Example 5

Plane stress analysis of a wrench

Contents   Page

1 Description of the problem

An example for analyzing stresses and deformations of a wrench is selected to illustrate some features of *ELPLA* for analyzing plane stress problems.

1.1 Loads and dimensions

The wrench shown in Figure 5-1 is 10 [mm] thick and about 200 [mm] in length. The wrench end is loaded with a pressure of 2 [N/mm] along 100 [mm] of its handle as shown.
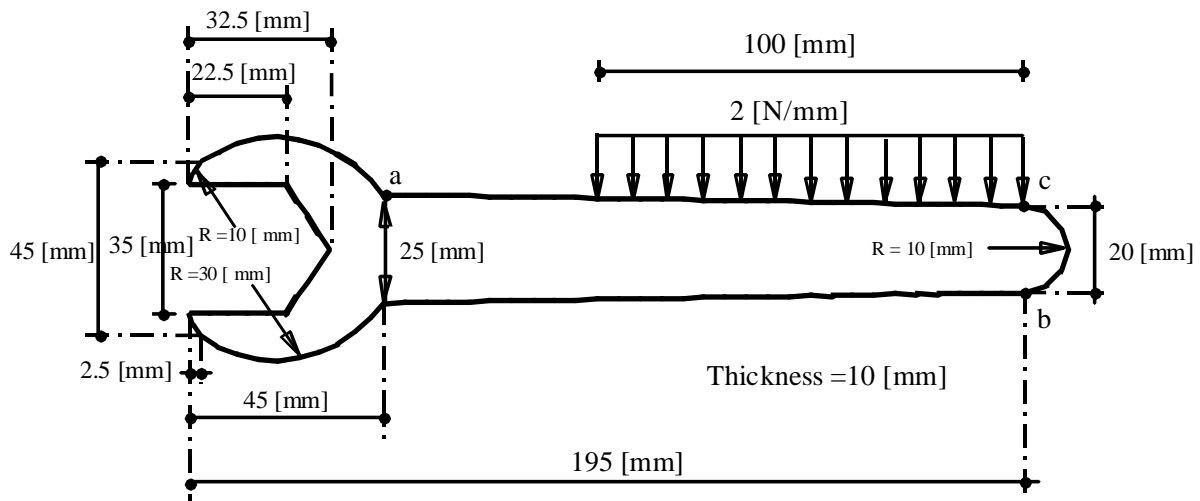

Figure 5-1   Wrench dimensions and loads

1.2 Wrench material

Material of the wrench has the following parameters:

Young's modulus E_b = 200000   [N/mm^2]
Poisson's ratio ν_b = 0.3   [-]

1.3 Analysis of the wrench

It is required to analyze the stresses and deformations of the wrench when it is being used to tighten a bolt. Consider that there is no horizontal or vertical displacement along the lines where the wrench contacts the bolt. This Tutorial Manual will not present the theoretical background of modeling the problem. For more information concerning the method of analysis a complete reference for numerical calculation methods are well documented in the User's Guide of *ELPLA*.

1.4 System units

Since the wrench dimensions are relatively small compared with those of foundations, it is preferred to use small units for measurement. To change the system units, choose "System of units" command from "Main data" menu of *ELPLA-Data*. The dialog box in Figure 5-2 appears. In this dialog box change the units for lengths and forces to [mm] and [N] respectively and then click "Save" button.
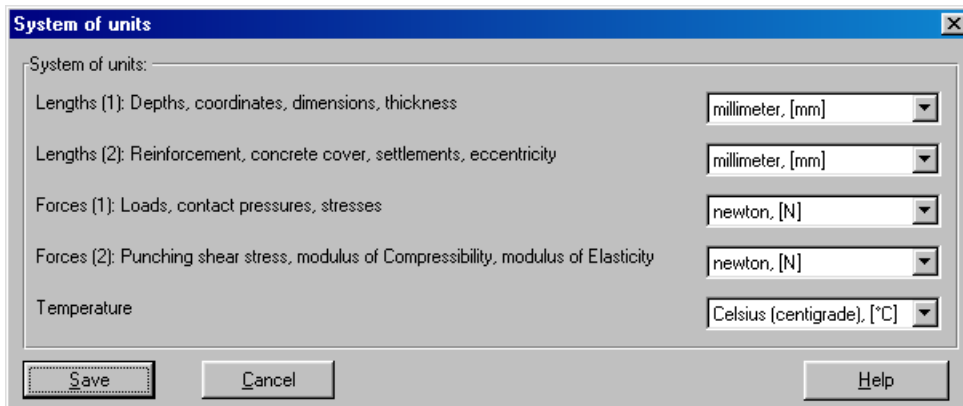
Figure 5-2   "System of units" dialog box

Also *ELPLA* checks the element size when a mesh is generated. Since the element size of slab foundation is relatively large compared with those of a wrench, it is required to disable the element size check. To uncheck the element size when generating FE-Net, choose the "Preferences" command from "Main data" menu of *ELPLA-Data*. The dialog box in Figure 5-3 appears. In this dialog box uncheck the "Check element size" option then click "Save" button.
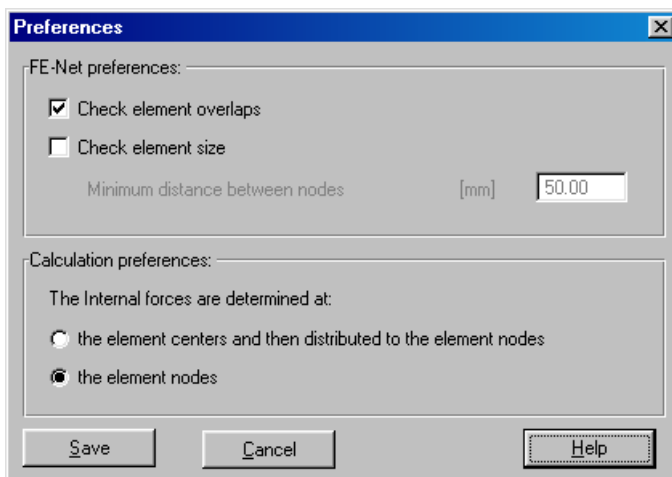

Figure 5-3   "Preferences" dialog box

2   Creating the project

In this section the user will learn how to create a project for analyzing a plane stress problem. The example will be processed step by step to show the possibilities and abilities of the program. To enter the data of the example, follow the instructions and steps in the next paragraphs.

2.1   Calculation method

To create the project, start the sub program *ELPLA-Data*. Choose the "New project" command from the "File" menu. The "Calculation methods" wizard appears, Figure 5-4. The wizard will guide you through the steps required to create the system data of the project. As shown in this Figure, the first form of the wizard is the "Analysis type" form.
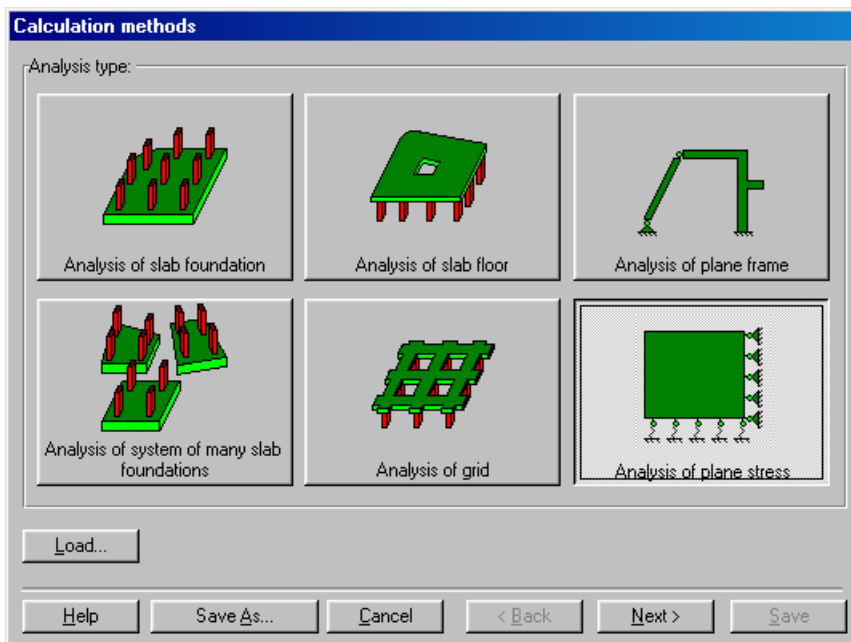
Figure 5-4　　"Calculation method" wizard with "Analysis type" form

In the "Analysis type" form in Figure 5-4, define the analysis type of the problem where *ELPLA* can deal with different structural systems. As the analysis type is a plane stress problem, select "Analysis of plane stress" option then click "Next" button to go to the next form.

The next form is the "System symmetry" (Figure 5-5). In this form choose "Unsymmetrical system" and click "Next" button.

The last form of the wizard assistant contains the "Option" list, Figure 5-6. In this list *ELPLA* displays some of the available options corresponding to the used numerical models, which differ from model to other.

In this list check "Supports/ Boundary conditions" check box then click "Save" button.
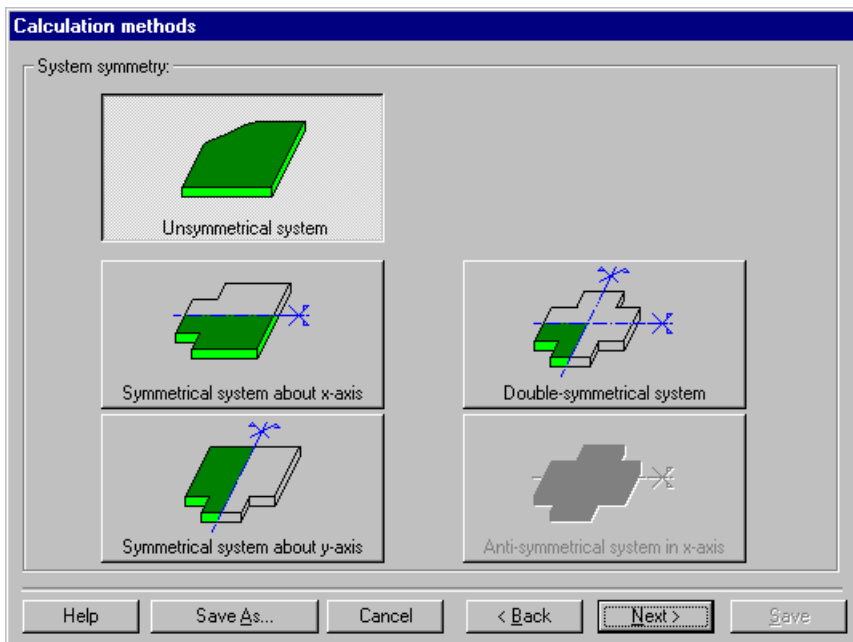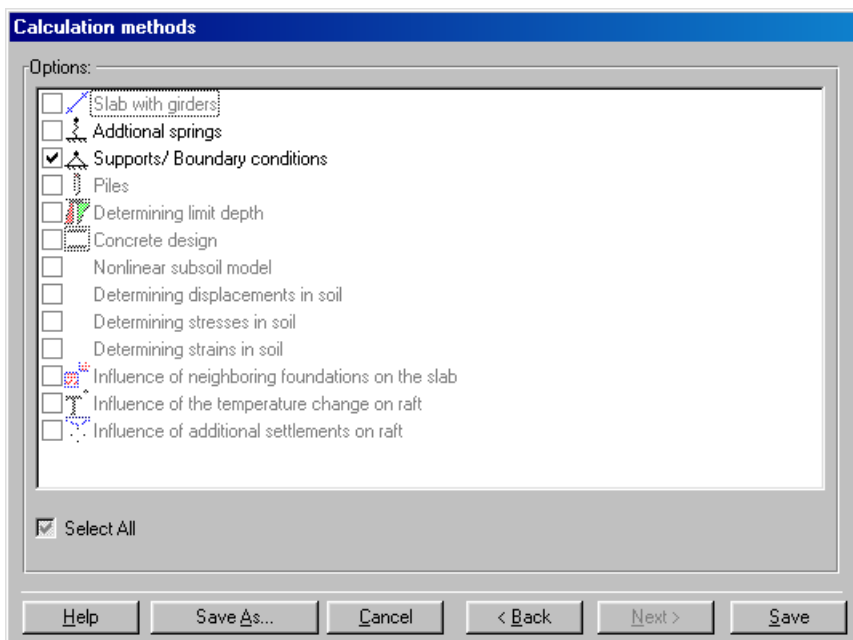
Figure 5-5   "System symmetry" form

Figure 5-6   "Options" list

After clicking "Save" button, the "Save as" dialog box appears, Figure 5-7.

In this dialog box, type a file name for the current project in the file name edit box. For example type "Wrench". *ELPLA* will use automatically this file name in all reading and writing processes. Then click "Save" button to complete the definition of the calculation method and the file name of the project.

ELPLA will activate the "Data" menu. Also the file name of the current project [Wrench] will be displayed instead of the word [Untitled] in the *ELPLA-Data* title bar.
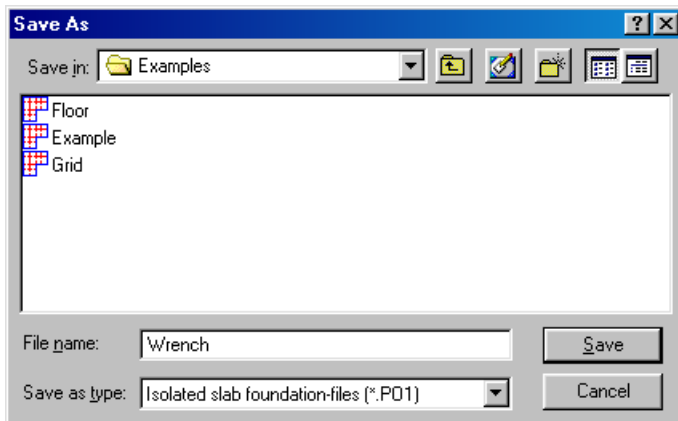
Figure 5-7   "Save as" dialog box

2.2   Project identification

To identify the project, choose "Project identification" command from "Data" menu of *ELPLA-Data*. The dialog box in Figure 5-8 appears.

In this dialog box

- Type the following line to describe the problem in the "Title" edit box:
  "Analysis of a Plane Stress Problem"
- Type the date of the project in the "Date" edit box
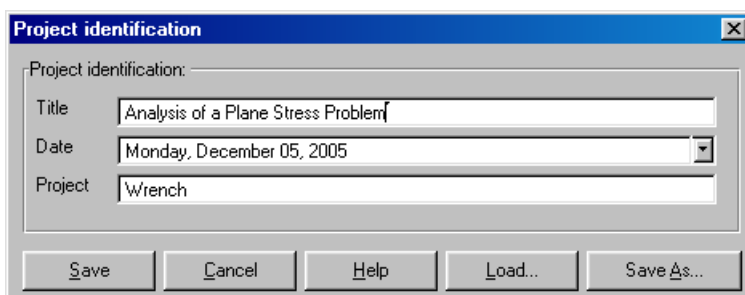- Type "Wrench" in the "Project" edit box
- Click "Save" button

Figure 5-8   "Project identification" dialog box

2.3   FE-Net data

To define the net for the wrench geometry, choose "FE-Net data" command from "Data" menu. The "FE-Net generation" wizard appears as shown in Figure 5-9. This wizard will guide you through the steps required to generate a FE-Net. The first form of the wizard is the "Slab type" form which contains a group of templates of different shapes of nets. These net templates are used to generate standard nets that have constant size in both *x*- and *y*-directions.

To generate the FE-Net, choose the irregular slab option then click "Next" button to go to the next form.
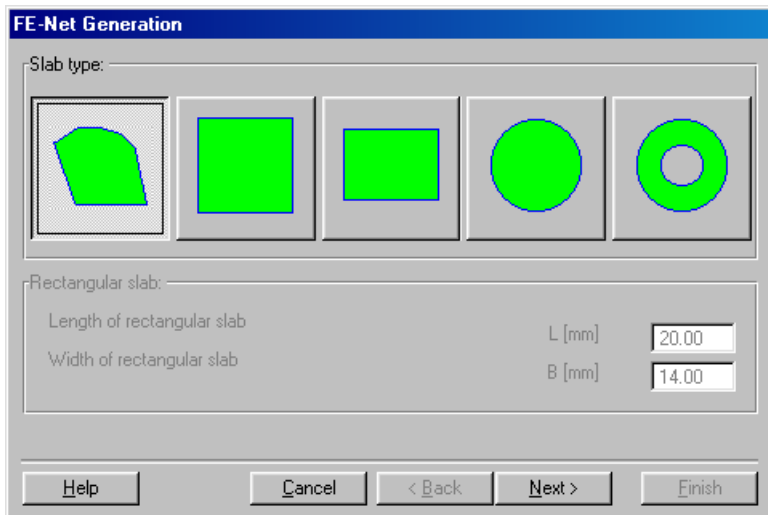
Figure 5-9   "FE-Net generation" wizard with "Slab type" form

After clicking "Next" in the "FE-Net generation" wizard, the following "Generation type" form appears, Figure 5-10. *ELPLA* can deal with various types of generations with triangle and/ or rectangular elements. In this form, choose the triangle elements option then click "Next".
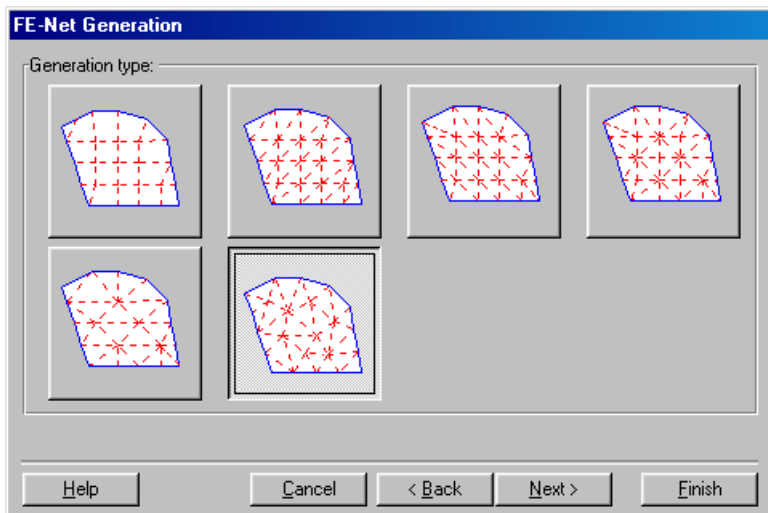

Figure 5-10   "Generation type" form

After clicking "Next" button in "Generation type" form, the following "FE-Net generation" dialog box in Figure 5-11 appears with the default generation parameters. In this dialog box click "Finish" button.
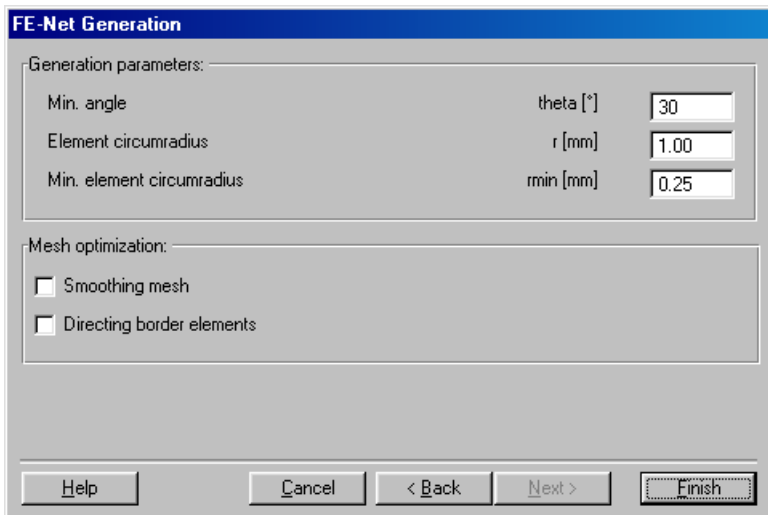
Figure 5-11   "FE-Net generation" dialog box

ELPLA will generate an imaginary net as shown in the following embedded program, Figure 5-12.
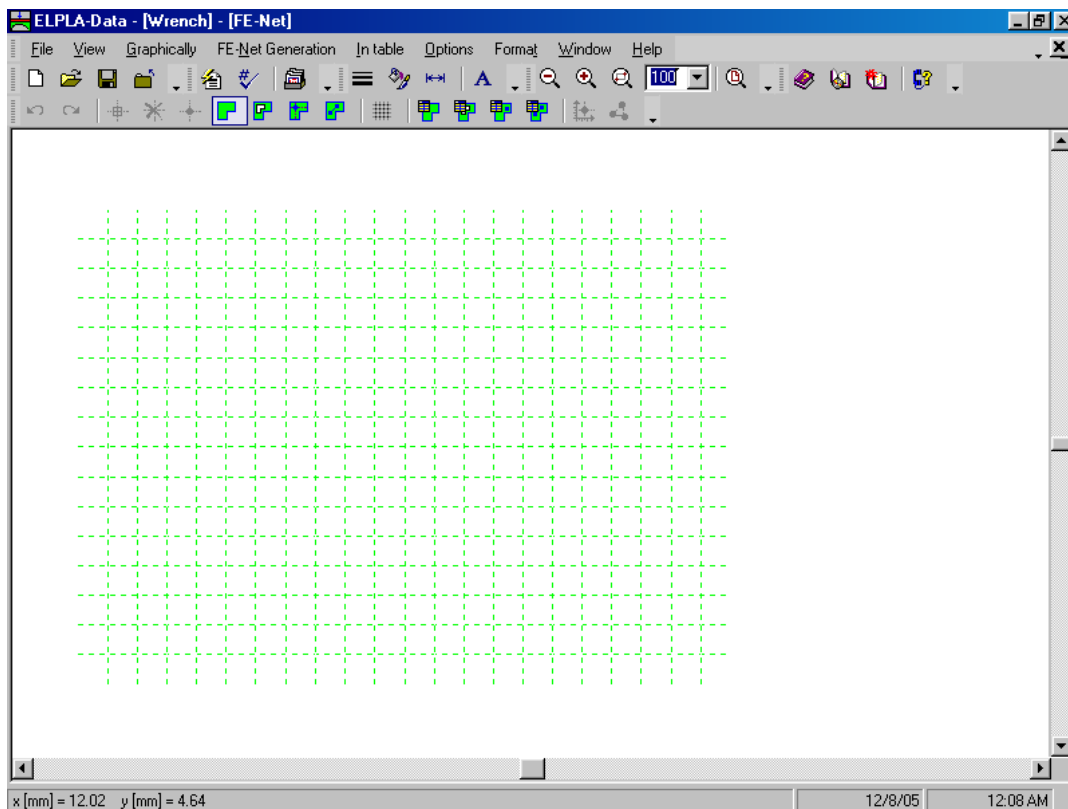

Figure 5-12   Imaginary net of a rectangular area on the screen

You can either draw the wrench directly on this imaginary net or input the wrench corner points in tabulated form. To input the FE-Net corner points, choose "Slab corners" command from "In Table" menu. The dialog box in Figure 5-13 appears.

To define the first segment of the wrench which connects points a and b (Figure 5-1)

- Type 45 in the "*x*1" edit box to define the *x*-coordinate of point a
- Type 32.5 in the "*y*1" edit box to define the *y*-coordinate of point a
- Type 190 in the "*x*2" edit box to define the *x*-coordinate of point b
- Type 30 in the "*y*2" edit box to define the *y*-coordinate of point b
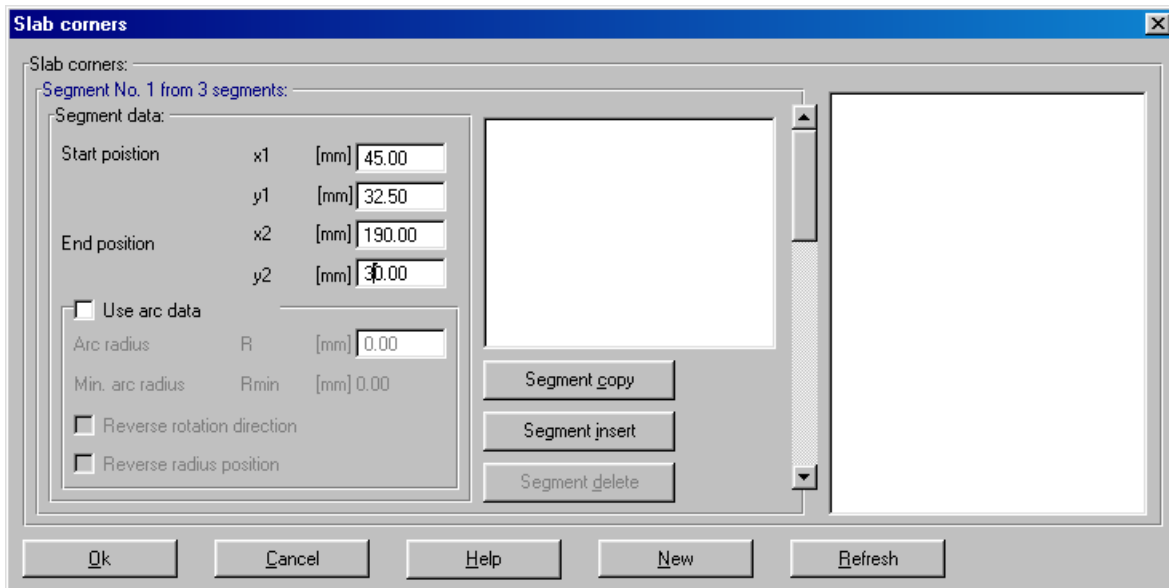

Figure 5-13   "Slab corners" dialog box

To define the second segment of the wrench which connects points b and c

- Use the Scroll bar to define segment No. 2
- Type 190 in the "*x*2" edit box to define the *x*-coordinate of point c
- Type 10 in the "*y*2" edit box to define the *y*-coordinate of point c
- Select the "Use arc data" option to convert the line segment to arc segment
- Type 10 in the "Arc radius" edit box to define the radius of the arc segment

In the "Slab corners" dialog box, *ELPLA* assumes that there are at least three segments with three corner points. Since the wrench drawing contains 11 segments, you can use the command "Segment insert" to insert the rest of the wrench segments. Use the corner points and arc information provided in Table 5-1 to complete the definition of the wrench corner points. Repeat the steps used for defining segment No. 1 to define any line segment and repeat those used for segment No. 2 to define any arc segment.

Table 5-1   Wrench Corner Points

Segment	Start position		End position		Arc Radius
	x_1	y_1	x_2	y_2	
[-]	[mm]	[mm]	[mm]	[mm]	[mm]
1	45	32.5	190	30	
2	190	30	190	10	10
3	190	10	45	7.5	
4	45	7.5	2.5	0	30
5	2.5	0	0	5	10
6	0	5	22.5	5	
7	22.5	5	32.5	20	
8	32.5	20	22.5	35	
9	22.5	35	0	35	
10	0	35	2.5	40	10
11	2.5	40	45	32.5	30

After completing the definition of the wrench corner points, the slab corner dialog box should be like this in Figure 5-14 where a small sketch of the wrench appears in the dialog box picture. Click the "OK" button to see the drawing of the wrench outlines as shown in Figure 5-15.
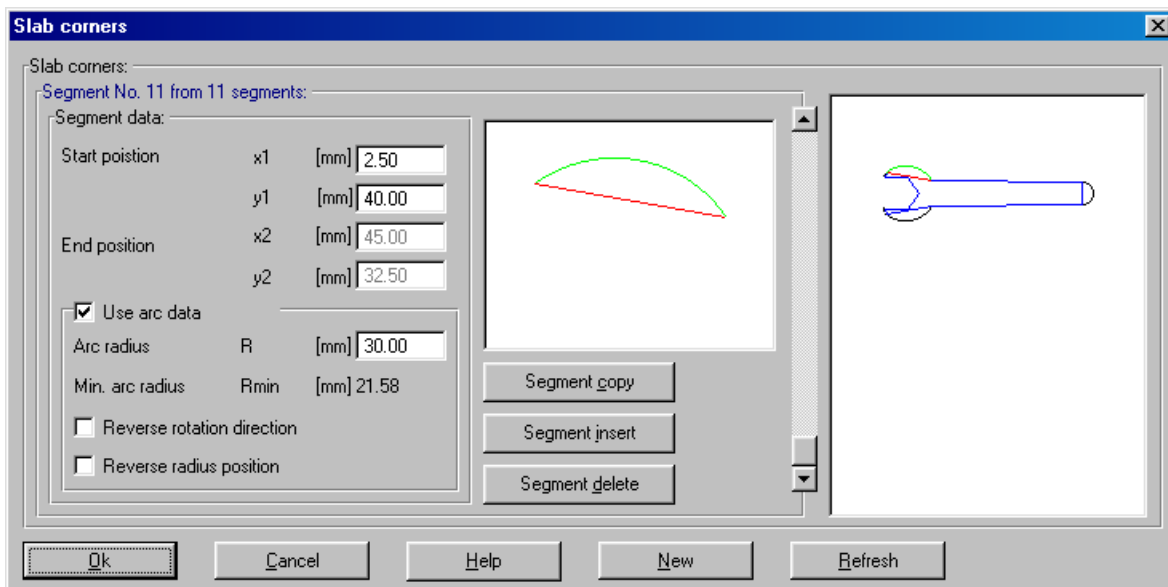

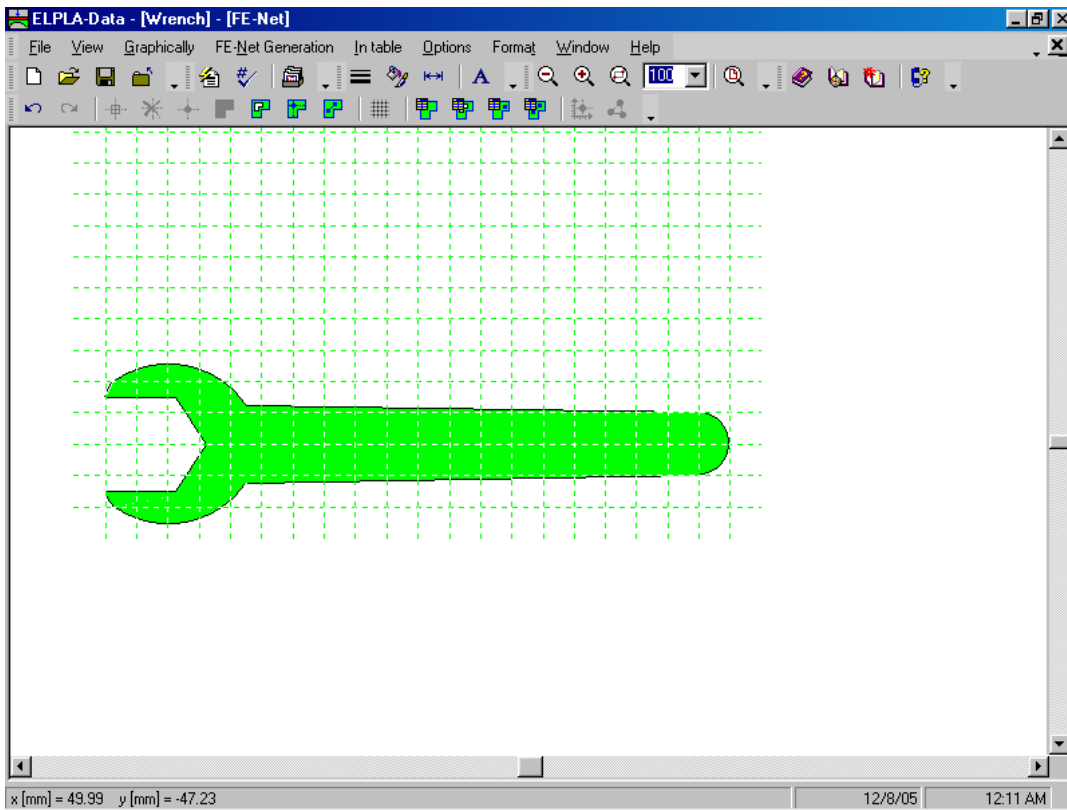Figure 5-14   "Slab corners" dialog box after defining the wrench corner points

Figure 5-15   Wrench outlines

To complete the FE-Generation of the problem, choose "Generating FE-Net" command from "FE-Generation" menu. The following "Generation of FE-Net" dialog box appears.
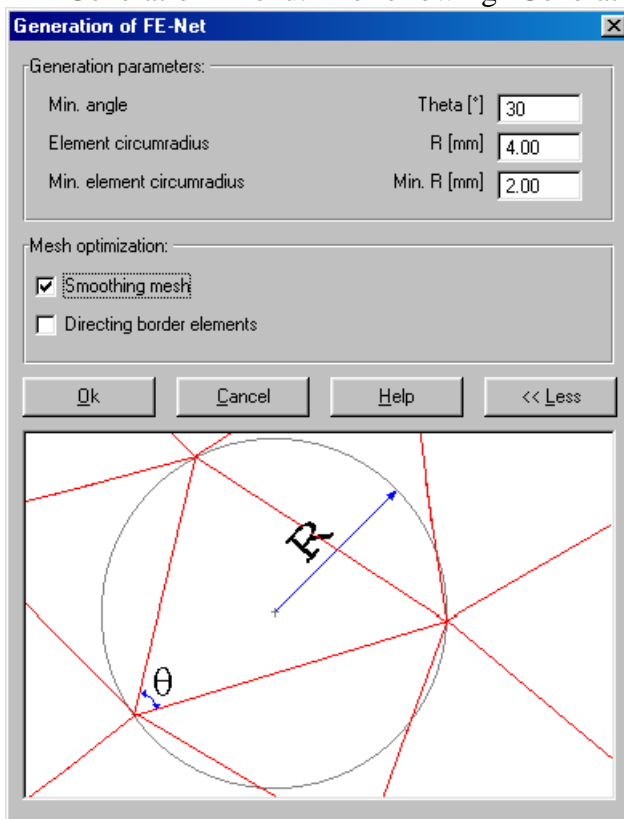

Figure 5-16   "Generation of FE-Net" dialog box

To enter the generation data

- Type 4 in the "Element circumradius" edit box to define the radius of circles containing elements
- Type 2 in the "Min. element circumradius" edit box to define the minimum radius of circles containing elements
- Check the "Smoothing mesh" option to optimize the dimension of FE-Net by making all elements having nearly the same area as possible as
- Click "OK" button

After clicking the "OK" button, FE-Net generation progress menu in Figure 5-17 appears in which various phases of the generation are progressively reported. After finishing the generation of the FE-Net and creating the wrench FE-Net as shown in Figure 5-18, do the following two steps:

- Choose "Save FE-Net" command from "File" menu in Figure 5-18 to save the data of the FE-Net
- Choose "Close FE-Net" command from "File" menu in Figure 5-18 to close the "FE-Net" embedded program and return to the main window of *ELPLA-Data*

Note that the sign "√" is typed automatically beside the "FE-Net data" command in the "Data" menu of *ELPLA-Data*.
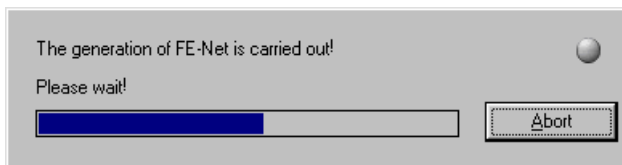

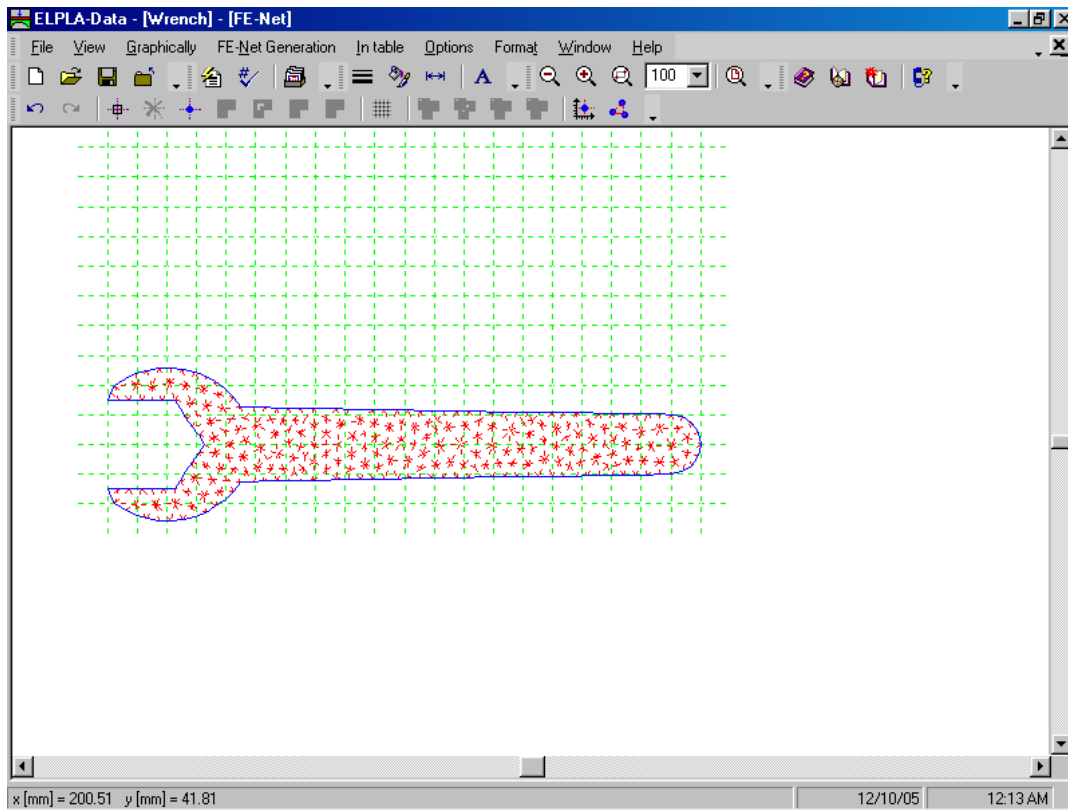Figure 5-17   "FE-Net generation" progress menu

Figure 5-18   Final FE-Net of the wrench

2.4   Supports/ Boundary conditions

To define supports, choose "Supports/ Boundary conditions" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 5-19 appears.
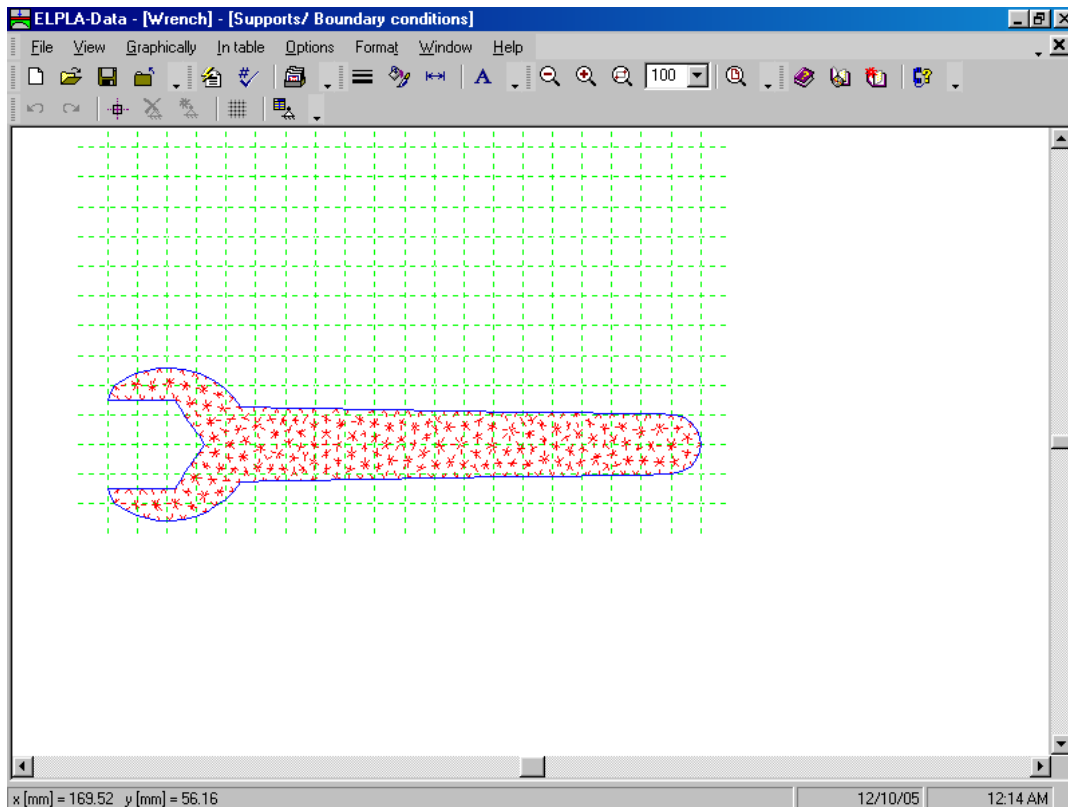
Figure 5-19   "Supports/ Boundary conditions" embedded program

Defining supports on the net

Defining supports or boundary conditions on the net may be carried out either graphically or numerically (in a table). In the current example the user will learn how to define supports on the net graphically. Considering that there are no horizontal or vertical displacements along the lines where the wrench contacts the bolt, all constrains will be applied only to the upper and bottom jaws of the wrench.

To define supports on the net

- Choose the "Zoom window" command from the "Window" menu to zoom the wrench head
- Choose "Select nodes" command from the "Graphically" menu in Figure 5-20. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on nods that have supports as shown in Figure 5-20
- After selecting nodes of supports, choose "Add boundaries" command from "Graphically" menu in Figure 5-20. "Supports/ Boundary conditions" dialog box in Figure 5-21 appears
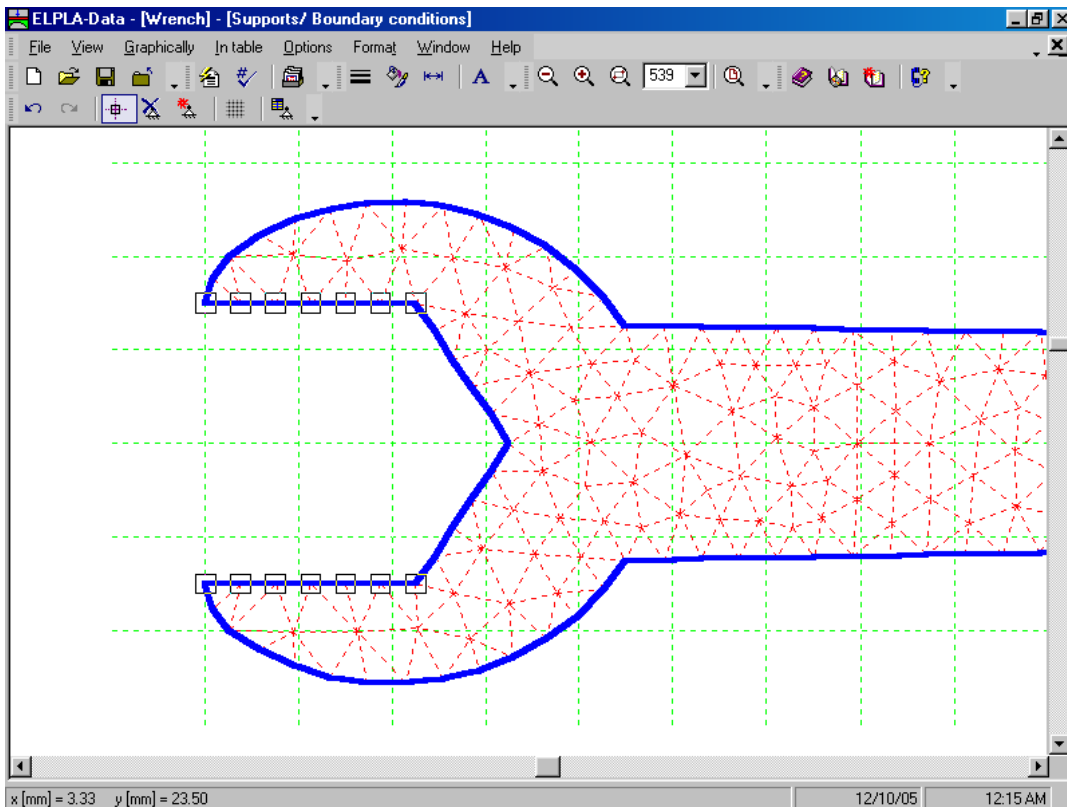
Figure 5-20   Selection of nodes that have supports

In this dialog box

- Type 0 in the "Displacement u" edit box to define the horizontal supports
- Type 0 in the "Displacement w" edit box to define the vertical supports
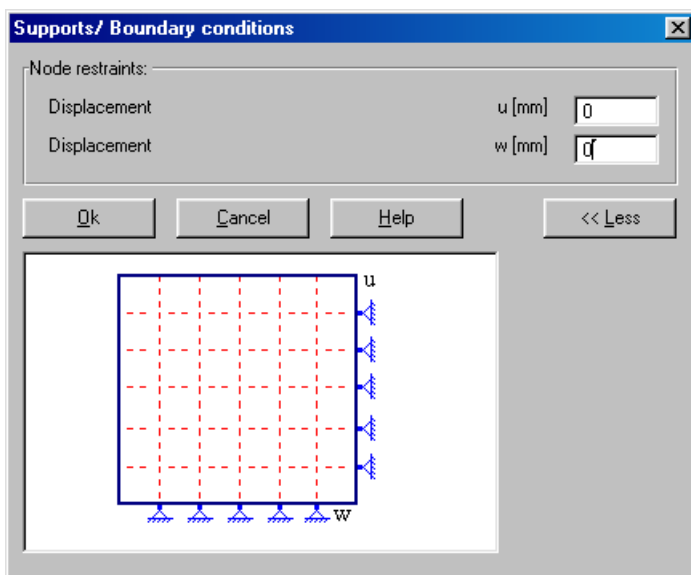- Click "OK" button

Figure 5-21   "Supports/ Boundary conditions" dialog box

After you have completed the definition of supports, the screen should look like the following Figure 5-22.
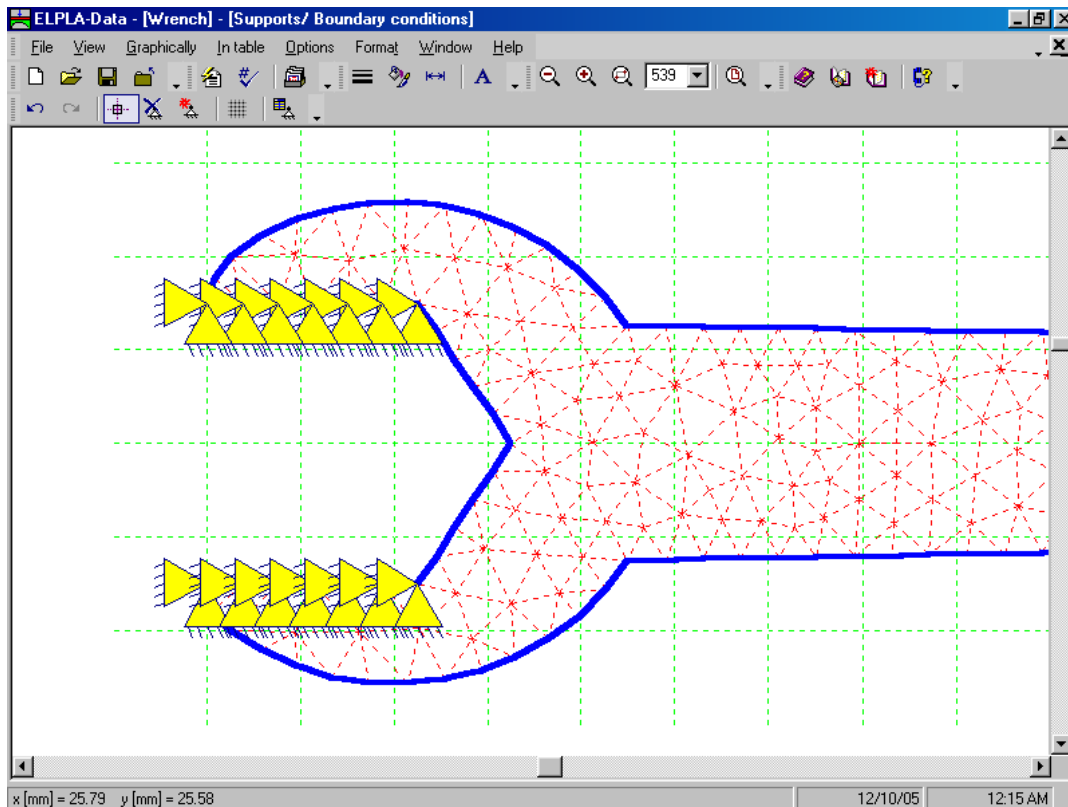
Figure 5-22   Supports on the screen

After entering supports, do the following two steps:

-   Choose "Save supports/ Boundary conditions" command from "File" menu in Figure 5-22 to save the data of supports
-   Choose "Close supports/ Boundary conditions" command from "File" menu in Figure 5-22 to close the "Supports/ Boundary conditions" embedded program and to return to the main window of *ELPLA-Data*

2.5   Slab properties

To define the material properties, choose "Slab properties" command from "Data" menu. The following embedded program in Figure 5-23 appears with default slab properties. The data of material properties for the current example, which are required to be defined, are wrench material and wrench thickness.

To enter the wrench material and thickness, choose "Element groups" command from "In Table" menu. The following list box in Figure 5-24 appears. In this list box, enter the E-Modulus and *Poisson's* ratio of the wrench material and wrench thickness. Then, click "OK" button to go to the next step.
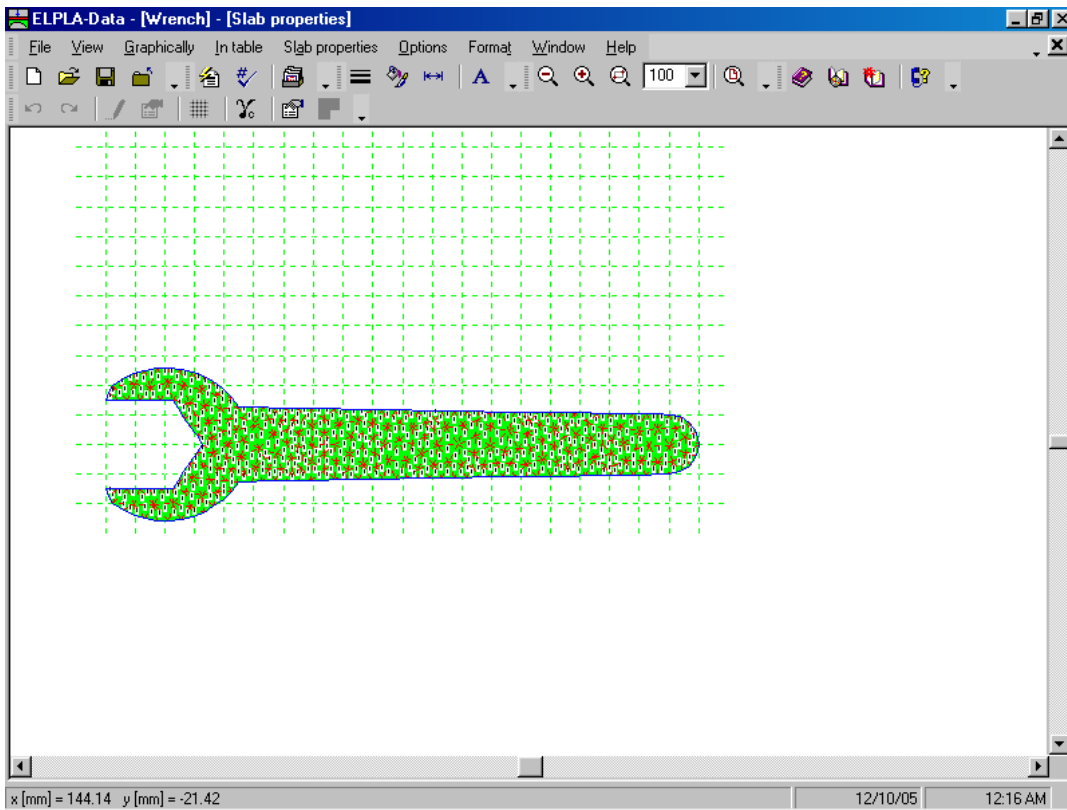
Figure 5-23   "Slab properties" embedded program
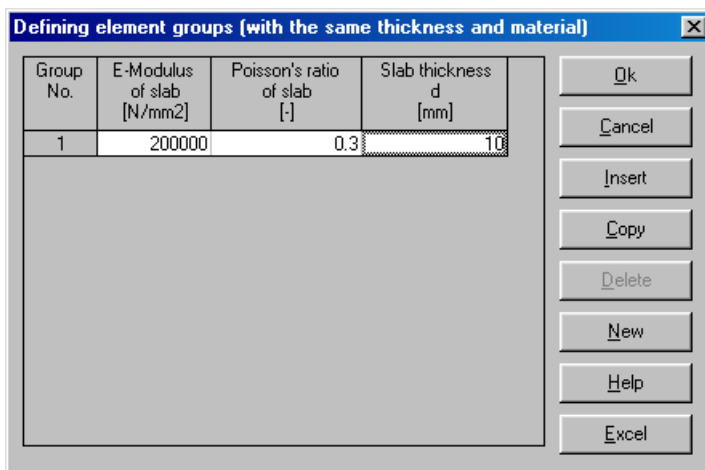

Figure 5-24   "Defining element groups" list box

To enter the unit weight of the wrench material, choose "Unit weight of slab material" command from "Slab properties" menu in the window of Figure 5-23. The following dialog box in Figure 5-25 with a default unit weight of 25 [kN/m^3] appears. To neglect the self-weight of the wrench in the analysis, type 0 in the "Unit weight of slab material" edit box then click "OK" button.
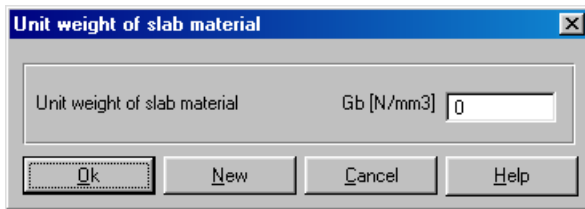
Figure 5-25   "Unit weight of slab material" dialog box

After entering the wrench material properties, do the following two steps:

- Choose "Save Slab properties" command from "File" menu in Figure 5-23 to save the foundation properties
- Choose "Close Slab properties" command from "File" menu in Figure 5-23 to close the "Slab properties" embedded program and return to the main window of *ELPLA-Data*

Note that the sign "√" is typed automatically beside "Slab properties" command in "Data" menu of *ELPLA-Data*.

2.6   Loads

To define the load applied to the wrench, choose "Loads" command from "Data" menu of *ELPLA-Data*. The following embedded program in Figure 5-26 appears.
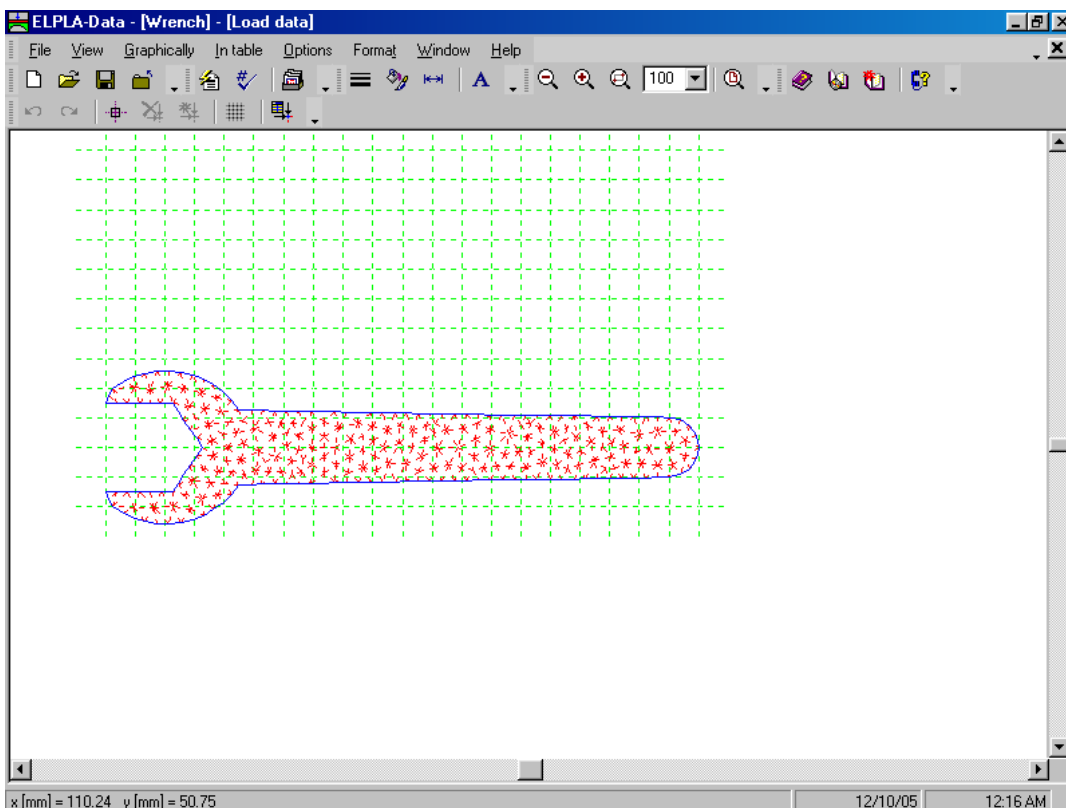

Figure 5-26   "Loads" embedded program

In plane stress problems, loads can be applied to nodes only. In this example the line load applied to the wrench will be converted to vertical concentrated loads applied to the wrench FE-Net nodes.

To enter the vertical concentrated load

- Choose the "Zoom window" command from the "Window" menu to zoom the wrench end as shown in Figure 5-27
- Choose "Select nodes" command from the "Graphically" menu in Figure 5-26. When "Select nodes" command is chosen, the cursor will change from an arrow to a cross hair
- Click the left mouse button on nodes that would have loads. Since the total force is 200 [N] over a length of 100 [mm] and the nodes are generated at about [4] mm spacing, choose 25 nodes from the wrench end as shown in Figure 5-27
- After selecting nodes of loads, choose "Add nodal loads" command from "Graphically" menu in Figure 5-27. "Nodal loads" dialog box in Figure 5-28 appears

In this dialog box

- Type 8 in the "Load *Pv*" edit box to define the vertical nodal loads
- Click "OK" button

After you have completed the definition of the concentrated loads, the screen should look like Figure 5-29.
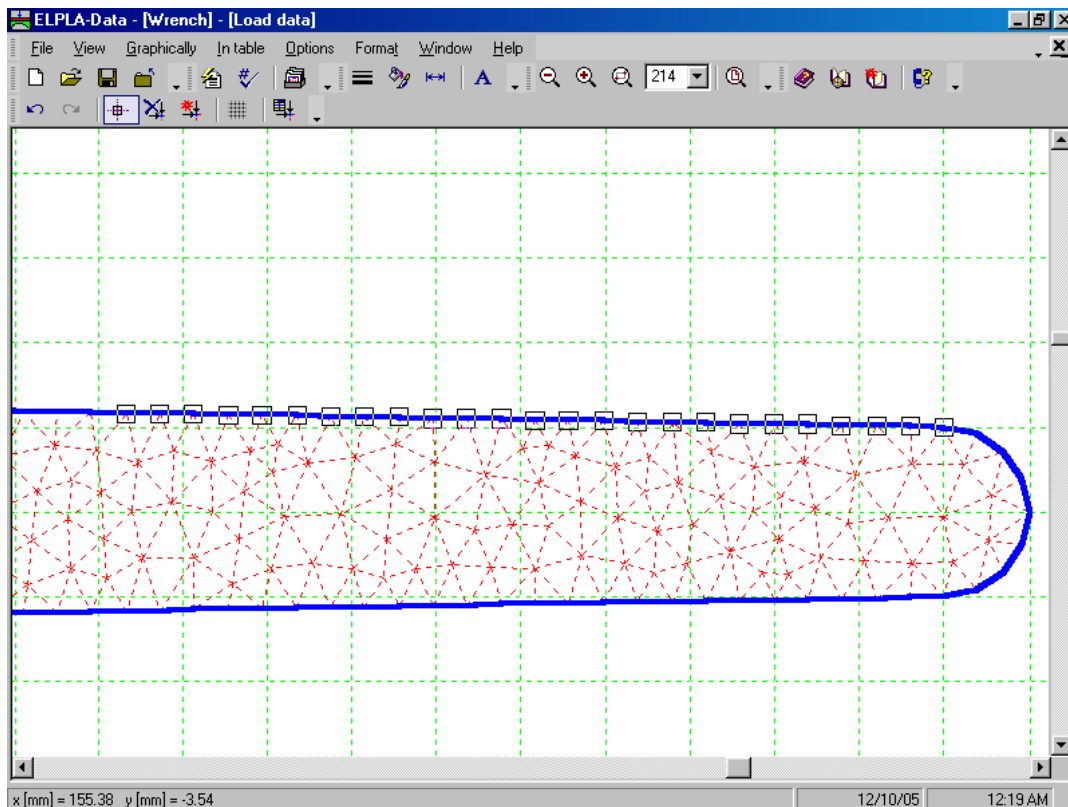

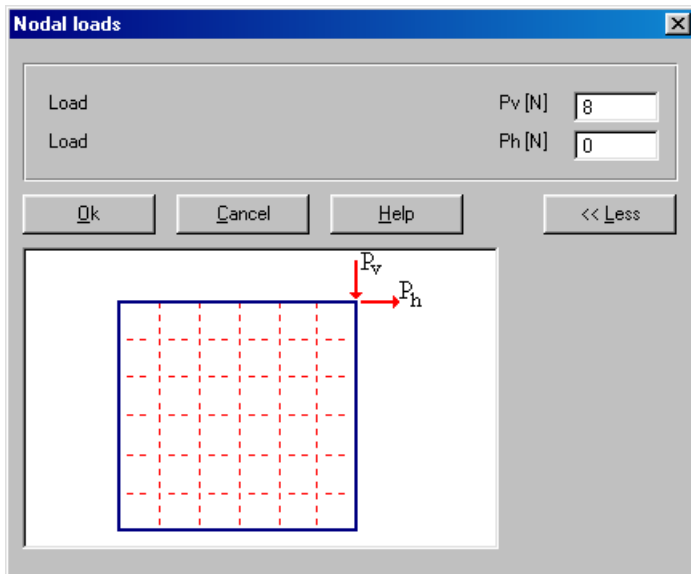Figure 5-27   Select load nodes
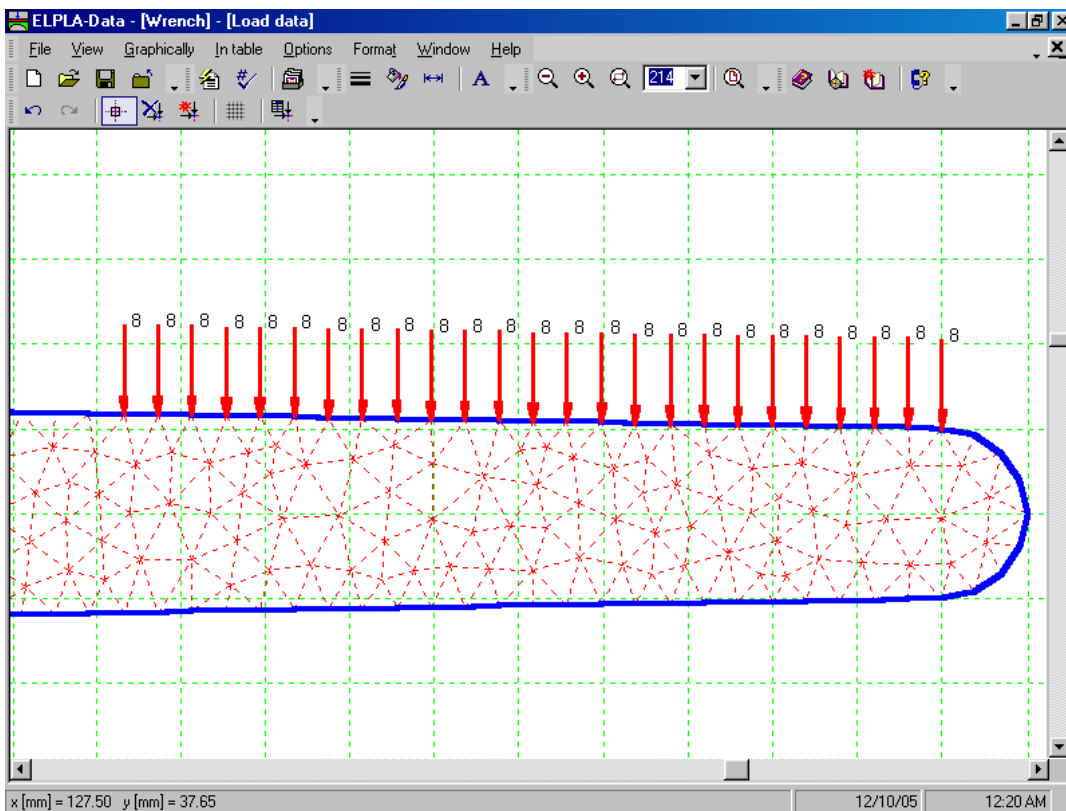
Figure 5-28   "Nodal loads" dialog box

Figure 5-29   Final loads

After finishing the definition of load data, do the following two steps:

-   Choose "Save loads" command from "File" menu in Figure 5-29 to save the load data
-   Choose "Close loads" command from "File" menu in Figure 5-29 to close the "Loads" embedded program and return to the main window of *ELPLA-Data*

Creating the project of the wrench is now complete. It is time to analyze this project. In the next section you will learn how to use *ELPLA* for analyzing projects.

3   Carrying out the calculations

3.1   Starting *ELPLA-Solver*

To analyze the problem, switch to *ELPLA-Solver*. This is done by clicking on "Solver" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Data*. Then, *ELPLA-Solver* window in Figure 5-30 appears.
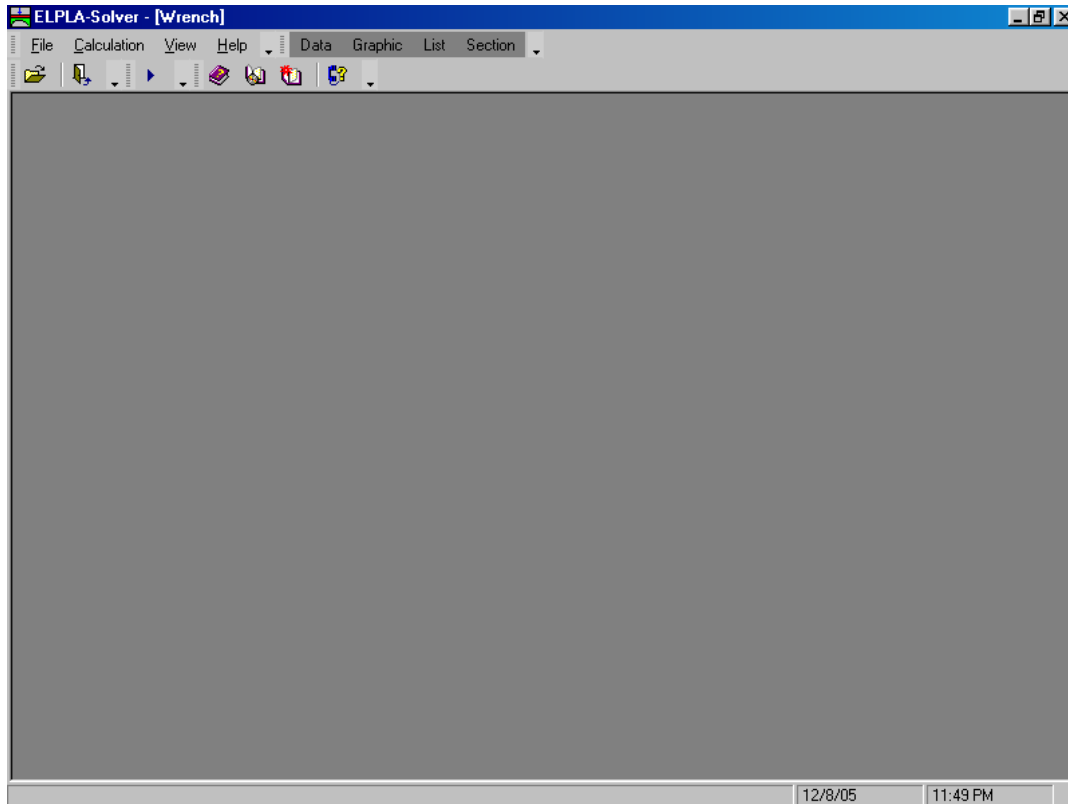

Figure 5-30   Opening screen of the sub program *ELPLA-Solver*

ELPLA-Solver automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Solver* window. Also, *ELPLA* will active the "Calculation" menu. This menu contains commands of all calculations. Commands of the calculation depend on the analysis type. For the current example, the items, which are required to be calculated, are:

- Preparing the calculation
- Analysis of plane stress

These calculation items can be carried out individually or in one time.

3.2   Carrying out all computations

To carry out all computations in one time, choose "Computation of all" command from "Calculation" menu in *ELPLA-Solver* window. The progress of all computations according to the defined analysis will be carried out automatically with displaying information through menus and messages.

Analysis progress

Analysis progress menu in Figure 5-31 appears in which various phases of the calculation are progressively reported as the program analyzes the problem. Also, a status bar on the screen down of the *ELPLA-Solver* window displays information about the progress of calculation.
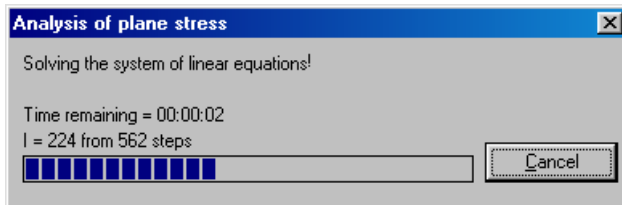

Figure 5-31   Analysis progress menu

4   Viewing data and result graphics

To view the data and results of a problem that has already been defined and analyzed graphically, switch to *ELPLA-Graphic*. This is done by clicking on "Graphic" in the menu bar of the sub programs at the upper-right corner of *ELPLA-Solver* window. *ELPLA-Graphic* window in Figure 5-32 appears. *ELPLA-Graphic* automatically opens the data file of the current example and displays the data file name in the title bar of *ELPLA-Graphic* window.
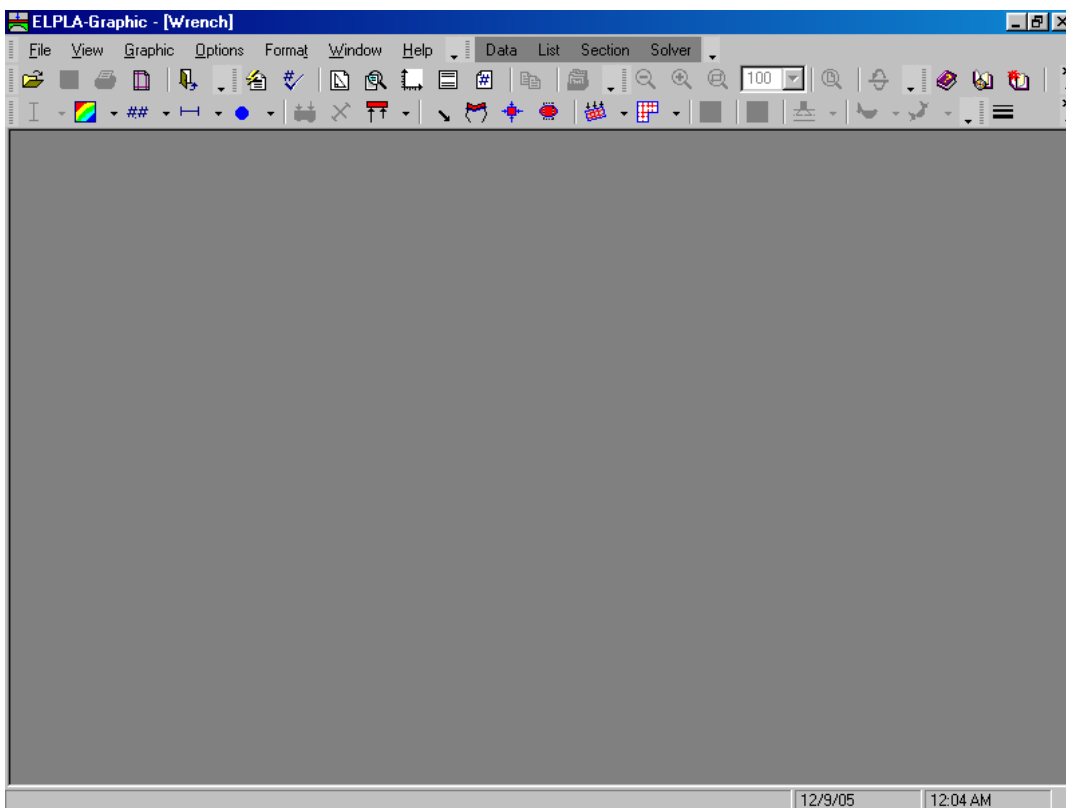

Figure 5-32   Opening screen of the sub program *ELPLA-Graphic*

ELPLA can show solid elements in three dimension view. To view elements with real dimension in 3D-view, choose "Plot parameters" command from "Options" menu of *ELPLA-Graphic*. The following "Plot parameters" dialog box in Figure 5-33 appears. In the "Solid elements" tab, check the following check boxes: "draw solid elements", "draw element border" and "color solid element", then click "OK".
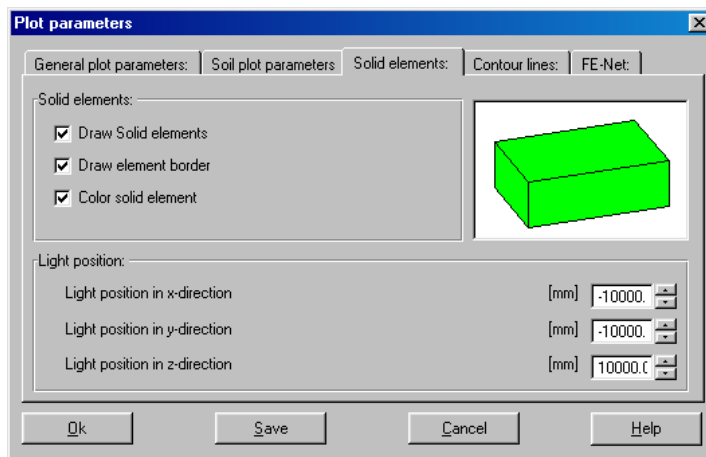

Figure 5-33   "Plot parameters" option box

To view the wrench in 3D-view, choose "Data in isometric view" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 5-34 appears. In this option box, select "Slab thickness" option, then click "OK" button. The wrench is now displayed as solid elements in 3D-view as shown in Figure 5-35.
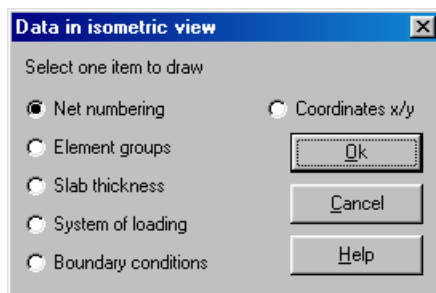

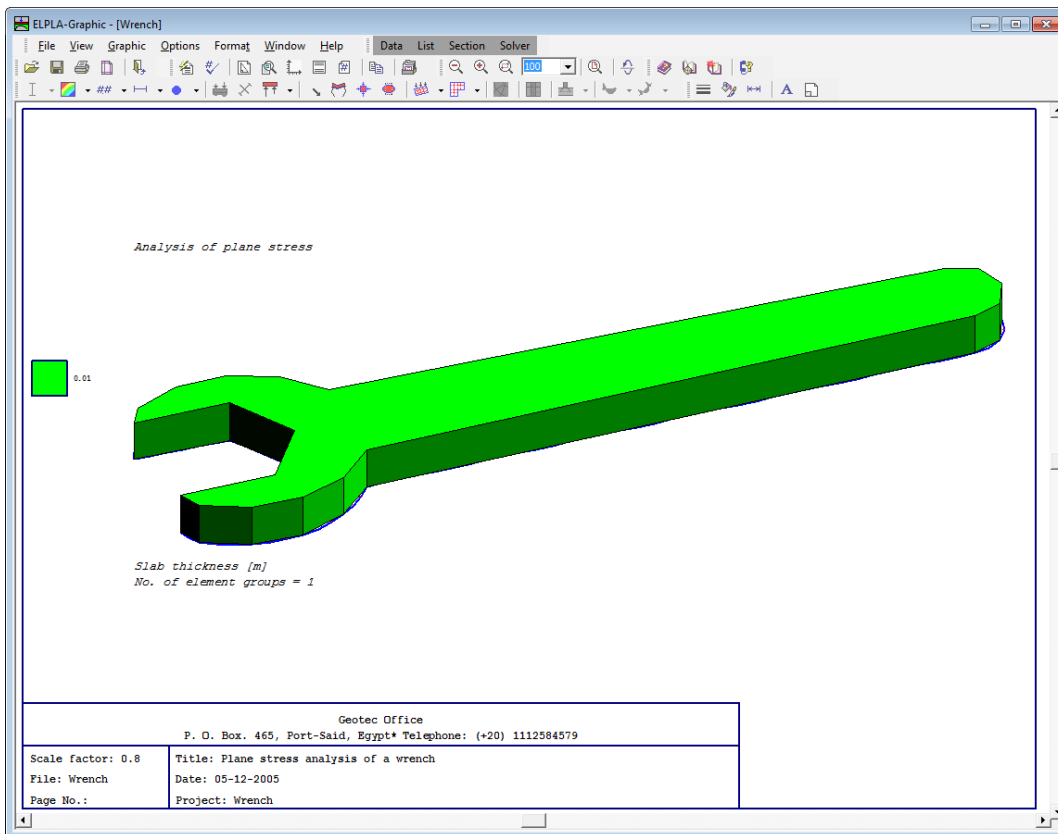Figure 5-34   "Data in isometric view" option box

Figure 5-35   Wrench in isometric view

To view the results as contour lines

- Choose "Results as contour lines" command from "Graphic" menu of *ELPLA-Graphic*. The following option box in Figure 5-36 appears
- In this option box, select "Maximum shear stresses max_Sigma" as a sample for the results to be displayed
- Click "OK" button

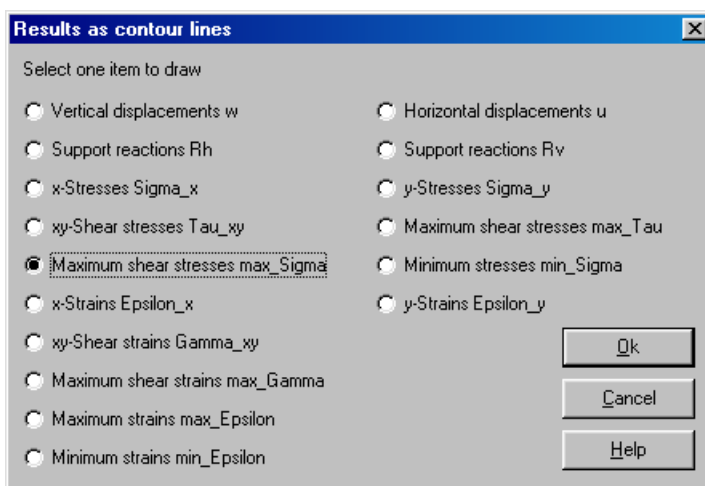The maximum shear stresses are now displayed as contour lines as shown in Figure 5-37.

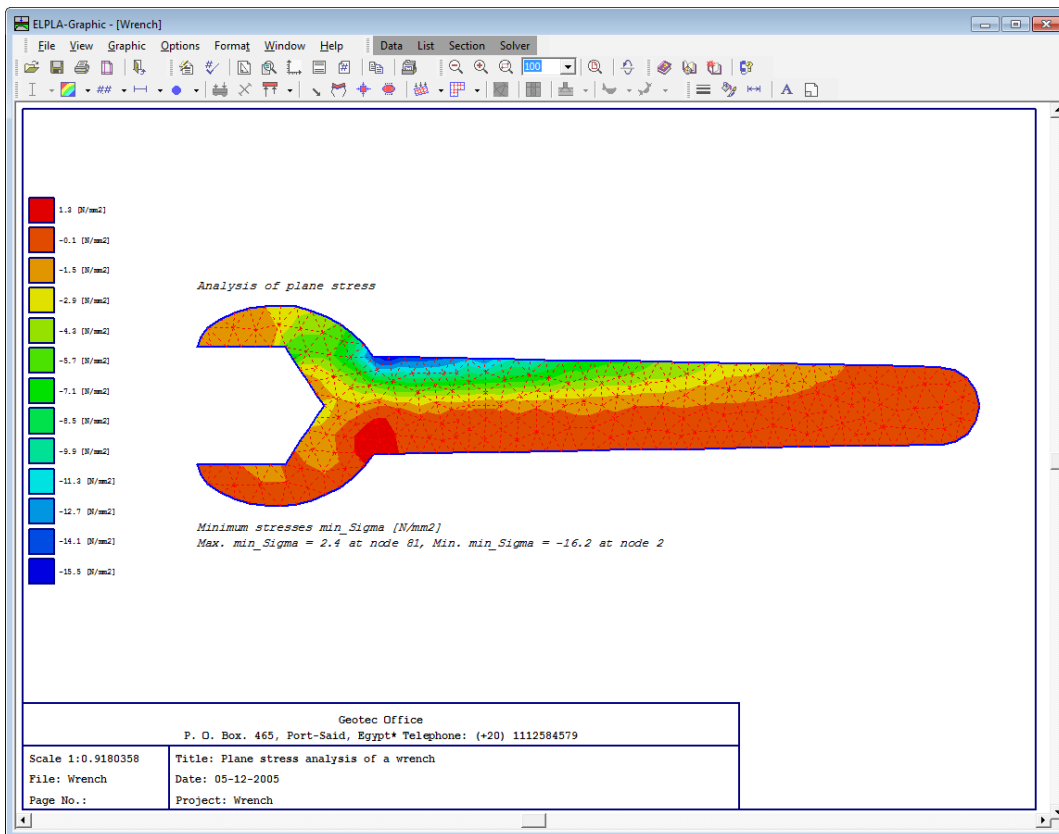Figure 5-36   "Results as contour lines" option box

Figure 5-37   Maximum shear stresses max_Sigma

5   Index

A

B

C

F

G

L

N

P

S

U