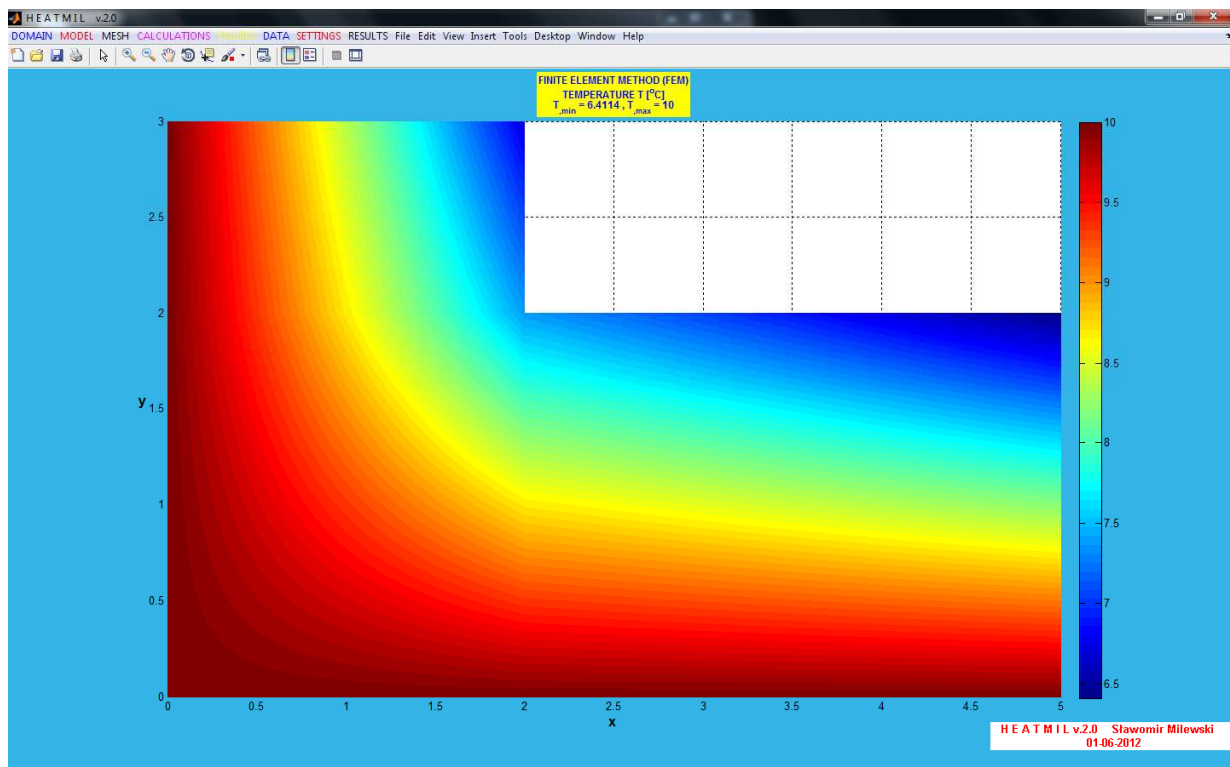# Analysis of the stationary heat flow problem

by means of the FEM / MFDM system

HEAT MIL

Created by:
Sławomir Milewski (slawek@L5.pk.edu.pl)
Electronic version:
http://www.L5.pk.edu.pl/~slawek

Cracow, March

1. Problem formulation

Model problem, with domain shown in Fig.1 together with the coarse FEM discretization (3 rectangular finite elements), is analyzed. Scalar temperature as well as vector heat flux fields are sought, assuming the stationary (time independent) heat flow through the given domain. Temperature along the edges 1-7 and 1-3 is 10 °C, whereas the heat flux is prescribed along the edges 5-8 and 5-6 with given value $-12 \, J / m^2 s$.
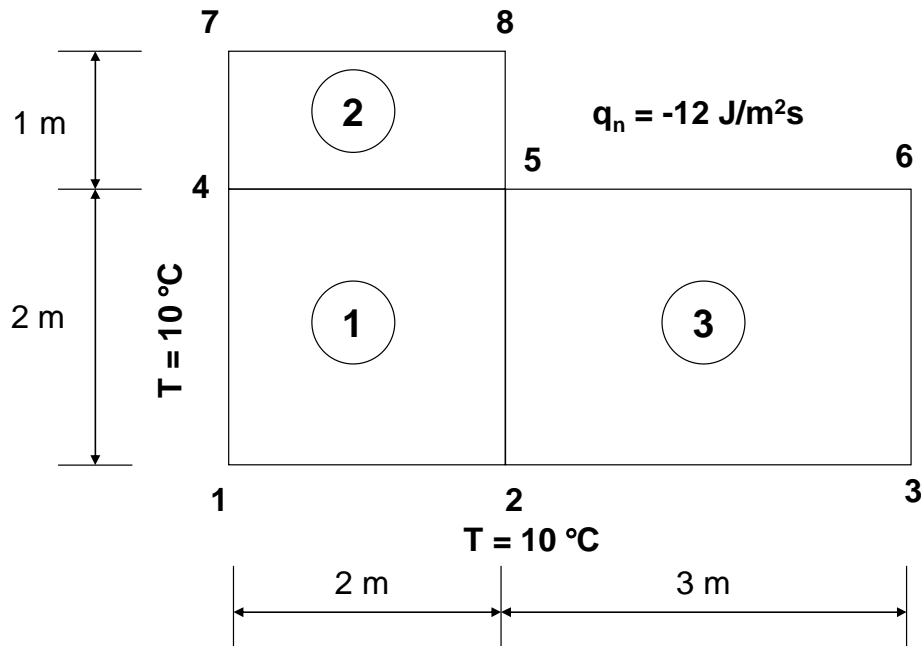
Fig. 1: Geometry, coarse FEM discretization and the heat state of the problem

Data:

material:	orthotropic, heat conductivity constants:
	$k_x = 4 \, J / \,^\circ Cms$ and $k_y = 7 \, J / \,^\circ Cms$
temperature:	edges 1-7 and 1-3, value $T = 10 \,^\circ C$
heat flux:	edges 5-8 and 5-6, value $q_n = -12 \, J / m^2 s$.
internal heat source:	none (0 $J / m^2 s$)

2. Start of the program and its components

„HEAT MIL" software, designed mainly for the stationary heat flow analysis by means of the FEM, may be found on the following webpage:

http://www.L5.pk.edu.pl/~slawek

in section "HEATMIL 2012". It contains the set of archived files (*heatmil.zip*), which has to be saved on the disc (right click on the blue link, followed by the option "Save target object as…"), and extracted to the specified localization. Subsequently, file named *heatmil* has to be run through the MATLAB Command Window.
Running of the Matlab program (recent version: Matlab2008) is possible by the shortcut placed on the desktop or by the reference localized in the fold "All programs" of the Windows operating system.
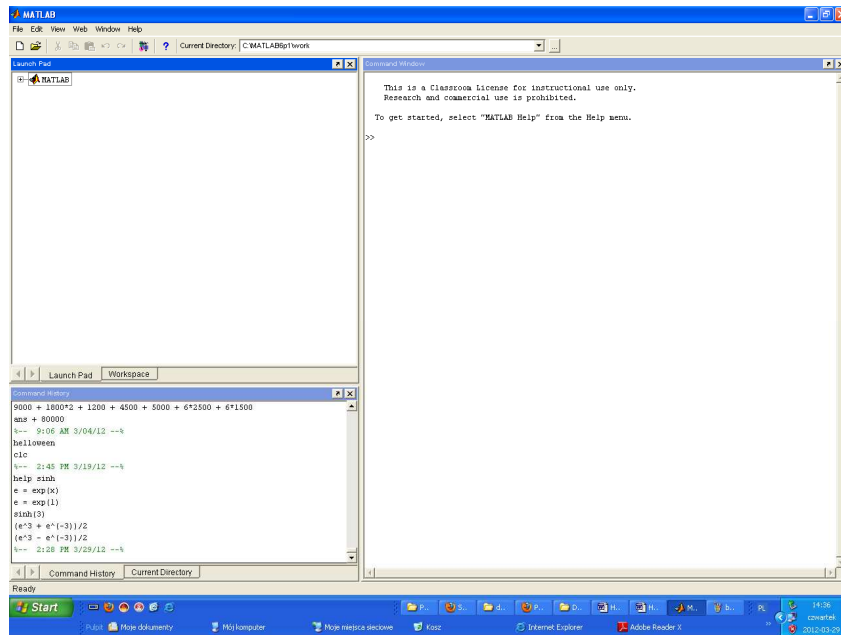
Fig. 2: Screen view after the start of the Matlab

Once the Matlab is started, one has to prescribe the "Current Directory" option from the main upper menu as the folder which contains extracted files of the program HEATMIL.

This selection may be done by means of the file browser icon , placed on the left of the directory name, on the main menu. After selection, enter the program name *heatmil* to the Command Line (near the flashing >> sign) in the Command Window of Matlab and press *Enter*:
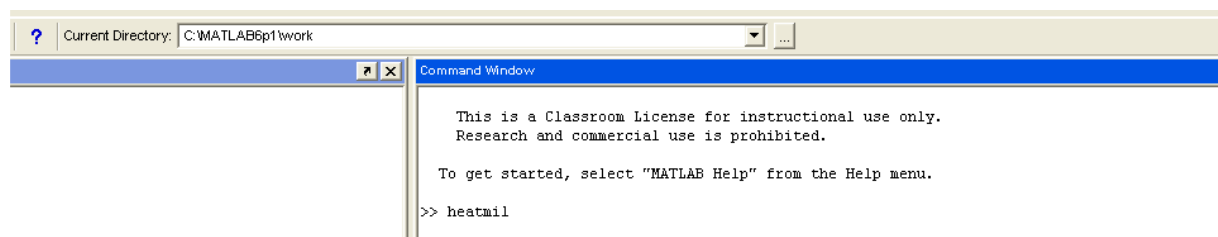
Program "HEATMIL.m" does not require ANY modifications in its code by the user – the only thing to do is to run this program in the Matlab environemnt.
Remark! In case of the accidental deletion or modification of the part of the code, one has to donwload it again and load by the Matlab Command Window.
After start of the program, one has to select the language (polish, english) of the program interface. In the present work, presented is the operation of the program in the enlish language version. After such selection, the main program menu will be visible (Fig.3) with several options (written using capital letters), corresponding to the various stages of building a discrete thermal model of the body, FEM/MFDM calculations as well as results visualization:

-           DOMAIN
-           MODEL
-           MESH
-           CALCULATIONS
-           FIGURES
-           DATA
-           SETTINGS
-           RESULTS

Remaining options placed in the upper menu (File, Edit, View, Insert, Tools, Desktop, Window, Help) are the built-in options of all figures created by Matlab.
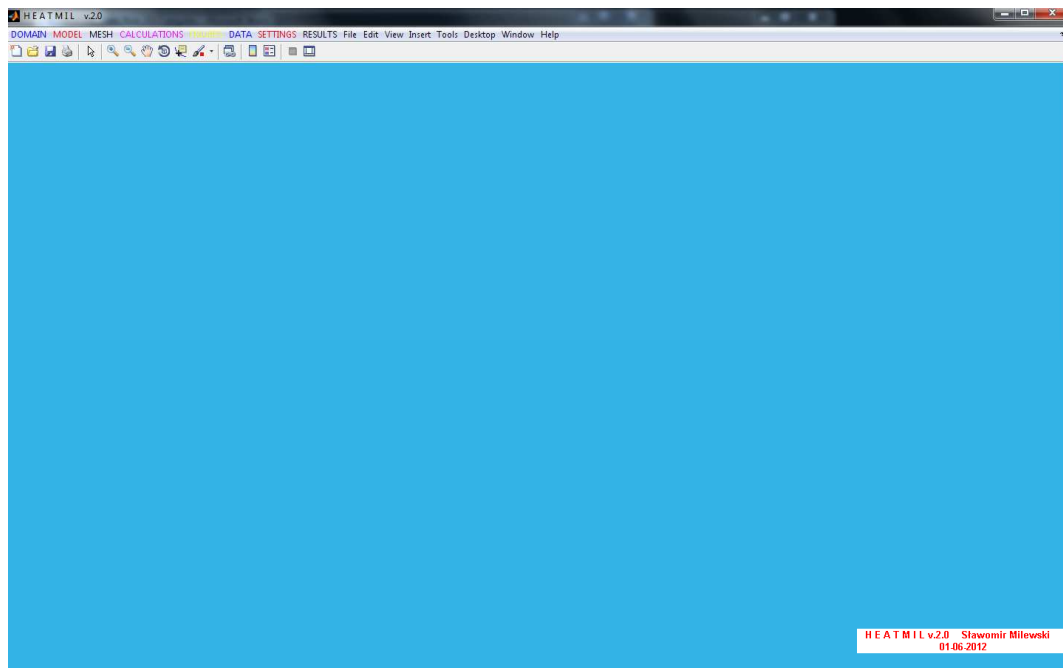

Fig. 3: View of the program "HEAT MIL" interface

The first task will be solved for a coarse mesh consisting of several (2,3,4) rectangular elements (bilinear interpolation of the temperature field), which may be confronted with the results of calculations conducted "by hand" (for instance, using Mathcad).

3. Geometry generation

Initially, all construction points of the boundary region will be defined - all of which result from the division of the domain into three finite elements (Fig. 1). They will serve both for the edge construction as well as for the manual generation of coarse FEM mesh (see Section 8).

Please select the option from the upper menu "DOMAIN -> EDGES POINTS". You will get the edit box for entering points. Please define the x and y coordinates of the first point and enter them into the white edit box next to the fields, "X (P1) =" and "Y (P1)". In the current task they will be the numbers 0 and 0 (see Fig.1 for details). After entering the coordinates, please press the "ADD NEW POINT." In the list on the left side ("ACTIVE EDGE POINTS"), provided data will be visible. After entering the coordinates of the first point, you may also check its position on the figure on the right hand side. In the same manner, please define the remaining seven points with given coordinates in accordance to the Fig.1.

Remark! If you made a mistake, you can use an option in the current edit box for point selection (check box). If you select a point on the list, press the button "DELETE SELECTED DATA". If you want to correct coordinates of one or more points, after entering the new coordinate values in the white boxes on the "ACTIVE EDGE POINTS", press the "UPDATE DATA" button.

When all eight points are constructed, figure and the list of active points should look as it is presented in Fig.5.
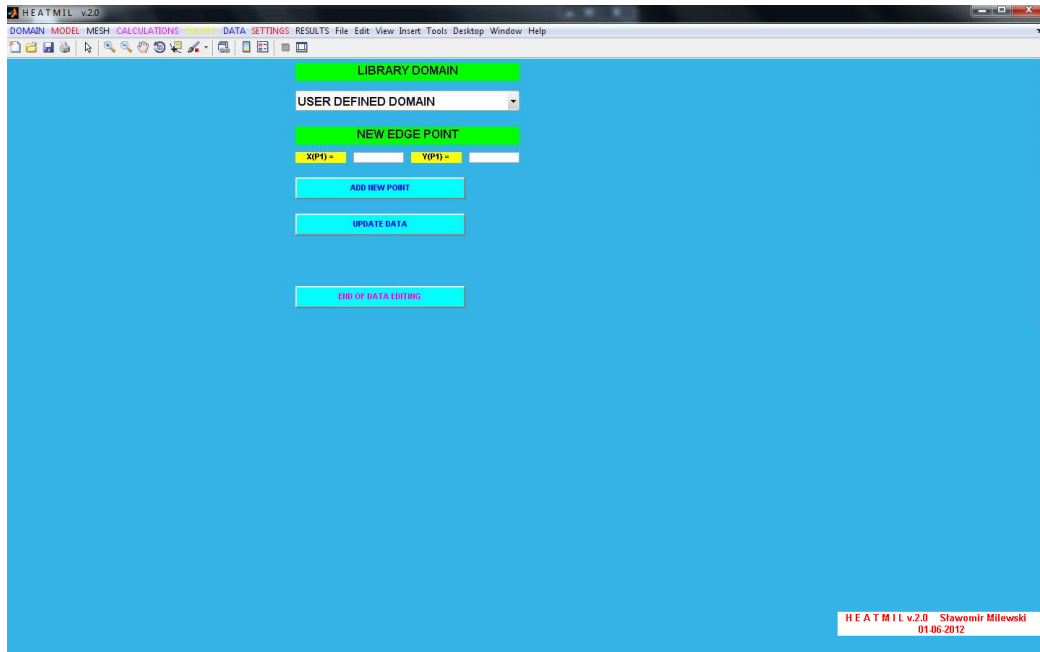

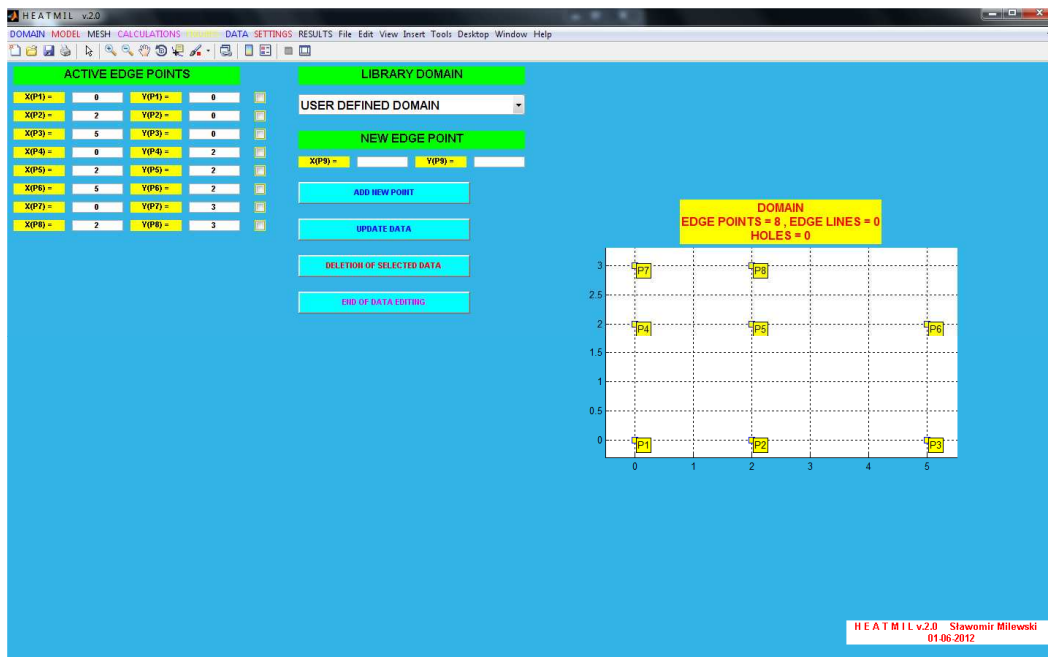**Fig. 4: Edit box for generation of the construction points of the domain boundary**

Fig. 5: Set of all construction points of the domain boundary

Generation of the edge lines is our next task. Select from the upper menu: "DOMAIN -> EDGE LINES". The appropriate edit box is presented in Fig.6. The "AUTOMATIC GENERATION OF THE EDGE LINES" enables fast creation of the boundary line - understood as a closed contour area (without holes in the current version of the program)

and therefore, it is possible only in case in which the edge points, forming the successive sections of the boundary line, are numbered sequentially (1, 2 , ..., 8, 1). However, in the considered example, the boundary line consists of the points in the sequence 1,2,3,6,5,8,7,4,1. Therefore, tt is necessary to build the contour domain line by line.
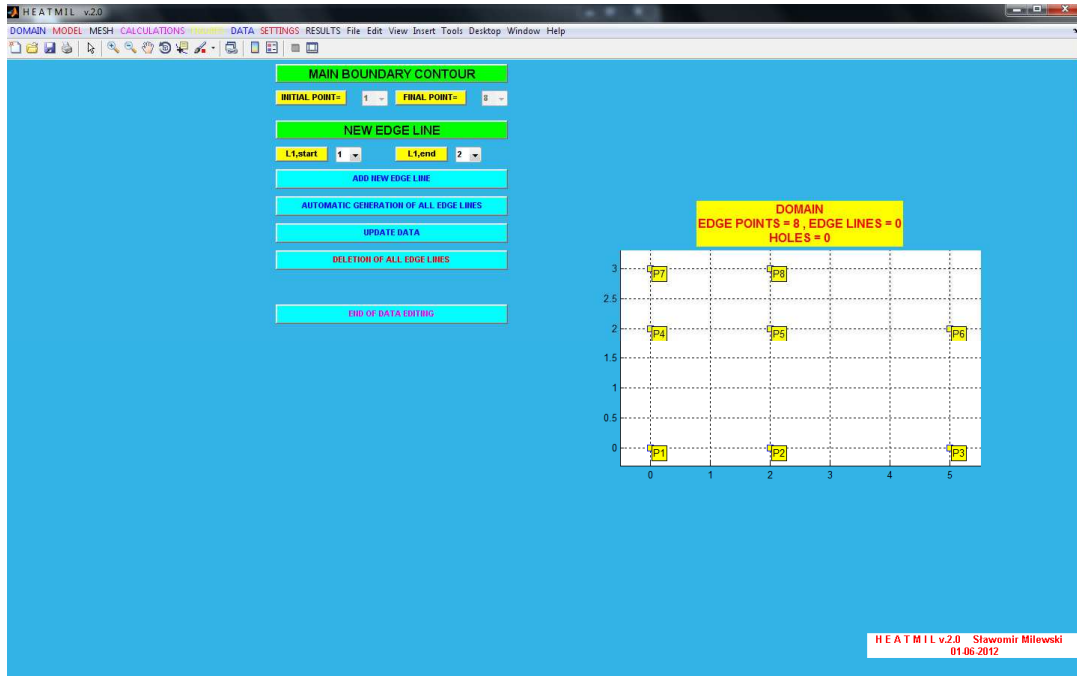

Fig. 6: Edit box for generation of the edge lines

In order to define the first line segment - for instance, between points 1 and 2, below the "NEW EDGE LINE" label, one may find two sets of points numbers. Select "1", the other "2", then press the "ADD NEW LINE'. In the list on the left side ("ACTIVE EDGE LINES") the line information appears. In the same way, enter the other lines, i.e., 2-3, 3-6, 6-5, 5-8, 8-7, 7-4 and finally 4-1.
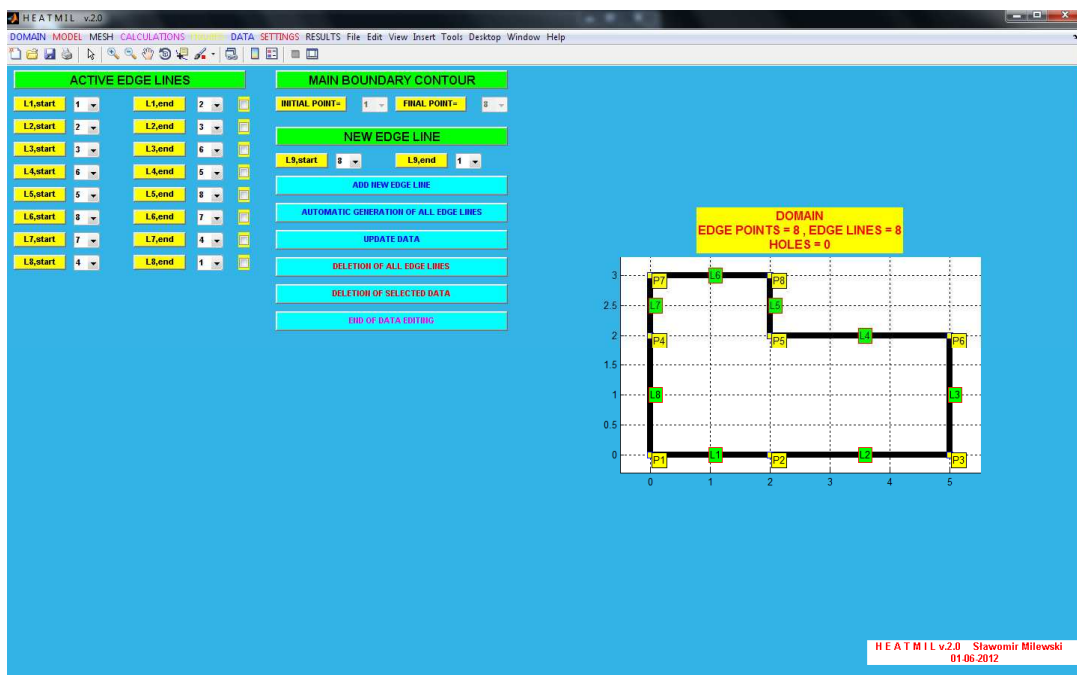

Fig. 7: Defined edge lines

Remark! If you made a mistake, you may remove the selected line (line selection + button "DELETE SELECTED DATA") or all lines (the button "DELETE ALL LINES"). In addition, the start and end numbers can be changed using the lists on the left side of the line information, after pressing the "UPDATE DATA" button.
The view of a edit box, after adding all lines is presented in Fig.7.

4. Definition of the material parameters

Please select "MODEL -> MATERIAL" option in the upper menu. You will get a table in which you have to define both thermal conductivity coefficients. After entering proper values ($k_x = 4$, $k_y = 7$), press the 'APPLY' button.

5. Definition of the heat generation intensity

In order to define a heat generation intensity function f , choose the option from the upper menu "MODEL-> LOAD" and enter the formula for f in the given table, which may depend on the spatial variables *x*, *y*. However, in the considered case, this intensity is equal to 0, which is the default state in the program. Confirm all changes by pressing 'APPLY' button.

6. Definition of the boundary conditions

In order to determine the boundary conditions of the essential (known temperature T) or a natural type (known heat flux q, given in the normal direction to the appropriate edge line), choose from the upper menu "MODEL -> BOUNDARY CONDITIONS" You will get a edit box, as in the Fig,8, with a list of all boundary lines ("BOUNDARY CONDITIONS"), and the figure of the domain in which successive lines will be marked with boundary conditions codes.
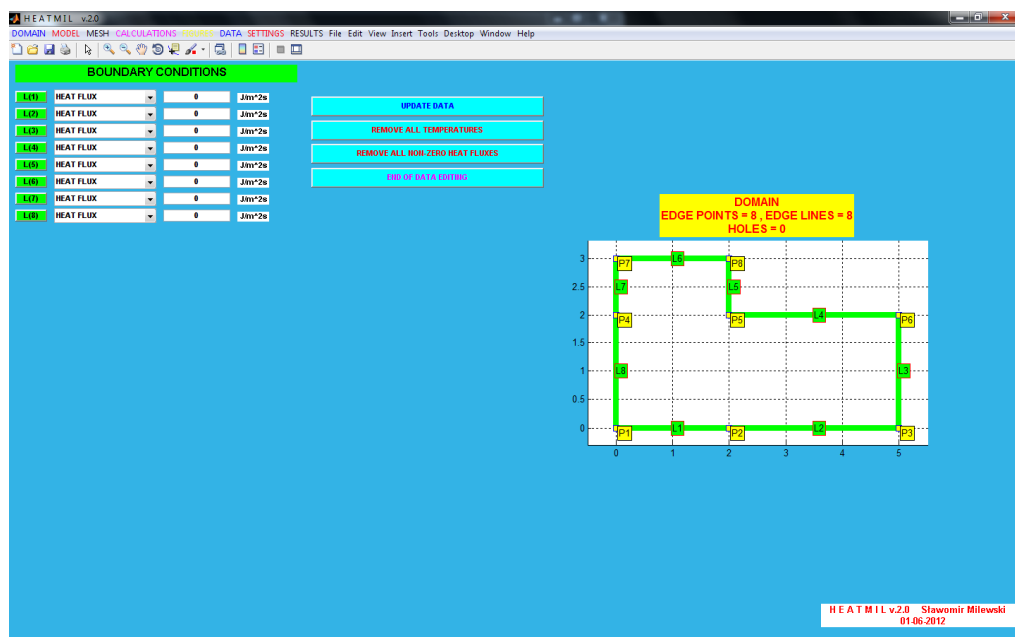

Fig. 8: Edit box for definition of the boundary conditions

By default, each generated edge line is assigned a homogeneous natural condition (heat flux with a zero value), which is marked with a green line. In order to change the

type of condition, select one of two options ("TEMPERATURE", "HEAT FLUX") on the list prescribed to each line, and then edit the white box to assign the appropriate value. Please, for the edges with numbers 1, 2, 7 and 8, select the "TEMPERATURE" field, and enter the value 10. For the lines with numbers 4 and 5, please enter the value -12 (not changing the 'HEAT FLUX' field). Once it is done, please press the "UPDATE DATA".
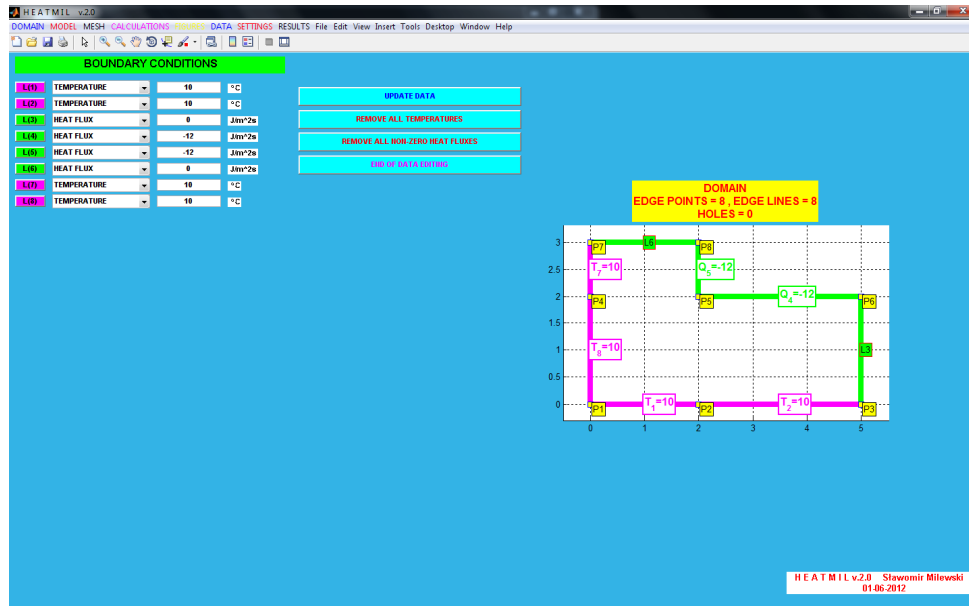

Fig. 9: The final form of the edit box for definition of the boundary conditions

After those operations, the edit box, containing the automatically refreshed figure, should look, as presented in Fig.9

7. Type of analysis and its parameters

Within the program, there is a possible choice between element analysis (FEM) and meshless analysis (MFDM). For each of these methods, one may set such parameters as the shape of elements (triangular and rectangular) and the degree of approximation (linear, parabolic). Additionally, one may choose between the stationary and non-stationary heat flow analysis. Some of these options are under current development. The default options in the program are set to:

- Stationary heat flow analysis
- FEM analysis
- The rectangular shape of elements
- Bilinear degree of interpolation (in the rectangular elements).

Change of these settings is possible (or shall be in the nearest future) by selecting from the upper menu "SETTINGS -> MODEL SETTINGS". In addition, it will be shown how to change the shape of elements into triangular ones during the mesh generation.

8. Manual generation of the coarse FEM mesh

Manual mesh generation (for several FEs) may be accomplished through the option from the upper menu "MESH -> MANUAL GENERATION". You will get the appropriate edit box (Fig. 10), and the figure, that shows generated mesh for FEM (element by element). By default, all pre-made boundary construction points are becoming nodes - there is no possibility to remove them. However, one may add new nodes on the edge or inside the domain, if necessary. In the considered task, all required

nodes are already in place (as the edge points). The implementation of additional nodes is described in section 12.
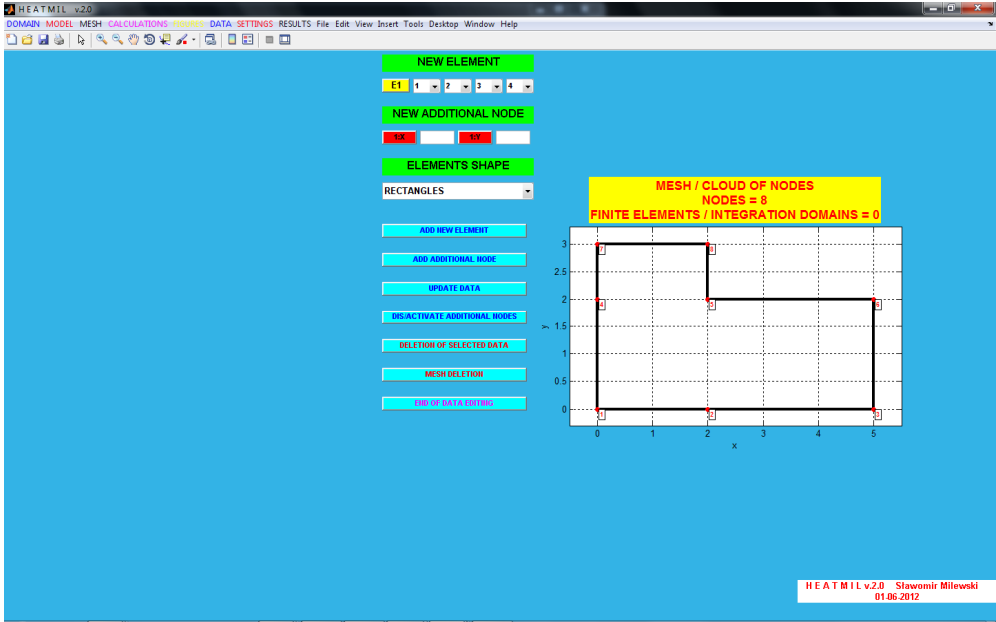

Fig. 10: The edit box for manual generation of finite element mesh

Below the label "NEW ELEMENT", there are four lists containing potential numbers of nodes. By selecting the appropriate numbers, one defines new elements. According to the Fig.1, the first element is built on nodes 1-2-5-4. Please select the relevant numbers from the list, in the provided order. The order of selection is essentially arbitrary, however the nodes should be pointed sequentially, along the sides of the rectangle (not diagonally).
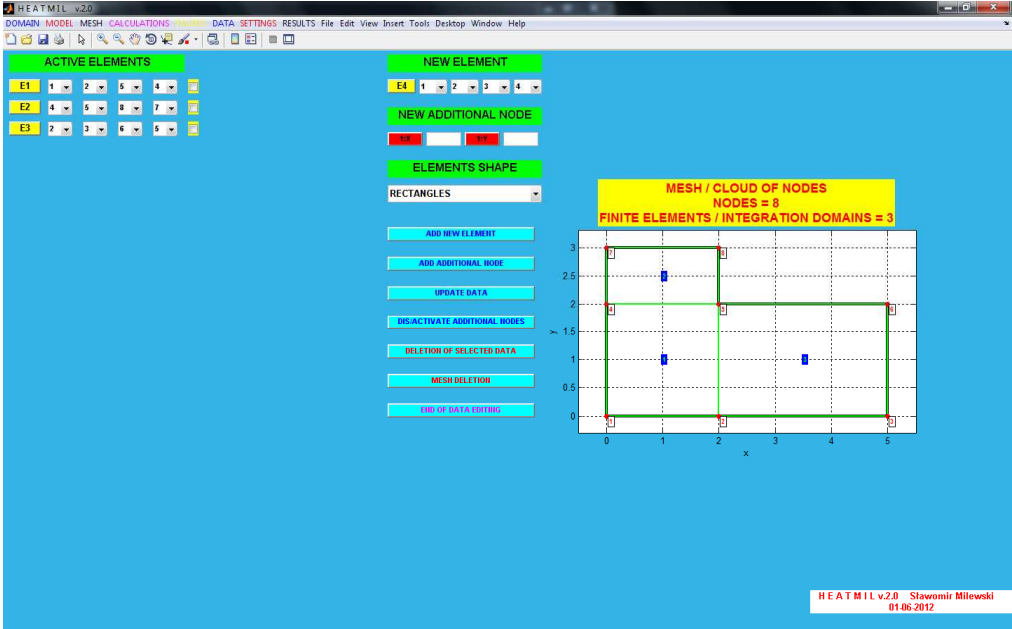

Fig. 11: The final form of the edit box for manual generation of the FE mesh

After selection the numbers from the lists, press "ADD NEW ELEMENT". Element shall appear in the list on the left side of the box ("ACTIVE ELEMENTS") as well as in the figure. In a similar manner, please specify element no. 2: 4-5-8-7 nodes as well as element no.3: the nodes 2-3-6-5. The final form of the edit box, after defining all elements, is

presented in the Fig.11.
As with points and lines, if misused, one may improve the numbers of element nodes,
delete the selected elements, or the whole mesh.

9. Calculations

Prior to the calculations, it is recommended to save the data of the model by using the
upper menu "DATA -> SAVE MODEL DATA TO THE FILE". Enter the file name
without any extension. The data is saved in a text file, it may be found within the current
directory ("Current Directory") of Matlab, the localication, from which the program was
loaded.
In order to perform the calculations we shall choose the "CALCULATIONS -> RUN"
from the main menu. If the model produces no error messages, calculations will be
performed and appropriate notification will appear.

10. Results overview

Viewing the results in graphical form is possible by selecting the appropriate option
from the upper menu "FIGURES". Let us start with the graph of the temperature
distribution. It will appear after selecting "FIGURES -> NUMERICAL SOLUTION
(TEMPERATURE)". In graph label, displayed are the minimum and maximum
temperatures of all nodes (Fig.12).
The graphs of heat flux components $q_{n,x}, q_{n,y}$, as well as the graph of the average value

$q_n = \sqrt{q_{n,x}^2 + q_{n,y}^2}$ may be produced by selecting one of three options from "FIGURES ->
DERIVATIVES OF NUMERICAL SOLUTION (HEAT FLUX)" Component values are
calculated by post-processing in elements (analytical differentation of the shape functions)
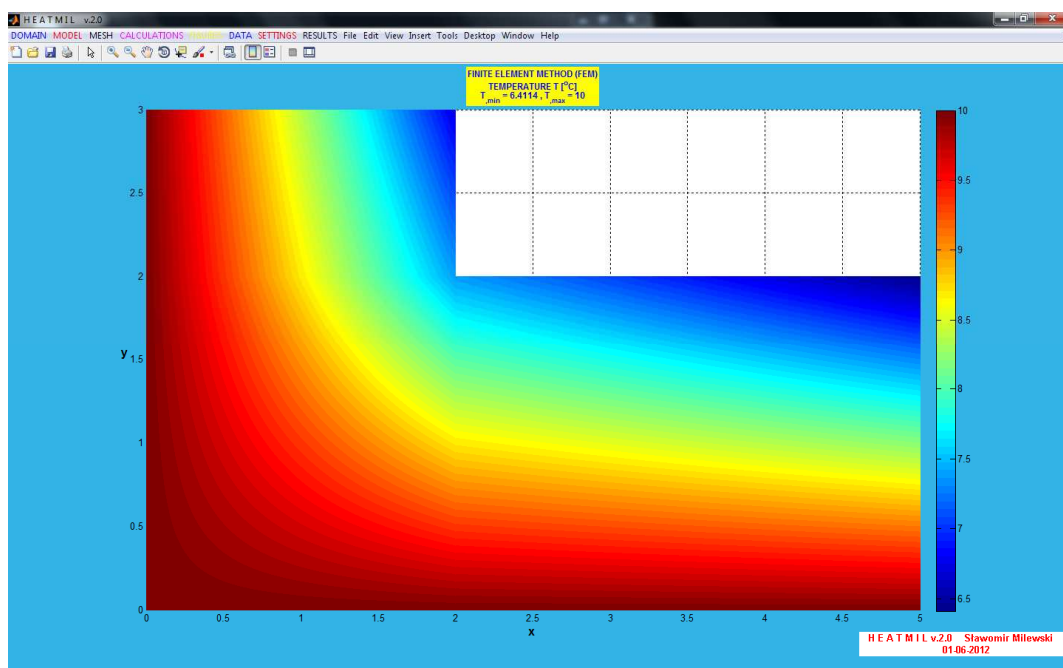using nodal solution only.

Fig. 12: Temperature distribution

10

The temperature distribution is prescribed here using continuous function (bilinear interpolation of nodal values), whereas the distribution of its derivatives is no longer a continuous one.

Graphs are presented in the plane (x, y), however the result may be viewed in 3D by selecting "Rotate 3D" (icon 🔄) located in the menu, for each of the images separately, and then by controlling the view by left or right mouse button.

All figures (this time by default, in the 3D view) may be called by selecting "FIGURES -> ALL GRAPHS OF THE NUMERICAL SOLUTION" – Fig.13.
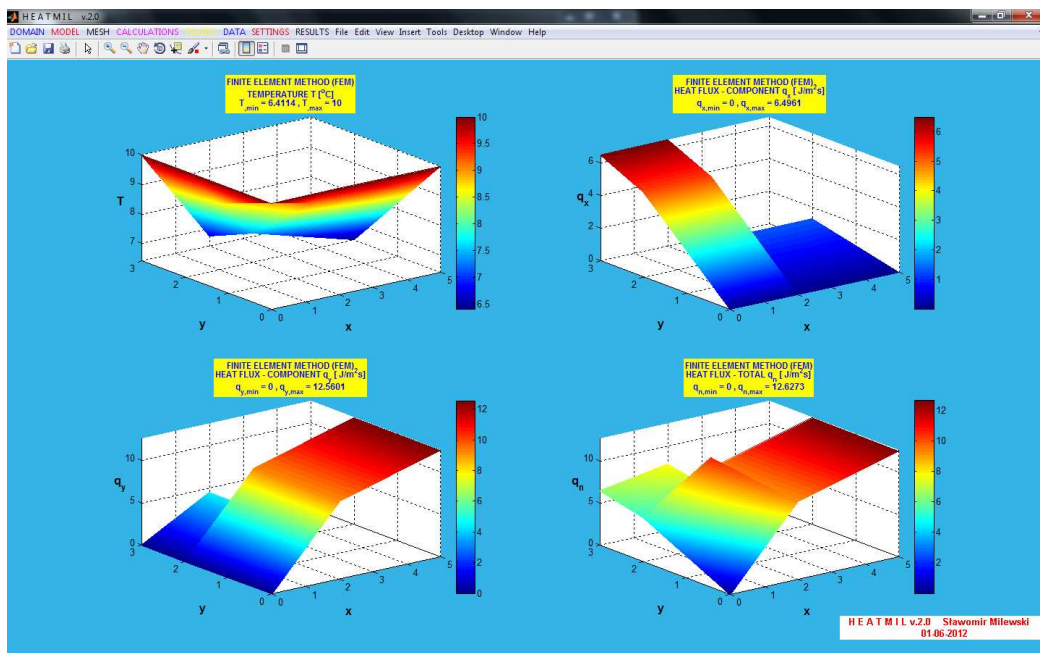

Fig. 13: The temperature distribution as well the heat flux components graphs

The analysis of the text results will be discussed next.

The "RESULTS -> NODAL RESULTS: TEMPERATURE AND REACTIONS" displays nodal primary (temperature T) and secondary unknowns (flux Q), calculated as "nodal reactions."

The "RESULTS -> ELEMENT RESULTS - HEAT FLUX" shows a list of elements with the values of heat flux components in five points of each element - in the four corners (at nodes of the element) and the center of gravity (center of the rectangle).

The "RESULTS -> RESULTS AT ARBITRARY POINT" allows to examine the solution value (temperature and heat flux) at the indicated node, element, or any point in the domain. Type x and y coordinates of the point, in which solution has to be calculated - a point has to be placed inside the domain or on its boundary. Please enter x = 4.5 and y = 1.9 for the considered example. The "UPDATE DATA" button pressed should display information about the temperature T = 6.8039 °C, as well as refresh the graph solution, in which the point should be marked.

The "RESULTS -> SAVE RESULTS TO FILE" option allows to save all the information about the model (data + results) to a text file. However, this procedure is not re-reading one - created in this way, the file can be applied for informational purposes only, outside the program. It has to be stressed, that reading and writing data to the model (without results) can be accessed via the "DATA -> SAVE MODEL DATA TO THE FILE" and "DATA -> LOAD MODEL DATA FROM THE FILE". After loading the data, calculations must be carried out again - the results are not written in a form to re-load.

11. Automatic generation of the fine FEM mesh

The option in the menu "MESH -> AUTOMATIC GENERATION" runs a small window in which one has to define the mesh density in the x direction and the mesh density in the y direction Default element shape is the rectangular one. Please enter a density of 3 (for x) and 5 (for y) and click 'APPLY'. Please, do not use higher densities for rectangular mesh (no larger than 10). After a while you should see the figure with dense FEM mesh containing 119 nodes and 96 elements - Fig.14. By right click, one may activate / deactivate the numbering of nodes and / or elements of the mesh.
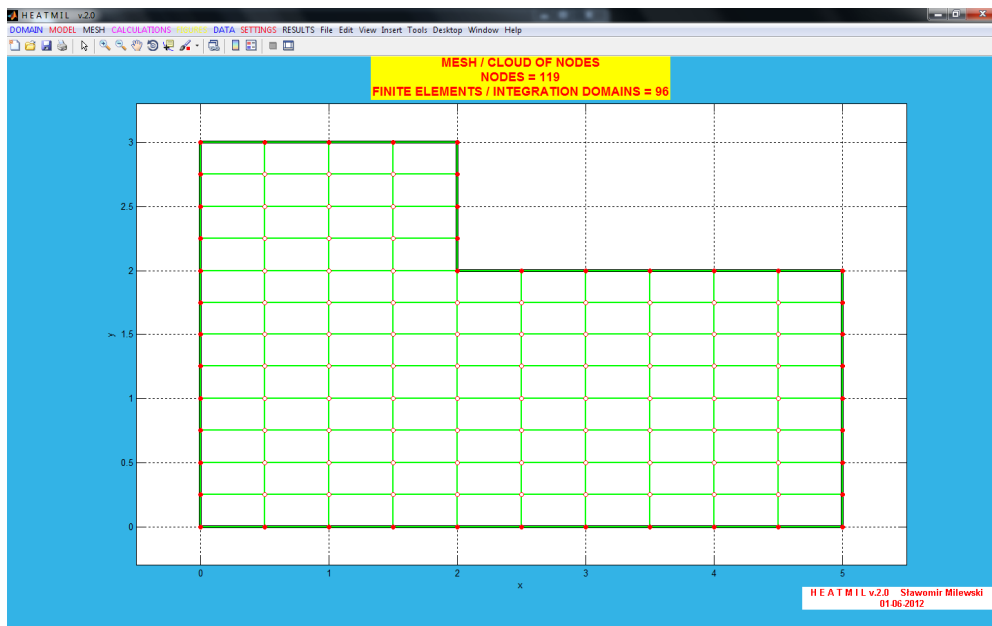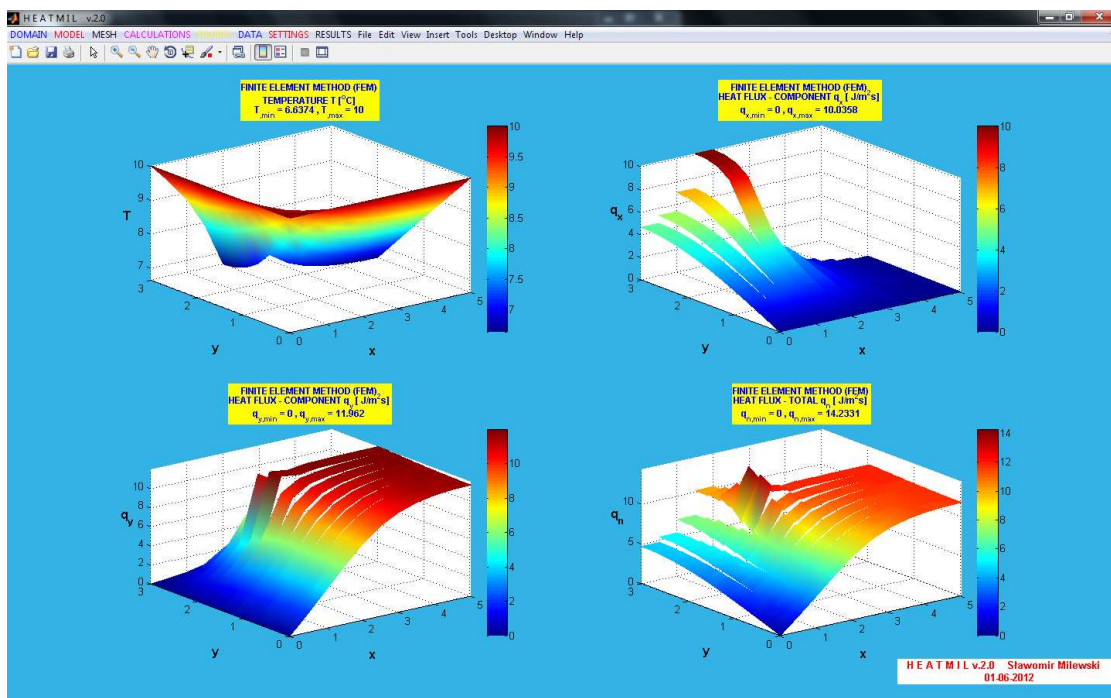

Fig. 14: Fine FEM mesh

Fig. 15: Temperature and het flux distributions for the denser FE mesh – without smoothing

Re-run the calculation by selecting the "CALCULATIONS -> RUN ', and after it is done, choose "GRAPHS -> ALL GRAPHS OF THE NUMERICAL SOLUTION", where one may browse all the results in the graphical form - the temperature distribution and flow components for denser mesh are presented in Fig.15.
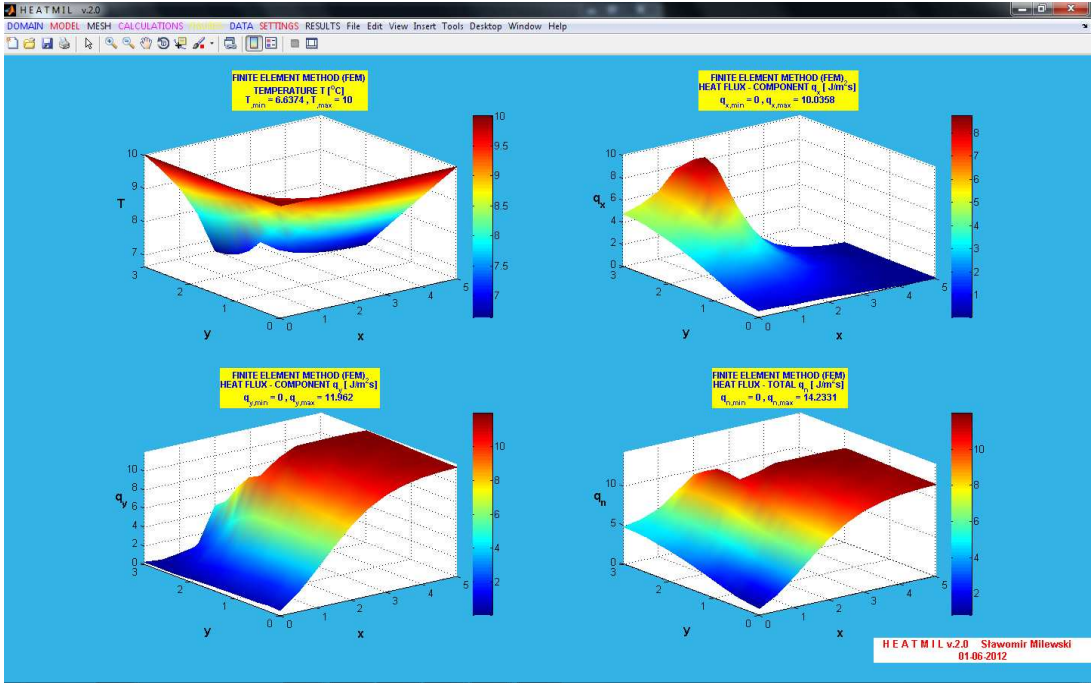

Fig.16: Temperature and het flux distributions for the denser FE mesh – with MWLS smoothing in the elements

In addition, by right-clicking on any of the graph axes, one may enable / disable smoothing of the graphs (which makes more sense especially in the case of the heat flux components graphs) using a MWLS technique (called Moving Weighted Least Squares), known for the meshless methods. Fig.16 shows the result.

12. Determination of the additional nodes for the manually generated FE mesh

If the mesh is generated manually, edge points, which become nodes automatically, may not be sufficient to ensure that the FEM mesh is build properly. For this purpose, one may introduce his own additional nodes and use them to build the mesh. Imagine that in the considered task, during the construction of the geometry, 2 points (coordinates (2,0)) and 4 (coordinates (0,2)) are missing – in fact, their presence is not required, since the entire lower horizontal line edge as well as the upper vertical line edge have the same boundary condition (temperature) prescibed, with the same value of 10 °C. The figure, showing the geometry with the specified boundary conditions, before the generation of the FEM mesh, is presented in Fig.17.
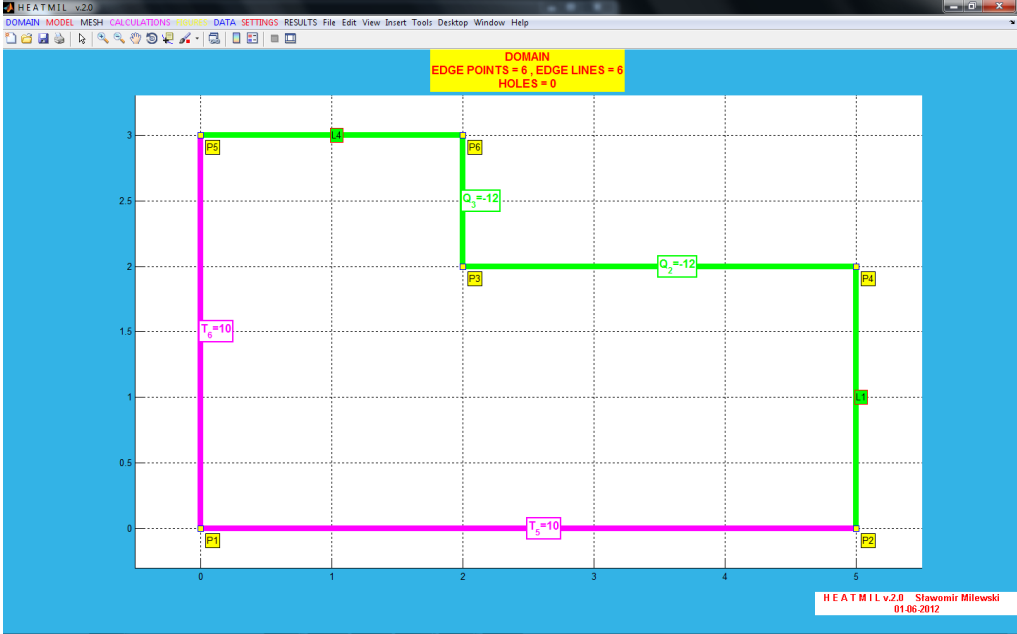
Fig. 17: Problem geometry without two obsolete edge points

Go to the generation of FEM mesh using manuall procedures. If one wants to build exactly the same grid of 3 elements, as before, one should generate two additional nodes – with the coordinates (2,0) and (0.2). For this purpose, fill in two text fields near the "NEW ADDITIONAL NODE" with coordinates of the first nodes: 2, 0, press the "ADD AN ADDITIONAL NODE" button, and then for the coordinates of the second node: 0 and 2, and again, press " ADD AN ADDITIONAL NODE ". Both points appear in the list on the left and at the right figure in the form of star marks, but they have a status of inactive. It is because such nodes may not always participate in the generation of the mesh. To activate them, simply select them from the list, then press "DE/ACTIVATE ADDITIONAL NODES". In the list of nodes, they will have the status: active. Specified numbers will be allocated to them: 7 and 8, which one may see the preview graph. The next steps are similar, as it was described in Section 8 – in order to construct three rectangular elements, one has to select four node numbers for each: the first element: nodes 1-7-3-8, the second element: nodes 8-3-6-5 and a third element: nodes 7-2-4-3. After this, the whole edit box should have the form, as in Fig.18.
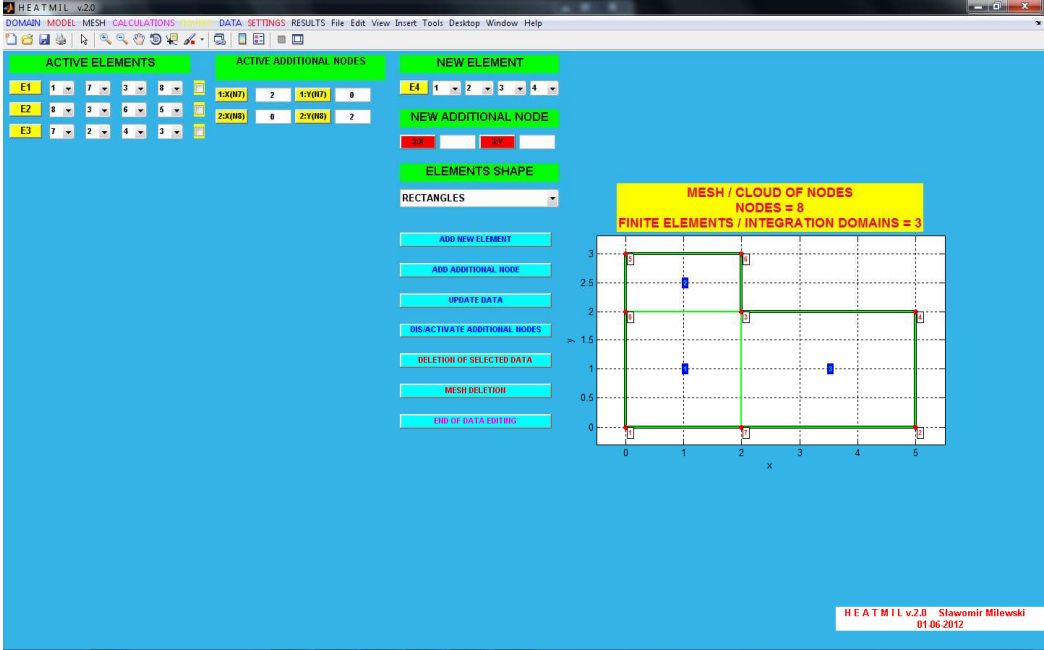
Fig. 18: FE mesh – three rectangles generated using two addtional nodes

For the generated mesh, we perform the calculations. Then we may view the results in the graphical or text form - of course, the solution should be the same as obtained previously (8 points for the edge).

13. Generation of the triangular mesh

Generation of a triangular mesh is possible in three manners. The first one is to convert a mesh of rectangles (providing such mesh has already been generated) to a mesh of triangles: the top menu: "MESH -> MESH CONVERSION". Backward conversion from triangular elements to rectangular ones is not possible.

The second method involves generation a trianglular mesh manually (as the mesh of rectangles). Removal of the current mesh (upper menu: "MESH -> MESH DELETION") is required, followed by the manually generattion of the triangular mesh (upper menu: "MESH -> MANUAL GENERATION"). In the "ELEMENT SHAPE" list, select "TRIANGLES" and then press the "UPDATE DATA" button. The rest of the procedure is similar as for the rectangular grid.

The third approach is the direct generation of the automatic mesh with triangular shaped elements. We will use this approach in the study for the geometry built on six points. Please choose from the upper menu: "MESH -> AUTOMATIC GENERATION", indicate the number of nodes 16x16 (the limitation here is not so strict as it is for rectangles), and then choose "ELEMENTS SHAPE" - "TRIANGLES" from the provided list, followed by the button 'APPLY'. Properly generated mesh - shown in Fig.19 - should contain 211 nodes and 360 elements.
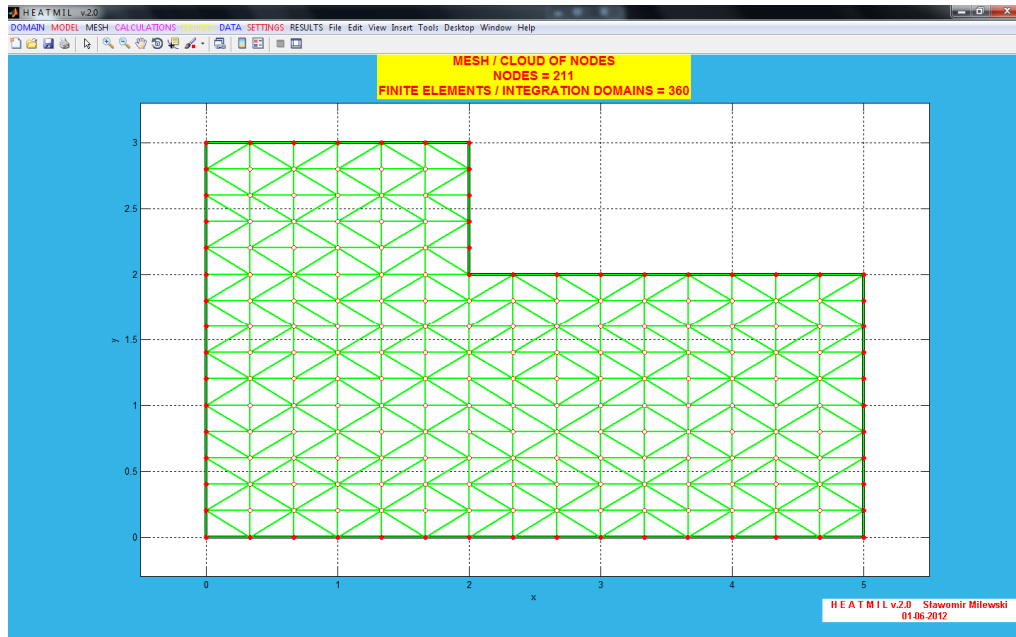
Fig.19: Exemplary triangular mesh, generated automatically

For such a new mesh, do not forget to restart the calculation. The maximum temperature for the current project should be 6.6467 °C. Fig.20 shows all available figures (temperature, heat flux) for the disabled option "RESULTS SMOOTHING (MWLS)". It should be noted that the lower degree of interpolation in the triangle (linear) corresponds to the constant values of the heat flux components in all finite elements.
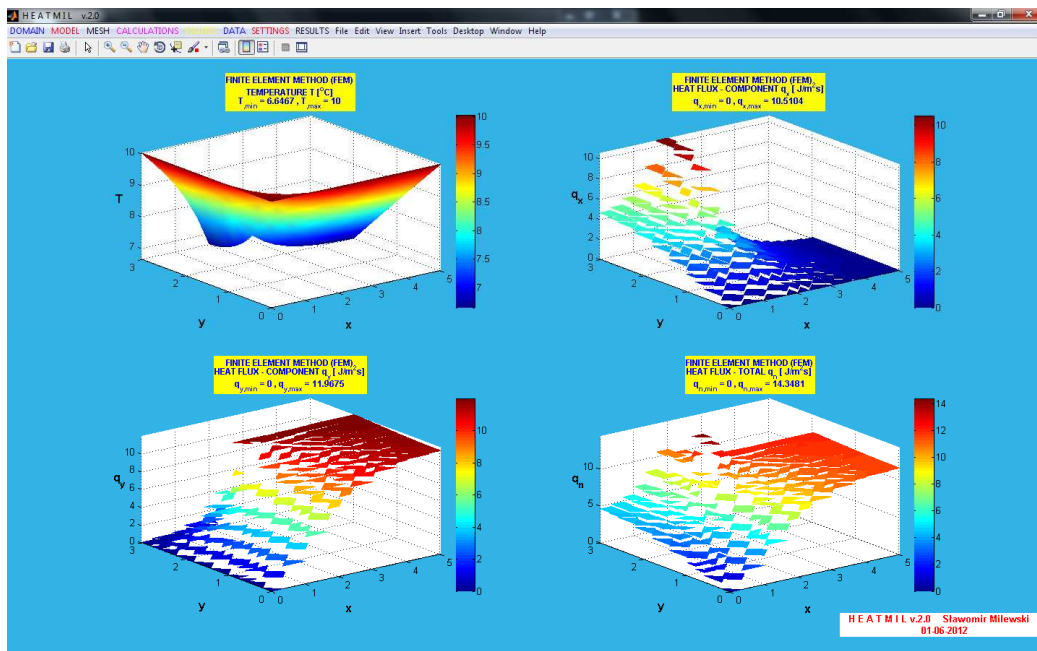

Fig. 20: Graphical results for the triangular mesh – without smoothing