Table of contents

6. Plate

This user's guide contains a number of useful hints, suggestions and recommendations, which helps the user to become familiar with the program module PLATE. By taking the user through various examples, it describes how to exploit it to its full potential starting with input data (pre-processing) and ending with both the graphical and text evaluation of obtained results (post-processing). It also contains a number of useful hints, which may make the user's life easier.

6.1 Input example – preprocessor

Ceiling plate:

parameters of the plate including openings are displayed in **Fig. 6.1**
thickness of the plate: d=0,2 m resp 0,25 m (the balcony part)
size of the steel-concrete beams 0,3 × 0,6 m
concrete B25 (Standard CSN 731001)
Loading: for details see chapter **5.1.7 Load input**

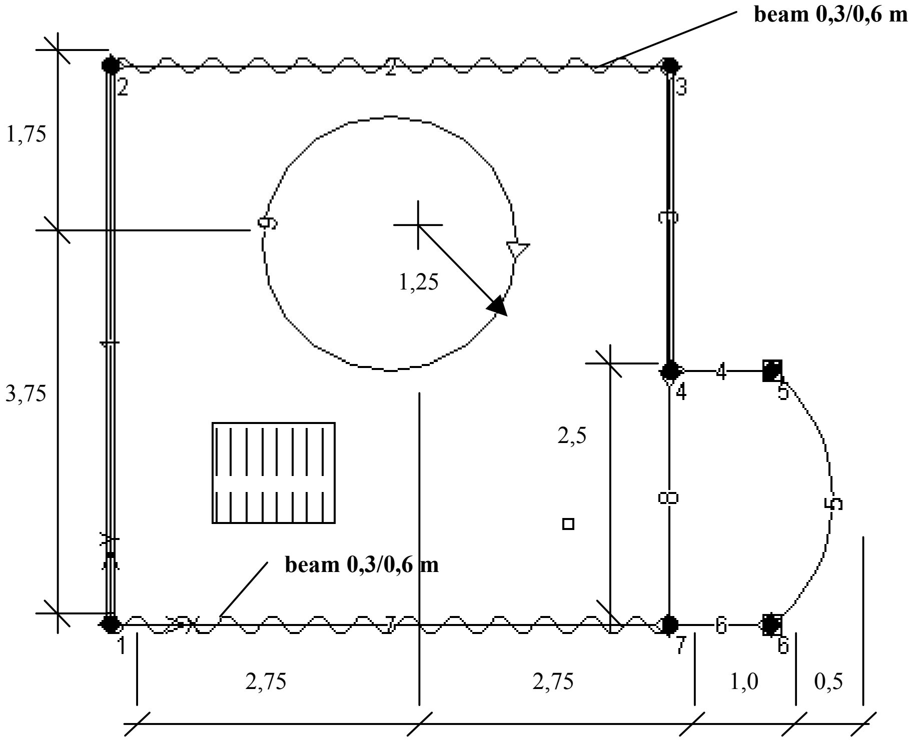


Figure 6.1 – Input data

The whole example will be processed gradually to show the possibilities and abilities of the program.

6.1.1 Geometry input

You begin with launching the module Plate. The following desktop appears (see **Fig. 6.2**). The menu bar for switching between individual input regimes (joints, lines, macroelements, load…) and to operate within these modes (edit, add, remove, create prototypes, etc.) appears on the right hand side of the desktop. Appearance of dialog windows in the lower part of the desktop depends on a given regime. In general, the input of individual plate parameters might be performed either numerically or graphically.

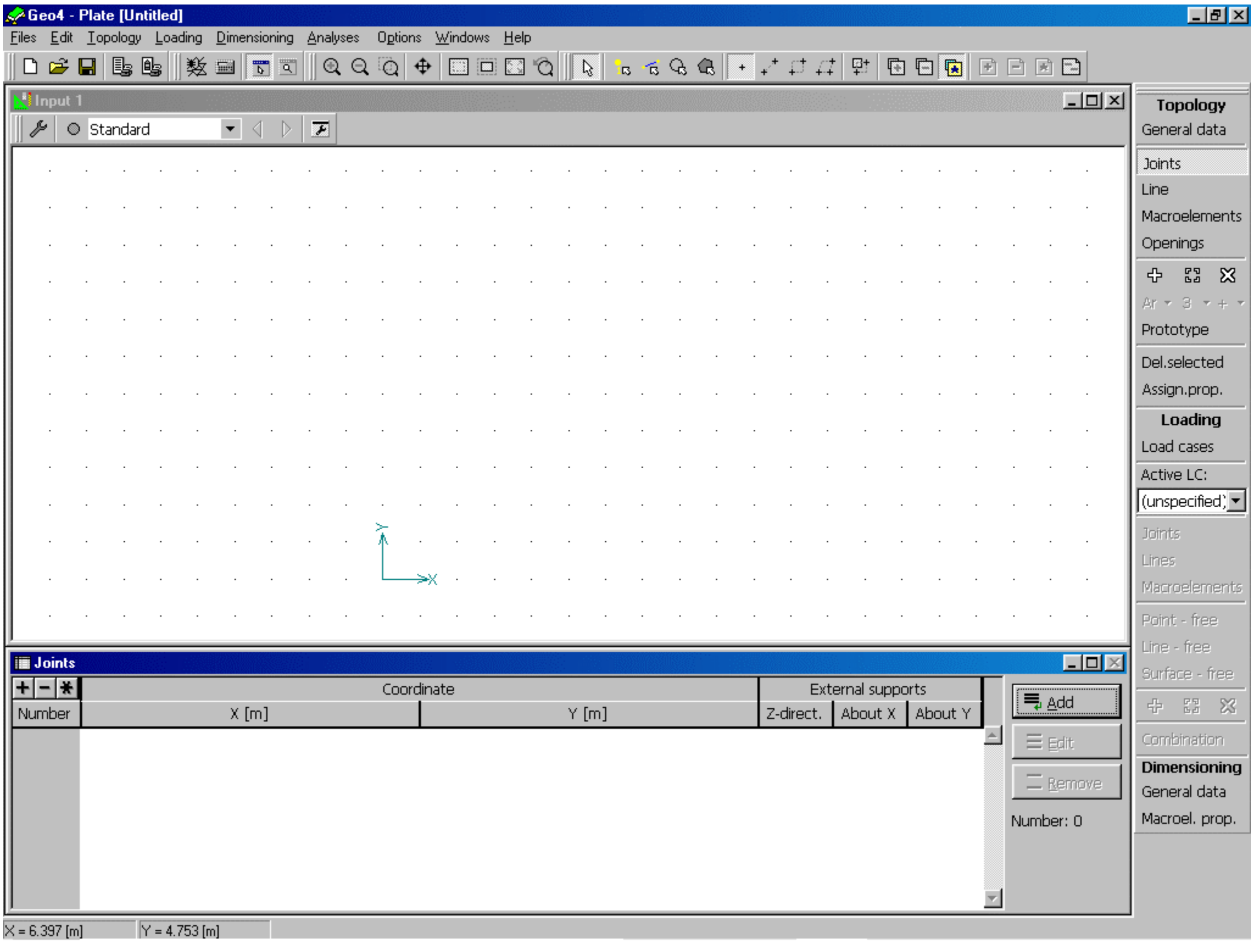


Figure 6.2 - Screen printout

6.1.2 General data

In the mode "**General data**", you input basic information about the task. You specify name of the task, its description, mesh type and a design **Standard** for materials and concrete dimensioning. For example, if you choose IS 456 (Indian standard), you will be able to select a material for macroelements and reinforcement only from materials that are defined by this standard.

6.1.3 Joints input

The Cartesian coordinate system is used. You start by inserting the topological points that characterize the shape of the plate. In the mode "**Joints**", you press the "**Add**" button. The "**New joint**" dialog window (**Fig 6.3**) appear that allows you to specify coordinates of individual points (after pressing the "**Add**" button the joint with corresponding dimensions appears in the chart "**Joints**"). The result is shown in **Fig. 6.4.**

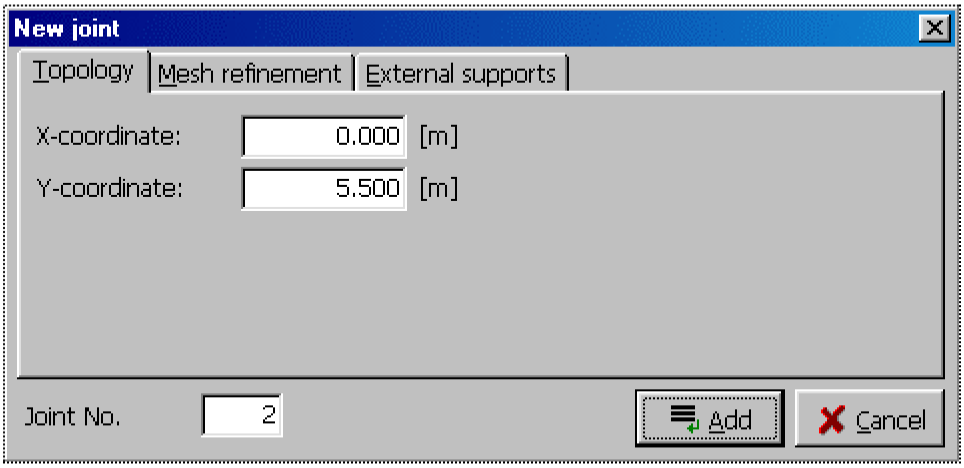


Figure 6.3 – New joint definition

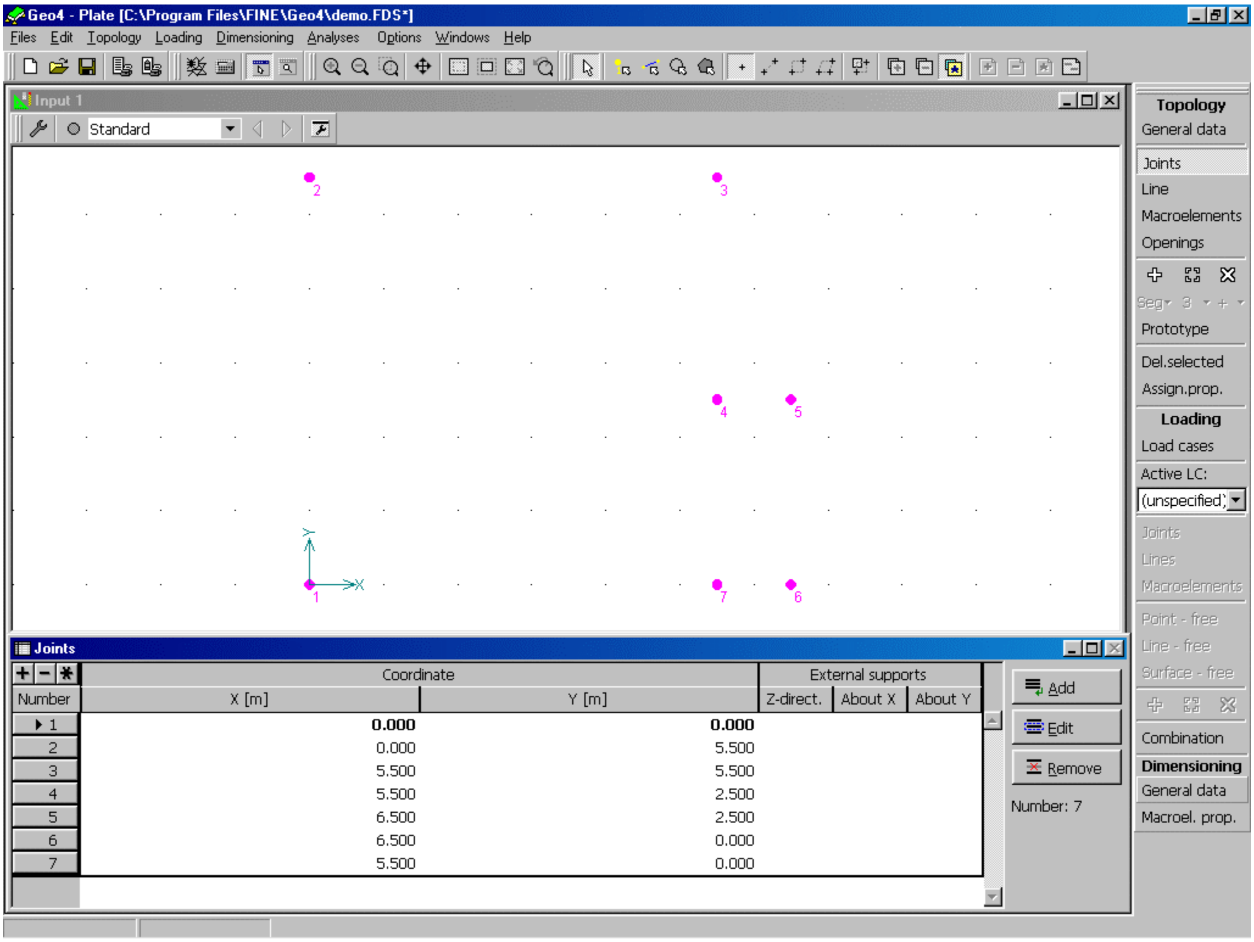


Figure 6.4 – Screen printout after inserting joints

Recall that points 5 and 6 also form the external support for the balcony part of the plate. You should therefore change their character using the "**Edit joint**" dialog window (see **Fig. 6.5**).

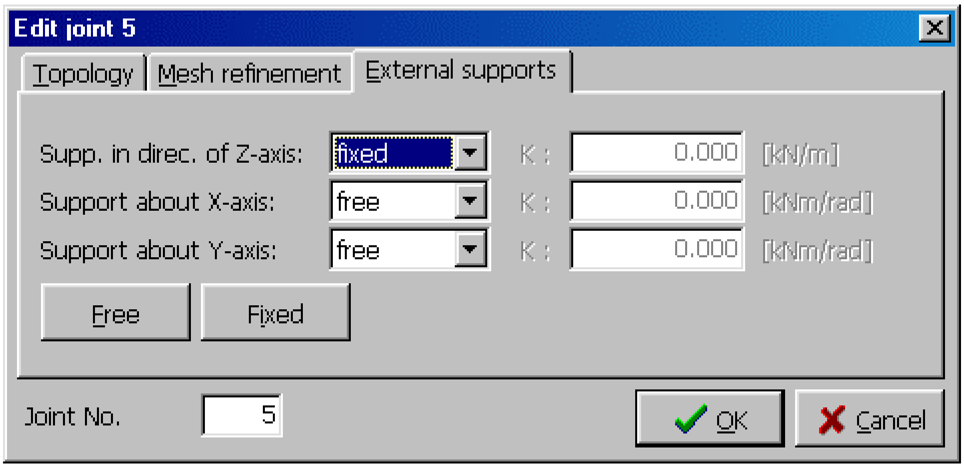


Figure 6.5 – Joint editing

To open the dialog window "**Edit joint**" you click the button with symbol "**Edit**" (on the menu bar on the right hand side of the desktop below the "**Openings**" mode) in the mode "**Joints**" and then use the mouse to indicate the joint you want to change. You make relevant changes and press "**OK**".

Complementary information

Type of the support can be fixed prior to the input of joints by creating a "**Prototype**". By pressing the "**Prototype**" in the mode "**Joints**", you open the "**Joint prototype**" dialog window (see picture). You may now adjust the required attributes and then by pressing "**OK**" to confirm them.

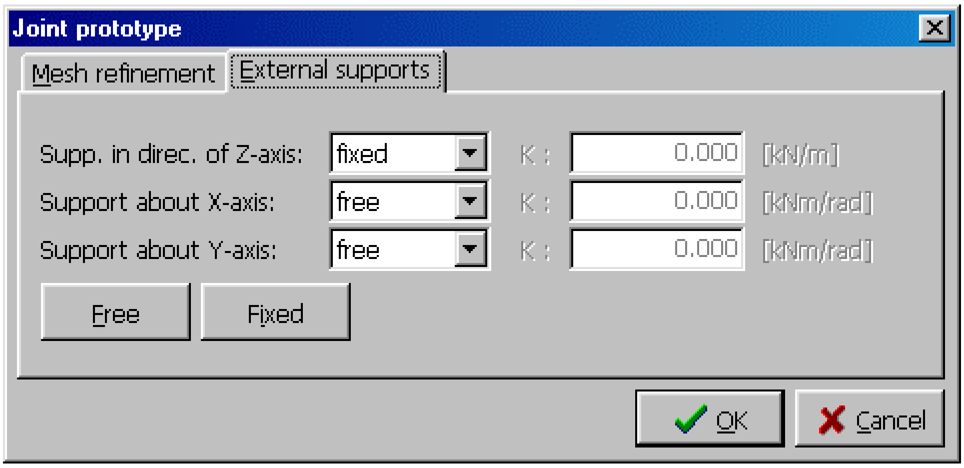


Figure 6.6 – Joint predefinition

From now on, an every new joint has a predefined character.

This process is recommended especially when a large number of joints have the same attributes. Note that the same approach proves useful when putting in lines and/or macroelements.

Complementary information

6.1.4 Lines input

To generate the final shape of the plate you connect the relevant joints using lines. The lines can be segments, arcs or circles. Recall that there are two options to create them – graphically and numerically. To exercise both options you shall create the segments and arcs graphically and the circles numerically. First, switch to the mode "**Lines**" and press the "**Add**" button. The "**Line prototype**" dialog window appears and you are prompt to specify attributes of a line and then by pressing "**OK**" to confirm them.

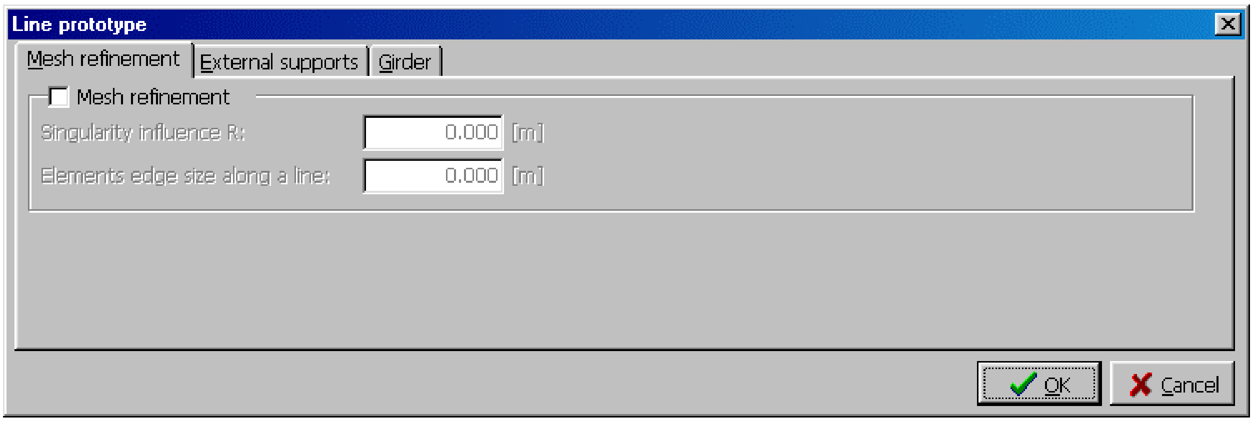


Figure 6.7 – Line predefinition

The same rules as for the joint prototype apply to the line prototype so that all lines you create next have the same properties (if you do not change them). In the present example, however, the lines are used for different purposes (to assign external supports or to specify steel-concrete beams. Therefore, it appears useful to put them in first (without their characterization) and then subjoin the required attributes.

To connect joints, you click the starting point using the left mouse button and by pulling the line move to the point, you want to connect (see **Fig. 6.8**). Then you click the end point. You repeat this operation until all the lines are created. **See Fig. 6.9** showing the list of the created lines. Points connected by individual lines, type of each line and character of the external supports are evident.

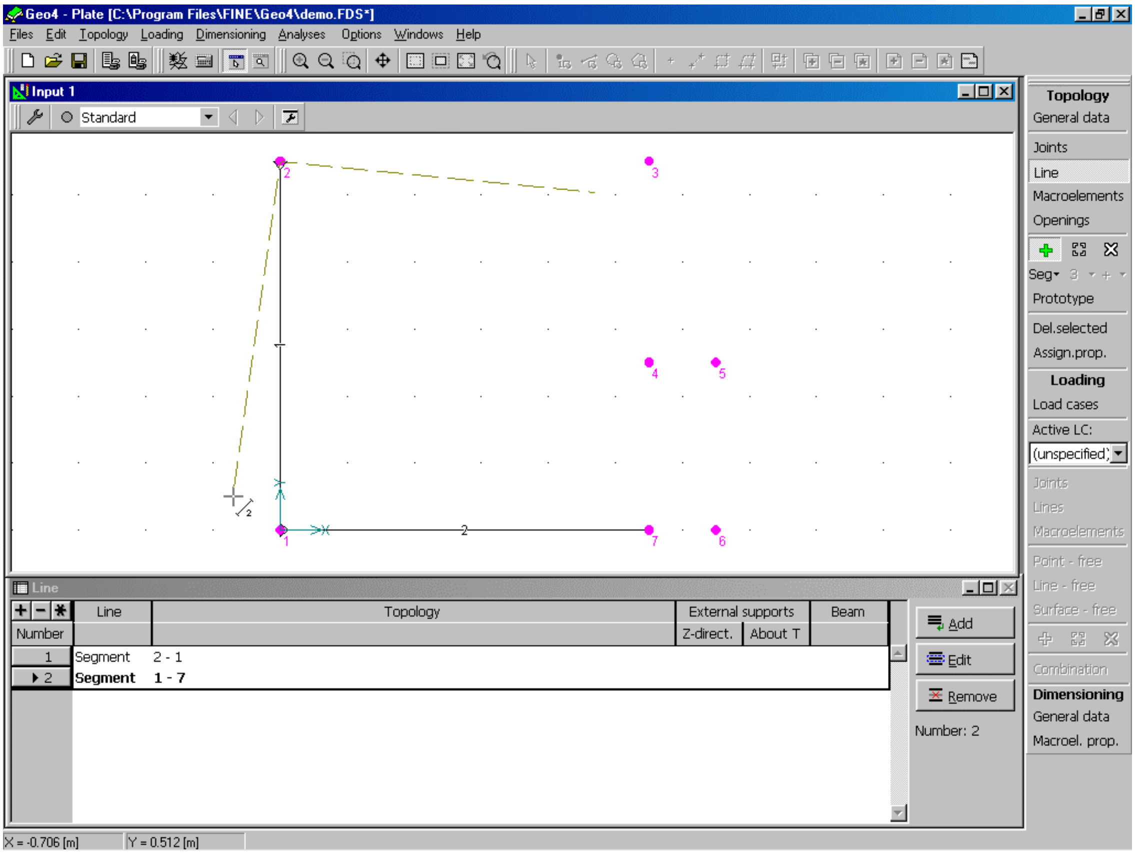


Figure 6.8 – Creating lines

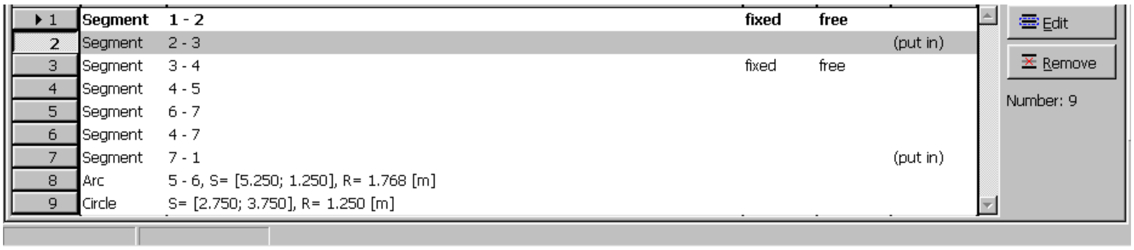


Figure 6.9 – List of lines

The point start with when creating lines can be selected arbitrarily. It is irrelevant whether you create the line No.1 by connecting points 1 and 2 or points 3 and 4. The choice of direction is also arbitrary (from point 1 to 2 or from 2 to 1).

You may now proceed with the graphical input mode to insert the line, which creates outline of the balcony part of the plate. Still in the regime "**Line**", press the button with symbol "**Add**". Using *combo boxes,* (see **Fig. 6.10**) choose the "**Arc**" option, the way in which you wish to create the selected line and the sense of input.

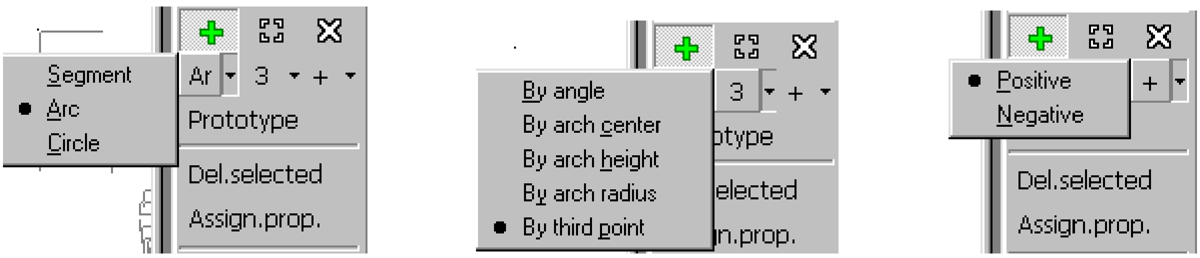


Figure 6.10 – Combo boxes for characterizing lines

You continue by putting in the arch location – the starting and end joints (5, 6) and its radius using the third point. The arc radius is 0,5 m.
You may proceed in the same way when creating an opening in the plate using the circular line (select the line type, coordinates of the centre and the radius).

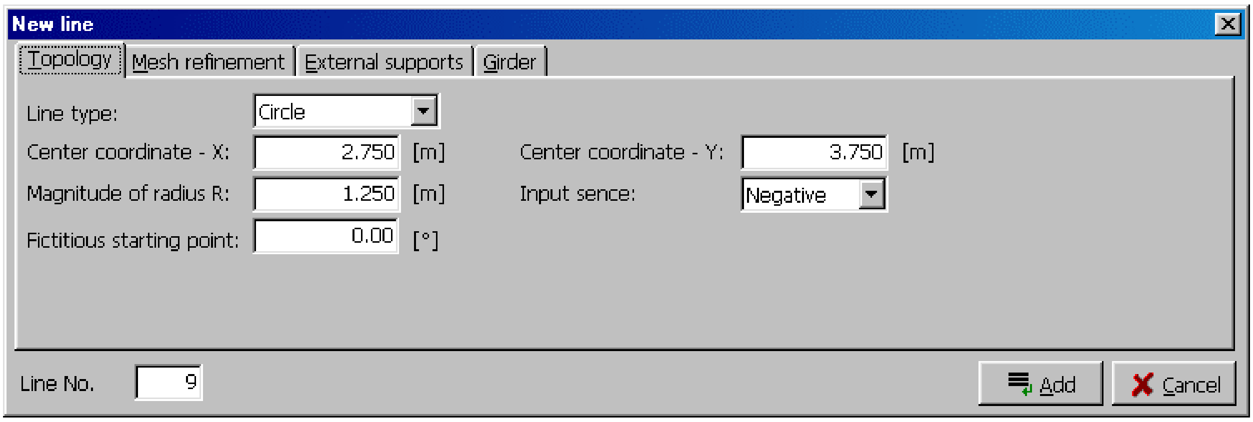


Figure 6.11 – Defining a circular line

Corresponding screen status appears in **Fig. 6.12**.

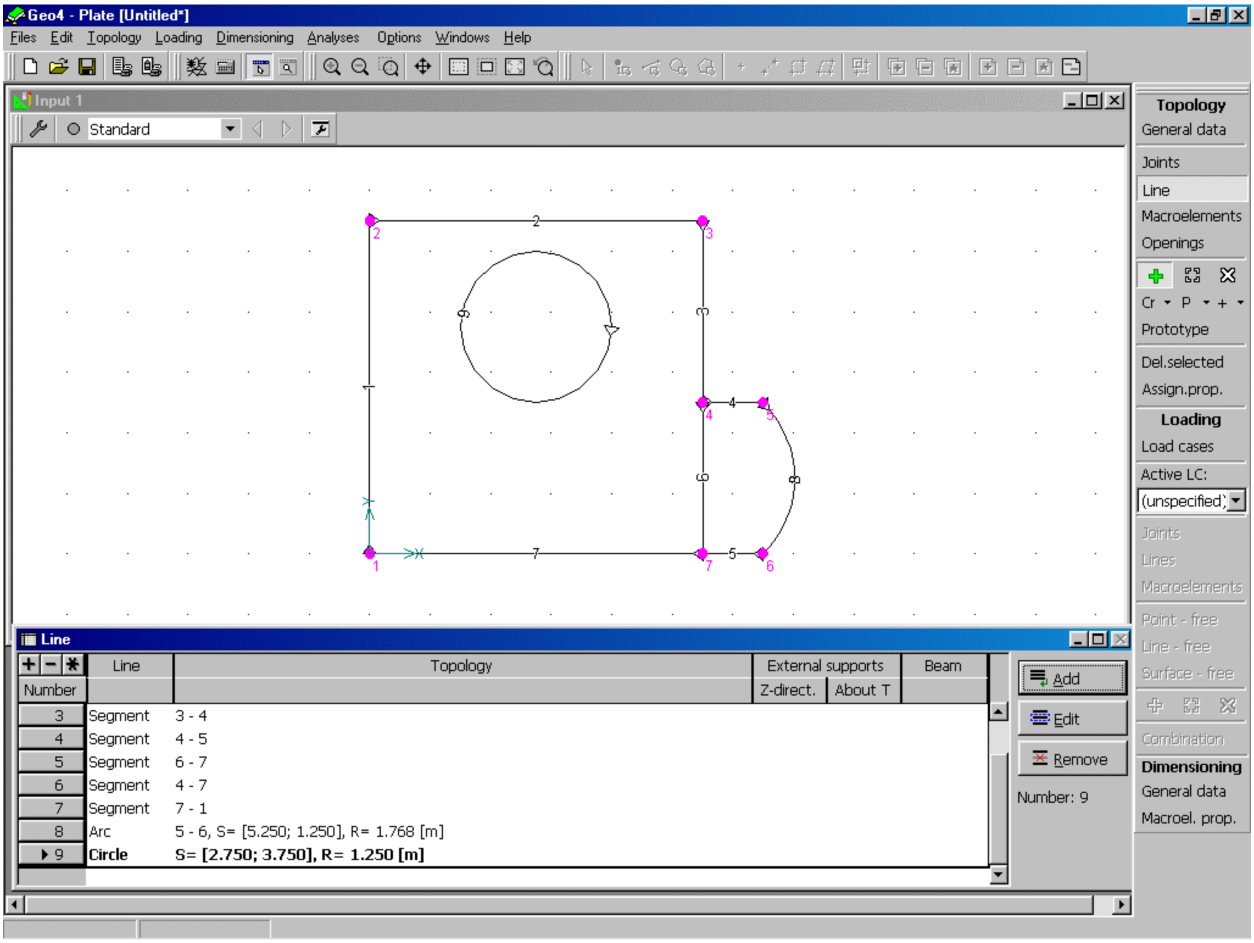
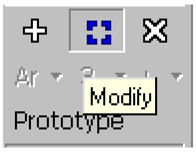


Figure 6.12 – Screen printout after inserting lines

6.1.5 Properties of lines input

Notice from the list of lines, Fig. 2.9, that lines 1 and 3 have external supports. In the regime "**Lines**", you press the button with symbol "**Edit**" and by clicking the selected line you open the dialog window "**Line edit**". Chose the "**External supports**" to define a support in the direction of Z-axis.



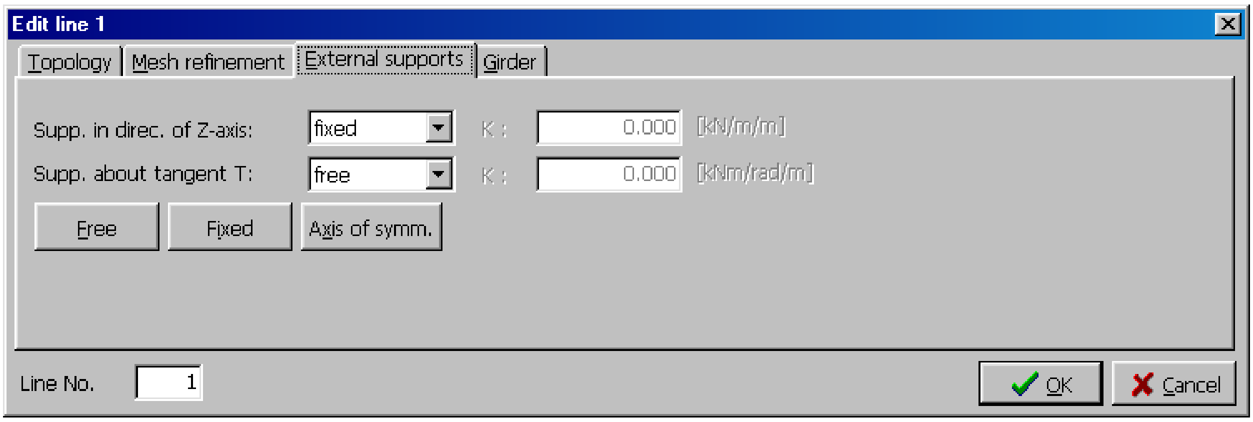


Figure 6.13 – External support input

You repeat this step for the line No. 3. The result is displayed in **Fig. 6.13**.

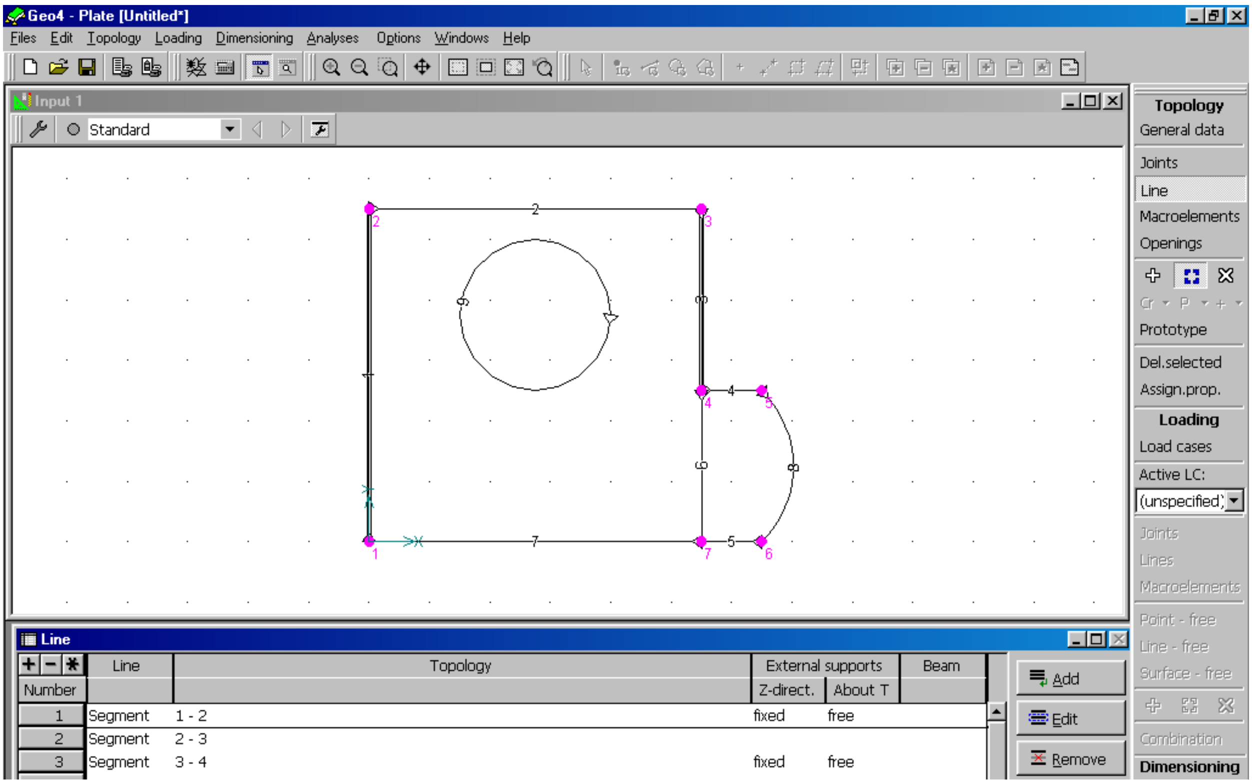


Figure 6.14 – Result of input of supports

We now postpone an input of girder beams after inserting macroelements – the reason for that is explained in Section "**Plate girder input**".

6.1.6 Macroelements and openings

Two options are available to define macroelements. In either way, you start by selecting the regime "**Macroelements**". In the first case, you proceed by clicking the button with the symbol "**Add**" to open the "**Macroelement prototype**" dialog box, which allows you to insert the macroelement thickness and material (either numerically or choosing from the catalogue of materials). Then you select the option "**Mesh refinement**" and specify the finite elements size (0,2 m in the present example). This data can influence the results and we shall focus on it in the theoretical part of this guide, in Section "Mesh generator". **(The mesh refinement can be set in the vicinity of lines and points – support and can be different from the mesh refinement on the whole macroelement!)**

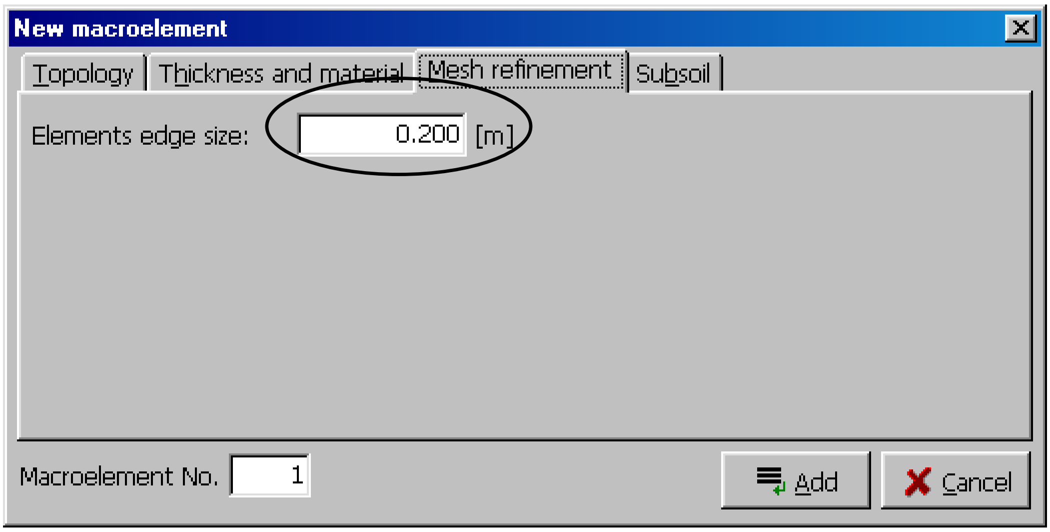


Figure 6.15 – Editing macroelement

To confirm your selection press the "**OK**" button. Then, using the left mouse button, you indicate the lines that outline the respective macroelement (lines No. 1, 2, 3, 7, 8 in the present example).

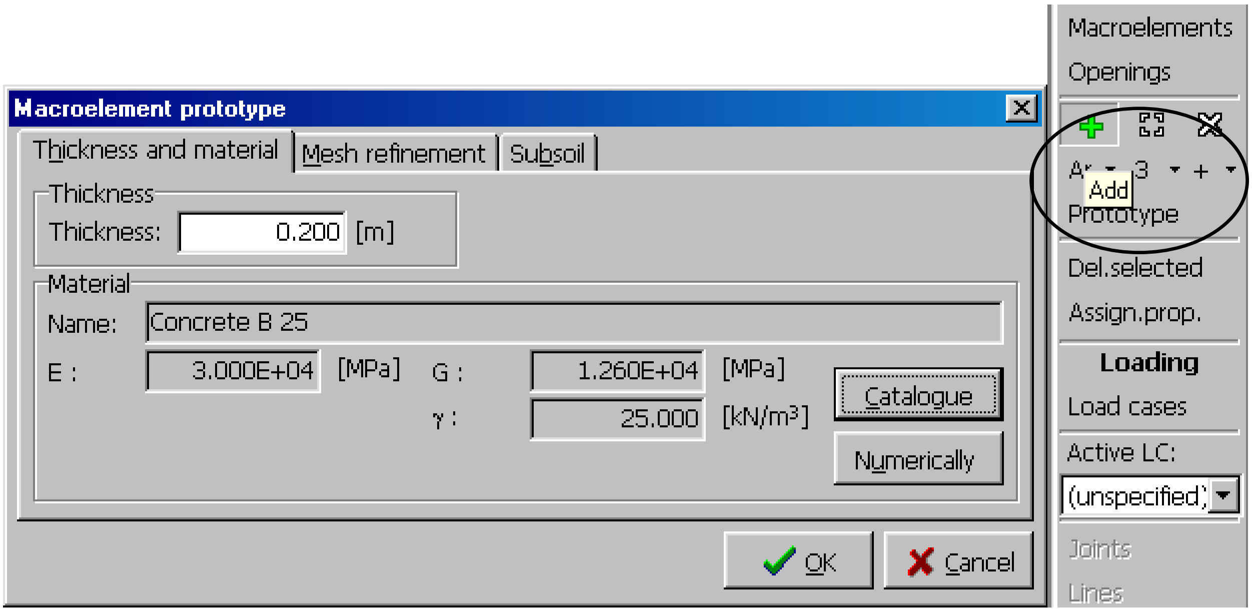


Figure 6.16 – Predefining macroelements

The macroelement will be marked with a gray color (see **Fig. 6.17**). You follow the same approach to specify the second macroelement – the balcony part (lines No. 4, 5, 6, 8).

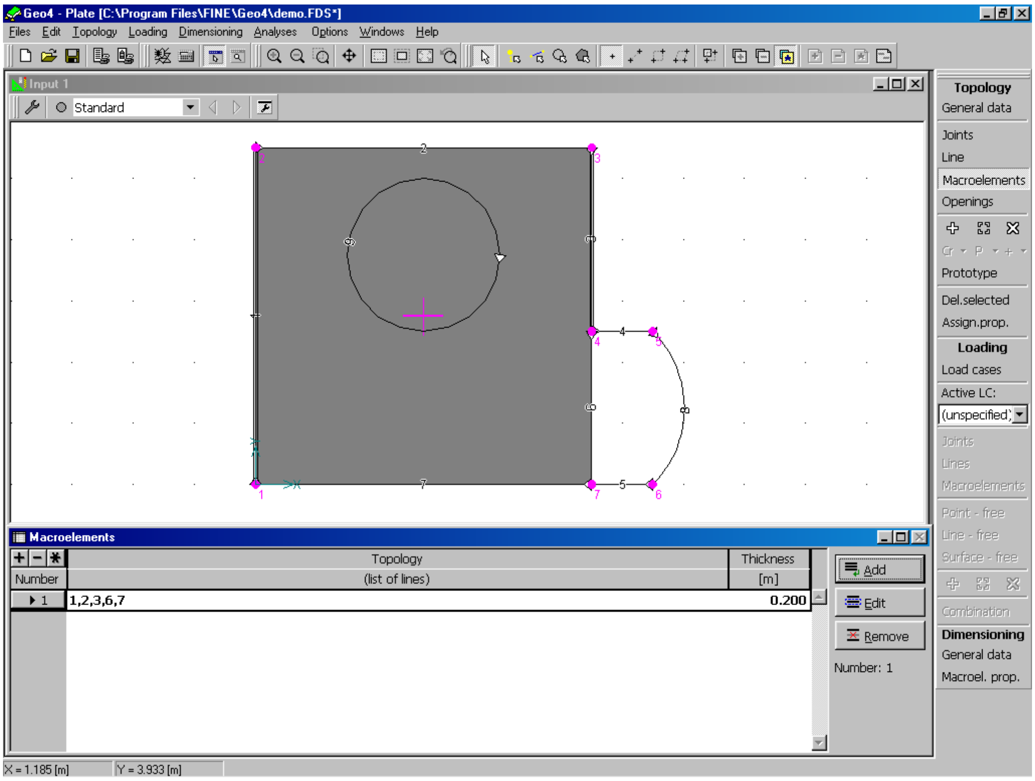


Figure 6.17 – Screen printout

The procedure for putting in an opening is similar. You switch to the mode "**Openings**", press the "**Add**" button and indicate the lines that outline the required opening (line No. 9 in the present example). **Fig. 6.18** shows the result.

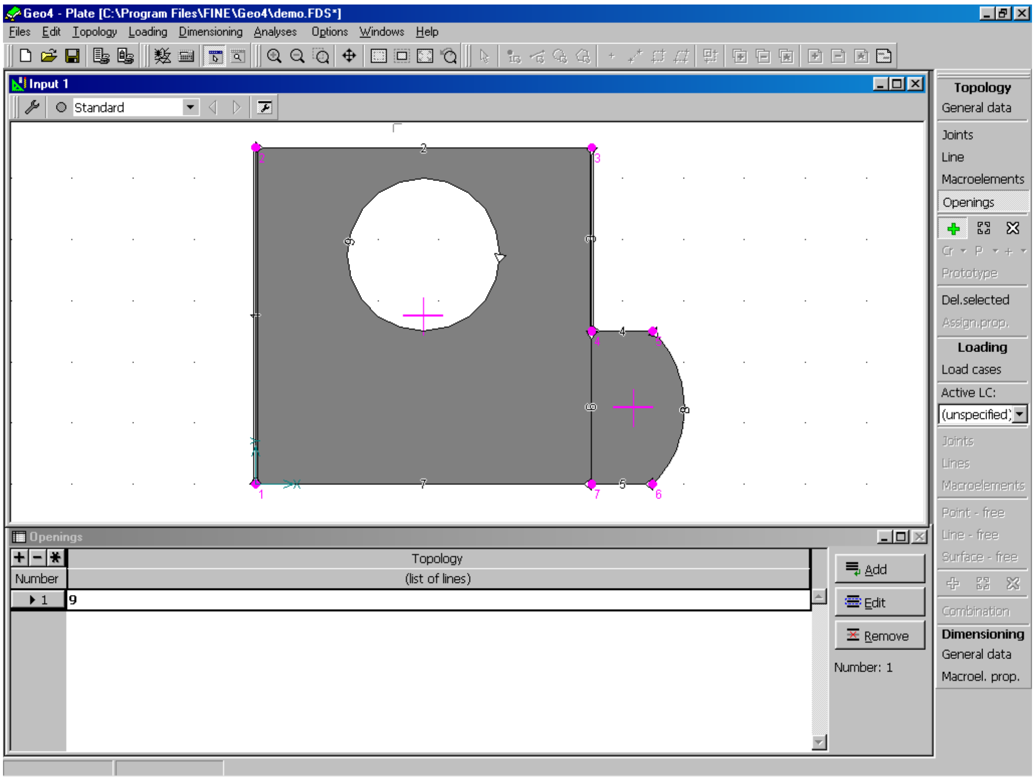


Figure 6.18 – Screen printout after inserting an opening

Complementary information

List of the lines that create the macroelement can be ordered also numerically. In such a case, you press the "**Add**" button in the mode "**Macroelements**" and in the "**New macro element**" dialog window, you enter the numbers of the selected lines. Then, using the option **Thickness and material**" you insert the macroelement thickness and material (either numerically or from the catalogue).

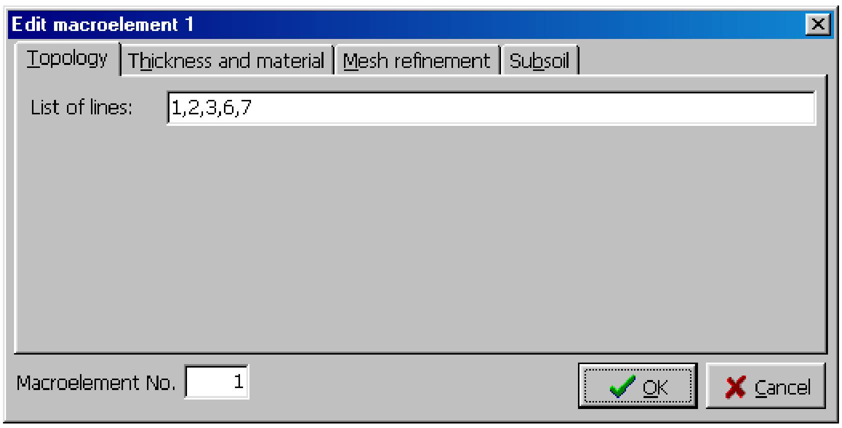


Figure 6.19 – Editing macroelement

Complementary information

6.1.7 Plate girder input

Plate girder beams are assumed located on the lines 2 and 7. To put in the girder beam, you follow a similar procedure as in Section **Input of properties of lines**. In the mode "**Lines**", you press the "**Edit**" button and clicking the line No. 2 you open the "**Line edit**" dialog window. Then you tick the variant "**Girder"** in the "**Girder**" option.

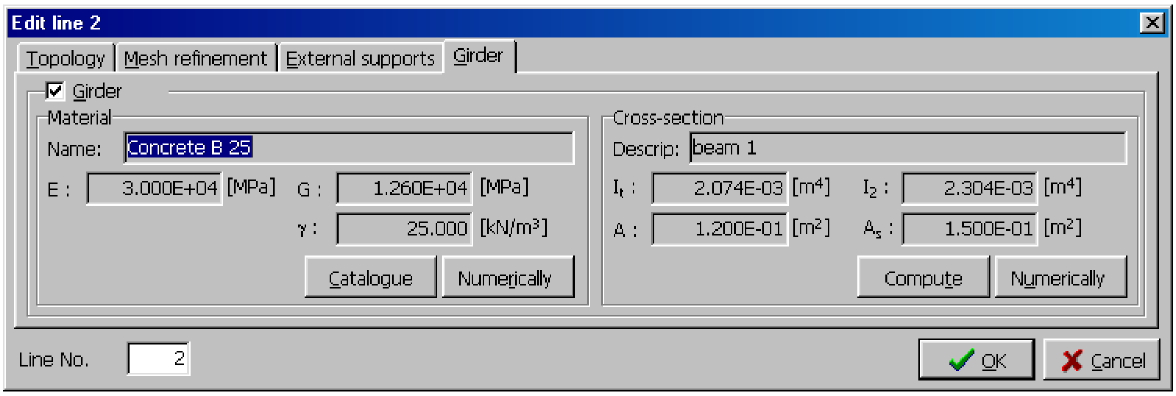


Figure 6.20– Girder definition

In this dialog window, you are prompt to insert the beam material (either numerically or from the catalogue – see **Fig. 6.20**) and its cross section.

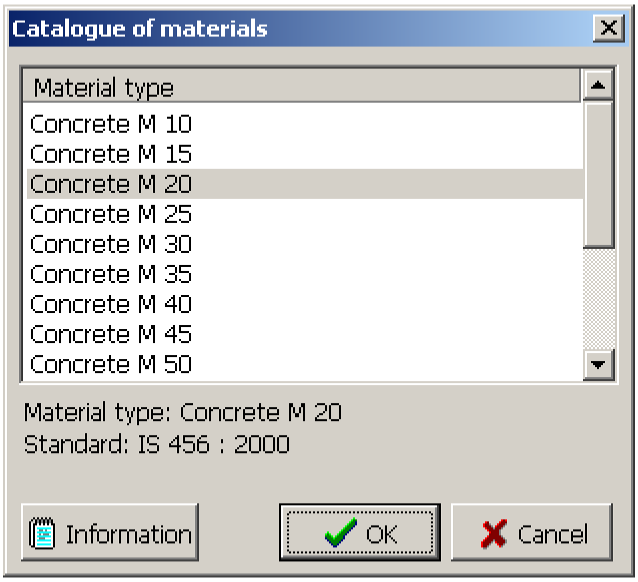


Figure 6.21 – Catalogue of materials

Note:
It is important to know that if you select this type of support (a beam with a plate) and then chose "**Compute**" to select the cross section, it is necessary to have the macroelements already put in (see the previous section). For computation, it is necessary to know the thickness of the plate connected to the beam. If you do not have the macroelements already put in, the alternatives in the dialog window "**Type of plate girder**" will not be available (see **Fig. 6.22**).

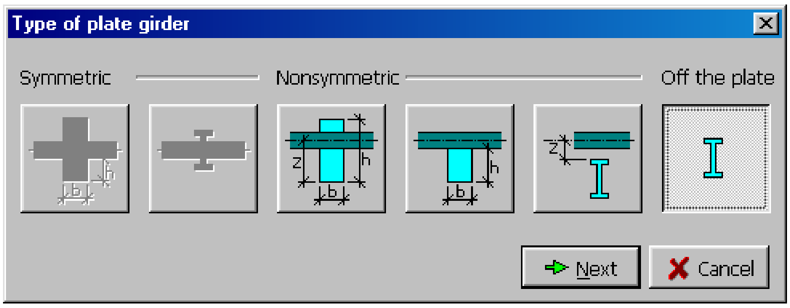


Figure 6.22 – Input of plate girder

If the macroelements are put in, the previous dialog window will change to the one displayed in **Fig. 6.23**.

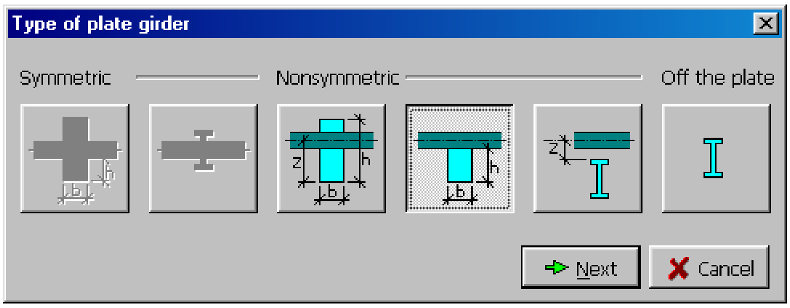


Figure 6.23 – Input of plate girder

Now you may choose the type of the beam and by pressing the "**Next**" button to open the "**Plate girder parameters**" following dialog window. This window serves to specify the beam name and size.

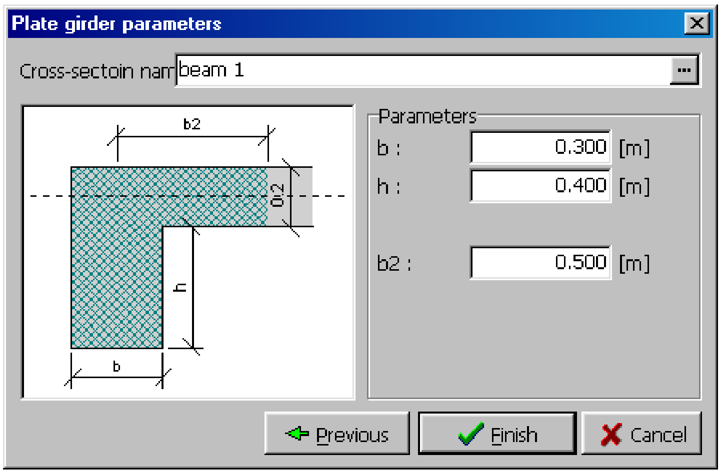


Figure 6.24 – Defining plate girder parameters

You close the input by pressing the "**Finish**" button. The resulting input of the girder beam on line No. 2 is displayed in **Fig. 6.25**. (*twiddle*).

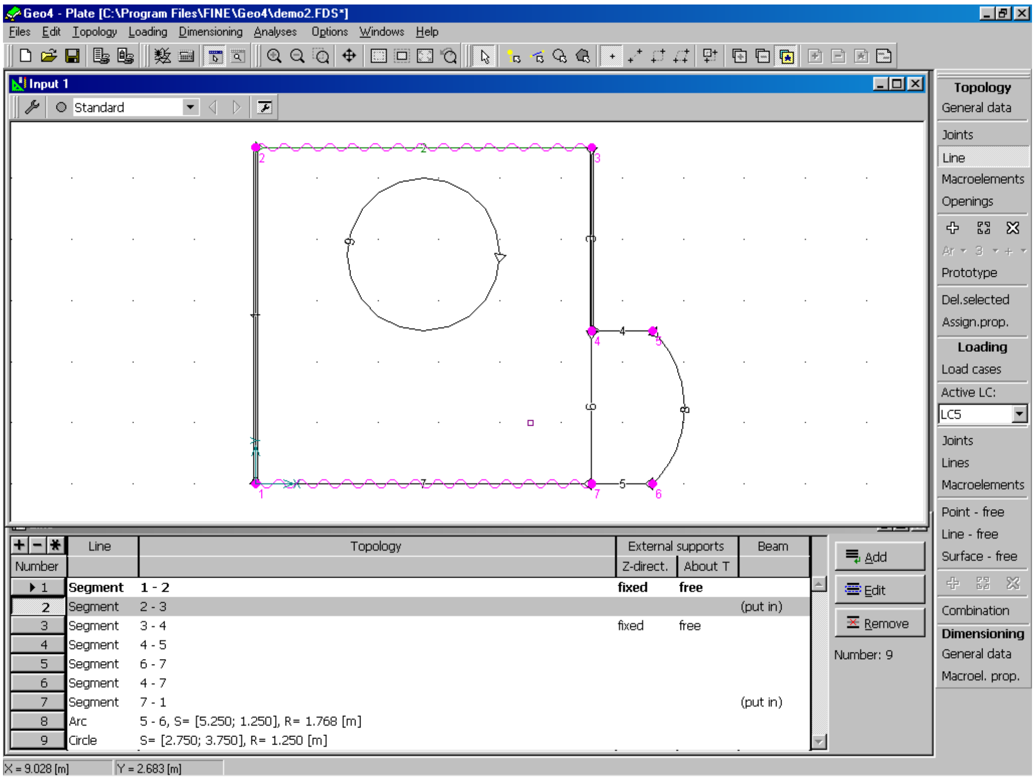


Figure 6.25 – Screen printout after inserting a plate girder

Evidently, input of a plate girder beam is a tedious operation. If you have a large number of lines having the same character, you can (apart from using prototype) employ the copying (assign of properties). You indicate the lines to which you want to assign new character (use the button on the tool bar). In this particular example, it is line No.7.

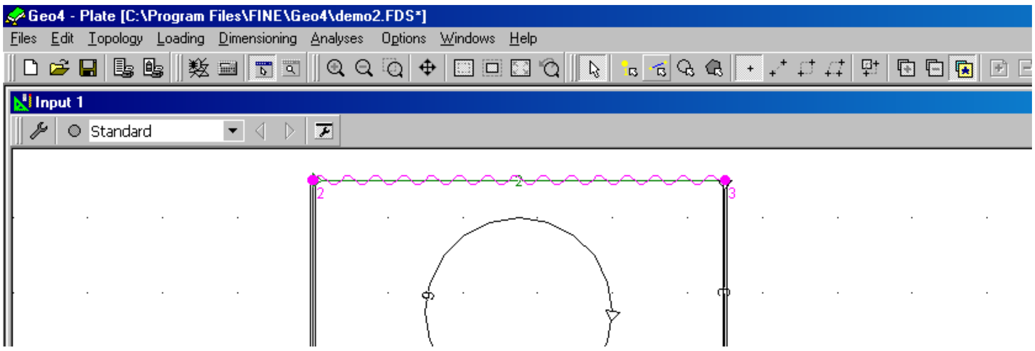


Figure 6.26 – Indicating lines

On the right on the menu bar with regimes, you press the "**Assign properties**" button. That will open the dialog window "**Select a master line**" where you can check the number of lines that will receive properties of the master line.

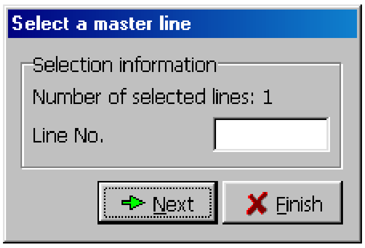


Figure 6.27 – Selection of master line when assigning properties

Then you indicate the master line (either numerically or with the mouse on the desktop) and press the "**Next**" button. In the opened dialog window "**Assign properties to selected elements**", you define the properties you want to copy and press the "**OK**" button.

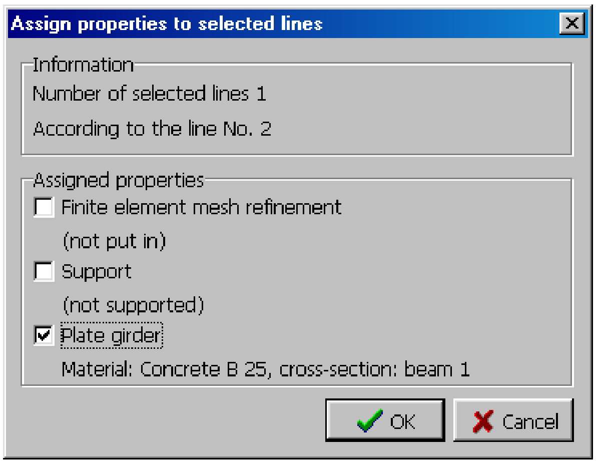


Figure 6.28 – Assigning properties to selected lines

Fig. 6.29 shows the results – two beams (on lines 2 and 7) with the same parameters were inserted. The complete plate geometry is also displayed.

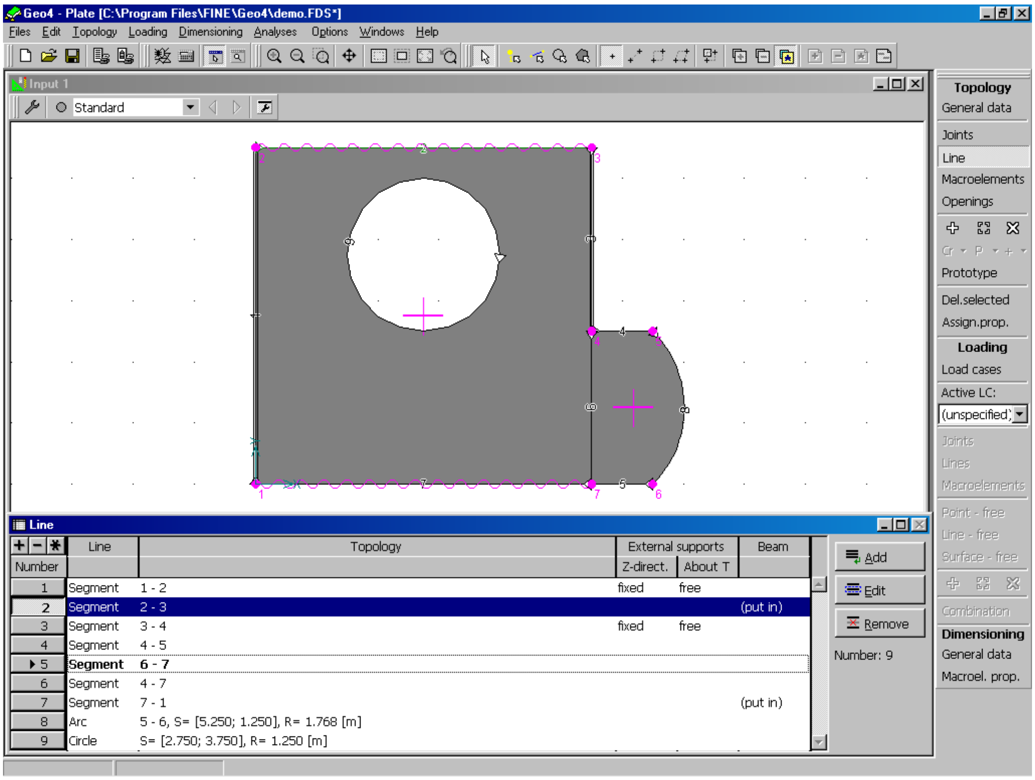


Figure 6.29 – Complete plate geometry

6.1.8 Load input

Load cases input

Two options are available to put in a load case (LC). The first variant is to define all the load cases and then assign a relevant load to a particular LC. The active LC (the LC you work with and to which you add particular loads) is represented on the menu bar on the right hand side of the desktop – there you also switch from one LC to another. The second option is to insert a LC and assign the load immediately.

In the mode "**Load cases**", you press the "**Add**" button in the lower dialog window. In the "**New load case**" dialog window, which will then open, you assign the name, the code and the coefficient of the LC. By pressing the "**Add**" button, the load case will be added to the list of load cases.

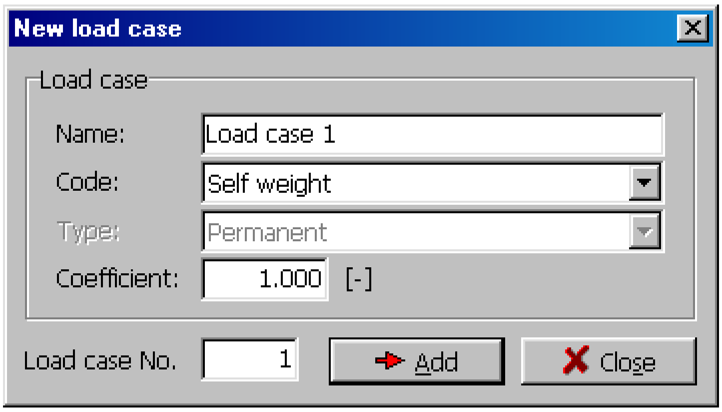


Figure 6.30 – Defining LC 1

The first **LC** offered by the program is the plate self-weight. This particular **LC** has some characteristics:

1) It can be put in only once.
2) The load of this LC will be recomputed with every change of the geometry (size, thickness of the macroelement, size of the beams).

LC 1 – self weight (**permanent**): **inputted**
Now you may add other load cases using the same procedure:
LC 2 –permanent – macroelements: Q1 = 1 kN/m2, Q2 = 1, 5 kN/m2 (the balcony part of the plate)

You add a new load case (see **Fig. 6.31**).

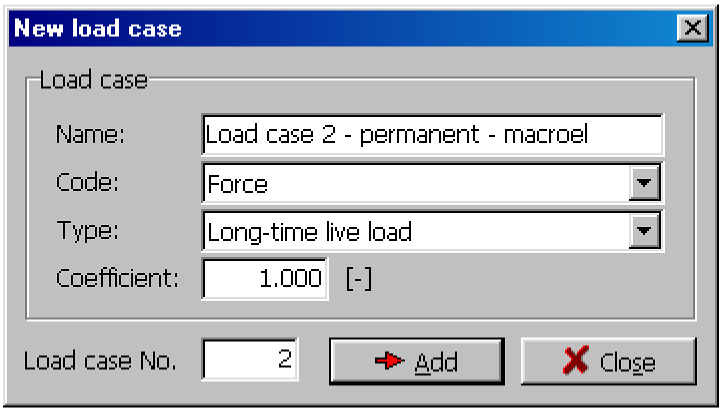


Figure 6.31 – Adding load cases

After closing the dialog window, you notice that a LC becomes automatically active.

When choosing the "**Macroelements**" option, the "**Load on a macroelement**" dialog window appears. By pressing the "**Add**" button, you open the dialog window "**Add mechanical load on a macroelement**". This dialog window serves to assign the load magnitude and the macroelement number. After pressing the "**Add**" button the program adds these items into the list.

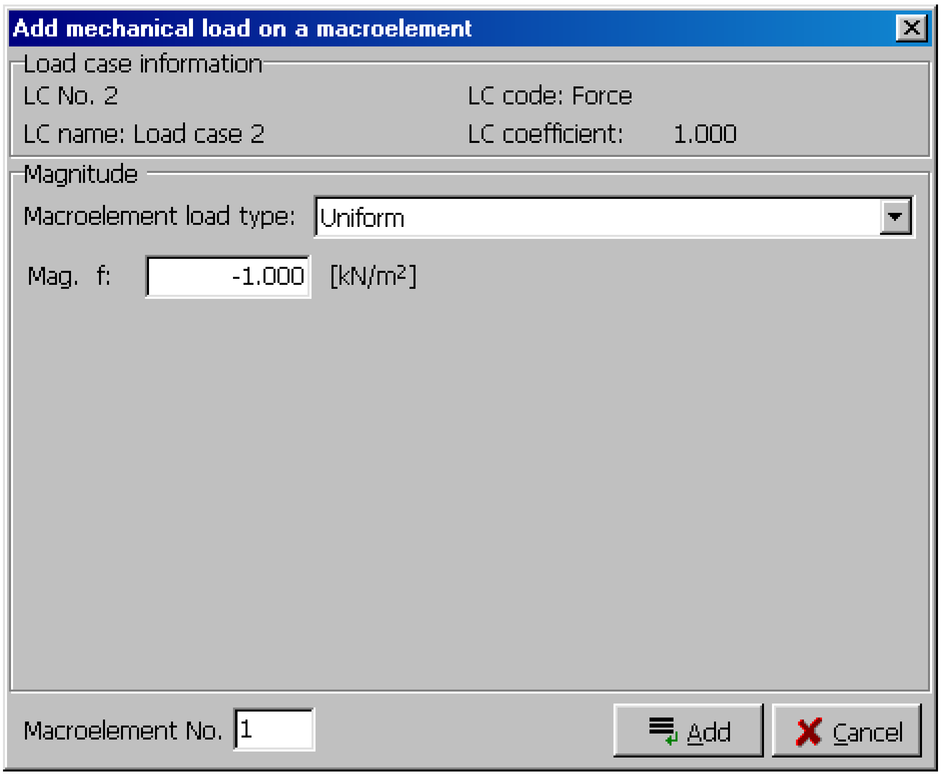


Figure 6.32 – Adding load cases

To input the load Q2 acting on the macroelement No. 2, you proceed as above. The result of the load input is found in the list located in the bottom part of the desktop, Fig. 2.32. Its graphical representation is also evident from **Fig. 6.32**.

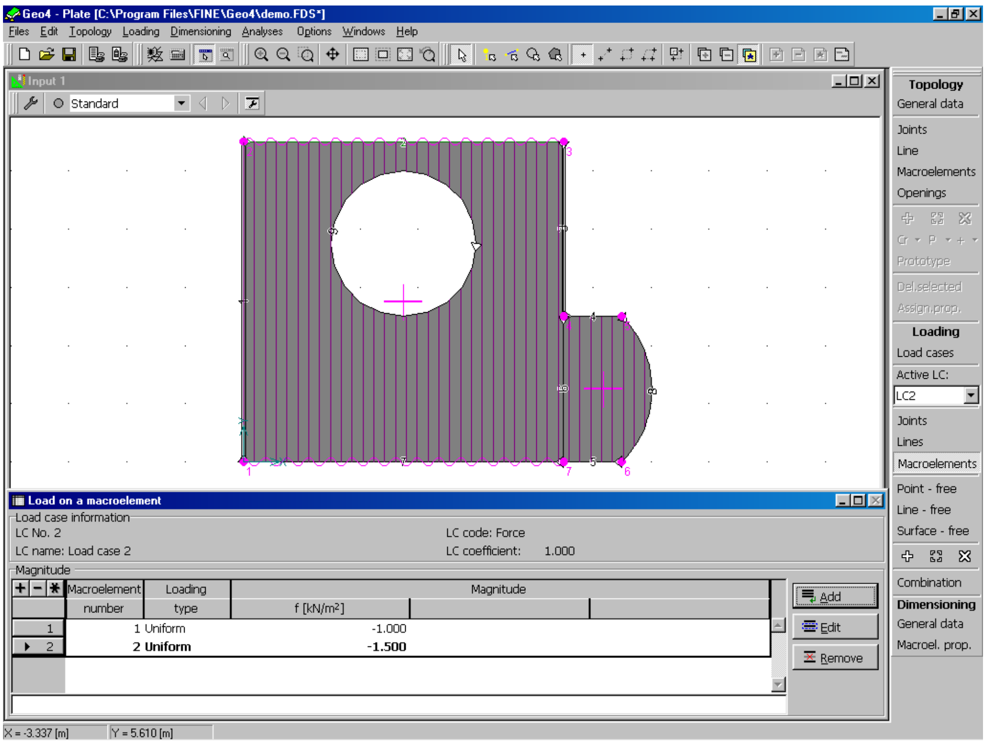


Figure 6.33 – Screen printout

LC 3 – live load - macroelements (load acting on macroelements): Q1 = 3.5 kN/m2, Q2 = 5.0 kN/m2 (the balcony).

The process of input is the same as for the previous load case (LC 2); only Q1 and Q2 change.

LC 4 – live load – lines (load acting on lines): line 2 = 12 kN/m, line 7 = 12 kN/m, and line 9 = 10 kN/m.

To add the load case No. 4, use the same approach as for the cases No. 2 and 3. First, select the "**Line**" option to open the dialog window "**Mechanical load on a line**". Pressing the "**Add**" button then opens the dialog window "**Add a mechanical load on a line**". You assign the magnitude**,** direction and type of the load and the line number. Then press the "**Add**" button to include the data into the list. Follow the same procedure to put in the load acting on lines No. 7 and 9.

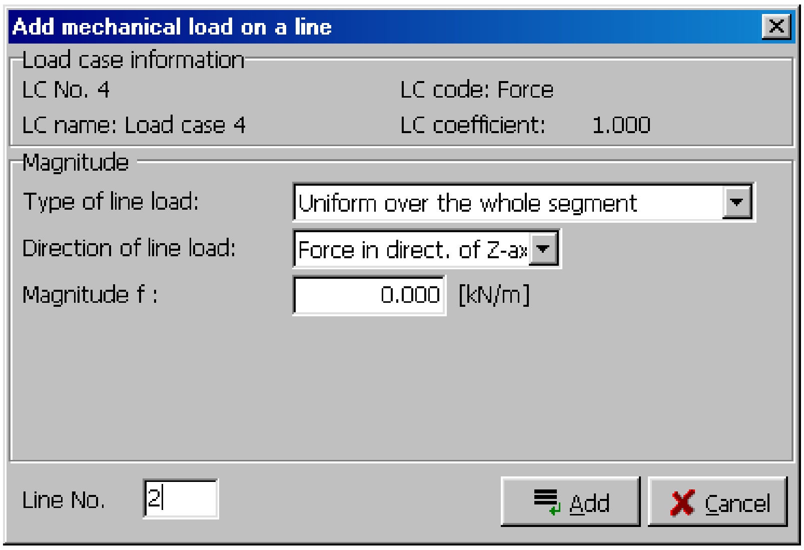


Figure 6.34 – Adding load

Again, the result of the load input is found in the list located in the bottom part of the desktop, **Fig. 6.32**. Its graphical representation is also shown in **Fig. 6.35**.

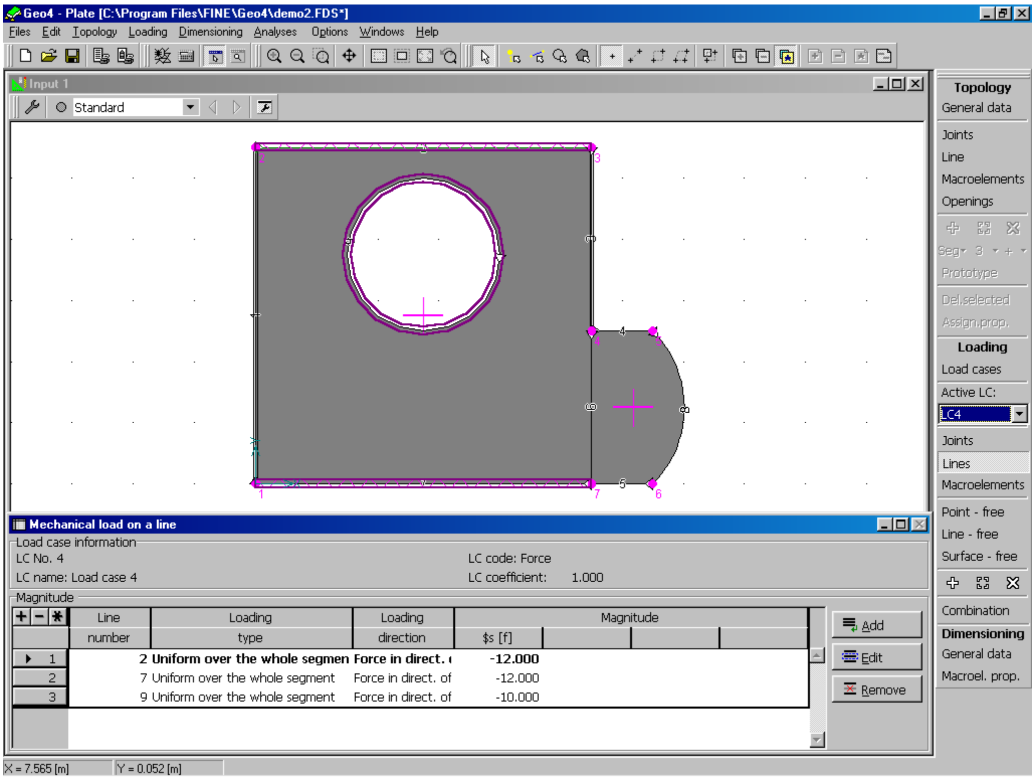


Figure 6.35 – Screen printout - active LC 4

LC 5 – live load – (free in a point, free on a macro element): point – free Fz = 11 kN (point of force application x = 4,5m, y = 1m), macroelement – free f = 2,2 kN/m2 (the macroelement is defined by points with the coordinates x, y – see the table below.

[1 , 1]
[2.2 , 1]
[2.2 , 2]
[1 , 2]

You define the load in a point in the following dialog window – you put in the point of force application (x = 4,5 m, y = 1 m) and its magnitude (Fz = 11 kN).

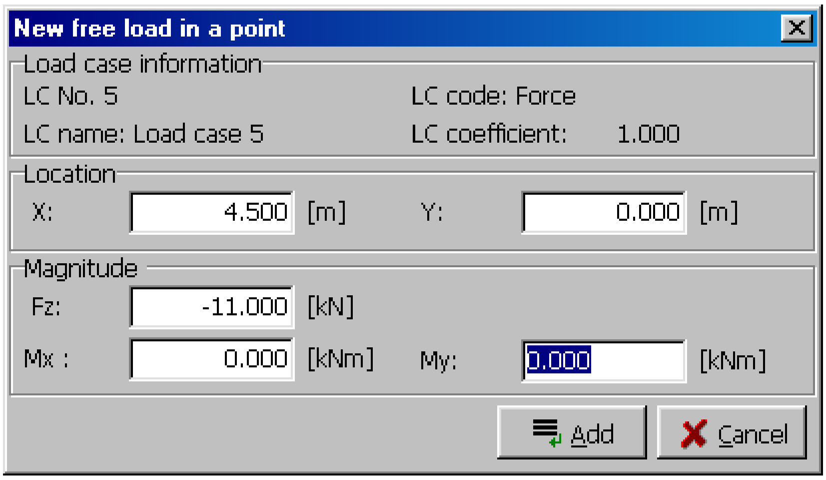


Figure 6.36 – Defining load in points

To input the free load acting on a surface element, you open the following dialog window displayed in **Fig. 5.37**. In this window, you define the macroelement using the coordinates of individual points (see the list) and the force magnitude (f = 2,2 kN/m2).

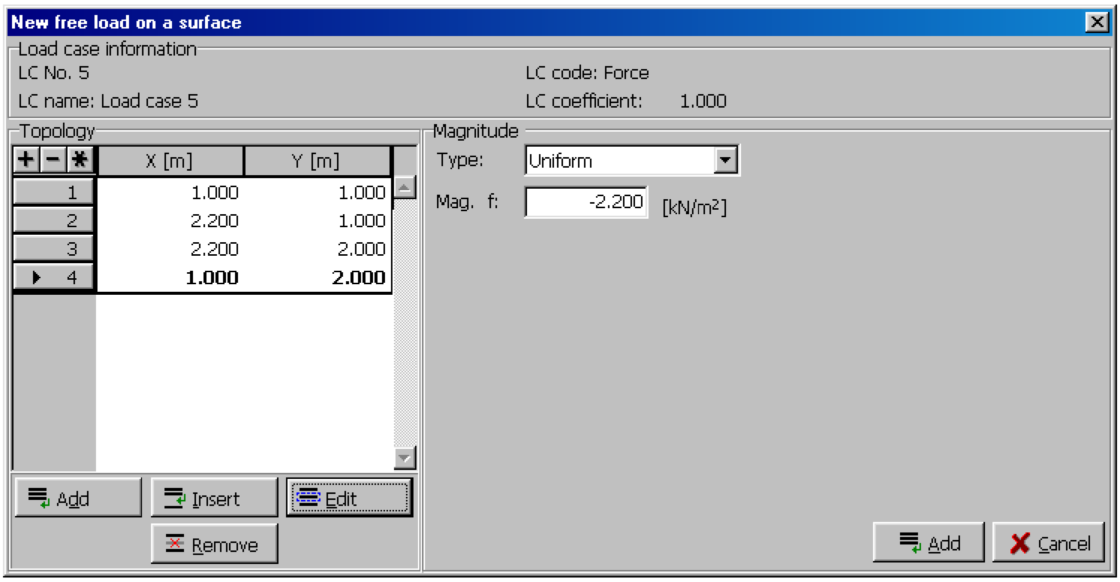


Figure 6.37 – Assigning load on a surface

The corresponding monitor status is shown in **Fig. 6.38**.

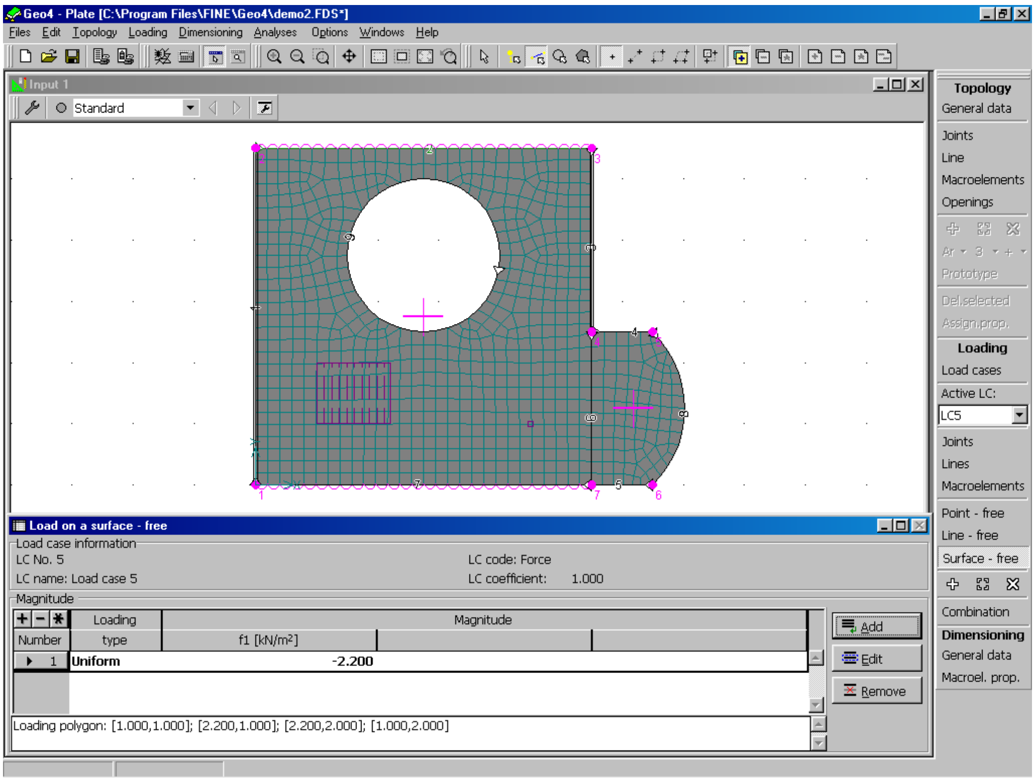


Figure 6.38 – Screen printout

6.1.9 Combinations of load cases input

Combinations of load cases are additional load cases on a construction resulting from summation of simple load cases multiplied by the coefficients of combination. A combination serves to model situations when more loads affect the construction at once (e.g., self weight + snow + wind + service load). Coefficients of combination can convey a reduced rate of probability that given loads will appear at the same time. Each combination is defined by its name and number. The character of combination is given by the code of combination. To put in a combination, it is necessary to establish which load cases appear in the combination and assign the coefficient of combination to each of these load cases. Two types of combinations exist depending on the codes – extreme and service. The extreme combination presents extreme effects of the load on a construction and mostly serves for verification according to the first limit state. In the extreme combination, every load case (LC) is multiplied by the coefficient of LC. In the service combination, the load cases are not multiplied by the coefficient of LC and the results can serve for verification according to the second limit state.

In the present example, you use one combination composed of all load cases. You create this combination by opening the "**New combination**" dialog window and then ticking all the load cases in the column "**Applicability – With**".

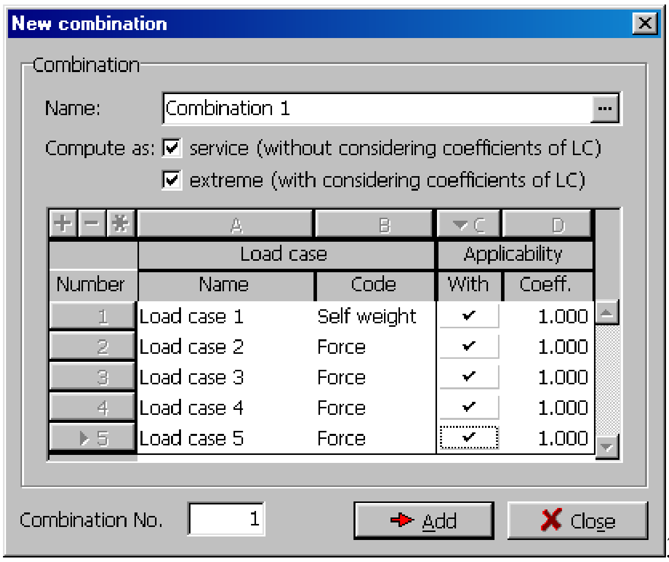


Figure 6.39 – Creating combination of load cases

6.2 Generating the finite elements mesh

6.2.1 The mesh generator – theoretical background

The mesh quality is essential for the accuracy of computations using the Finite Elements Method (FEM). Optimum triangular elements are equilateral triangles. Optimum quadrilateral elements are squares. Due to stability of used modern finite elements, there is no need to be concerned with elements that differ from the optimum shape.

The recommended process of creating the mesh:

- Generate a homogeneous mesh over all macroelements of the plate with mesh density comparable with the plate thickness (1x – 5x)
- In regions where high stress gradients can be expected (point supports, corners with internal angle > 180°), the mesh should be refined using singular points and lines. For singular points, it is possible to prescribe the element's edge size (density) in the centre and range of the mesh refinement. This range should be at least 2x - 3x the mesh density in the centre and both values (density, range) should be in a reasonable ratio to the mesh density prescribed in the neighbouring regions. This will provide a smooth transition among regions with different densities. Singular lines must be treated in a similar way.

You can improve the mesh quality by changing the input parameters. The final mesh quality is improved by the algorithm for mesh smoothing, which can be switched off (not recommended). The analysis of the plate problem is very fast so that large meshes with thousands of elements can be used in interactive working with the program.
The program can generate a finite elements mesh over an arbitrary plane region having boundary that consists of line segments, circular arcs and circles. The plate can be divided in one or more macroelements with constant thickness and homogeneous material properties and can contain an arbitrary number of openings. The openings must lay completely in one of the macroelements. A cutout can also come into being as an empty region surrounded by neighbouring macroelements. In this case, it is not input as an opening. It is also possible to put in internal points and lines, which are regarded as nodes and element sides of a mesh. For points and lines describing the plate geometry, it is possible to prescribe a mesh refinement in the same way as for singular points and lines (density, range).
The user can choose generating a triangular, quadrilateral or mixed mesh. The kernel of the algorithm consists of Delaunay triangulation supported by other methods for modification and optimisation of the mesh. The final operation of the generator is renumbering the nodes to achieve a minimum band with of the system of equations in the analysis.

6.2.2 Running the generator

The finite elements mesh generator can be run with the icon placed on the toolbar (see screenshot).

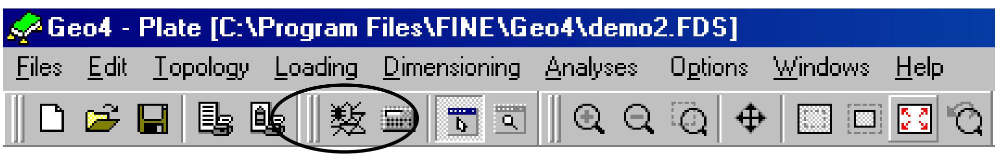


Figure 6.40 – Screen printout with generated finite element mesh

The following figure shows the generated finite element mesh.

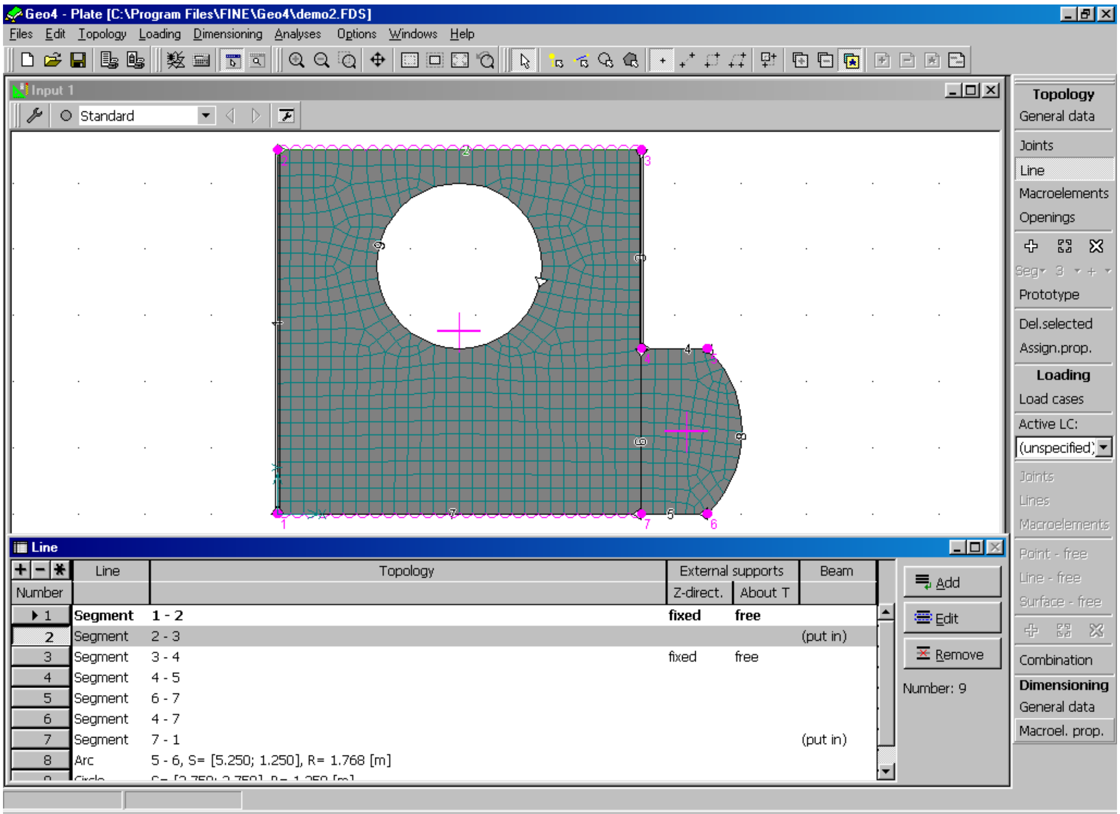


Figure 6.41 – Screen printout with generated finite element mesh

6.3 Analysis

6.3.1 Analysis – theoretical background

The first part of analysis consists of setting-up the global stiffness matrix. The matrix must comply with the boundary conditions (clamped, simply-supported or elastic) in points and lines and respect the subsoil interaction. Computing the right-hand sides of the linear system of equations based on loadings is the second step. The symmetric equation system with band structure is solved using the Cholesky method of decomposition of the stiffness matrix into upper und lower triangular matrices. The primary degrees of freedom are displacements w, φ_x a φ_y in the nodes. Internal forces m_x, m_y, m_{xy}, q_x and q_y together with derived characteristics m_1, m_2 are computed after solving the system of equations.

The right choice of finite elements is essential for the quality of results of the plate problem. The program Plate 2.0 uses the deformation variant of the FEM with triangle and quadrilateral elements DKMT and DKMQ (Discrete Kirchhoff-Mindlin Triangle and Quadrilateral).

The elements are based on the Kirchhoff theory of bending of thin plates, which can be considered as a special case of the Mindlin theory of thick plates using the following assumptions:

- change of the plane thickness in the z direction is negligible compared to the absolute value of the deformation w
- normals to the mid-surface of the plate before deformation remain straight after deformation but do not necessarily remain normal to it
- the normal stress σ_z is small compared to the stresses σ_x, σ_y

The DKMT and DKMQ elements have 9 and 12 degrees of freedom. There are three independent variables in each node:

- w – deflection
- φ_x – rotation about the x-axis
- φ_y – rotation about the y-axis

The elements fulfil the following criteria:

- the stiffness matrix has proper rank (no extra zero-energy modes)
- pass the patch test
- suitable for thin and thick plates
- good convergence characteristics
- computationally efficient

When choosing the mesh type, the quadrilateral elements can be slightly preferred, because they have (providing a high-quality mesh is used) generally better properties than the triangular ones. The program uses the Winkler-Pasternak model of interaction with foundation, which is characterized by the constants C1 a C2. The contact stress σ is computed as one of the results.

The plate can be reinforced with beams. A compatible one-dimensional element with w, φ_x and φ_y as degrees of freedom is implemented. The resulting internal forces are m_1, m_2 a q_3.

6.3.2 Start of analysis

You launch the analysis either with the button on the tool bar (see **Fig. 5.42**) or by pressing **F9**.

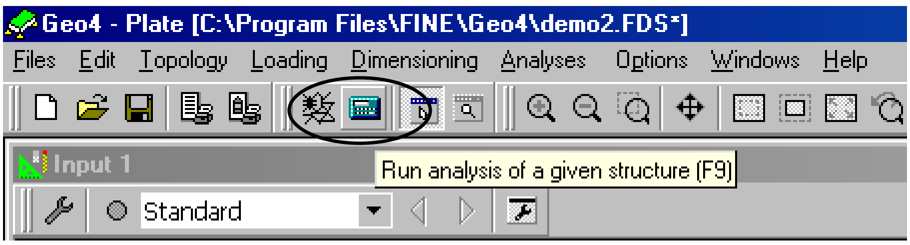


Figure 6.42 – Button for analysis of a given construction

6.4 Results – the postprocessor

When the analysis is completed, the program switches to the regime for presenting the results.

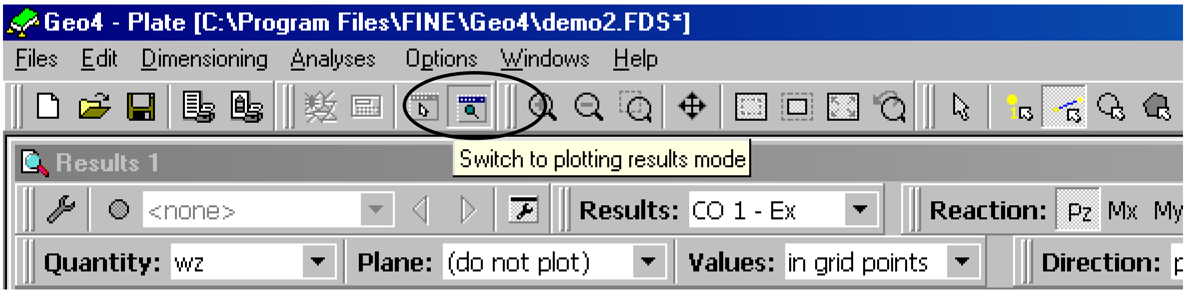


Figure 6.43 – Switching to regime for plotting results

Fig. 6.44 shows the desktop in the regime for presenting the results. Several options for presenting the results are discussed next.

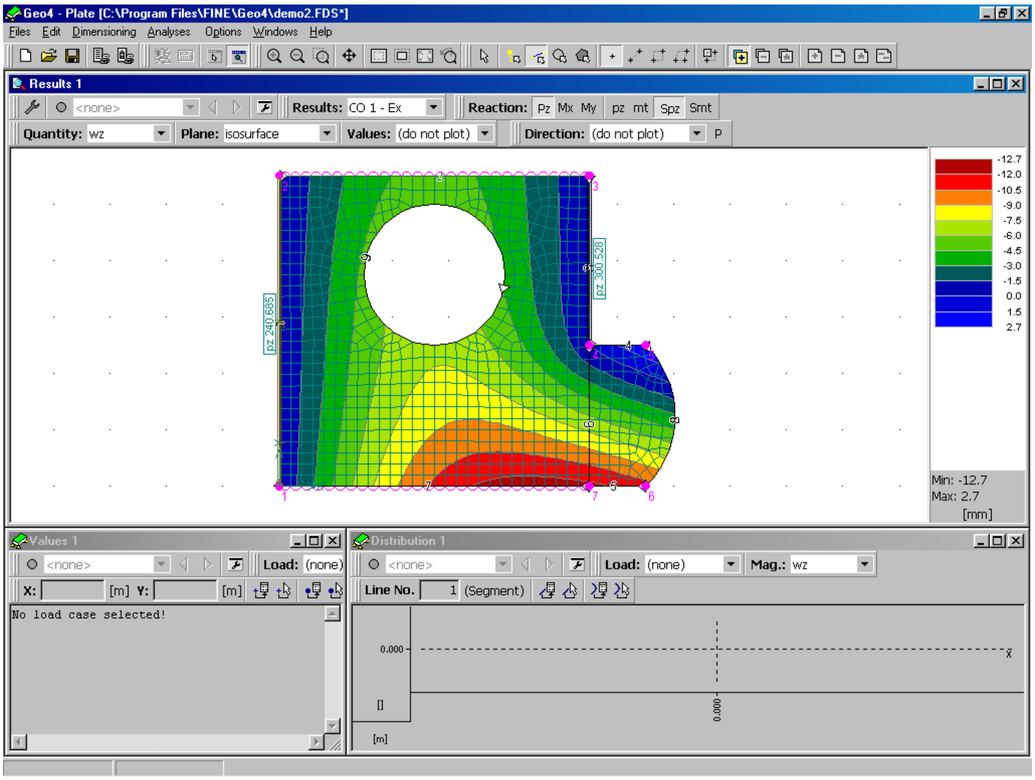


Figure 6.44 – Screen printout after switching

The program facilitates presentation of results in planes, along lines and in points. Standard layout of the postprocessor window is shown in the figure. Two menu bars are located above every output window (their functions will be described later). Size of the picture in the window (zoom) is determined on the standard horizontal menu bar for the active window.

Note:
During the work, it may happen that the windows will start to overlap, some of them may disappear etc. In that case, you can use the function "Arrange optimally" (in the menu "Windows") – the program will arrange windows in the same fashion as they were after the first switch to the postprocessor.

The most important options of results presentation are:
Results – determines combinations (LC, respectively) that are plotted. At the first launch, the first extreme combination is set.
Reactions – pressing these buttons determines which reaction forces should be plotted

Pz, Mx, My	- reactions in the point support
pz, mt	- reactions along the line support
Spz, Smt	- distributions of reactions along the line support

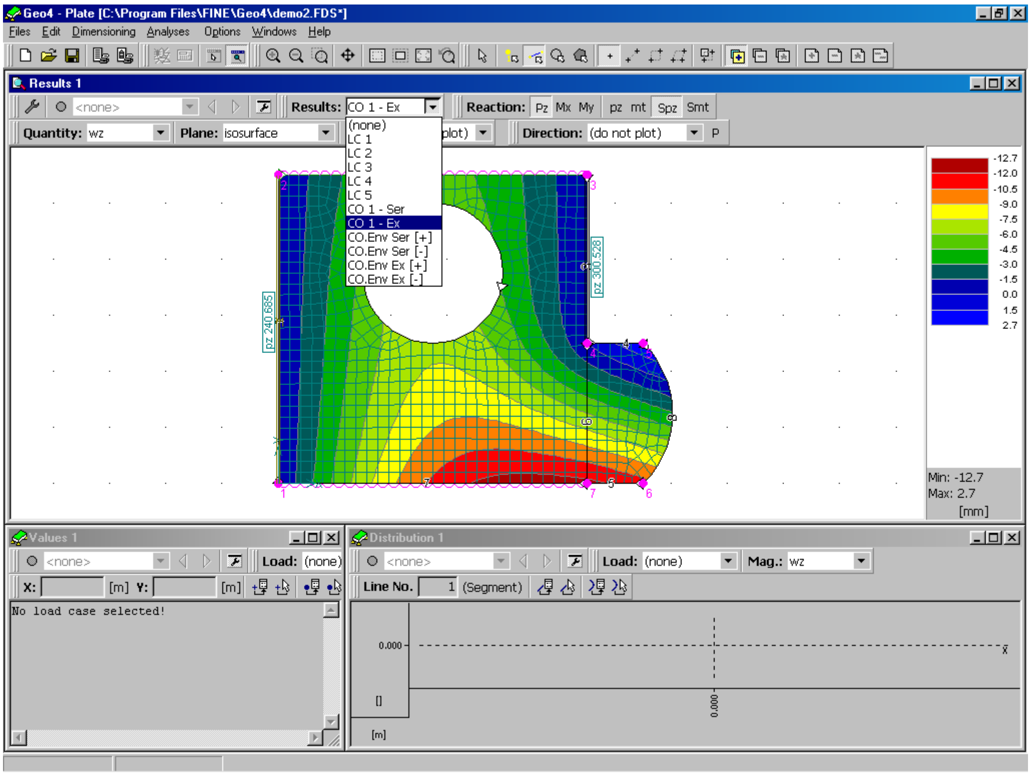


Figure 6.45 – Presenting results

Quantities – determines quantities that appear in the window

w_z	- displacement
φ_x, φ_y	- rotation
m_x, m_y, m_{xy}	- moments
q_x, q_y,	- shear forces
σ	- contact stress
m_1, m_2,	- principal moments
qmax	- principal shear force

Plane – determines the way of results presentation (isosurfaces, isolines, transitions…)

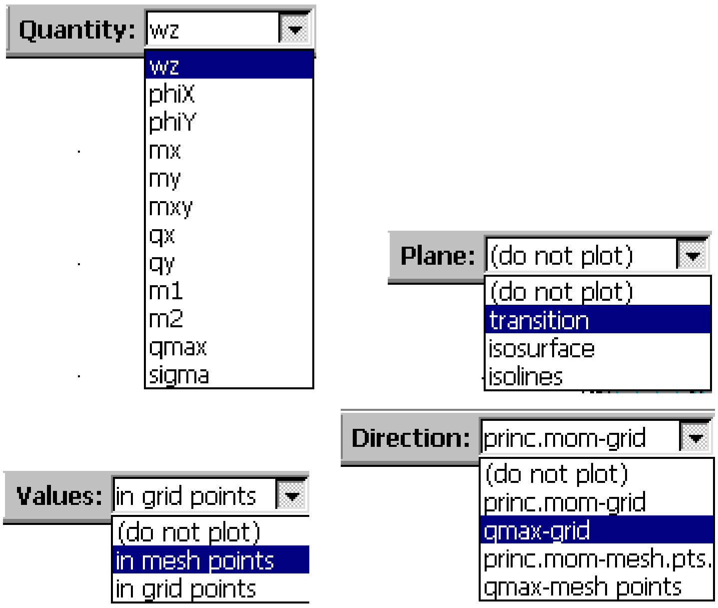


Figure 6.46 – Selection of plotting quantities

Quantities – determines magnitudes to be drawn – the most useful way of plotting is to use the preset grid. You set the grid from the menu "**Options**", "**Grid**".

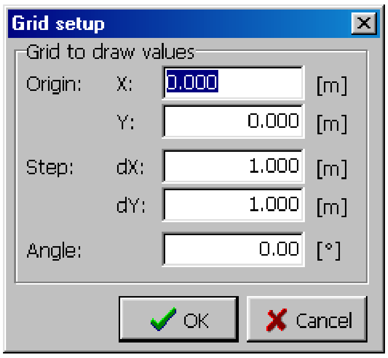


Figure 6.47 – Grid setup

Direction

Here you can set the plotting of direction of the principal moments (shear forces) in the grid or in the mesh points, respectively. **Fig. 6.46** shows the plot of principal moments in the grid:

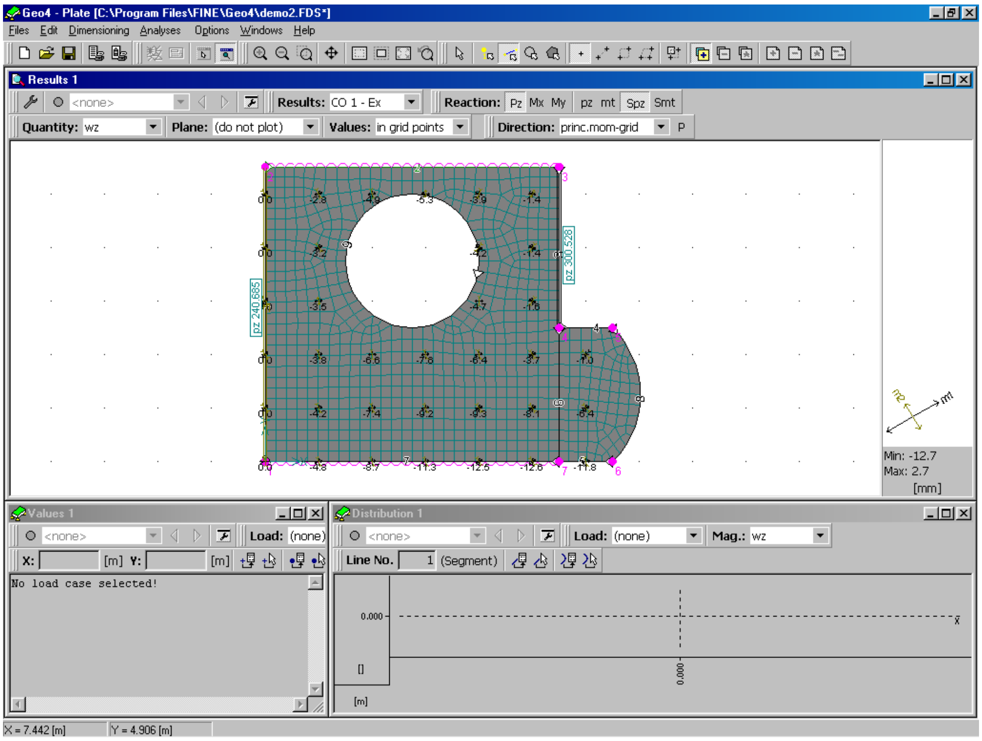


Figure 6.48 – Screen printout

The magnitudes of all constants can be found in every joint or point. To this end, use the "**Values**" dialog window in the left lower part of the desktop.

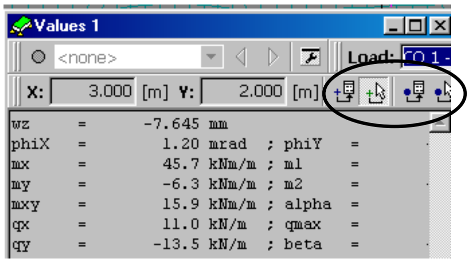


Figure 6.49 – Buttons for the "Magnitudes" dialog window

Fig. 6.49 displays the setting of magnitudes for the combination No. 1 – EX. in the point x = 2,5 m, y = 1,5 m). The buttons (indicated on the picture) serve to choose a joint or a point either graphically or using a dialog window.

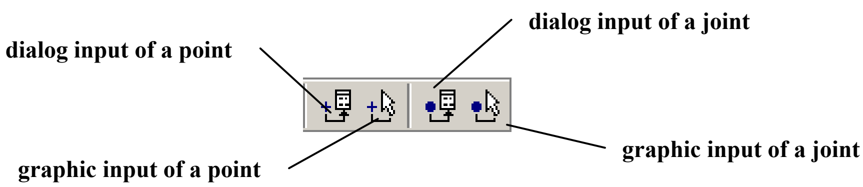


Figure 6.50 – Buttons for selecting points (joints)

Using similar procedure, you can display (again for the selected combination or particular LC) magnitudes on any line or segment (the "**Distribution**" dialog window in the right bottom corner of the desktop). Forces developed within girders can also be plotted along a line. If you want to plot forces in girders, you choose the relevant line and the required magnitude (e.g., m_2-girder). To select lines (similarly as for joints and points), use buttons in the dialog window – see **Fig. 5.51**.

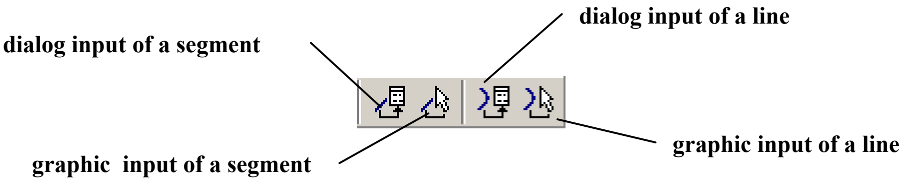


Figure 6.51 – Buttons for selecting lines (segments)

Fig. 6.52 shows the distribution of bending moment in the girder located on the line No. 2. It can be obtained by pressing the "**Choose line graphically**" button (the buttons indicated in the figure serve for working with a line – either using the dialog window or graphically) and then choosing the line No. 2. To proceed, select the quantity (**m2-girder – bending moment in a girder**) and the dialog window displays the moment distribution.

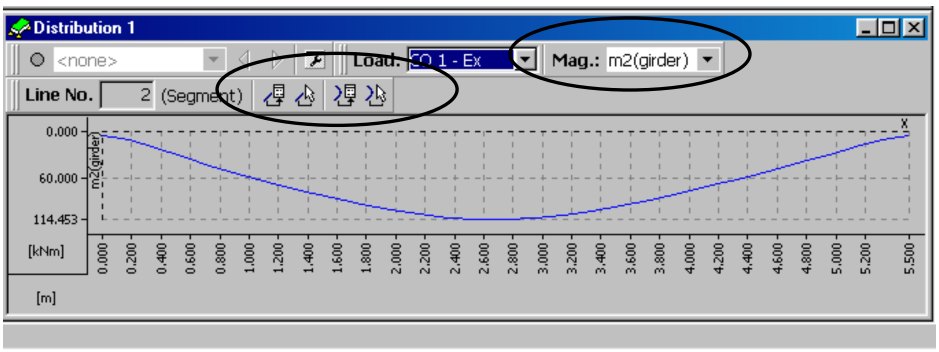


Figure 6.52 – Distribution of bending moment

You can apply the same process to get the distribution of other quantities.

m_1 (girder) – twisting moment on a girder
m_2 (girder) – bending moment on a girder
q_3 (girder) – shear force on a girder

Selecting the desired quantities to be shown on the desktop simplifies the plotting. The "**Plotting setup**" dialog window, in which you choose the required setup, will appear by clicking the button indicated on the following figure (it is always located on the menu bar in the window).

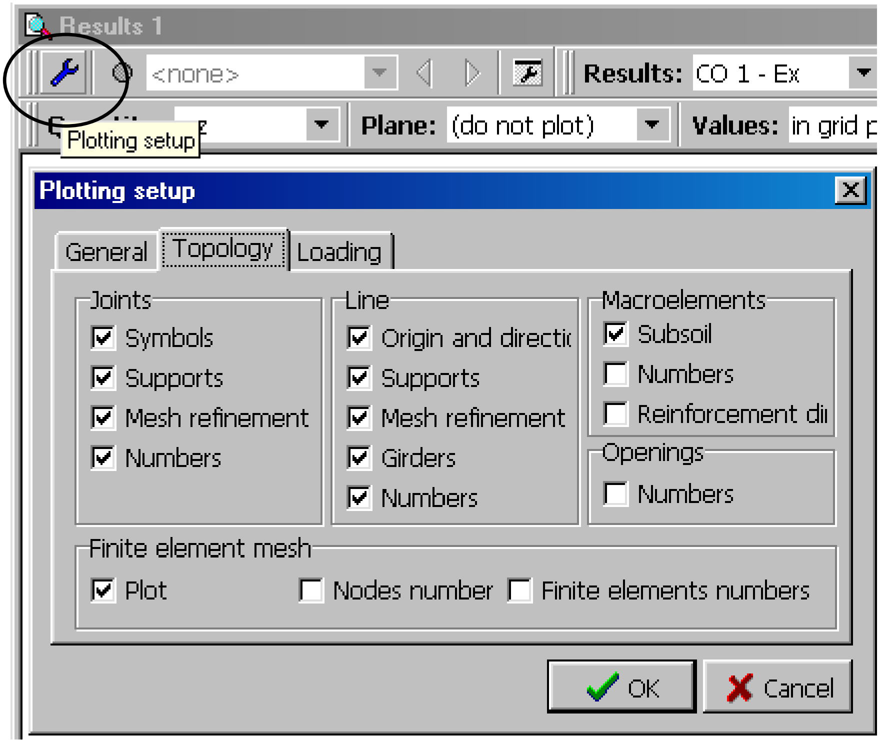


Figure 6.53 – Button for choosing setups

Every menu bar has a button for saving the setup of particular dialog window (plus its description). Therefore, you can create arbitrary number of views in every dialog window. Then you can simply switch from one to another without redefining the setup.

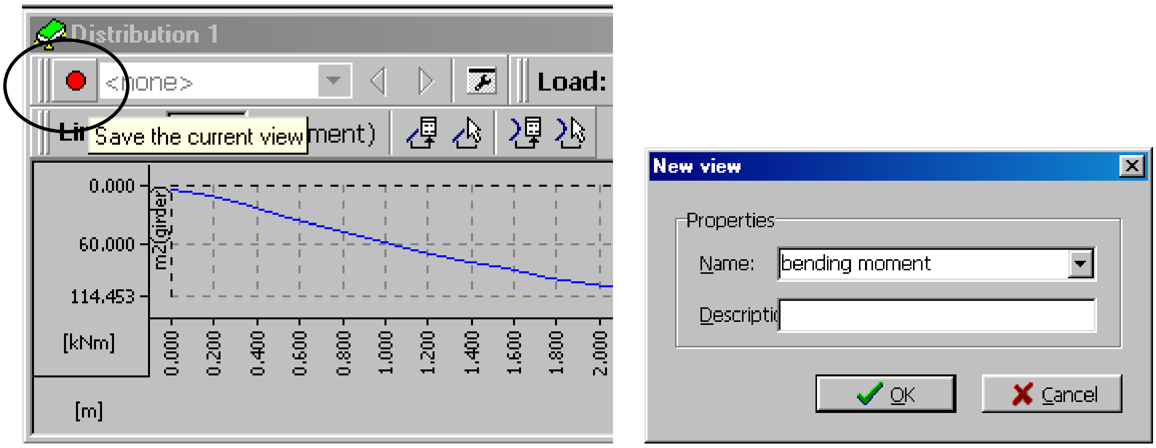


Figure 6.54 – Saving the current view, creating a new view

6.5 Print – text and graphic outputs

6.5.1 Text output

The dialog window "**Plate – output document**" will appear after clicking "**Text printout**", see **Fig. 6.55**.

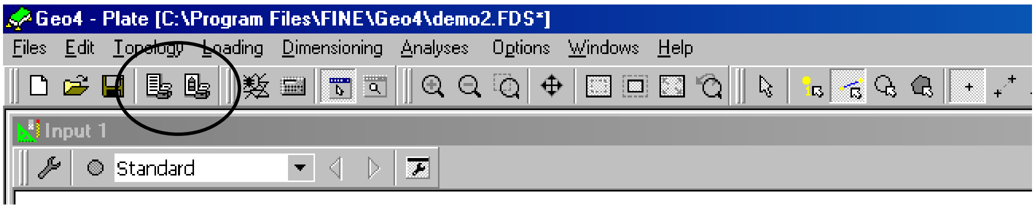


Figure 6.55 – Menu bar – button for printing

You tick the data you want to include into your output protocol on the left hand side of this dialog window, and by using the button "**Generate**" you create the protocol, **Fig. 5.56**. The button "**Save document**" on the menu bar is used to save the generated output protocol (in format **.RTF*). Using the "**Graphic printout**", you send the document to the printer. The last button – "**Setup of logo, header setup, footer setup** – serves for setting up the page appearance.

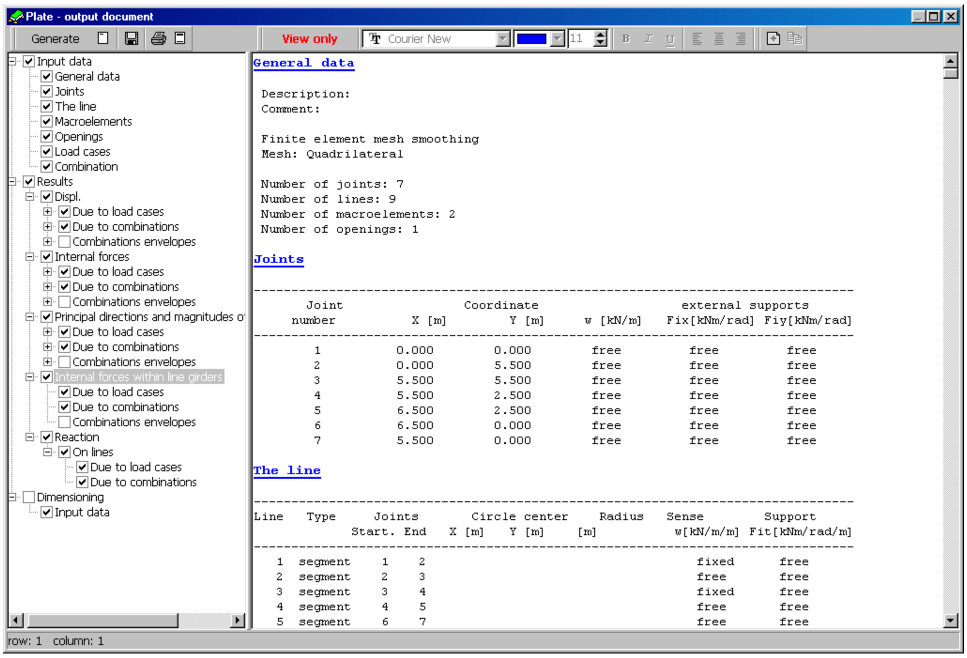


Figure 6.56 – Output protocol

Elements on the menu bar above the generated text serve for basic editing of the document.

6.5.2 Graphic output

Rules for the graphic printout are simple. The active window from the desktop is always printed. The printer prints corresponding status of the monitor.

Fig. 6.57 shows the active dialog window "**Distribution 1**". (If printing the graphic output, this dialog window appears on the printed page.)

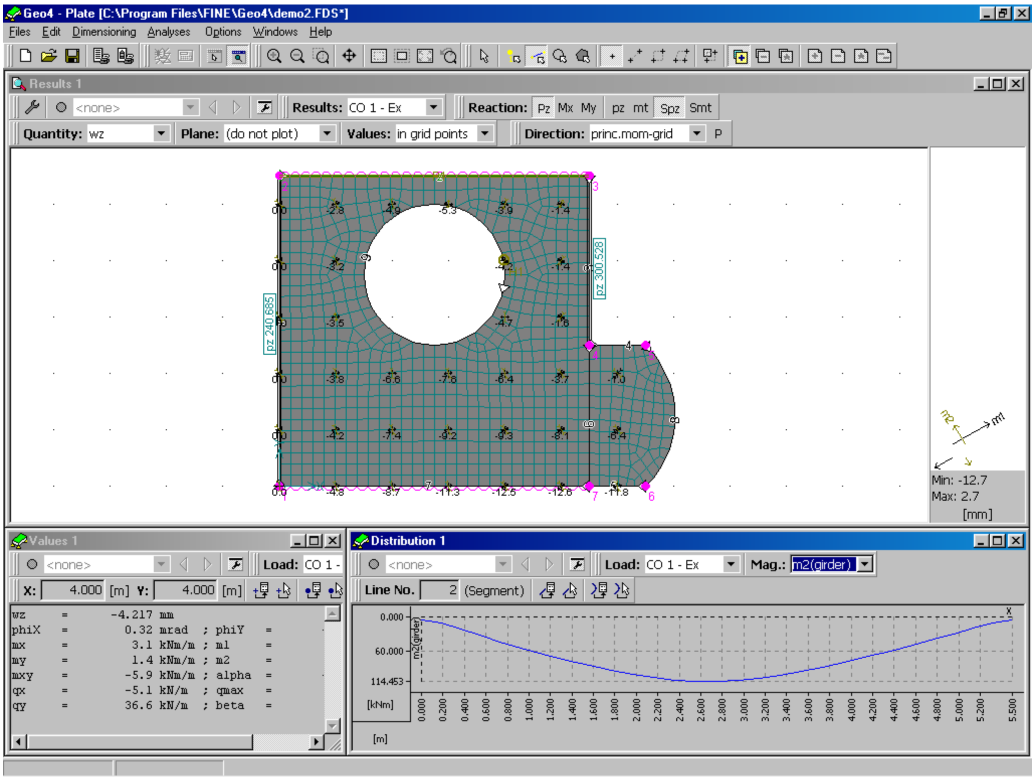


Figure 6.57 – Final screen printout

6.6 Module „Reinforcement"

6.6.1 Input

The module reinforcement allows you to compute the required area of the plate longitudinal reinforcement and the required area of shear reinforcement represented by stirrups. The basic parameters of reinforcement are inputted in the "General data" dialog window.

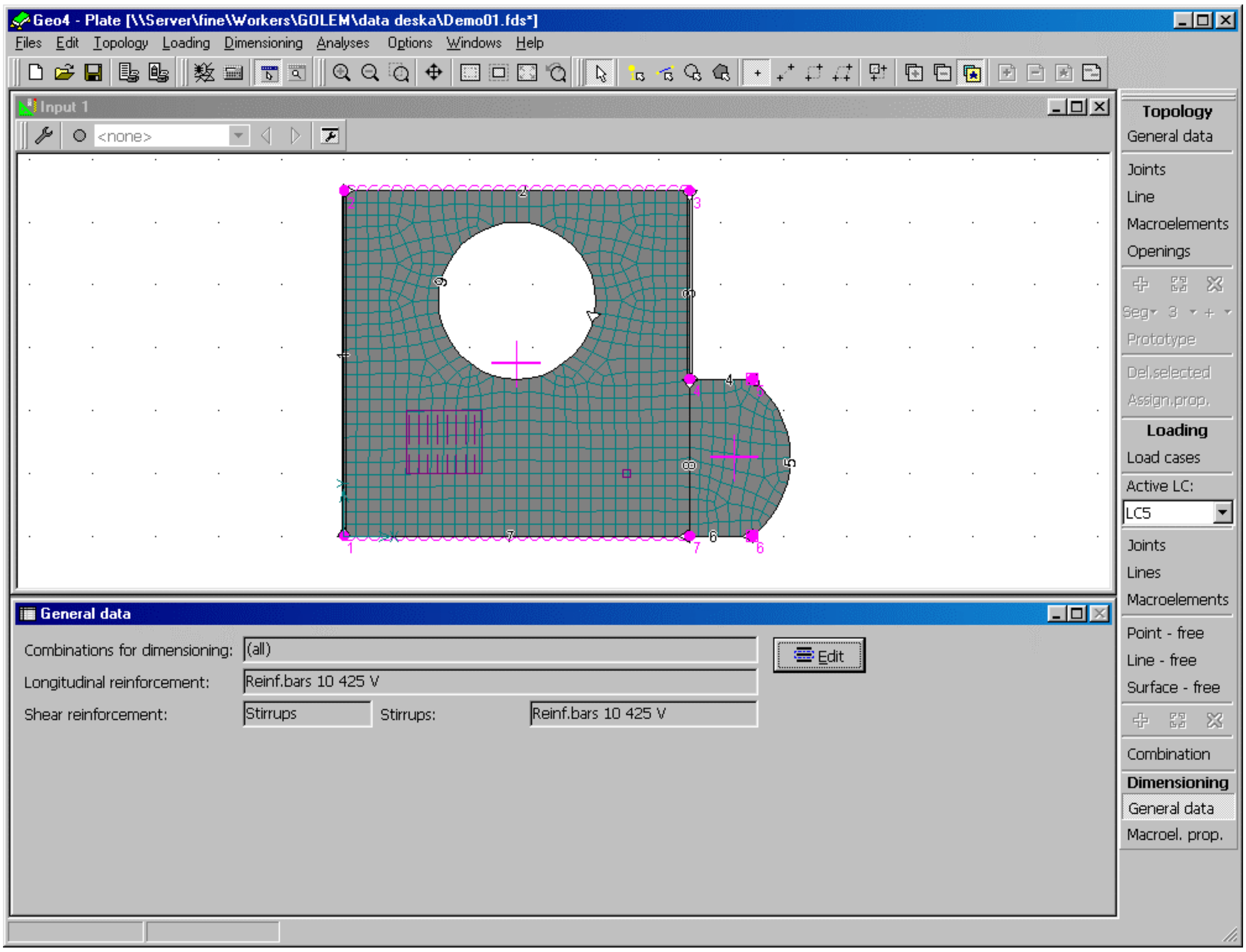


Figure 6.58 – Dimensioning – general data

You are required to specify the material of longitudinal and shear reinforcement together with the selected combination for which the dimensioning is carried out. Default setup assumes the dimensioning analysis to be performed for all combinations – free (all) – therefore, it takes into account the envelope of combinations.

The "**Macroel. prop.**" window serves to specify directions of reinforcement bars within individual macroelements and centroids of bars measured from the edge of concrete. Reinforcement bars in both directions must be mutually perpendicular.

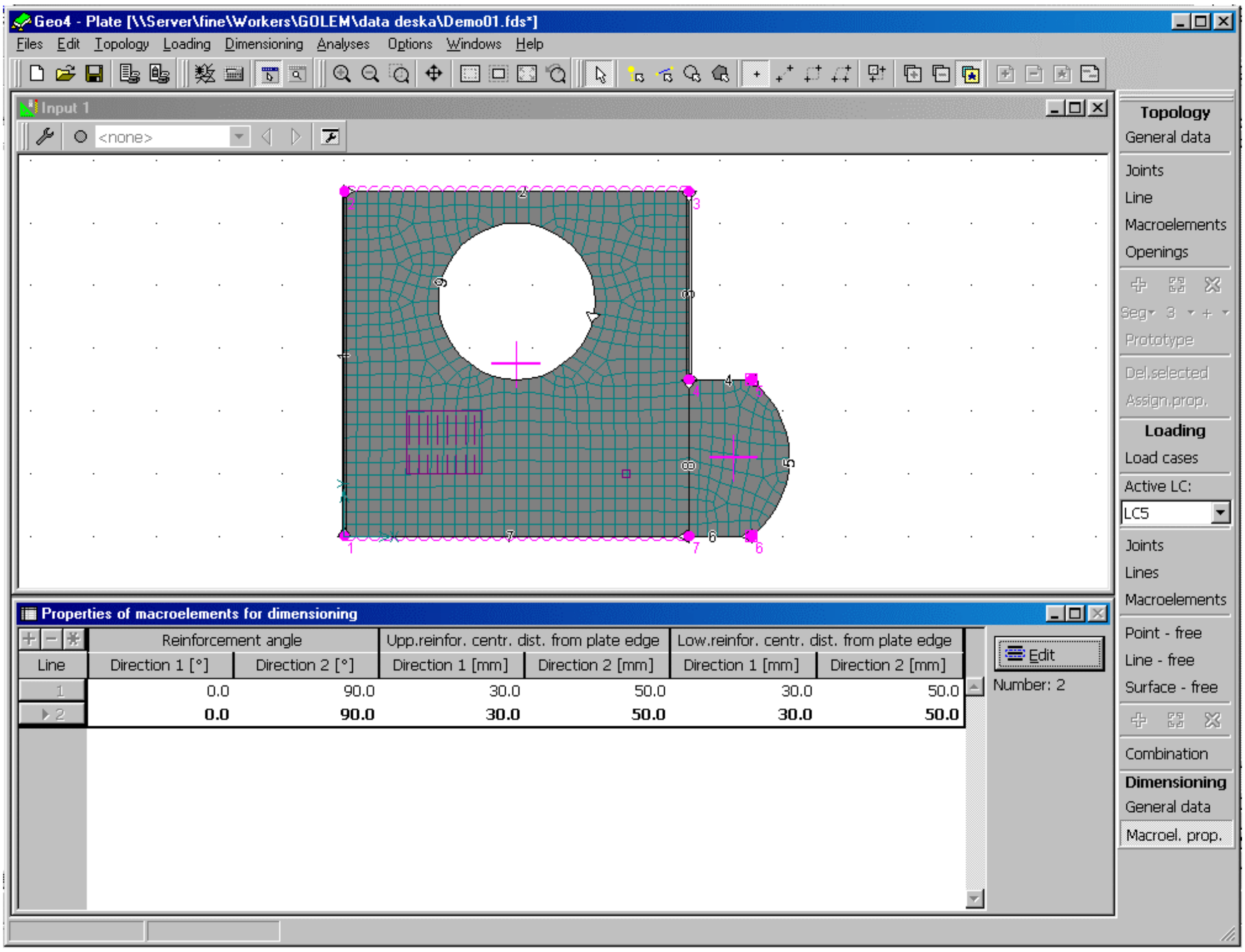


Figure 6.59 – Dimensioning – properties of reinforcement

6.6.2 Analysis

The programs first transforms moments m_x and m_y into the directions of reinforcement bars. The dimensioning moments are then provided

$$M_{1,\dim} = m_u \pm abs(m_{uv})$$
$$M_{2,\dim} = m_v \pm abs(m_{uv})$$

$$m_u = m_x \cos^2\alpha + m_y.\sin^2\alpha + 2.m_{xy}\sin\alpha\cos\alpha$$
$$m_v = m_x \sin^2\alpha + m_y.\cos^2\alpha - 2.m_{xy}\sin\alpha\cos\alpha$$
$$m_{uv} = -m_x\sin\alpha.\cos\alpha + m_y\sin\alpha.\cos\alpha + m_{xy}(\sin^2\alpha - \cos^2\alpha)$$

Depending on the above moments the program then determines the required area of the upper and lower reinforcement in individual directions together with the degree of reinforcement. The actual analysis depends on the selected standard – computational steps related to various standards are described in the appendix.

The program also determines the maximum shear forces developed in the plate, computes the shear force transmitted by concrete and the maximum allowable shear force. Providing the shear force exceeds the shear strength of plain concrete the programs determines the required area of stirrups.

6.6.3 Postprocesing

Postprocessing of final reinforcement distributions is carried out in analogy with other plate variables. First, switch to the module for plotting the results of the dimensioning analysis.

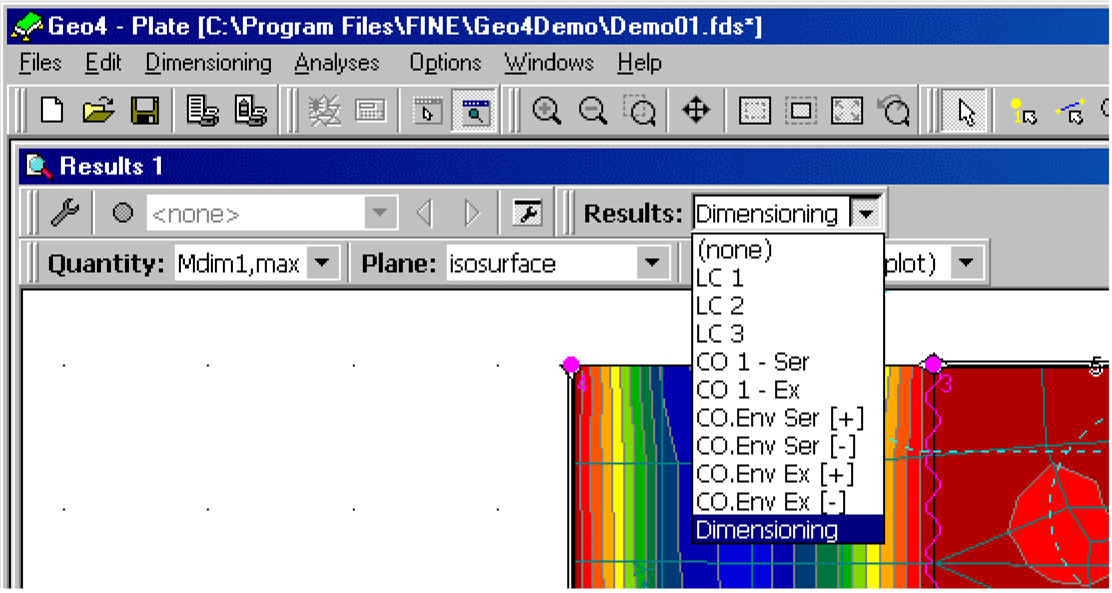


Figure 6.60 – Results of dimensioing

The next step requires selection of a given variable to be shown.

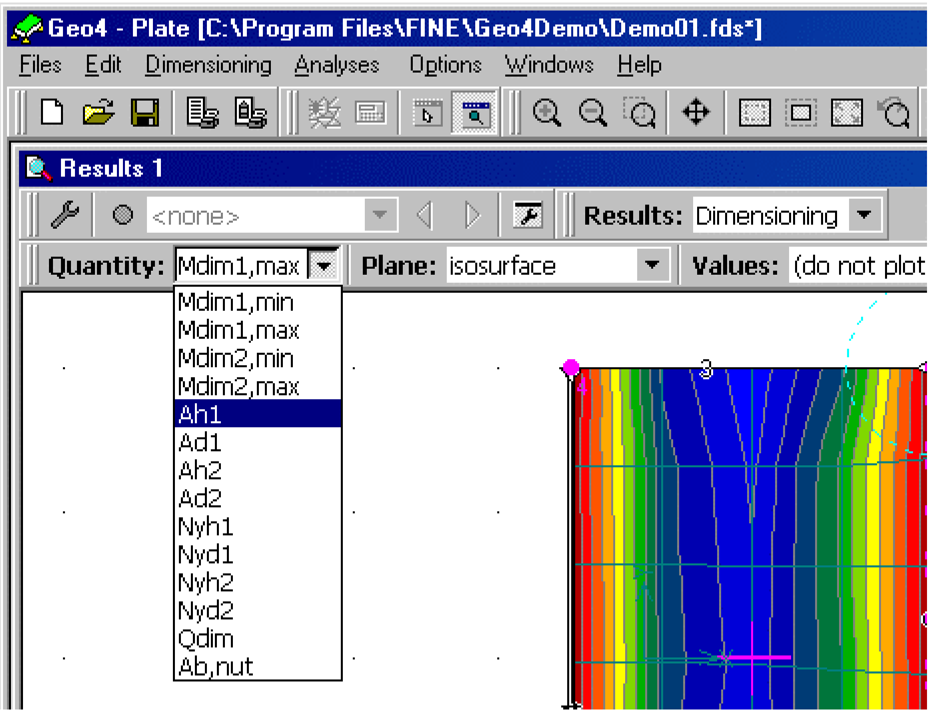


Figure 6.61– Selection of variables for plotting

Results for a given point provided in a text format are also well arranged, Fig. 5.62

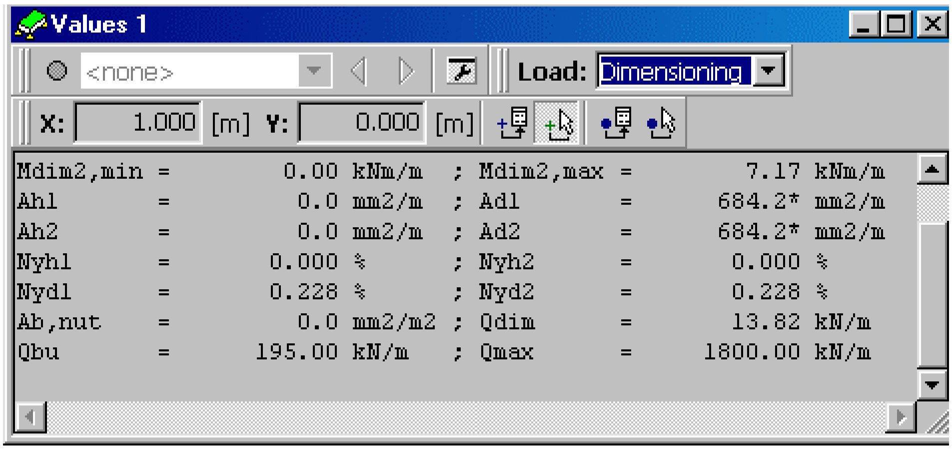


Figure 6.62 – Results pertinent to a given point