





















THERMAL ANALYSIS 2D

User's Manual

CONTENTS

1.	Application version history	4
2.	Starting program	5
3.	One file principle	5
4.	Work in more windows	5
5.	Introduction screen	6
6.	Window arrangement	7
7.	User account settings 	7
7.1.	Help settings	7
7.2.	Author signature	7
7.3.	Send to calculation	7
7.4.	Default catalogues	7
7.5.	Other settings	8
8.	Messages	9
8.1.	Incoming messages	9
8.2.	Bin	9
9.	User groups	9
10.	Work with the file system	10
10.1.	File – New	10
10.2.	File – Open	10
10.3.	File – Save	12
10.4.	File – Save as	12
10.5.	File – Close	12
11.	Using program	13
12.	Recommended input procedure	13
13.	Details 	14
13.1.	Details catalogue 	14
13.2.	Detail settings 	14
14.	Import and export 	16
15.	Drawing tool 	18
15.1.	Information bar	19
15.2.	Tool bar	20
16.	Selection and editing tool 	21
17.	Settings 	22
18.	Materials 	23
19.	Boundary conditions 	23
20.	Picture 	24
21.	Interior surface temperature factor f_{Rsi}	25
22.	Linear thermal transmittance Ψ	26

22.1.	2 boundary conditions.....	28
22.2.	3 boundary conditions.....	28
22.3.	Floor slab on grade	31
22.3.1.	Simplified procedure according to EN ISO 13370	33
22.3.2.	Calculation of the heat flow via the ground	34
23.	Texts 	35
24.	Calculation.....	36
24.1.	Calculation start.....	36
24.2.	Loading the results	36
24.3.	Calculation archive	37
25.	Results.....	37
25.1.	Details 	38
25.2.	Data 	38
25.3.	Settings 	38
25.4.	Image export 	40
25.4.1.	Generate images.....	40
25.5.	Protocol 	41
25.6.	Protocol export 	41
26.	Validation according to EN ISO 10211 and EN ISO 10077-2.....	42
27.	Program updates.....	44
28.	Troubleshooting.....	44

1. APPLICATION VERSION HISTORY

Version	Date of issue	Significant changes
1.1.1	28.11.2017	Update of the computational kernel (resolved issues in case of details with big difference between the maximum and minimum length of the path)
1.1.0	22.11.2017	Update of the computational kernel Addition of graphic output of heat flow calculation results
1.0.2	15.11.2017	Fixed double counting of the surcharge when calculating $f_{R_{Si,cr}}$
1.0.1	7.11.2017	Addition of possibility of automatic move of the imported DXF file
1.0.0	30.10.2017	The first version of the program Thermal Analysis 2D released

2. STARTING PROGRAM

There are two ways to run the program.

- a) Via the website www.deksoft.eu
- b) Via another program.

All programs run directly in the browser window, no installation is needed.

3. ONE FILE PRINCIPLE

All computing programs share one file. There is no need to create a separate file for each of the programs to work within a single project (building). For example, if you already have a file in Energetics or Thermal Analysis 1D, you do not need to create a new file in Thermal Analysis 2D but just open an existing file.

4. WORK IN MORE WINDOWS


Within one computer, it is possible to run multiple windows with programs from www.deksoft.eu. This allows multiple input files to be opened at the same time. **CAUTION: When you open the same file in multiple windows, the synchronization feature is restricted and changes may not be synchronised correctly.**

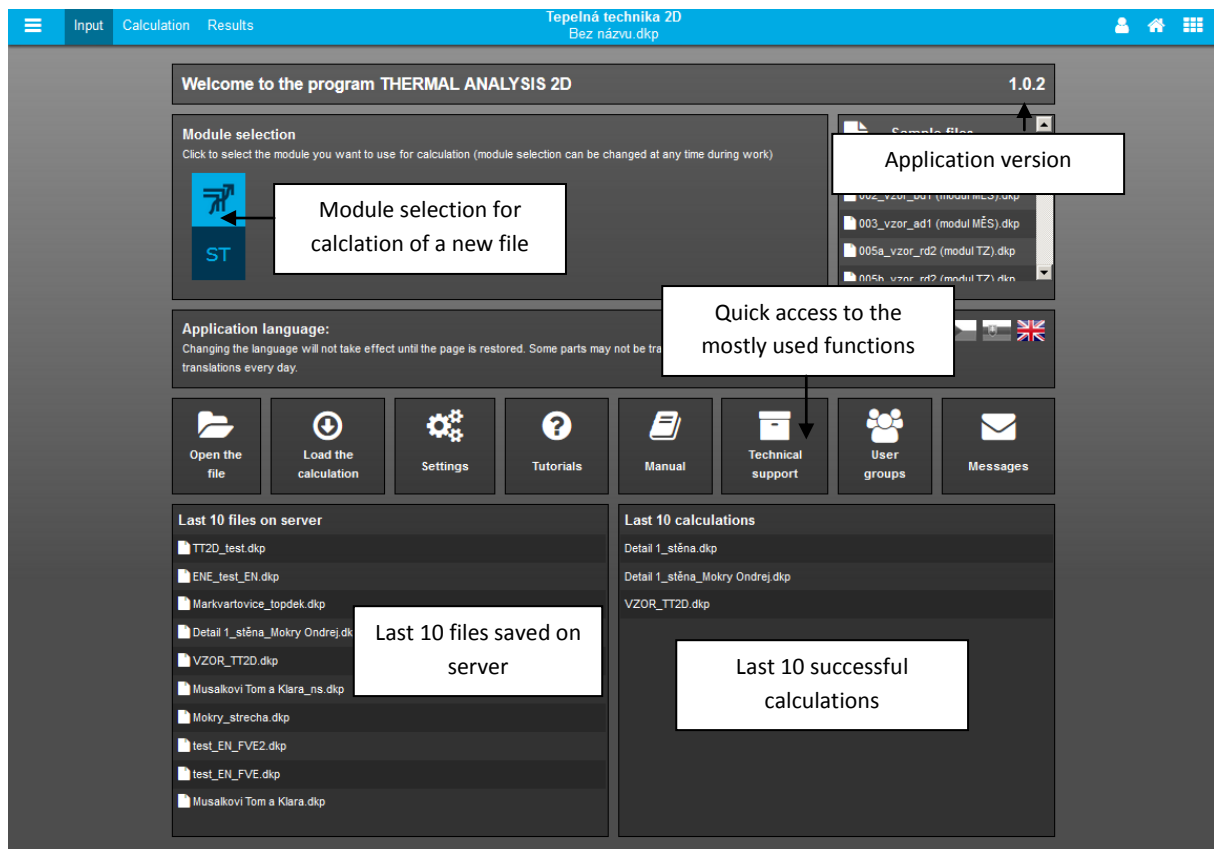
5. INTRODUCTION SCREEN

When you start the application or when you open a new file, you will see an introduction screen that provides quick access to the mostly used features and a clear choice of computing module.

At present following module is available:

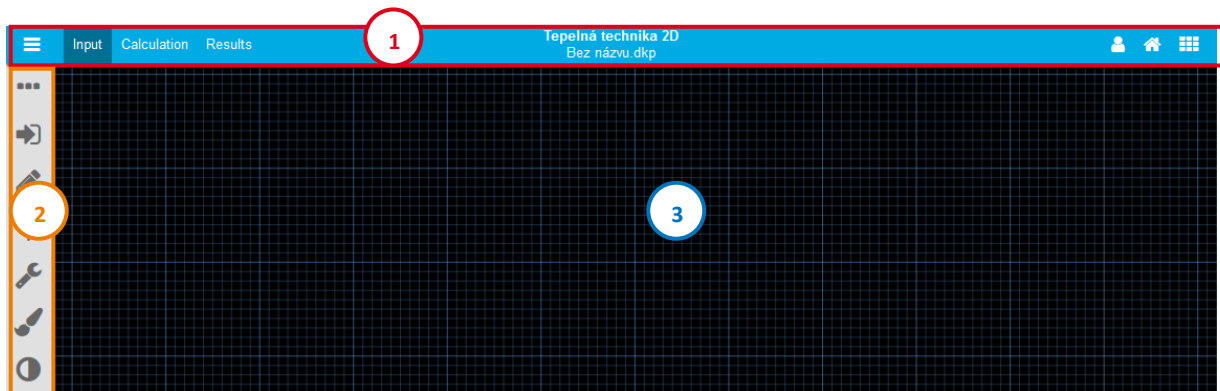
- a) Stationary calculation – steady state (ST)

In case you do not have a license for one of the modules, a symbol  and restricted access information will be displayed above the module selection icon.



6. WINDOW ARRANGEMENT

1. Top bar / 2. Navigation in program / 3. Window for drawing or showing results



7. USER ACCOUNT SETTINGS

You can access the user profile settings by going to the user menu in the top bar and selecting the Settings option.



7.1. HELP SETTINGS

This option allows you to enable and disable the help system in DEKSOFT programs.

7.2. AUTHOR SIGNATURE

All programs allow you to use the function of automatically fill in identification data of the author that is filled in this section.

7.3. SEND TO CALCULATION

This option allows you to set the behaviour of the programs when you send the file to the calculation. It allows you to change the following parameters:

- a) File saving when sending to calculation
- b) Number of displayed files sent for calculation in the top menu
- c) Behaviour when overwriting unsaved data

7.4. DEFAULT CATALOGUES

In this section you can choose which catalogues will be displayed as default after the catalogue is invoked in the Thermal Analysis 2D. So you can set up the catalogue you most often use to speed up the overall work in the application.

7.5. OTHER SETTINGS

In this section, you can set the auto save interval or completely disable auto-save (value set to 0). **CAUTION: auto save function only works for files stored on the server.**

You can also change the default language setting. **CAUTION: Translations are only available in some applications (see www.deksoft.eu for more information). To fully reflect your language change, you need to refresh the page.**

In the other settings section, you can also choose an alternative look of the application. In the current version, the following features are available. **CAUTION: Bright looks only apply during text input. Drawing detail window is always with a dark background.**



Default



Default, dark top bar



Grey



Blue



Purple

8. MESSAGES

Messages can alert you for the news in building physics applications. You can invoke the modal message window by going to the user menu and clicking Messages. The red number highlights the number of new messages.



8.1. INCOMING MESSAGES

This section is automatically opened when you click Messages. Received messages that have not been removed are displayed. Unread messages are bold. Clicking on the appropriate line will open the message.

8.2. BIN

Deleted messages are displayed in the bin. Messages from the bin are automatically completely erased after 30 days.

9. USER GROUPS

User groups allow users to share catalogues. For more information about working with user groups, see the separate manual.

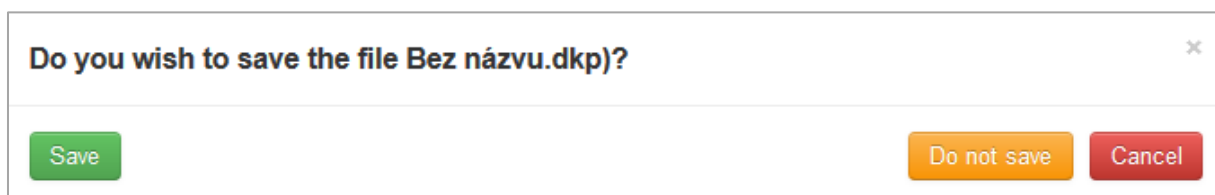
10. WORK WITH THE FILE SYSTEM

To work with the file system, use the menu in the top bar. You will be informed of all events by using the notification information in the lower right corner.



10.1. FILE – NEW

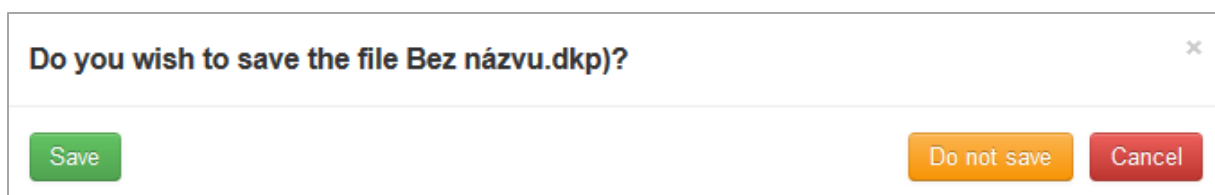
Creates a new file for work with programs. While working in the program, you may first be asked if you want to save the currently used file. In this case, the new file will only be created after saving an existing file or choosing the Do not save option.



CAUTION: We recommend that you save the new (created) file as soon as possible to the server to use the AutoSave feature.

10.2. FILE – OPEN

This item is used to open an existing file. While working in the program, you may first be asked if you want to save the currently used file. In this case, the new file will only be created after saving an existing file or choosing the **Do not save** option.



In the next step, you can choose whether to open a file from a server or from a local computer. By choosing **Local computer**, a system explorer will appear in which you can search for the file you want. Selecting **Server** displays the structure of your files and directories from which you can select the file you want.



TIP: In the modal window File open, you can click the table header to sort the file by name or date.

CAUTION: Opening files from a local computer is only allowed to users with a valid license.

File open ✕

Searching in saved files

Creation of a new folder

File selection for delete or move functions

File sorting

New folder

	Name	Created	Changed	
Shared files ← Shared storage of user groups				
<input type="checkbox"/>	2013			Delete file / folder
<input type="checkbox"/>	2014			Rename file / folder
<input type="checkbox"/>	2015			Delete file / folder
<input type="checkbox"/>	2016			Delete file / folder
<input type="checkbox"/>	2017			Delete file / folder
<input checked="" type="checkbox"/>	Bez názvu.dkp	25.11.2013 07:16	11.08.2014 14:39	Delete marked files
<input type="checkbox"/>		29.09.2016 12:19	24.05.2017 09:46	Copy marked files
<input type="checkbox"/>	Markvan	14.05.2017 12:49	14.11.2017 14:46	Move marked files
<input type="checkbox"/>	PENB Náměšť na Hané	08.04.2016 11:02	26.08.2016 09:52	Delete marked files

Delete marked files

Vložení vybraných souborů

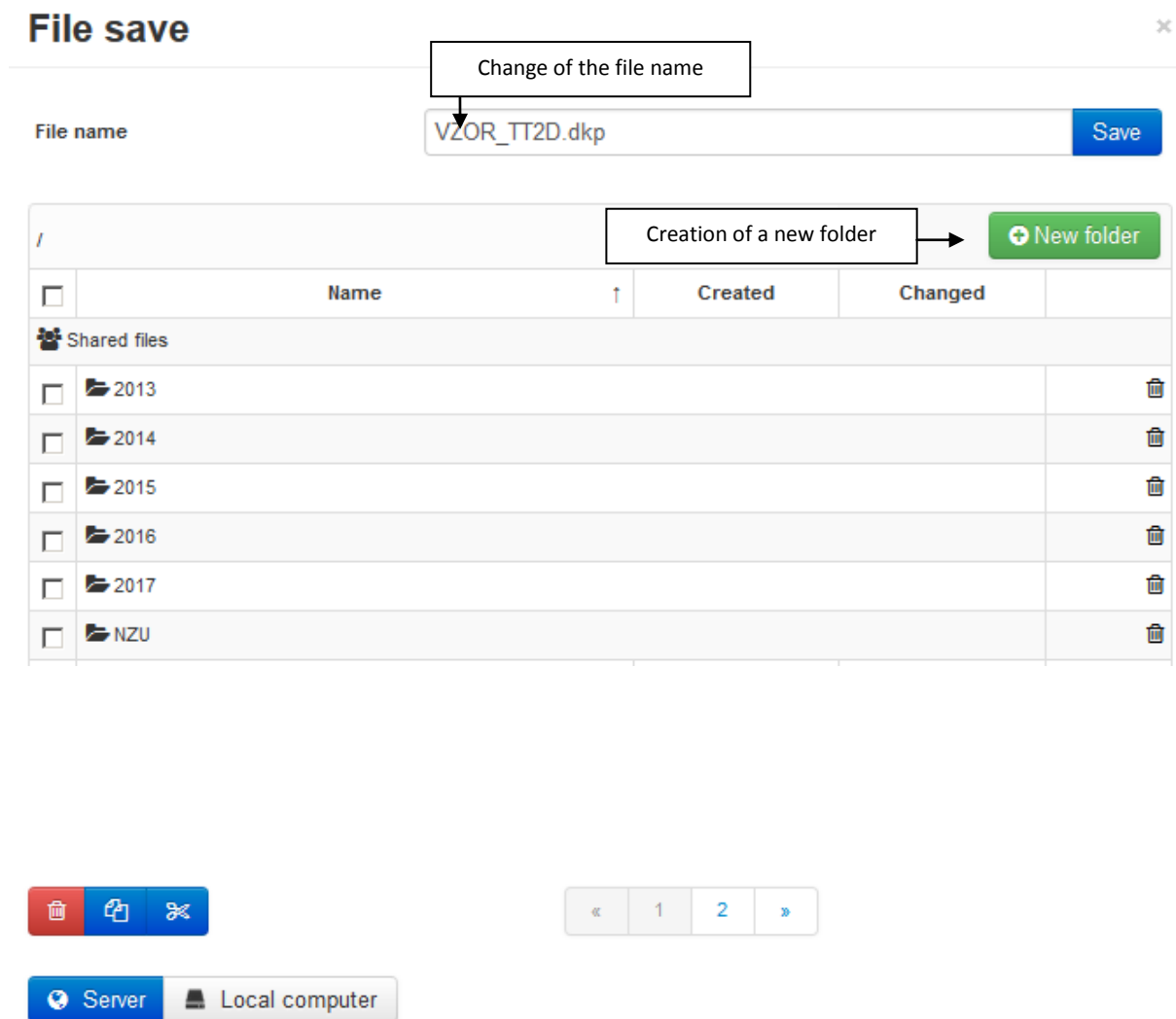
Server

Local computer

Change your storage selection

10.3. FILE – SAVE

If the file was previously saved, the current version of the input will be saved. During first attempt of saving the file, the selection window will appear where you want to save the file (**Server** or **Local computer**). When selecting **Local computer**, the file is downloaded according to the settings of a default Internet browser (most often automatically downloaded to the Downloads folder). When selecting **Server**, a modal window opens, where you can create folders or rename the file. Save the file with the **Save** button.



10.4. FILE – SAVE AS

This option allows you to save a copy of the file. The **Save as** window is the same as the **Save** window.

CAUTION: In case of saving to server, the newly saved file will be opened after the saving is finished. When saving to a local hard drive (Local computer), the original file remains open.

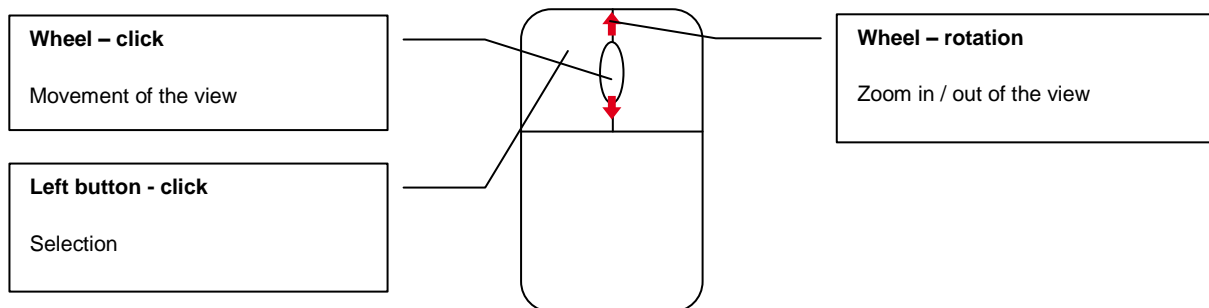
10.5. FILE – CLOSE

This command closes the currently used program. Before closing, you may be asked if you want to save the currently used file.

11. USING PROGRAM

The basic functions of the program can be activated using the toolbar on the left side of the screen.

For movement within the 2D model, the use of a computer mouse fitted with a wheel is the most appropriate option. Computer mouse control is illustrated in the following figure.



12. RECOMMENDED INPUT PROCEDURE

The program is designed for input from the top of the page to the bottom (both in the sections and in navigation).

The recommended input procedure in the Thermal Analysis 2D is as follows:





- 1) Create detail geometry (by drawing directly in a program using Drawing tool / by importing a DXF file)
- 2) Definition of materials and their assignment to models
- 3) Definition of the boundary conditions and assignment of them to paths
- 4) Defining the required additional assessments (temperature factor, linear thermal transmittance)
- 5) Entering the text input for the protocol
- 6) Performing the calculation
- 7) Processing of results

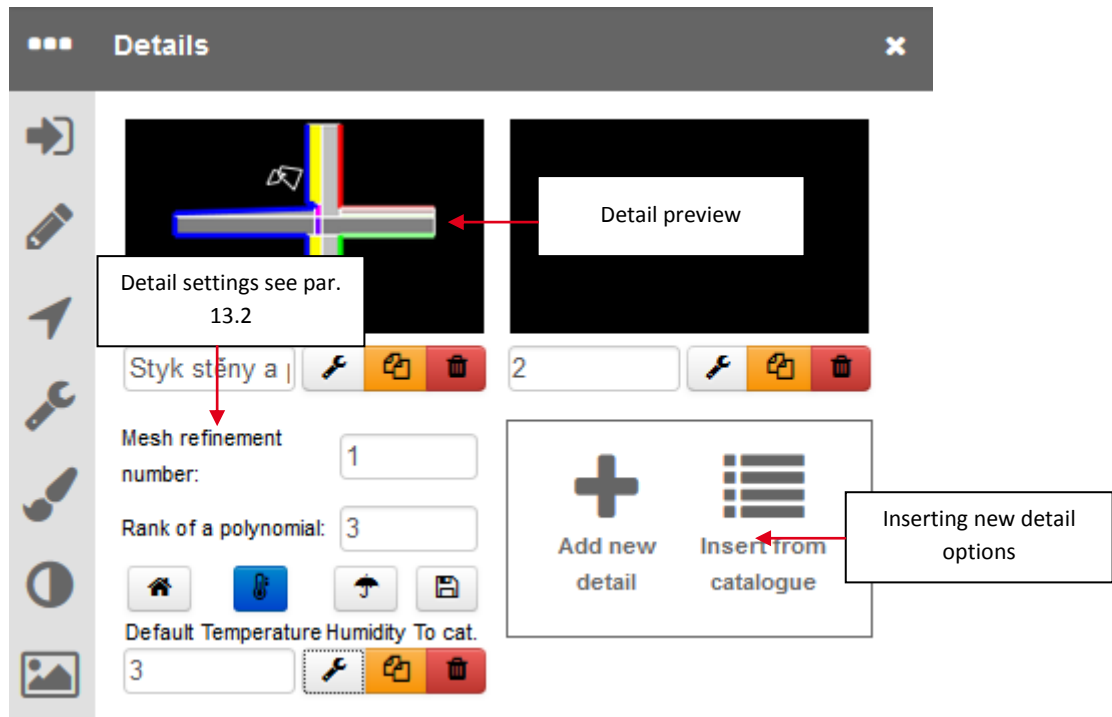
Video

At <https://stavebni-fyzika.cz/programy/teptech2d/vyukove-materialy> you will find a sample video with the input procedure of the Thermal Analysis 2D.

13. DETAILS ...


In the details section, you can do the following:

- Adding new details to the DKP file (using the button + **Add new detail**)
- Inserting detail from the detail catalogue (using the button  **Insert from catalogue** - see chapter 13.1)
- Adjust detail settings (using the button  **Settings** - see chapter 13.2)
- Copy detail (using the button  **Copy**)
- Delete detail (using the button  **Delete**)
- Change the name of the detail in the text box



13.1. DETAILS CATALOGUE


The catalogue of details allows access to previously stored user details or to the typical details prepared by the building materials manufacturers. Within the catalogue you can use sharing between users. For details on how to share catalogues with user groups, see the separate manual.

NOTE: At present, it is not possible to save the detail input and the results of the calculation of the linear thermal transmittance at the same time. We recommend that you save the detail input and then manually fill in the values of the calculated linear thermal transmittance in the **Catalogue edit** (button  in the upper right corner).

When entering the detail from the catalogue, it is checked whether the materials used in detail already exist in the input. If so, the user will be able to choose whether he wants to use existing materials or to insert materials from the catalogue to new positions. The same applies when inserting boundary conditions.


You can save details in the catalogue in the detail settings section.


13.2. DETAIL SETTINGS


By clicking on the icon  below the detail preview, the specific settings for that detail will show up.


The **mesh refinement number** affects the number of finite elements to which the detail will be divided. It is possible to enter values in the interval $<0; 5>$, the default value is 1. **CAUTION: We do not recommend choosing values higher than 2. If higher values are selected for more complex details, the server computational memory limits may be exceeded and the calculation may not be performed. Such high values are not necessary for the assessment of building details.**


Rank of a polynomial affects the approximation of the solution on each mesh element. In the case of the 1st degree polynomial, the solution is linear, for the 2nd degree it is approximated by the quadratic function, etc. With the increasing rank of the polynomial, the accuracy of the solution increases, but also the memory demands and the calculation time increase. It is possible to enter values in the interval $<1; 10>$, the default value is 3.

The **Set as default** icon  allows you to set which detail will be displayed as default when you open the .dkp file.

The **Calculate temperature** icon  allows you to choose whether to display the results of the calculation of the temperature field of the detail. If the button is active (blue background colour), the calculation results of the detail temperature field will be available.

The **Calculate humidity** icon  allows you to choose whether you want to display the results of the calculation of the humidity field of the detail. If the button is active (blue background colour), the results of the detail humidity field will be available.

The **Heat flow** icon  allows you to choose whether you want to display the results of the heat flow calculation. If the button is active (blue background color), heat flow calculation results will be available.

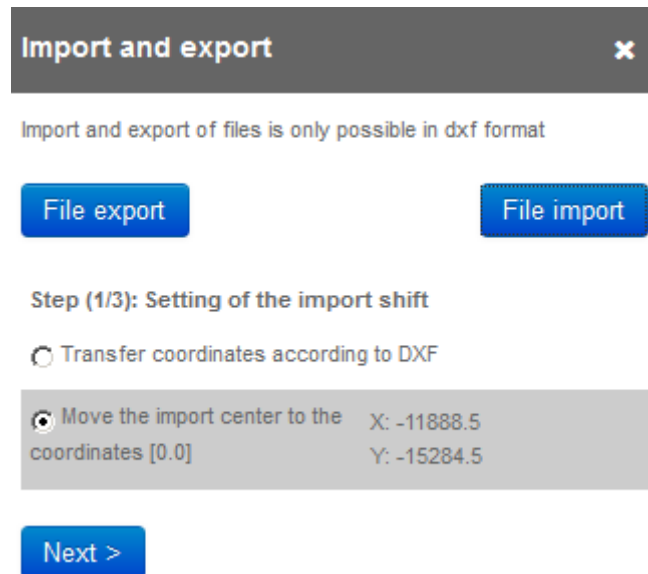
The icon **To cat.**  you can save the selected detail in the catalogue. The saved detail can then be used in other files. Detailed geometry including specified materials, boundary conditions, settings and input of additional calculations are stored in the catalogue.

14. IMPORT AND EXPORT ➡

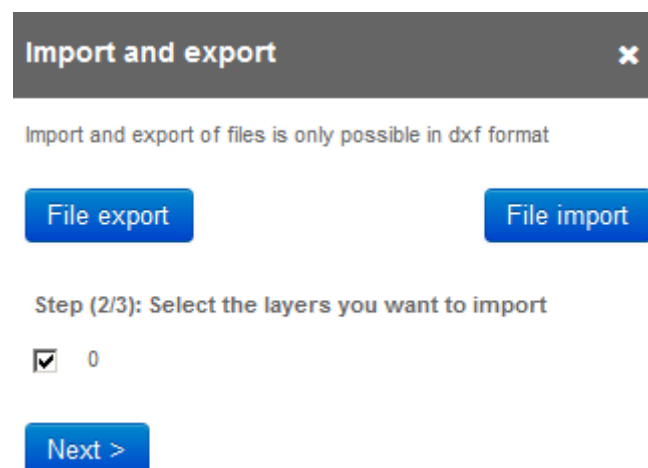
In the current version of the program, it is possible to import DXF files. In the next versions of the program we are preparing an extension of the import options.

Importing a DXF file is possible in the following steps:

- Select a DXF file** on a local disc after clicking on the File import button - after uploading the file, a preview is shown in the program environment with other setup options.
- Step (1/3): Setting of the import shift.** In the case of a small DXF file shift, DXF coordinate consistency is automatically selected. If the shift is more pronounced, the detail is moved to the origin coordinates (0,0). Leaving a large shift of the detail can lead to display errors.



- Step (2/3): Select the layers you want to import.** Select layers to import from DXF file. In this step, it is possible to omit unnecessary layers from the import to achieve greater clarity of detail and more reliable import. After selecting the layers you want to import, click **Next >**.



- Step (3/3): Import settings.** In the last step, the import settings are available. If you use the default settings, DXF is inserted only as the basis for manual geometry creation (the default option for details that were not modified for import into TA 2D). If the detail is well prepared for import, you can use advanced settings to allow direct import of imported geometry without further editing.

Keep the division into layers according to the original - when using this option, original layers from DXF file will be imported into separate layers. Original layers will not be affected after import.

Automatically divide into areas - using this option, the geometry will be subdivided into sub-areas that can then be directly assigned with material properties.

Merge overlapping paths - multiple overlapping paths (lines) will be deleted when you use this option. Work with the imported file will then be smoother. This option is recommended to be enabled when you select option Automatically divide into areas.

Split paths at intersections - when you use this option, the paths (lines) are automatically divided in their intersections. This option is recommended to be enabled when you select option Automatically divide into areas.

Remove unfinished paths - unfinished paths can not be directly used to enter material properties. Using this option the unfinished paths will be removed during import.

Import and export ✕

Import and export of files is only possible in dxf format

[File export](#) [File import](#)

Step (3/3): Import settings

Keep the division into layers according to the original	<input type="checkbox"/>
Automatically divide into areas ^{1) 2)}	<input type="checkbox"/>
Merge overlapping paths ²⁾	<input type="checkbox"/>
Split paths at intersections ²⁾	<input type="checkbox"/>
Remove unfinished paths ¹⁾	<input type="checkbox"/>

1) In some cases, errors may remain in the model after import, especially if there are more non-overlapping parts. We recommend using it along with the Split paths at intersections and Merge overlapping paths.

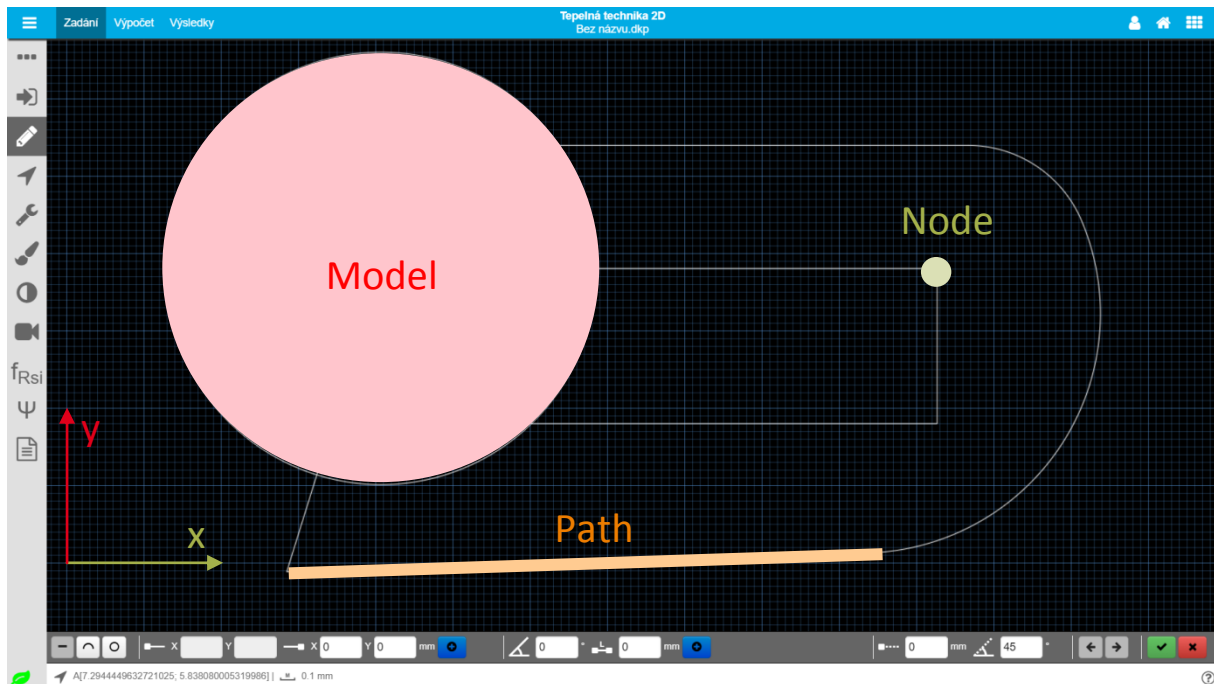
2) Computer performance demanding function, import may take longer time.

[< Back](#) [Complete import](#)

15. DRAWING TOOL





In addition to importing function, detail geometry can be entered directly in the program using the drawing function. The geometry of the detail in the Thermal Analysis 2D environment consists of the following elements:

- Model** - a closed area that can be assigned with material properties consisting of paths.
- Path** - Line or arc that is the link of two nodes and which can be assigned with the boundary condition properties.
- Node** - the point at which the path begins or ends.




Drawing in Thermal Analysis 2D is done by entering individual paths. Once the path is closed, the model is automatically created.

Drawing can take place in two ways:

- By manual drawing using the toolbar** (see chapter 15.2)
- By entering the nodes using the mouse.** When drawing with the mouse, you can use the following keys:
 - Ctrl** - while drawing with the mouse the set increment in the toolbar be used (icon )
 - Shift** - while drawing with the mouse the set angle in the toolbar be used (icon )
 - Ctrl + Alt** - attachment to the grid
 - Enter** - automatically complete the model (the equivalent of the button )
 - Esc** - cancel drawing (equivalent of button )

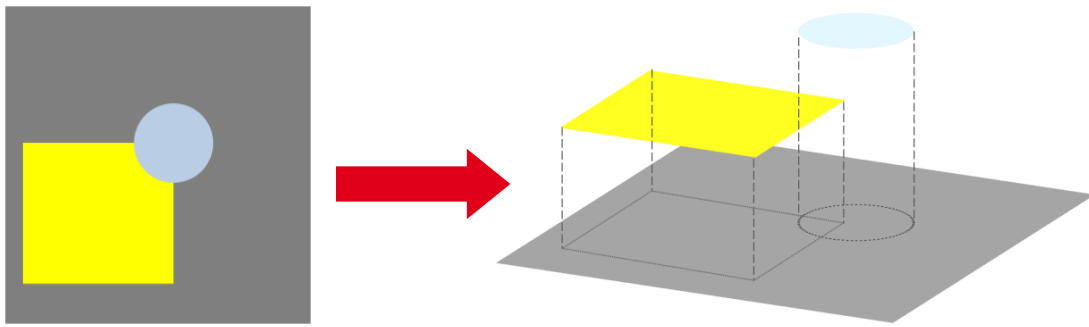
For advanced users: Placement of the first point at the origin of the coordinates

If you want the first node of path to be located at the origin of the coordinates, just press the button  in the manual coordinate input section of the toolbar.



Each model is placed into a separate layer. If multiple layers are superimposed in the detail, the material or boundary condition that is currently visible is entered into the calculation. This feature can be used to create

more complex details (see the following figure). You can change the order of layers using the Selection and Editing tool (see Chapter 16).



After activating the drawing tool, two bars are displayed at the bottom of the screen - dark tool bar and light information bar.

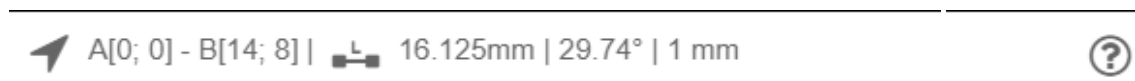
15.1. INFORMATION BAR

If the path is not open (before you start drawing), three information items are displayed in the Information bar:

- Current cursor position** - coordinates of the current position in the format [x; y]
- Current grid scale**
- Help for drawing keys**






If the path is in drawing more, the following information is also displayed:

- Initial (A) and End (B) path node** - coordinates of the initial node in the format [x; y]
- The length of the currently created path**
- Current path angle**





15.2. TOOL BAR

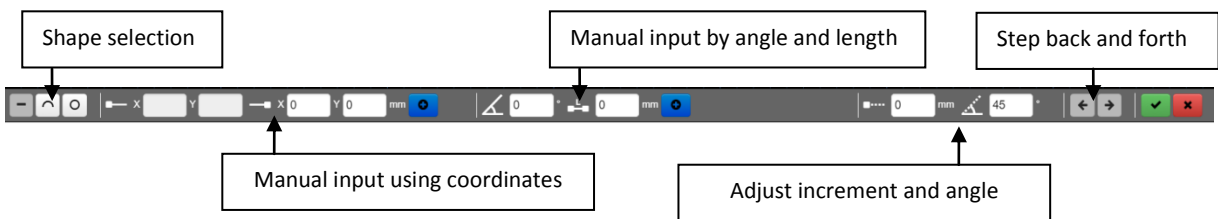
When entering by mouse, use the toolbar to select:

- a) **Shape type** (line / arc / circle)
- b) **Increment settings** ■....
- c) **Adjustment of the angle** 
- d) **Step back** ← - removes the last added node
- e) **Step forward** → - returns the last returned node back
- f) **Selection whether to draw a longer**  **or shorter**  **arc between nodes** (only when entering an arc)
- g) **Whether the arc is to be concave**  **or convex**  (only when entering an arc)

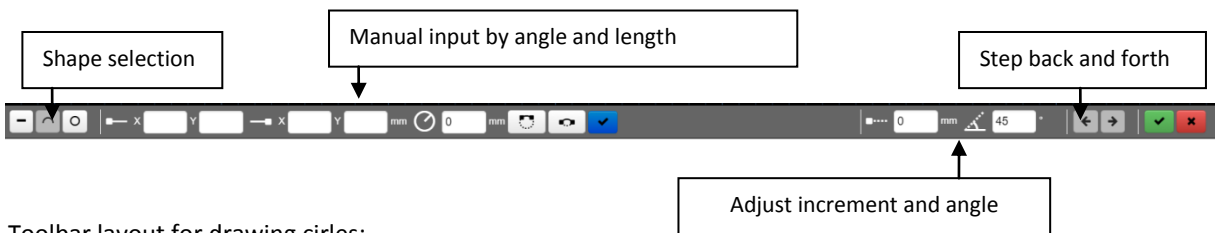
For manual input, you can also use the input field:

- h) **Coordinates of the start** ■— **and end** —■ **points**
- i) **Angle**  **and length** ■— of the path (this is an alternative way of entering, it can not be combined with the coordinates input)
- j) **Arc radius**  (only for arc and circle input)
- k) **Automatic route closure** ✓
- l) **Cancel drawing** ✕

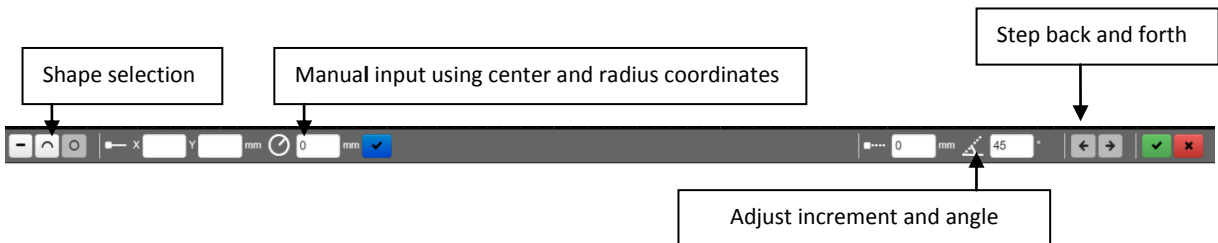
Toolbar layout for drawing lines:






Toolbar layout for drawing arcs:














Toolbar layout for drawing circles:



16. SELECTION AND EDITING TOOL ↗

When selecting and editing is activated, you first need to choose the selection mode (model  / path  / node ) in the toolbar. You can use the selection and editing function to perform the following operations:

- a) **Move model / path / node** 
- b) **Rotate the model / path / node** 
- c) **Change of model material** 
- d) **Changing / removing the boundary condition for the path** 
- e) **Changing the order of the layers (level up  / level down  / to front  / to back )**
- f) **Merging models** 
- g) **Split the path and create a new node** 
- h) **Remove path from model** 

TIP: When selecting a model, information about the selected model's area will be displayed in the information bar.

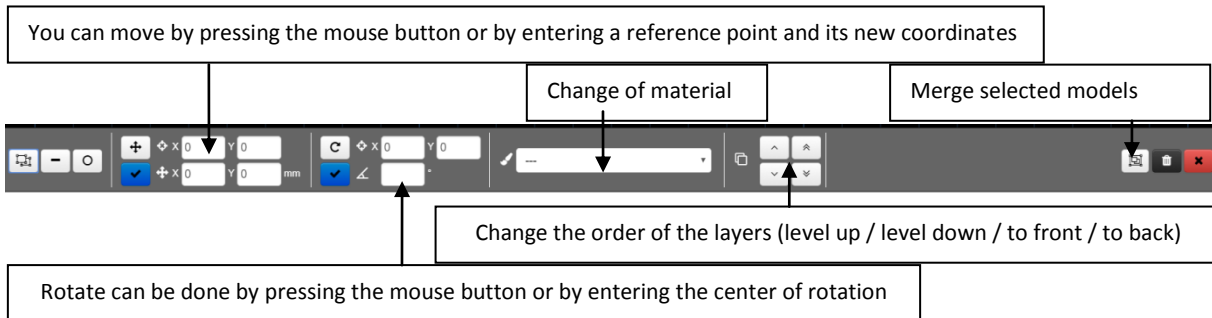
More elements can be selected at the same time by dragging the mouse over the area or by clicking on the area:

- a) Dragging from right to left selects each element that is at least partially in the selection
- b) Dragging from left to top selects the elements that are in the selection across their entire surface
- c) Ctrl + clicks - add the element to the selection
- d) Ctrl + Shift + clicks - removing the area from the selection

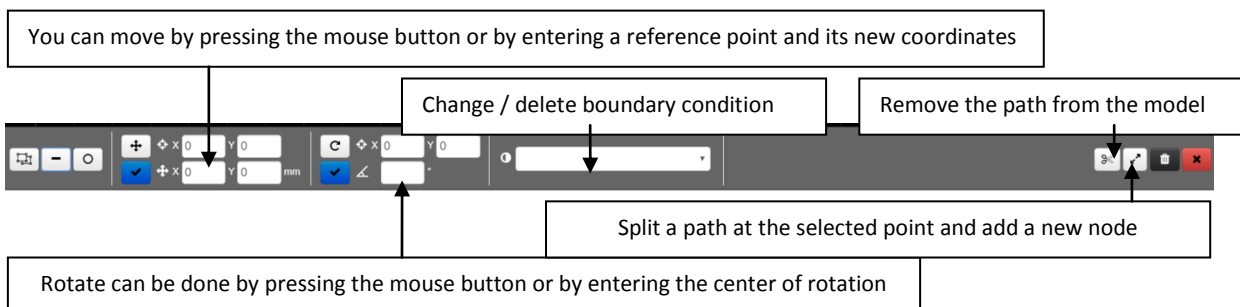
Appearance of the toolbar before choosing the selection method:



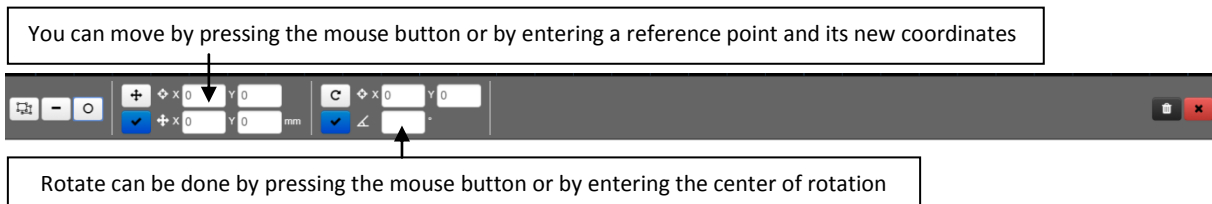
Appearance of the toolbar when selecting a model:



Appearance of the toolbar when choosing a path:



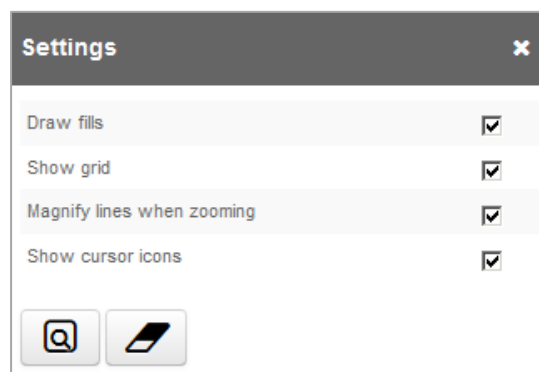
Appearance of the toolbar when choosing a node:



17. SETTINGS

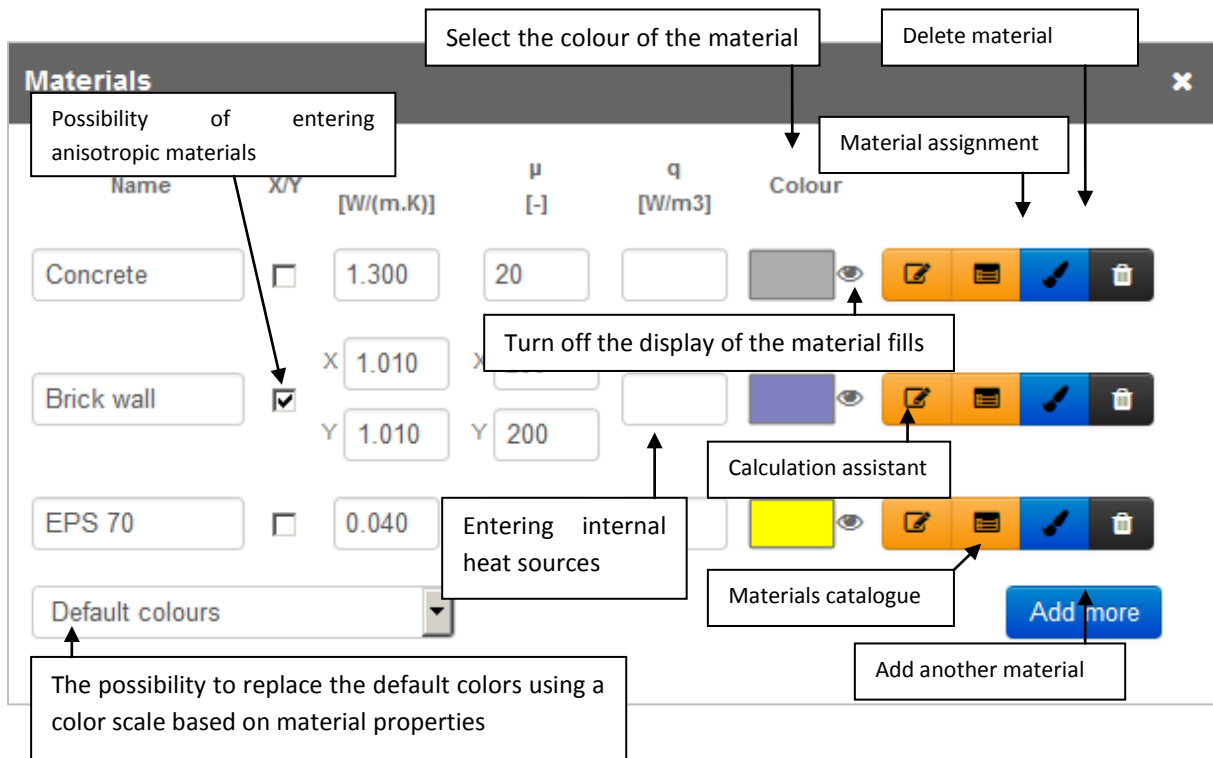
The following can be done in the setup menu.

- a) The option **Draw fills** allows you to turn off the display of model fillings in detail preview. For example, if you need to adjust the detail geometry, colour fills can reduce the clarity of the drawing. By default, this option is active.
- b) The option **Show grid** allows you to turn off the display of the grid in the background.
- c) The option **Magnify lines when zooming** allows you to set whether the zoom thickness should be increased. By default, this option is inactive.
- d) The option **Show cursor icons** allows you to display information icons next to the cursor when the drawing tool is active (information about snap option, close the model). This feature may reduce the response speed. By default, this option is inactive.
- e) The button **Customize window size** maximizes the detail view in the current drawing field size.
- f) The button **Clear scene** clears all details of the detail geometry.



18. MATERIALS

The menu **Materials** is used to specify the material characteristics and their assignment to the created models. You can enter any number of materials in the input using the **Add more** button. The features are described in the following figure.




If anisotropic materials option (materials with different properties in different directions) is activated, a separate fields for entering the thermal conductivity and the diffusion resistance factor in the x (horizontal) and y (vertical) directions are displayed.

The following calculation assistants are available in the current version of the program:

- Calculation of thermal conductivity and diffusion resistance of air layers according to ČSN EN ISO 6946
- Consideration of the quality of the vapour barrier
- Calculation of design thermal conductivity from the declared value

For details on each of the calculation assistants, refer to the Thermal Analysis 1D User's Manual.


The catalogue of materials is the same as in the program Thermal Analysis 1D. All materials, including user catalogues, are available here. For details on how to create user catalogues and how to share them among users, see the separate manual.

Using the Material assignment feature, you can directly assign the material to the model in the geometry input. Clicking on the icon  can be used to select models in the graphical input. **TIP: You can drag and drop the window to any position by clicking on the grey area of the header and then dragging it.** If it is not possible to select a particular model in the assignment, it is probably due to its placement in the lower layer. The order of models can be changed by selecting and editing tool (see chapter 16).

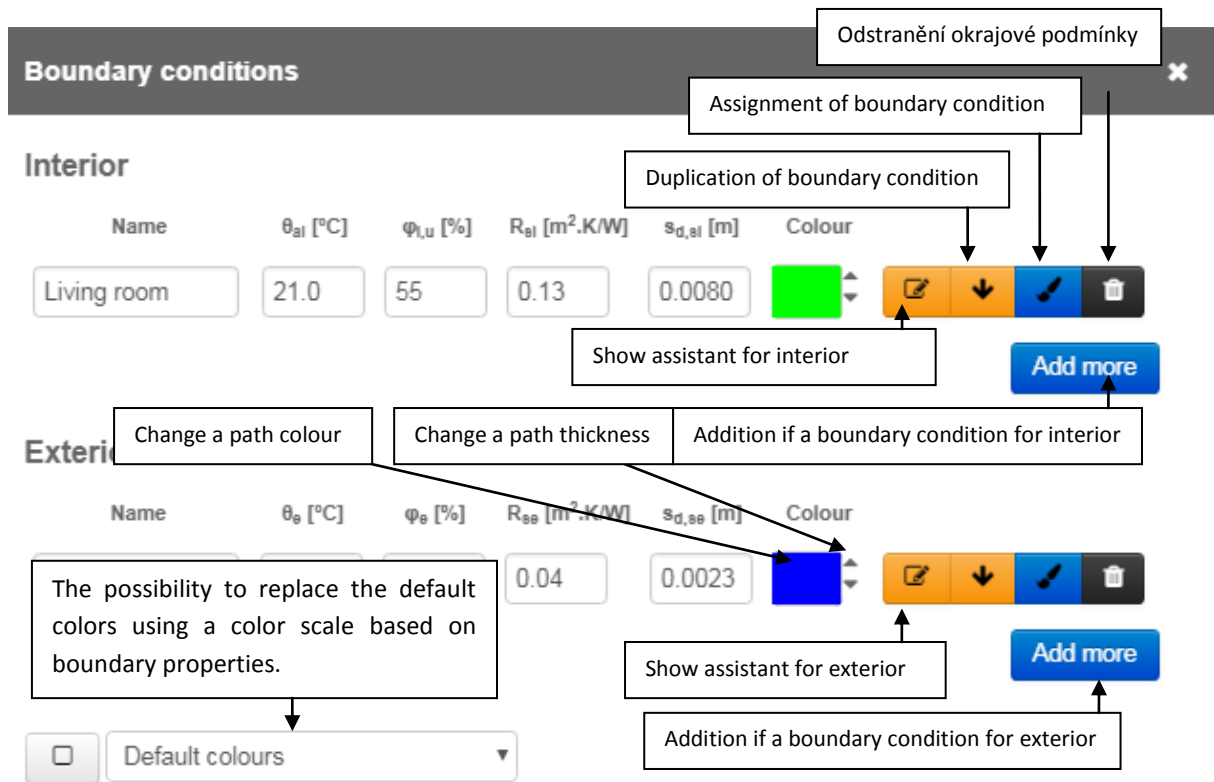
When deleting the material that is used to enter some details, a modal window with Warning about the use of the material will be displayed. Therefore, it is not possible to remove the material in use with just one click.

19. BOUNDARY CONDITIONS


The menu **Boundary conditions** is used to specify boundary conditions and assign them to the paths created. You can enter any number of boundary conditions using the **Add more** button. The features are described in



the following figure. **TIP: To specify the boundary conditions, we recommend using a wizard that can be called up using the icon .**

CAUTION: If the path on the boundary of the detail is not assigned with a boundary condition it is automatically considered as an adiabatic boundary.



The screenshot shows the 'Boundary conditions' dialog box with two sections: 'Interior' and 'Exterior'. The 'Interior' section has a table with columns: Name, θ_{ai} [°C], $\phi_{t,u}$ [%], R_{si} [m².K/W], $s_{d,si}$ [m], and Colour. A row for 'Living room' is shown with values 21.0, 55, 0.13, 0.0080, and a green color swatch. The 'Exterior' section has a similar table with columns: Name, θ_{se} [°C], ϕ_{se} [%], R_{se} [m².K/W], $s_{d,se}$ [m], and Colour. A row is shown with values 0.04, 0.0023, and a blue color swatch. Callouts point to various icons: 'Odstranění okrajové podmínky' (trash icon), 'Assignment of boundary condition' (pencil icon), 'Duplication of boundary condition' (down arrow icon), 'Show assistant for interior' (pencil icon), 'Change a path colour' (color swatch), 'Change a path thickness' (double arrow icon), 'Addition if a boundary condition for interior' (pencil icon), 'The possibility to replace the default colors using a color scale based on boundary properties.' (color scale dropdown), 'Show assistant for exterior' (pencil icon), and 'Addition if a boundary condition for exterior' (pencil icon). There are also 'Add more' buttons for both sections.

Using the boundary assignment function, you can directly assign the boundary condition to the path in the geometry input. When you click the icon , you first need to select the model which is the path part of, and then select a specific path within the selected model. **TIP: You can drag and drop the window to any position by clicking on the grey area of the header and then dragging it.** If it is not possible to select a particular model in the assignment, it is probably due to its placement in the lower layer. The order of models can be changed by selecting and editing tool (see chapter 16).

When entering boundary conditions, the fills of the models are automatically turned off for greater clarity. The display of fills when entering boundary conditions can be adjusted using the button / .


For advanced users: Specify the boundary condition for only part of the path


In certain situations, the boundary condition is only required for part of the path. If you already know about it when creating geometry, just add the node at the dividing point of the path. If the path was not split when creating geometry, you can additionally add the node using the Selection and editing tool (see Chapter 16).

20. PICTURE

The menu **Picture** lets you save the current view of the detail input. The captured picture can be used to display in the protocol or it can be saved to the local hard drive.

You can save the current view of the detail by using the **Save current view** button.

Use the icon  to return to the already saved pictures in the input.

Use the icon  to view the created picture. The created pictures have inverted black and white colours for easier use in printed documents. **TIP: You can save the file to the local hard drive by using the context menu by right-clicking the mouse over the picture view.**

21. INTERIOR SURFACE TEMPERATURE FACTOR f_{Rsi}

You can enter the data for the automatic temperature factor evaluation by using the menu **Interior surface temperature factor**. **CAUTION: Before calculating the internal surface temperature factor, make sure that the resistance values are correctly selected for heat transfer in the boundary conditions. Their values for calculating interior surface temperature factor f_{Rsi} and the linear thermal transmittance are different.**

Calculate the temperature factor of interior surfaces is YES / NO option. If YES is selected, another input fields are displayed, and the f_{Rsi} evaluation is displayed in the protocol.

Choose, for which boundary conditions you want to set the requirement by selecting from the drop-down list, where all the boundary conditions for the interior / exterior are displayed. In case of more boundary conditions with the same temperature (eg differing only with the heat transfer resistance value), simply select any condition. The evaluation will then automatically consider all boundary conditions with the same temperature.

Space in which air humidity is permanently treated by air-conditioning is YES / NO option. This option affects the determination of design relative humidity to determine the requirement for the internal surface temperature factor.

Highest allowed relative air humidity is a choice from a drop-down list between 80% relative humidity (mold growth risk) - standard choice for building constructions and 100% (risk of condensation) - a standard choice for wall openings (windows, doors).

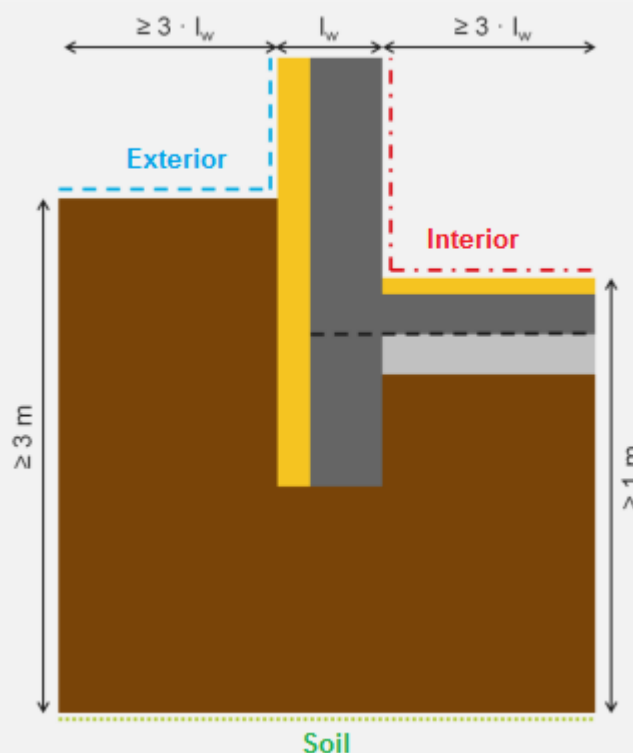
For advanced users: Evaluation of surface temperatures for floor slabs on grade

In the case of floor evaluation, it is necessary to model a block of soil with the following dimensions:

Direction	Distance to the central element
Horizontal distance to the vertical plane, inside the building	At least three times the thickness of the wall
Horizontal distance to the vertical plane, outside the building	At least three times the thickness of the wall
Vertical distance to the horizontal plane below ground level	At least 3 m
Vertical distance to the horizontal plane below the floor (applies only in case when the floor level is more than 2m below ground level)	At least 1 m

*Note: The values given in the table apply only to the calculation of surface temperatures. To determine the linear thermal transmittance, different dimensions need to be selected (see Chapter **Chyba! Nenalezen droj odkazů.**).*

According to the national annex in EN ISO 10211, the temperature in the Czech Republic is 5° C for the horizontal cutting plane through the soil.



22. LINEAR THERMAL TRANSMITTANCE Ψ

Using the **Linear thermal transmittance** menu, you can enter data for automatic evaluation of the Linear thermal transmittance. **CAUTION: Before calculating the linear thermal transmittance, make sure that the resistance values for the heat transfer in the boundary conditions are correctly selected. The values for calculating the interior surface temperature factor and the linear thermal transmittance are different.**

Calculate the linear thermal transmittance is YES / NO option. If YES is selected, an additional input fields are displayed, and the linear thermal transmittance is displayed in the protocol.

The **Detail type** is selected from the drop-down list:

- a) 2 boundary conditions - For details, see chapter 22.1
- b) 3 boundary conditions - For details, see chapter 22.2
- c) Floor slab on grade - For details, see chapter **Chyba! Nenalezen zdroj odkazů.**

Dimension system is a selection from the drop-down list:

- a) Exterior
- b) Interior
- c) Total interior


The choice should correspond to the subsequent use of the results. For calculations of energy demand in the Czech Republic, a system of external dimensions is used as standard. In order to accurately include the effect of linear heat bridges into energy calculations, it is therefore necessary to select the option Exterior.


The **Requirement according to ČSN 73 0540-2** is a selection from the drop-down list:

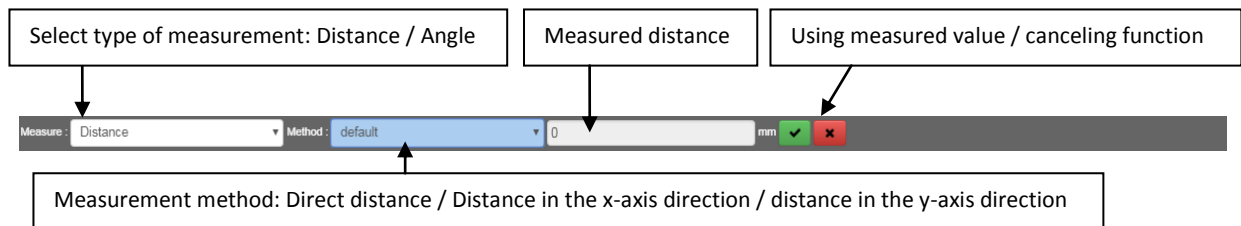
- a) External wall connecting to the next construction except for the opening (window/door)
- b) External wall connecting to the opening (window/door)
- c) Roof connecting to the opening (window/door)
- d) Not to evaluate

22.1. 2 BOUNDARY CONDITIONS

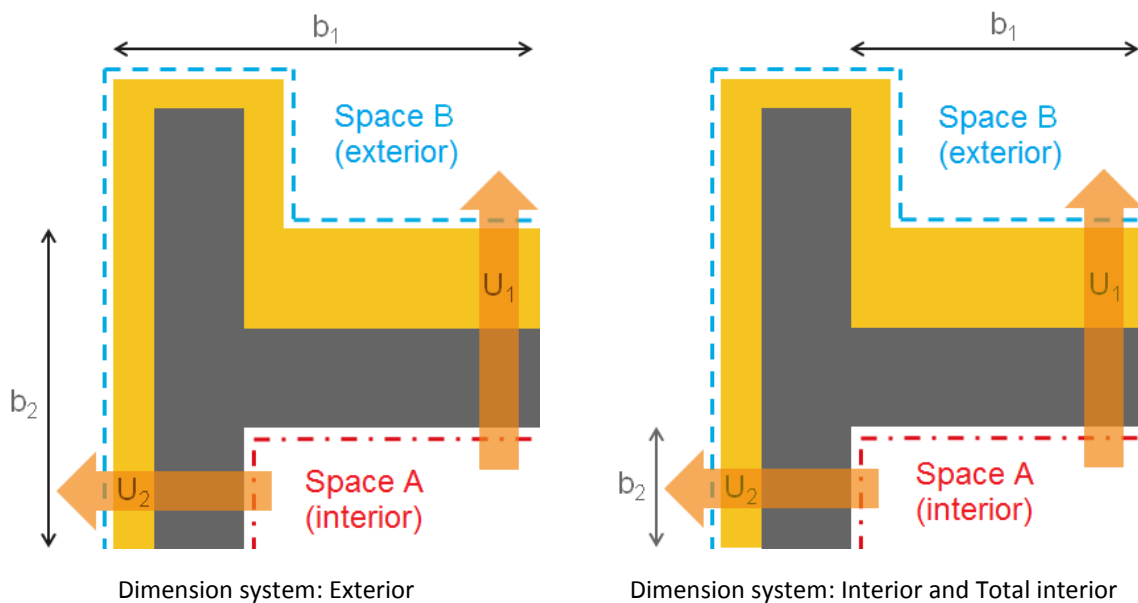
When selecting 2 boundary conditions (the most common situation where only one interior and exterior is in the detail), the following input options are displayed. The parameters are also shown in the figures below for greater clarification.

U-value of construction 1 (2) is the calculated heat transfer coefficient of construction 1 (2). The value of the U-value can be taken from Thermal Analysis 1D using the icon . Clicking on this icon opens a modal window that selects all of the structures you have entered in Thermal Analysis 1D.

Dimension b for construction 1 (2) is the dimension of construction 1 (2) for calculating the linear thermal transmittance. Dimension b should match the selected dimensional system. Click on the icon  to see the measurement bar at the bottom of the screen (see figure below). Distance measurement can be done either by selecting two points or selecting a path.



TIP: If there is only one type of exterior structure in the detail, it is necessary to divide it into two parts with the same U-value so that $b = b_1 + b_2$.



For advanced users: Principle of calculation

The linear thermal transmittance is calculated according to the following equation:

$$\Psi = L_{2D} - b_1 \cdot U_1 - b_2 \cdot U_2$$

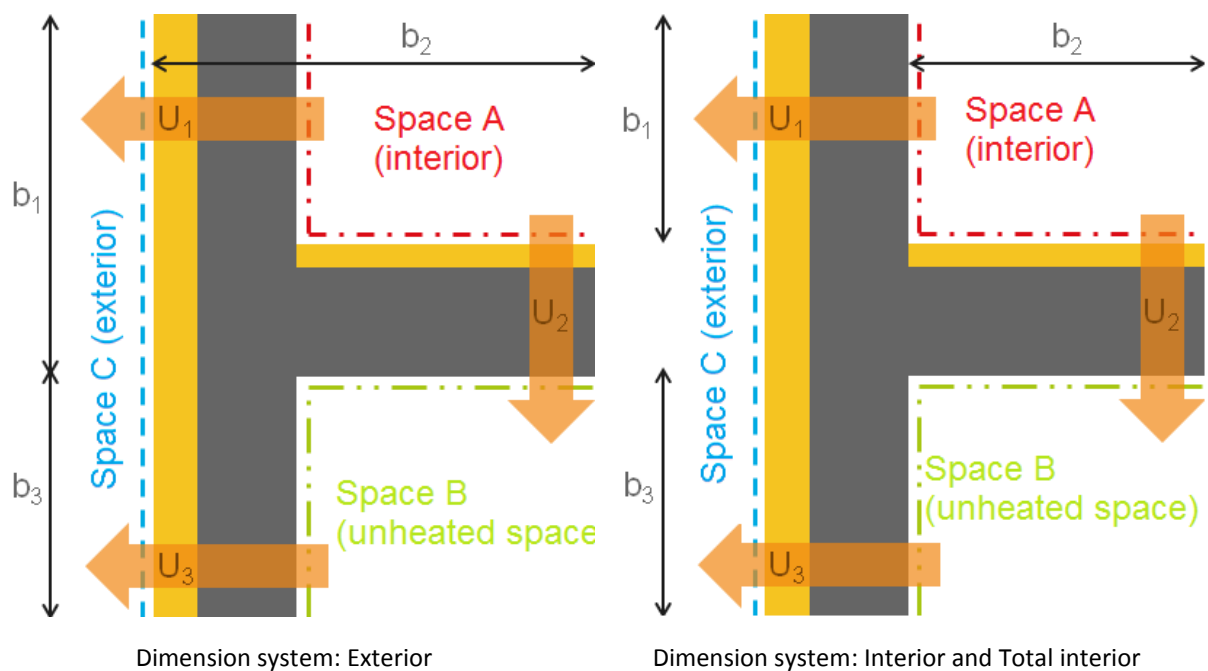
22.2. 3 BOUNDARY CONDITIONS

When selecting 3 boundary conditions, the following input options are displayed. The parameters are also shown in the figures below for greater clarification.

U-value of construction 1 (2, 3) is the calculated heat transfer coefficient of construction 1 (2, 3). The value of the U-value can be taken from Thermal Analysis 1D using the icon ↶.

Dimension b for construction 1 (2, 3) is the dimension of construction 1 (2, 3) for calculating the linear thermal transmittance. Dimension b should match the selected dimensional system. Click on the icon ↻ to use the measurement bar.

Boundary conditions selection serves to identify 3 environments that are present within detail. Typically, the following types are assumed to be standard: interior / unheated space / exterior. In case of more boundary conditions with the same temperature (eg differing only with the heat transfer resistance value), simply select any condition. The evaluation will then automatically consider all boundary conditions with the same temperature.



For advanced users: Principle of calculation

For the assessment of detail with three boundary conditions, a special 3 calculations are performed in addition to the standard calculation to determine the partial flows between environments. The calculation procedure is based on the standard EN ISO 10211. In these calculations, the temperature entered in the boundary conditions is replaced by a temperature of 0°C and 1°C based on the following table. The result is always three values of the linear thermal transmittance for each pair of environments in the input.

Calculation number	Temperature [°C]		
	Space A (interior)	Space B (unheated space)	Space C (exterior)
1	1	0	0
2	0	1	0
3	0	0	1

Note: The final thermal coupling coefficient values L_{2D} from the calculations are numbered according to the calculation number $L_{2D,1}$, $L_{2D,2}$, $L_{2D,3}$.

The next step is to determine the thermal coupling coefficient between the environments:

$$L_{2D,AB} = \frac{L_{2D,1} + L_{2D,2} - L_{2D,3}}{2}$$

$$L_{2D,AC} = \frac{L_{2D,1} + L_{2D,3} - L_{2D,2}}{2}$$

$$L_{2D,BC} = \frac{L_{2D,2} + L_{2D,3} - L_{2D,1}}{2}$$

From thermal coupling coefficient, it is possible to determine the linear thermal transmittance:

$$\Psi_{AB} = L_{2D,AB} - U_2 \cdot b_2$$

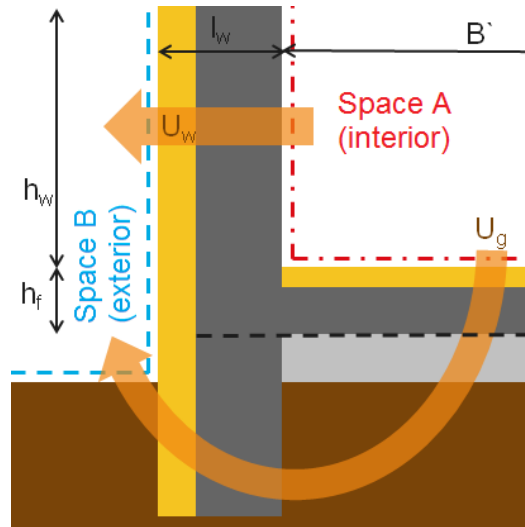
$$\Psi_{AC} = L_{2D,AC} - U_1 \cdot b_1$$

$$\Psi_{BC} = L_{2D,BC} - U_3 \cdot b_3$$

Comparison with the requirements of ČSN 73 0540-2 is automatically performed only for the linear thermal transmittance between the heated space and the exterior.

22.3. FLOOR SLAB ON GRADE

When selecting **Floor slab on grade**, the following input data values are displayed. The parameters are also shown in the figure below for greater clarification.



For all dimension systems

In the case of Floor slab on grade, EN ISO 10211 permits two methods of calculation:

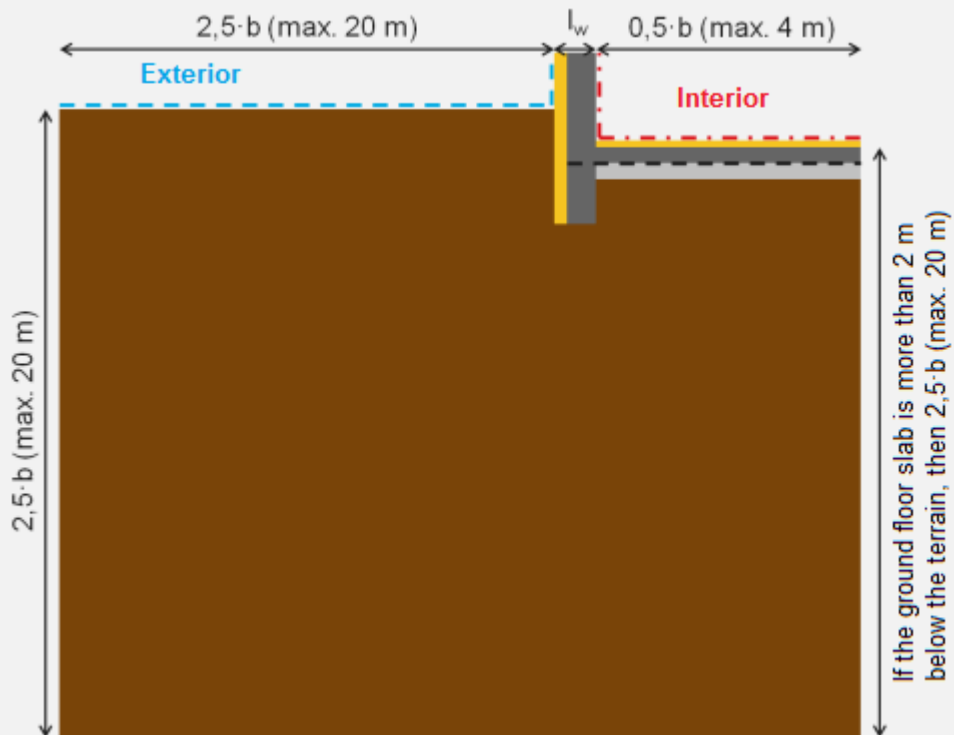
- Simplified procedure according to EN ISO 13370 - details are given in chapter 22.3.1
- Calculation of heat flow via the ground - details are given in chapter 22.3.2

For advanced users: The size of the block of the ground

In the case of floor assessments, it is necessary to model a block of the ground with the following dimensions:

Direction	Distance to the central element
Horizontal distance to the vertical plane, inside the building	0,5 x characteristic floor dimension (max. 4 m)
Horizontal distance to the vertical plane, outside the building	2,5 x characteristic floor dimension (max. 20 m)
Vertical distance to the horizontal plane below ground level	2,5 x characteristic floor dimension (max. 20 m)
Vertical distance to the horizontal plane below the floor (applies only in case when the floor level is more than 2m below ground level)	2,5 x characteristic floor dimension (max. 20 m)

Note: The values given in the table apply only to the calculation of the linear thermal transmittance and surface temperatures.



22.3.1. SIMPLIFIED PROCEDURE ACCORDING TO EN ISO 13370


The simplified procedure does not require the calculation of the second detail to determine the heat flow through the ground. For calculation, however, it is necessary to have a **U-value** of the floor on grade U_g determined according to ČSN EN ISO 13370. **CAUTION: This is not the value determined when calculating the U-value in Thermal Analysis 1D, which is determined according to ČSN EN ISO 6946.**


The **U-value of the wall** U_w in $W/(m^2.K)$ is the calculated heat transfer coefficient of the wall structure. The value of the heat transfer coefficient can be taken from Thermal Analysis 1D using the icon ↶.


The **U-value of the floor according to EN ISO 13370** U_g in $W/(m^2.K)$ is the value of the coefficient of heat transfer, including the influence of the soil determined according to ČSN EN ISO 13370.

For advanced users: Obtaining the U_g value from the EPC protocol

If you have an energy performance certificate for a building in which EN ISO 13370 was used to determine the heat loss via the ground, you can obtain the U_g value by multiplying the specified U-value without the influence of soil (according to ČSN EN ISO 6946) by the value of the correction factor b.

The **dimension h_w** in m is the distance from the floor surface to the cutting plane of the wall. Click on the icon  to use the measurement bar.

The **dimension h_f** in m of the floor structure is the structure thickness considered in the calculation of the U-value. Click on the icon  to use the measurement bar.

The **thickness l_w** in m is the thickness of the wall. Click on the icon  to use the measurement bar.

Dimension B' in m is a characteristic floor dimension according to ČSN EN ISO 13370. **TIP: If the calculation is with floor width of 4 m (ie $B' = 8$ m), the result can be used for any floor of larger dimensions.**

For advanced users: Principle of calculation

The calculation of the linear thermal transmittance is performed for the internal and total internal dimension system according to the following equation:

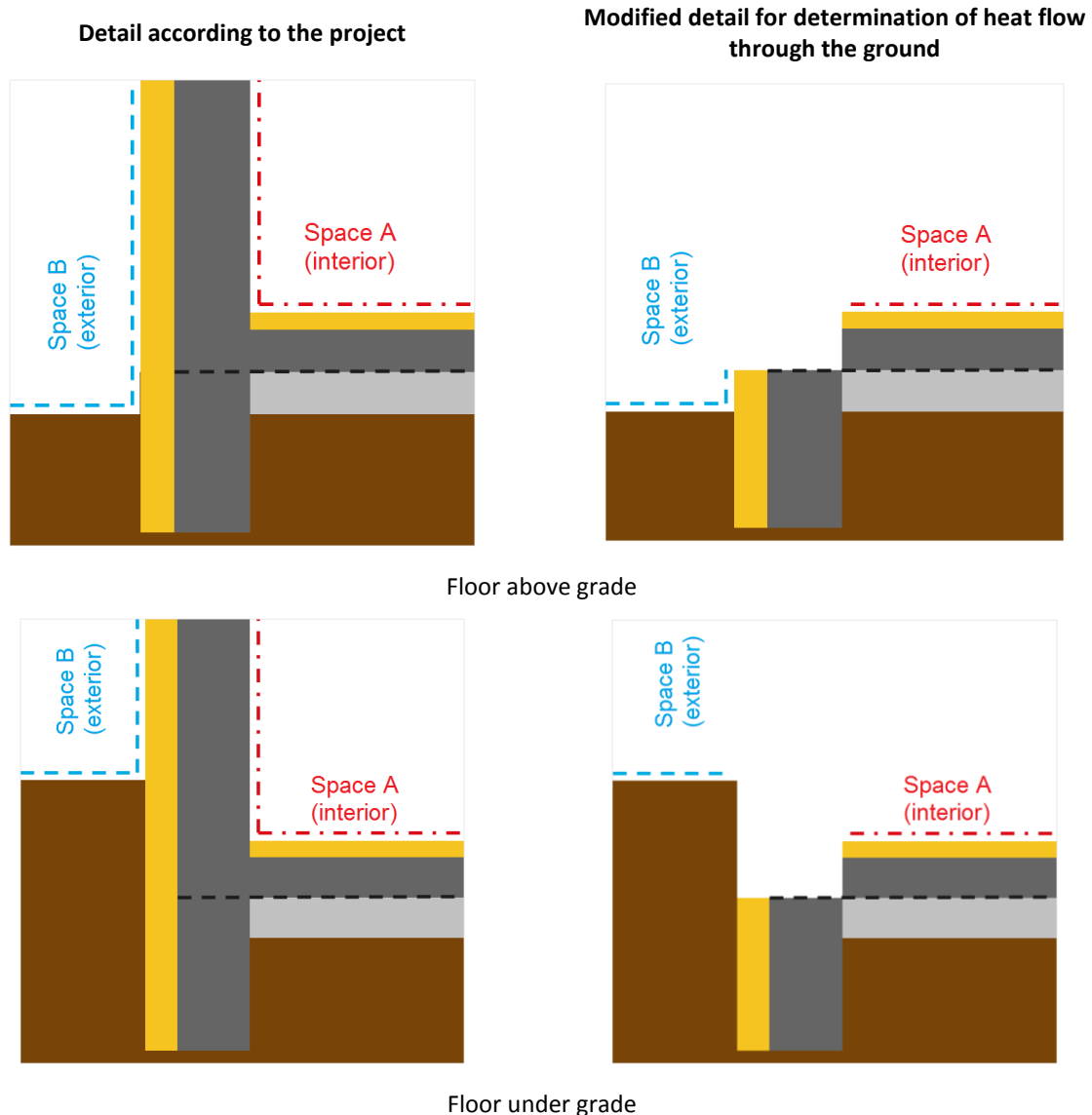
$$\Psi = L_{2D} - h_w \cdot U_w - 0,5 \cdot B' \cdot U_g$$

The calculation of the linear thermal transmittance is performed for the outer dimension system according to the following equation:

$$\Psi = L_{2D} - (h_w + h_f) \cdot U_w - 0,5 \cdot (B' + 2 \cdot l_w) \cdot U_g$$

22.3.2. CALCULATION OF THE HEAT FLOW VIA THE GROUND

A more detailed calculation of the heat flow through the ground requires the creation of 2 details. One detail according to the project and the second detail with an adjustment for the determination of the heat flow through the ground without the influence of the perimeter wall. The calculation of the linear heat transfer calculation is made only for details according to the project. The diagram of creating a second detail for calculating the heat flow through the ground is shown in the following figures.



Note: The schemes are based on ISO 10211:2017

Detail with calculation of heat flow through the ground is a drop-down menu with a list of all details in the DKP file. The detail should meet the requirements of the above mentioned schemes.

The **U-value of the wall U_w** in $W/(m^2.K)$ is the calculated heat transfer coefficient of the wall structure. The value of the U-value can be taken from Thermal Analysis 1D using the icon ↶.

The **dimension h_w** in m is the distance from the floor surface to the cutting plane of the wall. Click on the icon ↷ to use the measurement bar.

The **dimension h_f** in m of the floor structure is the structure thickness considered in the calculation of the U-value. Click on the icon ↷ to use the measurement bar.

The **thickness l_w** in m is the thickness of the wall. Click on the icon ↷ to use the measurement bar.

Dimension B' in m is a characteristic floor dimension according to ČSN EN ISO 13370. **TIP: If the calculation is with floor width of 4 m (ie B' = 8 m), the result can be used for any floor of larger dimensions.**

For advanced users: Principle of calculation

The calculation is made for details according to the project (value L_{2D}) and the heat flow through the ground (value $L_{2D,a}$).


The calculation of the linear thermal transmittance is performed for the internal and total internal dimension system according to the following equation:


$$\Psi = L_{2D} - h_w \cdot U_w - L_{2D,a}$$

The calculation of the linear thermal transmittance is performed for the outer dimension system according to the following equation:

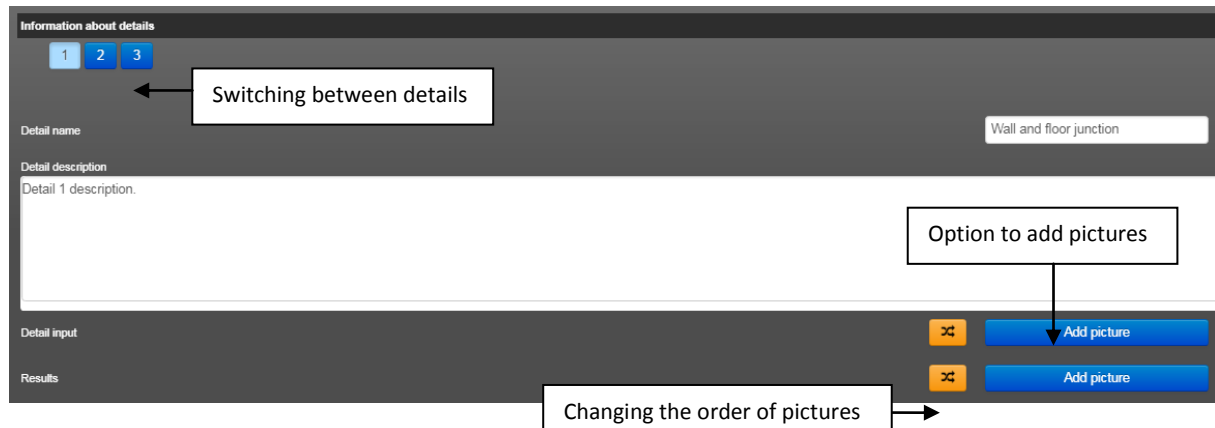
$$\Psi = L_{2D} - (h_w + h_f) \cdot U_w - \frac{0,5 \cdot B' + l_w}{0,5 \cdot B'} \cdot L_{2D,a}$$



23. TEXTS

In the Texts section, you can enter the identification information for the input to the protocol and select the images to display. Completing this section does not affect the calculation. Clicking on the icon  will replace the graphical input with the **Basic information** input form.

If the identification data have already been entered in any of the other DEKSOFT programs, it will be automatically moved to the Thermal Analysis 2D program. It is also possible to use the signature input function (button  in the section **Author identification**). Signature information can be entered in: User menu > Settings > Signature of the author of the document.

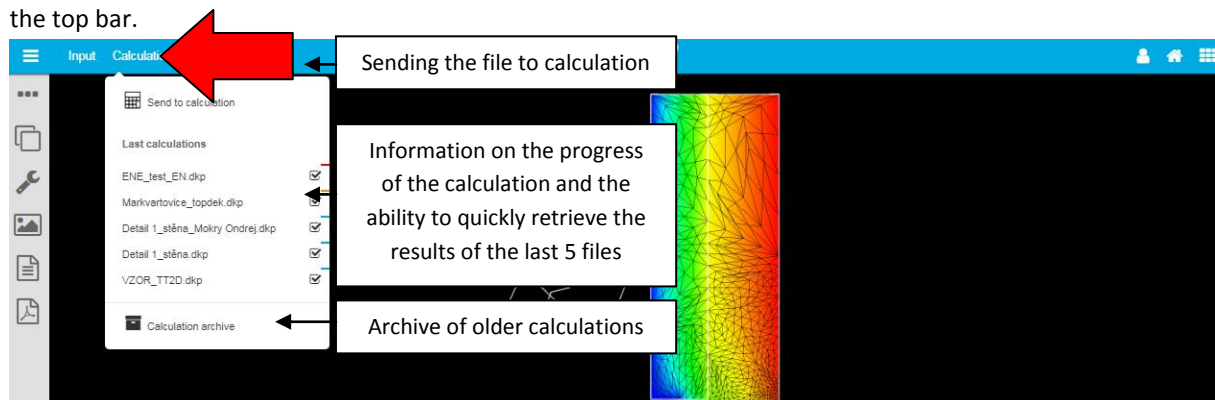
In section **Information about details**, you can edit the detail name, add a more detailed description, and upload the images for display in the protocol.






Clicking the button **Add picture** will display a window with the option to add picture. There are two options: **Load a saved view** (select from saved images within the input) or Upload a custom picture (a selection of images stored on the local hard drive  or on the server .

24. CALCULATION

You can view the options for working with the calculation by moving the mouse or by clicking at Calculation in the top bar.



24.1. CALCULATION START

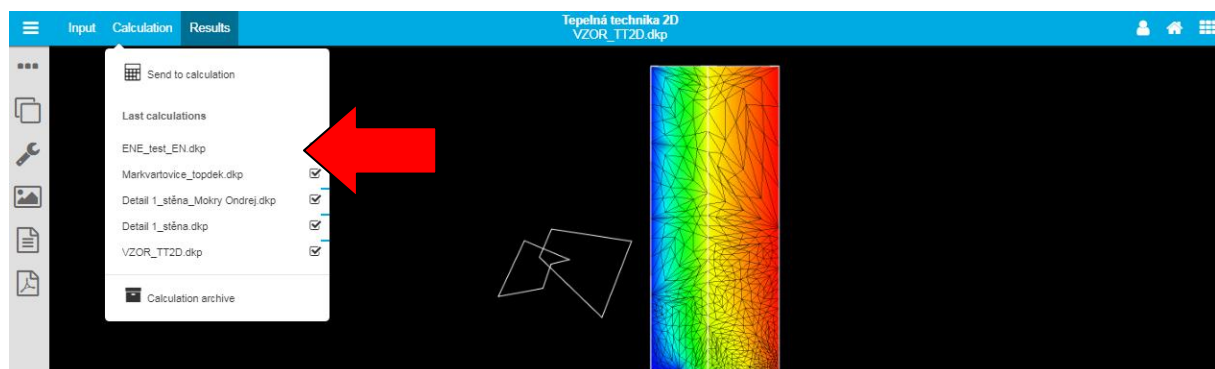
Calculation is started by using the option Send to calculation. After submitting the calculation file, a file name appears in the Last 5 Calculations section, an icon  appears on the right-hand side indicating that the file is pending or send to calculation is in progress. When the calculation is complete, the status icon changes to . If an error has occurred during the calculation, the icon  will appear.

CAUTION: Once you have made a change in the input, it is always necessary to recalculate the file!

In case of larger calculations, the progress bar is active (both for files and the total under section **Calculation** in the top bar). The program, for which the current calculation is performed, is indicated by a coloured rectangle next to the file name. The colour of the rectangle corresponds to the colour of the program.

24.2. LOADING THE RESULTS




Once the calculation has been completed successfully, it is necessary to load the calculated calculation to display the results. **Loading the calculation is done by clicking the file name** in the **Last 5 calculations** section. To load the results of older calculations, you can use the **Calculation archive** option. Calculations that can be loaded for the file in DEKSOFT programs are indicated by a coloured rectangle on the right side. The rectangle colour matches the colour of the program icon.



Alternatively, the results can be loaded by using the button in the information panel in the bottom right corner.



24.3. CALCULATION ARCHIVE

The **Calculation archive** allows you to access all calculated files in a separate modal window. You can load the calculation using the button  or by clicking on the calculation name. Use the icon  to attach a note to a specific version of the calculation. The button  deletes the results.

TIP: In the results modal window, you can sort the calculations by name or date by clicking on the table header.

When you load the calculation, the specific entry for which the calculation was performed is always opened. The calculations that can be loaded for each file in each application are marked with a colour rectangle next to the file name. The rectangle colour matches the application icon colour.

25. RESULTS

There are two main types of outputs available:

- a) Graphical display
- b) Protocol

To switch between different views, use the left navigation bar. The different functions are described in the following chapters.

CAUTION: Sometimes, the following Cautions may appear when loading the results.


If the result files are not generated by the calculation, this Caution will be displayed. The most common cause is choosing a too detailed mesh. Try lower values in Mesh refinement number settings. If the problems persist, send the DKP file to info@deksoft.eu.

For larger file sizes, this Caution will appear before the results are loaded, allowing you to choose whether to really load the results. This Caution is due to the limitation of data retrieval, while large result sets may have longer display times.

If you cancel, you will be able to download the results in the image format instead of the interactive view.

25.1. DETAILS ...

In the section Details, you can switch between the details for which the calculation was performed. If the calculation of the temperature or humidity is not selected for the detail, the detail will not be displayed in the menu.

Click the button  to download the results file in VTK format.

25.2. DATA

In the section Data, you can switch between temperature and humidity distribution results display.

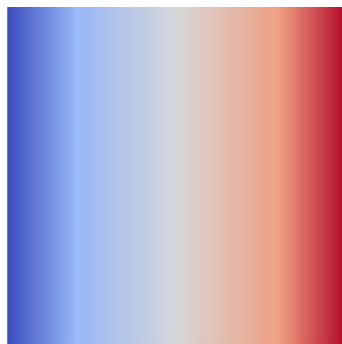
25.3. SETTINGS

In the section Settings, you can influence the display of results. The following options are available.

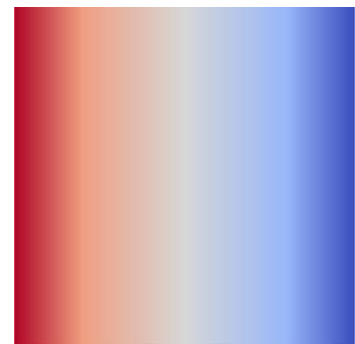
- Show input geometry** allows you to turn off the display of paths from the input. By default, this option is active.
- Show fills** allows you to turn off the display of the colour scale. By default, this option is active.
- Show borders** allows you to display the contours of the detail. By default, this option is inactive.
- Show calculation mesh** allows you to turn off the display of the calculation mesh. By default, this option is active.
- Show calculation mesh in colour** allows you to display the calculation mesh with the colour with key according to the displayed value. By default, this option is inactive.
- Show minimum values of boundary conditions** allows you to display the points in which the minimum value is reached for each of the boundary conditions specified. By default, this option is active.
- Colour range** allows you to choose between different views. The options are shown in the figures below.



Rainbow



Cold - warm



Warm - cold

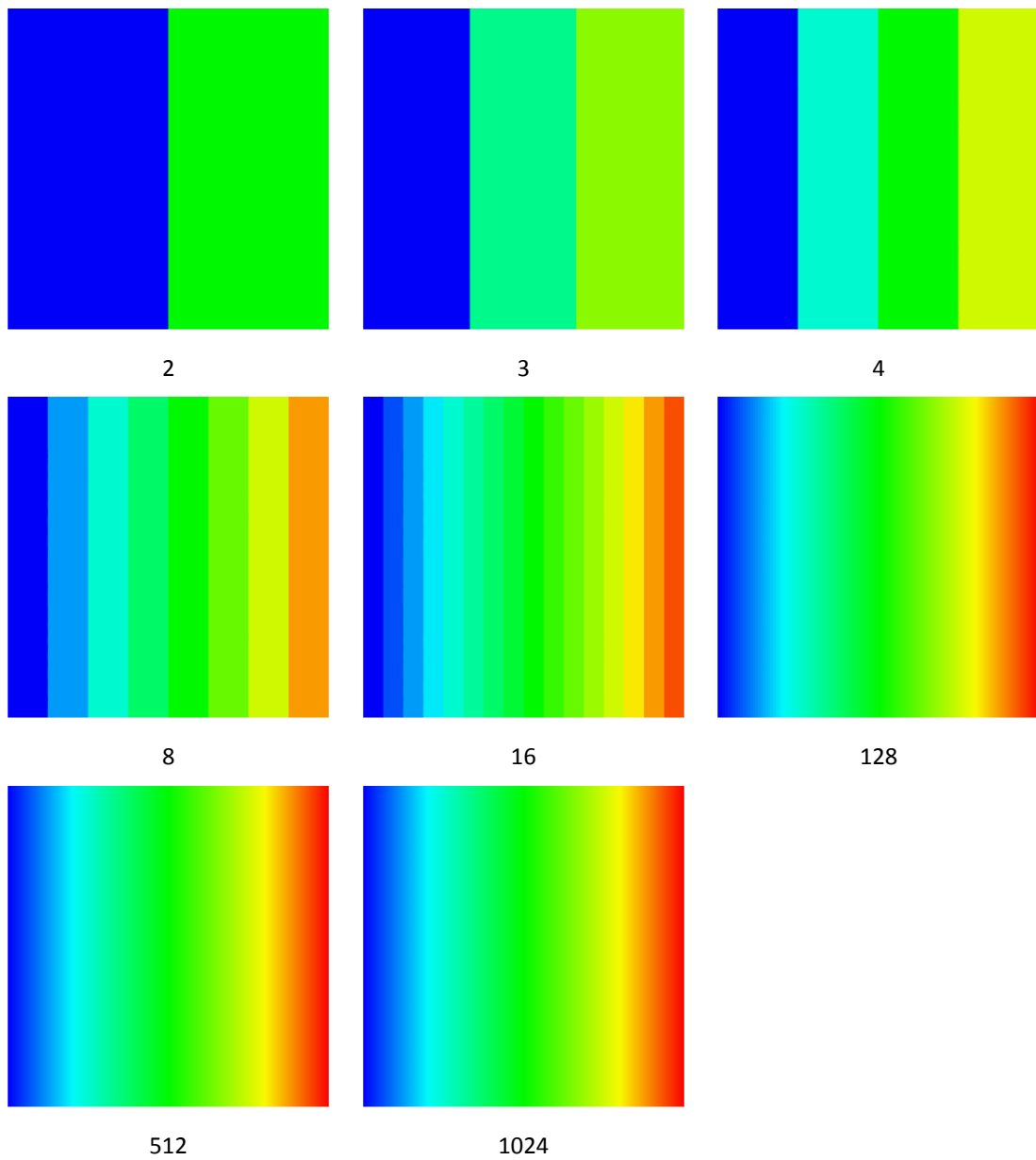


Black body



Shades of grey

- Number of colours** allows you to set the number of colours used in the colour palette. The options are shown in the figures below.





- i) The button **Customize window size** maximizes the detail display in the current field size to display the results.

25.4. IMAGE EXPORT

Image export is used to store outputs in PNG graphics to the hard drive. The saved outputs can be loaded in the input to the text part for display in the protocol.

There are two ways to save images.

- a) **Current view** button  that saves an image corresponding to the current view in the program. During the export, the background colour is automatically inverted for an easier use in printed outputs.
- b) **Generate image** button , allowing you to create images according to user settings, regardless of the current view. Thanks to this export, which allows additional values interpolation, it is possible to remove the imperfections of the standard display (at lower mesh density, "teeth" may appear in the outputs). At the same time, it allows for a larger range of outputs (eg display of isolines). For details, see chapter 25.4.1.

25.4.1. GENERATE IMAGES

The following options are available in the image export section.

Settings

Width x height in px allows you to adjust the size of the generated image.

Background allows you to set the background colour. By default, the white background is set.

Interpolate values before viewing allows for additional interpolation of values for more uniform colour field and contours. By default, this option is active.

Replace the background colour by transparency allows you to generate an image without background. By default, this option is inactive.

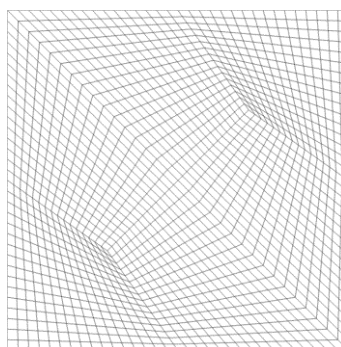
Crop image allows you to crop the image to the size of the detail. By default, this option is active.

Layers

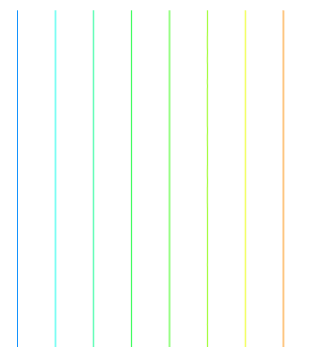
In the layer section you can paste various layers into the image. Layers as shown below are currently available. Layers can be combined with each other. A description of the basic setting options is shown in the figure below.



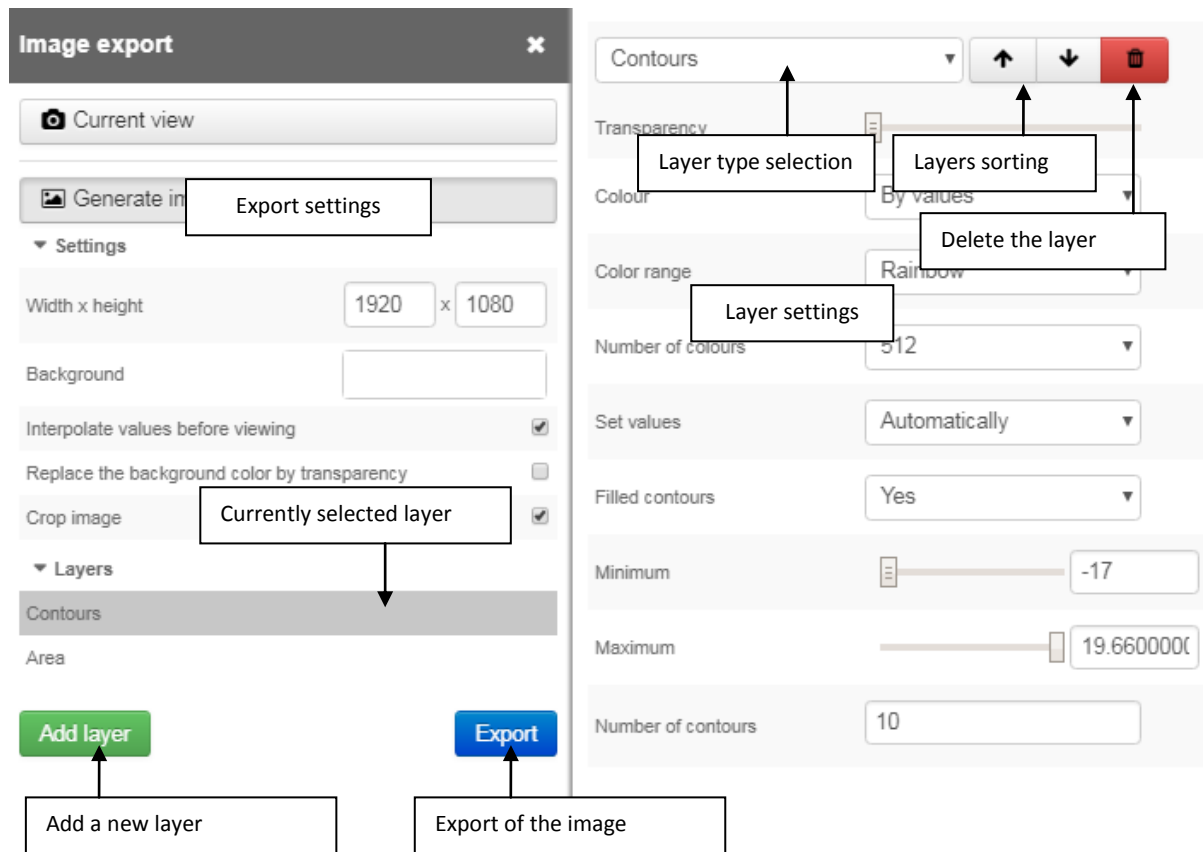
Area



Calculation mesh



Contours



25.5. PROTOCOL

The section **Protocol** allows you to preview the protocol before generating PDF and to access basic numeric results.

25.6. PROTOCOL EXPORT

The section **Protocol export** allows you to export the protocol to PDF.

26. VALIDATION ACCORDING TO EN ISO 10211 AND EN ISO 10077-2

The program Thermal analysis 2D has been successfully tested according to European standards ČSN EN ISO 10211 and ČSN EN ISO 10077-2. The test results are shown in the tables below. If you are interested in more detailed testing information, please contact us at info@deksoft.eu.

In the case of validation according to ČSN EN ISO 10077-2, the thermal conductivity of the air layers was determined by the iterative calculation in Excel based on the results of the calculations in the program Thermal analysis 2D.

Validation testing according to ČSN EN ISO 10211:2009					
Case 1					
Checkpoint	Temperature according to standard	Acceptable deviation	Calculation result	Deviation	Evaluation
-	°C	°C	°C	°C	-
1	15.1	0.1	15.09	0.01	Complies
2	14.7	0.1	14.73	0.03	Complies
3	13.4	0.1	13.38	0.02	Complies
4	9.7	0.1	9.66	0.04	Complies
5	10.8	0.1	10.81	0.01	Complies
6	10.3	0.1	10.32	0.02	Complies
7	8.6	0.1	8.64	0.04	Complies
8	5.3	0.1	5.25	0.05	Complies
9	7.5	0.1	7.47	0.03	Complies
10	7.0	0.1	7.01	0.01	Complies
11	5.6	0.1	5.61	0.01	Complies
12	3.2	0.1	3.19	0.01	Complies
13	5.0	0.1	5.00	0.00	Complies
14	4.7	0.1	4.66	0.04	Complies
15	3.6	0.1	3.64	0.04	Complies
16	2.0	0.1	2.01	0.01	Complies
17	3.2	0.1	3.22	0.02	Complies
18	3.0	0.1	2.99	0.01	Complies
19	2.3	0.1	2.31	0.01	Complies
20	1.3	0.1	1.26	0.04	Complies
21	1.9	0.1	1.91	0.01	Complies
22	1.8	0.1	1.77	0.03	Complies
23	1.4	0.1	1.36	0.04	Complies
24	0.7	0.1	0.74	0.04	Complies
25	0.9	0.1	0.89	0.01	Complies
26	0.8	0.1	0.82	0.02	Complies
27	0.6	0.1	0.63	0.03	Complies
28	0.3	0.1	0.34	0.04	Complies

Case 2					
Checkpoint	Temperature according to standard	Acceptable deviation	Calculation result	Deviation	Evaluation
-	°C	°C	°C	°C	-
A	7.1	0.1	7.06	0.04	Complies
B	0.8	0.1	0.76	0.04	Complies
C	7.9	0.1	7.90	0.00	Complies
D	6.3	0.1	6.27	0.03	Complies
E	0.8	0.1	0.83	0.03	Complies
F	16.4	0.1	16.41	0.01	Complies
G	16.3	0.1	16.33	0.03	Complies
H	16.8	0.1	16.77	0.03	Complies
I	18.3	0.1	18.33	0.03	Complies
Heat flow	W/m	W/m	W/m	W/m	-
Case 2	9.5	0.1	9.49	0.01	Complies

Validation testing according to ČSN EN ISO 10077-2:2012					
Picture	L_{2D} according to the standard	Acceptable deviation	Calculation result L_{2D}	Deviation	Evaluation
-	W/(m.K)	%	W/(m.K)	%	-
D.1	0.550	3	0.552	0.4	Complies
D.2	0.263	3	0.262	0.4	Complies
D.3	0.424	3	0.427	0.7	Complies
D.4	0.346	3	0.346	0.0	Complies
D.5	0.408	3	0.412	1.0	Complies
D.6	0.659	3	0.654	0.8	Complies
D.7	0.285	3	0.282	1.1	Complies
D.8	0.181	3	0.184	1.7	Complies
D.9	0.207	3	0.207	0.0	Complies
D.10	0.481	3	0.486	1.0	Complies

27. PROGRAM UPDATES

Program update process is a great benefit of web application format. You enter the program using a web browser and the program itself runs on high-performance servers. You are always sure to use the latest version of the application without the need for any installation, or to watch the release date of the new version.

28. TROUBLESHOOTING

The program Thermal Analysis 2D has been extensively tested. There should be no more serious problems when working with the program. If problems still occur, we recommend following the below mentioned points.

- a) Do you use a supported web browser?

Supported browsers include: [Mozilla Firefox](#), [Google Chrome](#), [Apple Safari](#) and [Opera](#)

- b) Have you tried to close and reopen the web browser?
- c) Have you tried to restart your computer?
- d) Have you tried to clear the browser cache?
 - For Google Chrome use Ctrl + Shift + Del and *Empty Cache*
 - For Mozilla Firefox using Ctrl + Shift + Del and *Cache*
 - For Opera, use Ctrl + Shift + Del and *Delete cache*

If all questions are answered yes and the problem persists, please send a short description with your file to info@deksoft.eu. We will try to find the cause of the problem as quickly as possible and remove it.

You can always find the latest version of the documents at www.deksoft.eu.

If you did not find the necessary information in the document, please contact us at info@deksoft.eu.