The main geometry of this section is defined with coordinates given in Table 6.2.

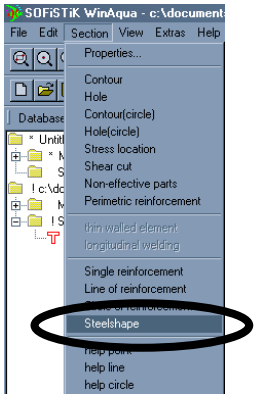
Table 6-2

Point	Coordinate	
	Y	Z
1	0.5	-0.25
2	0.5	-0.15
3	0.075	-0.15

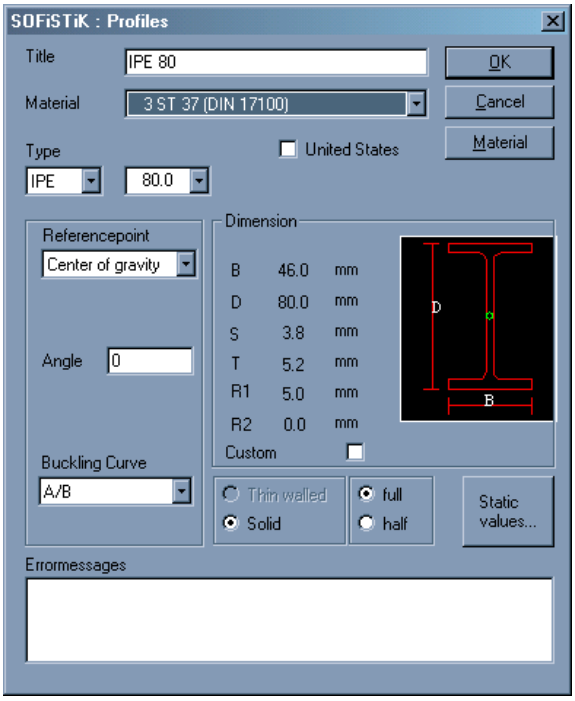
6.3. Solving the problem with WinAqua

6.3.1. Drawing composite section in WinAqua

Create a new database and import two materials (B25 and ST 37) for the composite section. The "**Section Thick walled**" should be created in the database. To import steel shape section user should use "**Steelshape**" command from "**Selection**" menu. This command can be run from the icon  placed in "**Edit section**" toolbar, too.



After starting "**Steel shape**" command "**Profile**" window will appear.



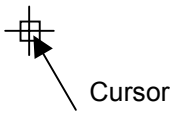
All parameters that define steel profile can be set in this window:

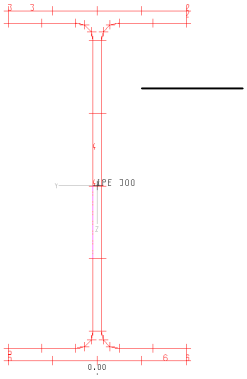
Title: Here user defines a profile name. WinAqua automatically gives a name in the title text box (depending on the selected profile).

Material: From this list box user can select and set structural steel material for steel profile. If there is no structural steel material defined in the database then this list box will be empty, so WinAqua cannot create steel profile!

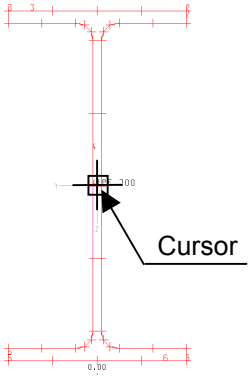
Type: With this option user can select type and dimension of a steel profile (IPE,HEA,HEB,HEM,U,I etc.)

Reference point: With reference point WinAqua defines base point for inserting the profile. For example, if reference point is top center then base point for placement of a section will be as shown on the next figure.



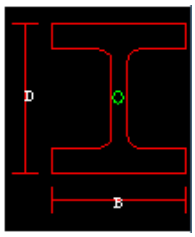


If reference point is set to center of gravity then base point for placement of a section will be as shown on the next figure.

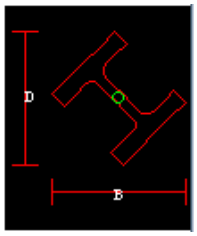


Angle: Defines angle of the section.

If the Angle=0^{o} then WinAqua will insert section as it is shown on the next figure.

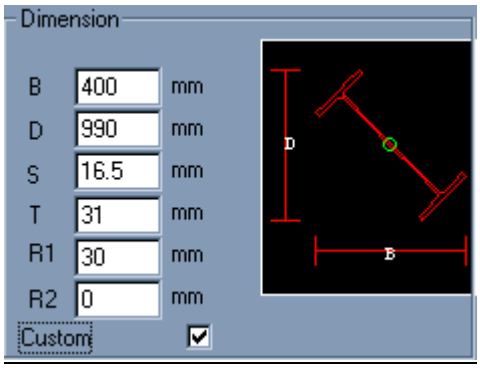


If the Angle=45^{o} then WinAqua will insert section as it is shown on the next figure.



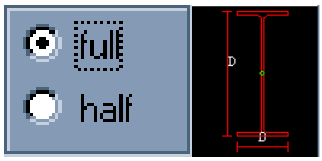
Buckling curve: User can set buckling curve for selected profile (D, A/B, B/C, C/D). Usually WinAqua automatically sets buckling curve when user selects appropriate steel profile.

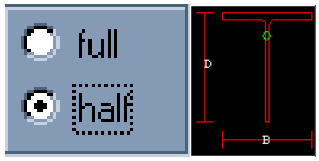
Option Custom: This option gives user opportunity to define his own dimension for the steel profile.



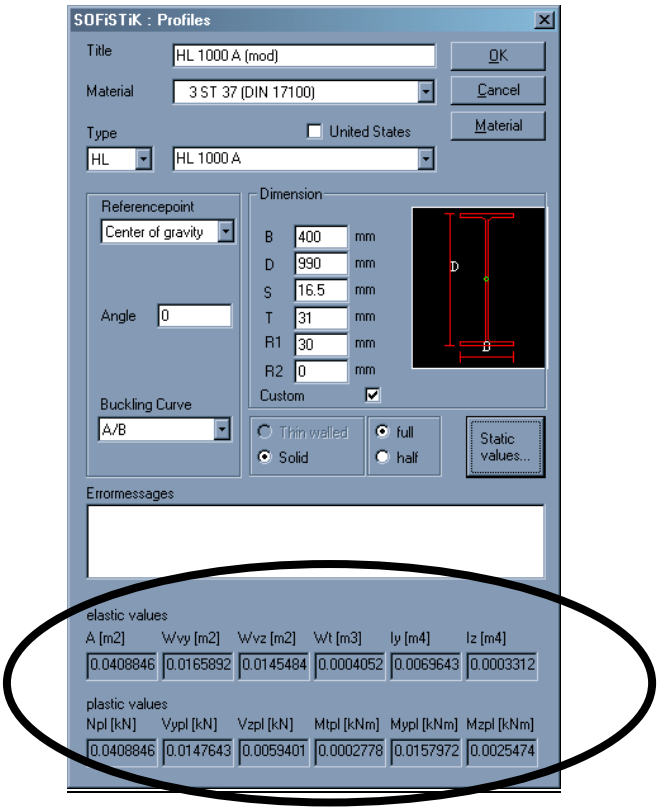
Option button "full" and "half"

If the option button is set to "**full**" then WinAqua will import a full steel profile. If the option button is set to "**half**" then WinAqua will import a half steel profile as it is shown on the next figure.





Button static value: This button gives the main elastic and plastic geometric characteristics for the selected profile.

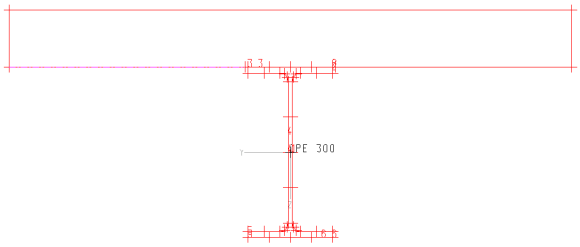


By clicking on the button "**OK**" WinAqua will ask the user to define the position point for the steel profile.

In this example user should select profile IPE300 with base point "**center of gravity**" and he has to place the profile in a coordinate **0,0**.

Drawing concrete plate will be done by using "**Contour**" command.

Command:
New Polygon, first point [m] ***0.5,-0.25***
New Polygon, next point [m] ***0.5,-0.15***
New Polygon, next point [m] ***0.075,-0.15***
[Press right mouse button and select command Mirror all points]
first point of axis or axis [m] ***0,0***
last point of axis or axis [m] ***0,0.15***



The calculation process will be same as it was explained in Example 6, paragraph **6.1 Composite Section (step 2)**.