flixo

Version 8

The thermal bridge analysis- and reporting application

flixo
by Infomind GmbH

The thermal bridge analysis- and reporting application

is a tool to analyze the thermal properties of a design or construction, and to document those results.

calculates and gives an overview of temperatures and heat flux densities; it shows temperatures in specific places along with other global thermal properties.

Table of Contents

Part I

1 New

1.1 What's new in version 8

A selection of the new features and improvements of the flixo version 8:

Model and Results

- Psi-Value Tool extended with the kind "1 construction"
- min./max. temperature optionally takes into account the surfaces of air cavities that are open to the outside
- optional display of the scale and depiction of a reference measure at output
- optional display of units in the result object field function
- automatic measurements of the calibration panels within the Uf-Value calculation
- materials additionally possess a standard surface property
- objects can be hidden or shown manually
- additional possibilities for displaying areas and materials
- all texts and names in documents are multilingual. The displayed language of a document can be customized
- databases that are not based on national norms or external data providers are multilingual
- layer assignment of objects can be customized
- support of new algorithms of the norm EN ISO 10077-2:2016
- review of the grid to see whether all areas have been divided
- vectorized and parallelized algorithms to speed up the calculations
- calculation protocol

DXF-Import

- changed rules of import for BLOCK according to AutoCAD

User Interface

- alphabetical or index view for materials in the materials flyout
- new delete tool
- the defining of guidelines now works as in CAD-programs
- edit tool with extended input through keyboard
- stretching of selected objects with the edit tool
- direct adding and deleting of corners with keyboard shortcuts and mouse clicks through the edit tool
- additional possibilities of selection and inversion of the selection
- customized look&feel

1.2 What's new in version 7

Below a selection of the new features and improvements of the flixo version 7:

Model and Results

- Multiple model pages and multiple master pages per document (see context menu in the lower tabs).
- Frame U-Value and Joint U-Value can be calculated either automatically or user-defined.
- The wizard for the Psi-Value calculation allows additional adjustments in the calculation. The reference point can be outside of the construction.
- The calculations of PSI-Value, Uf-Value and UTJ-Value are more flexible.
- Automatic backup and recovery functions of open files after problems.
- Roller Shutter Box U-Value calculation according to EN ISO 10077-2.
- Automatically meet of the accuracy and termination criteria required by the standard EN ISO 10211. Relative heat flow error now also as a termination criterion.
- Wizard to calculate the influence of screws according to prEN ISO 12631
- Optimized calculation of the heat flow.
- You can define for results optionally the number of decimal points or the number of significant digits.
- Graphical objects are taken into account by the Cavity Wizard.
- Advanced options when you assign a template to a document.
- Remove the mesh and the results of the calculation.
- Air cavities according to EN ISO 10077-2:2012 can be calculated also anisotropic; Anisotropic calculated air cavities are shown in the material table.

User Interface

- Reports and documents are protected similar to Excel. Actions and functions can be restricted specifically per file.
- Start page with different links to open recently used files and to create new files.
- Context help flyout with detailed explanations of the use of the active tool.
- The order of the tabs can be customized, colors can be assigned to the tabs.
- Support the ESC button similar to AutoCAD.
- Advanced object capturing features.
- Materials and boundary conditions used in the document are highlighted.
- The order of the components in the properties of the PSI-value with 3 components has been adapted: the components are listed counter-clockwise.
- On the model pages the edit tool and the select, move, scale tool are two separate tools, on the report page the two tools can be grouped together.
- flixo can be used with different profiles. The default template and inactive tools can be set per profile.

Miscellaneous

- Many new field functions: reference to a calculated result, Relative Heat Flow Error, Sum of absolute Heat Flow, Current Program Version, Program Version Document, EN ISO 10077-2 Version.

- Emissivities are listed in the boundary condition and material tables if flixo calculates them.
- The external temperature can be adjusted in the fRSi calculation.
- Database (Materials, Boundary conditions, Components) are split in user-defined parts and parts which are maintained by infomind or third-party. The non-user-defined database can be updated over the Internet.
- Materials of www.baubook.at are integrated into flixo.
- Non-empty database sections can be deleted too.
- Various settings of the dialogs are proposed as default when you next open the dialog.
- Parameter objects can contain groups and graphic objects.

1.3 What's new in version 6

Below a selection of the new features and improvements of the flixo version 6:

Model and Results

- Psi-Value tool with plenty of new application possibilities, an optimized visualization of the Psi-value results with automatic measurements and display of the calculation formulas, graphical representation of the reference point
- Psi Joint Value calculation according to EN ISO 12631
- Spacer or Edge Psi-Value calculation according to EN ISO 10077-2
- Psi-Value calculation with 3 Constructions (e.g. Psi-value of a specific window - wall junction)
- Key values for the calculation of the Psi-value can manually be adjusted in the Properties flyout, the heat flow can be defined as a component key value (e.g. for the Psi-value calculation for base)
- U Joint Value calculation according to EN ISO 12631
- Equivalent U-Value with formulas and help objects comparable to the frame U-value
- The frame dimension calculation for the Frame U-Value and Joint U-Value can be based on the projection of the colder or the warmer side or on the bigger of the two sides
- For Frame U-Value and Joint U-Value objects the reference points for determining the frame dimension can be moved by the Select, Move, Scale tool
- Linear shaped heat sources or sinks
- Screws according to EN ISO 12631: new 3-dimensional equivalent object with an automatic calculation of the equivalent conductance and air cavities, the 3-dimensional equivalent object can also have a rectangular cross section
- Averaged surface properties of cavities can either be adjusted like before directly to the cavity material or newly be calculated by flixo for the 4 main directions based on the current surface properties
- Cavity surface properties can either be set directly with the Boundary Condition tool or automatically assigned by the Cavity EN ISO 10077-2 tool based on the material emissivities table
- Beside the condensation zone flixo can highlight zones where the humidity exceed the critical surface humidity

DXF Import

- If desired materials can be assigned automatically to the domains based on the layer assignment in the DXF file
- It's possible to convert only a partial part of a DXF file (cf. Crop tool)
- Accelerated deletion of lines and polylines
- Optionally DXF layers which are marked as hidden in the DXF file can be ignored, if they should be imported they are initially hidden in flixo too and not visible
- Zoom possibilities for layer preview
- The DXF object 3DFACE gets recognized and imported

User Interface

- Tools to create new objects like flow objects etc. can be configured as Multiple Use tool or as Simple Use tool
- For tools which create new objects, the Select, Move, Scale tool can temporarily be activated by pressing the **Alt** key, after releasing the **Alt** key the previous tool is active again
- The Select, Move, Scale tool inherit additionally all functions of the Edit tool from the previous versions, the Edit Tool is obsolete
- Most of the properties from the selected objects can be adjusted directly in the Properties flyout like e.g. the Result object
- Document flyout in order to search and manage the flixo documents, documents of the version 6 have a preview
- Message bar with comments and contextual commands above the document window, the zoom to conflict tool is replaced by specific message bars
- The commands **Calculate** and **Batch Calculations...** are moved to the menu **File**
- The ***polylines of the set boundary conditions*** can be displayed optionally as a preview, the display occurs time delayed without influencing the continuation of the work
- Cutting regions can be limited to ***selected objects***

Miscellaneous

- The applied boundary conditions on Glazing objects remain also after editing the settings in the dialog window
- Display of the total width of the Glazing object in the dialog window
- The calculation of the construction can also be started on the report pages
- When copying an object the surface temperatures of a Min./Max. Temperature object are copied into the clipboard in ***CSV*** format and can for example be inserted in a table calculation program with the menu command **Edit.Insert Content** and additionally be displayed as a diagram
- Beside the orthogonal dimension also ***coordinate dimension*** can be created by the Dimension tool
- Components can be displayed ***filtered*** similar to the materials in the Component flyout
- ***True lengths of surface lines*** are displayed for the Min./Max. Temperature object and Heat Flow object in the Properties flyout
- The capture spots on defining the boundary conditions can be limited to ***vertices with change of direction***

- Additional key values of surface temperatures can be displayed individually (cf. Min./Max. Temperature style and Temperature style), the places after decimal point can be configured for the fRsi-value if the temperature is not displayed
- The places after decimal point from material and boundary condition entries can be configured in the Table Legends style, optionally instead of the name the use type name can be used in the column legend
- Creating new entries (like materials, boundary conditions etc.) by selecting the entry <New> in the drop-down list
- Several flixo files can be selected and opened with File Open dialog window
- Boundary Conditions can be shown in the report pages with the same line widths like in the model, the set up is done in the Result Object Style dialog window
- DXF files can be dragged into the application window and imported automatically
- Isotherms can be exported as DXF file too

1.4 What's new in version 5

A selection of new features and an update of flixo version 5:

Model and Results

- Streamlines to illustrate thermal bridges
- Measuring tool to measure the distance between two points in the model
- Distinction of objects as graphic objects, which cannot be taken into account when calculated, but which as result objects (e.g. materials) can optionally be displayed
- Simplified rotation of 90°
- Rotate tool and an extended selection mode with an angle constriction to a multiplier of the element angle
- Cut tool with additional numerical margin entries
- By Min./Max. Temperature objects the fRsi factor and humidities can optionally be hidden
- Edges inside an air cavity can optionally be ignored
- Scaling factor during DXF import can be determined
- As a CSV file temperature fields can be exported
- Additional model features: Interior warmer than exterior for correct application of surface temperatures in hot weather climates

User Interface

- New arrangement of elements and Flyout domains to individually adapt the interface
- Toolbox flyout broken down into ***Tools*** list, ***Object*** list, and ***Tool*** properties
- Layers display of the current page in list form
- Styles display in list form with a filter function to facilitate the search of a style
- German, English, French and Italian User interface languages
- Keyword search in the menu
- All open documents can be **saved** with a single command, all open windows can be **closed** with a single command
- DXF import options can be activated shortly before importing

1.5 What's new in version 4.1

Here are some of the new features and improvements in flixo version 4.1:

Models and Results

- Automatic filling of not fully enclosed or not materialized cavities according to EN ISO 10077-2 with a new wizard.
- Defining of parametric objects constructed only by material layers with a new wizard and the application of these objects in the construction
- Defining of glass units with a given U-value with a new wizard and the application of these objects in the construction.
- Comprehensive replacement of materials in all domains by another material as well as comprehensive replacements of boundary conditions of all boundary condition objects by another one.
- Data can be imported and exported from and to other applications by using the Building SVG Interface. Beside the geometry other information describing the construction (e.g. material properties) will be considered while importing. Depending on the program, the export data can be processed further (e.g. a direct presentation of results in an Internet Browser).
- Optionally the fRsi factor including surface and critical room humidity can be shown for any surface point.
- The "greatest density" can be used as a criteria for the automatic assignment of materials from domains with non-defined material properties.

Miscellaneous

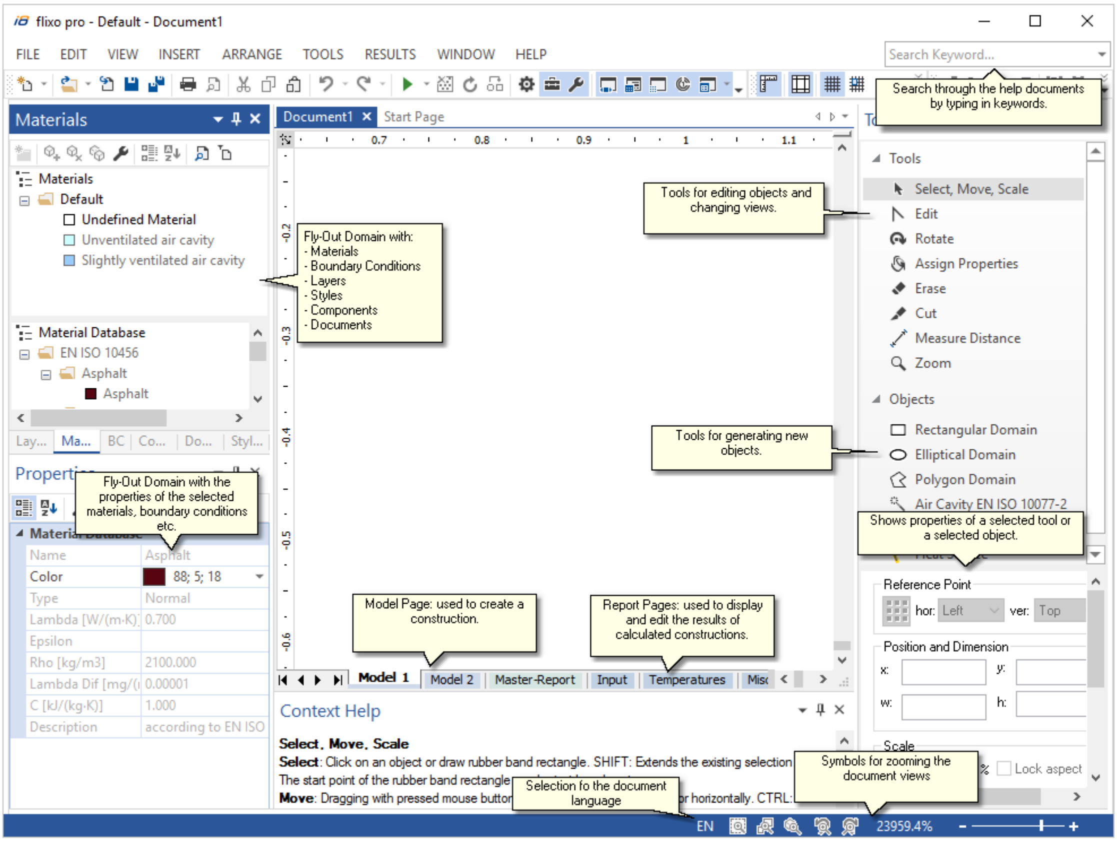
- Library with numerous components (like window, walls, floors etc.) for fast generation of constructions using parametric components.
- Simple duplication of materials, boundary conditions, and style properties.
- By pressing the **ALT** key the Fill tool becomes a **pipette** tool with which material, boundary condition, and style properties can be selected from objects and easily assigned to other objects.
- Numerous new options for customizing the automatic saving, interrupting and switching to the first report page.
- All objects of a specific object type can be selected. E.g. with this feature all boundary conditions can be easily selected and deleted - if needed.
- For guide lines and field functions the corresponding dialogs for the creation will be shown when accessing the property command.

Part II

2 Use

2.1 User Interface

Here is a basic screen shot of flixo with the most important components. You can click on the individual symbols of the figure to jump to the corresponding documentation.



2.2 Program Structure

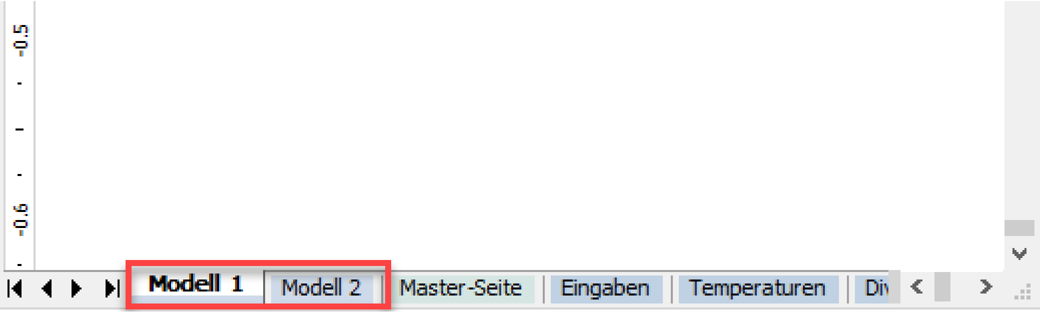
flixo is structured into two different components: ***Models*** and ***Reports***.

Models

In the model part the physical construction is defined. Material domains and boundary conditions are the most important objects you will be working within this input component. Constructions are generally made from different material domains and boundary conditions possessing different physical characteristics.

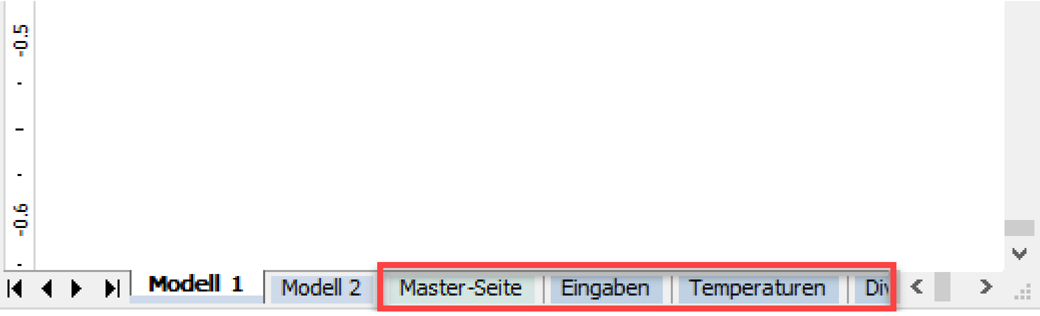
You can insert several models into one file or delete existing model pages (see context menu of the tabs).

Material domains can be created, changed and moved. They can even overlap. Essential for the future analysis of these domains is the visibility of such domains: a domain that is covered by another domain will not be included in the calculation.



Reports

The report component lets you perform various thermal queries on the model and allows you to create a report based on those results.



2.3 Workflow

Typicalflixo work flow is divided into the following parts:

- In the model pages, the constructions geometry with all its physical properties is entered.
- Then the construction is analysed and calculated.
- Finally, the results are evaluated and a report is complied in order to present the results as desired.

In every part, similar basic functions will be implemented like the determination of layers, materials, font styles, etc.

2.4 Defining the Construction

You can enter the geometry of the construction either directly in flixo or you can import it from CAD products and complete it on the model page afterwards.

To enter the construction, use the tools and objects on the Model page. You can switch to the Model page by clicking on the corresponding tab just below the work area

.

With the following tools existing objects can be edited:

- Select, Move, Scale
- Edit
- Rotate
- Assign Properties
- Delete
- Cut
- Measure Distance
- Zoom

With the following tools new objects can be created:

- Rectangular Domain
- Elliptical Domain
- Polygon Domain
- Air Cavity EN ISO 10077-2, frame , pro
- Boundary Condition
- Heat Source, pro

The geometry of the construction can also directly be imported from CAD products and adjusted in the model page.

Functions like logical operations, Align object, the use of guidelines and grids facilitate the data entry. If the construction is more complex and is entered on multiple layers or if it needs to be defined with previously designed elements, it would best you get to know the Layers flyout and Component flyout before hand.

For a complete definition of the construction, you must enter the materials and boundary conditions. The materials assignment can be done with the Assign Properties tool or with the Boundary Condition tool. Alternatively, materials and boundary conditions can be assigned by dragging them from the materials database or table, or respectively from the boundary conditions database or table.

A step-by-step explanation of these techniques can be found in tutorial 1 and tutorial 2, and the import of CAD product constructions can be found in tutorial 3.

2.5 Calculate

In the menu **Tools.Options,** and in the category **Document>Calculate**, the settings of the calculation parameters can be adjusted. Subsequently, the calculation of the temperature distribution in the construction can be started with the menu command **File.Calculate** ( ).

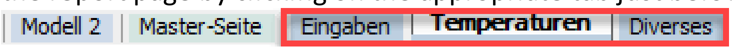
Model calculation is explained in tutorial 1.

The progress of the computational process is displayed in a Status window, in which the calculation can also be interrupted. After ending the process, the window has to be closed (depending on the general program settings cf. menu **Tools.Options**, category **Application>General**) by clicking on OK.

The next step then is to prepare the report.

2.6 Preparing a Report

After you have defined a physical model with materials and boundary conditions and the calculation is complete, you can generate a report based on the analysis. You can switch to the report page by clicking on the appropriate tab just below the work area:



All reports are based on a template, in which the general appearance of the reports can be determined based on standardized report pages. The creation of such templates are explained in tutorial 6.

There are two types of report pages: 1. Master Reports; 2. normal pages:

- On the ***Master pages*** you can insert objects that should appear on all normal pages.
- On the normal ***Report pages*** (Report 1, Report 2, ...) you generate specific analyses.

With the Result Object tool ( ) you can create individual result objects. You can display various object properties once you have selected them with the Select, Move, Scale tool ( ):

- Isotherms
- Temperature fields
- Stream Lines
- Heat flux fields
- Uf-value, if the object is a window frame according to EN ISO 10077-2
- Usb-value, if the object is a roller shutter box according to EN ISO 10077-2
- U joint value according to prEN ISO 12631
- Legends of the materials and boundary conditions used in the model
- Materials and boundary conditions used in the model
- Graphical objects

The visible view of the result object can be adjusted with the Crop tool ( ) in order to display, for example, only the critical domain.

The choice of the visible object properties can be done using the Properties Flyout, the context menu (right click), the Results menu or with the Results Object properties dialog window, which can be activated by double clicking on the result object.

Aside from the object properties, you can calculate and display different results with the results tools:

- Temperature tool
- Min./Max. Temperature tool
- Heat Flux tool
- Heat Flow tool
- U-Value tool (Parallel Layers, Equivalent U-Value, Frame Uf-Value, UTJ Joint, Roller Shutter Box U-Value)
- Psi-Value tool (1 Component, 2 Components, 3 Components, Edge/Spacer)

You can activate the results tools from the Toolbox flyout or through the Results menu.

To display additional information, and to prepare and arrange reports, the following tools are at your disposal:

- Dimension tool
- Information tool
- Line tool
- Polyline tool
- Rectangle tool
- Ellipse tool
- Text tool

Activating the arrangement tools can either be done from the Toolbox flyout or via the Insert menu. With the menu command **Insert.Picture...** you can insert pictures into the report. With the menu command **Insert.Field...** you can insert fields for page numbers, current saved date, etc., which will be automatically refreshed. With the menu command **Insert.Object**, you can insert text and spreadsheet objects.

The properties of the tools can be defined in the options dialog window (cf. **Tools.Options...** or category **Application>Tools**). The properties of the generated results (i.e. local temperatures) can be determined in the Properties flyout and the graphic properties in the Styles dialog window. The result styles can be found in the Styles flyout.

After finishing a report, you can either print the report or display it as a preview before printing. You can find those commands under the File menu.

2.7 Settings

You can change the appearance of flixo to suit your needs:

User Interface

- ***Flyout domains*** (i.e. Materials, Boundary Conditions, Toolbox, Properties, Context Help) can be positioned anywhere on the application window and they can be grouped together. Flyouts can always be displayed, they can always be hidden, or they can appear as soon as the mouse cursor is moved over the tab.
- The ***toolbars*** can be arranged, displayed, and hidden as desired. Additional commands can be added to or removed from the toolbar with the menu command **Tools.Customize**.

Options

Additional application specific settings such as units, calculation priorities, tool characteristics, database paths can be adjusted in the Options dialog window (cf. menu command **Tools.Options**, category **Application** or icon in the standard toolbar).

These settings will automatically be loaded and applied the next time flixo is started.

Other **document specific** settings such as numbers of visible decimal places, font size, calculation precision etc. apply for the current document. These settings will be adapted in the template on which the document is based. These settings can be adjusted in the following manner:

- In the Options dialog window , category Document.
- In the Styles dialog window, for example activated from the Styles flyout, the appearance and the visible precision of the individual results can be configurated.

The creation of a template is explained in tutorial 6.

Object specific properties refer to an object like local temperature object, Psi-value object, etc. You can adjust these settings as follows:

- In the Properties flyout.
- In the Object Properties dialog window.
- In the Style dialog windows. For example, in the Style flyout you can define the appearance and the precision of the results.

Coordinate System

The coordinate system is a right-handed coordinate system. The units as well as the origin can be adjusted.

The **coordinate origin** can be adjusted by clicking on the icon in the upper left hand corner of both rulers and by moving the mouse cursor in the direction of the new origin. At this point, the mouse cursor symbolizes the zero point (neutral point) of the coordinate system, from which the blue lines make up the axes of the system. As soon as the you release the mouse button the zero point is moved (note the adjusted settings in the rulers).

Part III

3 Tutorials

The following tutorials will help you to get to know our product. In each tutorial, a file is generated in individual steps, whereby the results are available as flixo files. You can either perform the procedures yourself, or just read the text, watch the tutorial film, and examine the results. The individual steps are illustrated with help of short films.

The tutorials have been generated with flixo version 7. With version 8 new functions have been added and the look of flixo has been customized. However, the tutorials can be performed without restrictions using version 8.

Your browser settings must allow the showing of Flash films in order to view them.

Basic Techniques

This tutorial illustrates the entry of a simple model with flixo, the definition of materials, the assignment of boundary conditions, and the generation of a report.

Advanced Techniques

After having acquired the basic techniques, this tutorial explains the entry of more complex constructions. This example explains how to generate a window model from individual components.

DXF Import

energy plus, frame, pro

In this tutorial, another flixo application is explained. This application allows you to use documents, which have been generated with CAD software (e.g. AutoCAD).

Window Frame Analysis and Wizards

frame, pro

In this tutorial, the different wizards, which facilitate data entry for the materials table and database are explained. Furthermore, the modeling and analysis of window frames according to European norms is described.

Psi-Value and U-Value calculation

In this tutorial different Psi-values (Psi-value with 2 components, Psi-value with 3 components and edge Psi-value) as well U-values (U joint value according to EN ISO 12631, equivalent U-value, U-value for constructions with two layers and roller shutterbox U-value) are calculated.

Document Templates

In this tutorial, the creation and arrangement of document templates is illustrated. With document templates you can easily change the appearance and characteristics of documents based on the template. If, for example, your company logo should appear on all report pages, then a template containing this logo can be created.

Parametric Objects

energy plus, frame, pro

In this tutorial, you will adjust the lengths of parametric edges, so that the component can exactly be inserted into a new situation. In a second step, the tutorial illustrates how you can define edges as a parameter of complex objects.

Screws according to EN ISO 12631

pro

This tutorial illustrates the definition of a periodically existing screw as an equivalent 3-dimensional object. Important and relevant points for the calculation are emphasized.

3.1 Basic Techniques

In this introductory tutorial, you will create a simple flixo document directly within the flixo editor to analyze a wall/floor junction. First, you will define the necessary settings; next, enter the geometry of the model. Then, you will define the materials and the boundary conditions. Lastly, you will start calculating the thermal characteristics, and then display the results in a report.

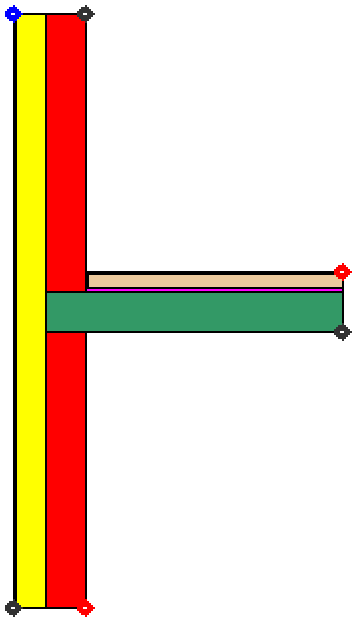


Figure 1: Model of a wall/floor junction

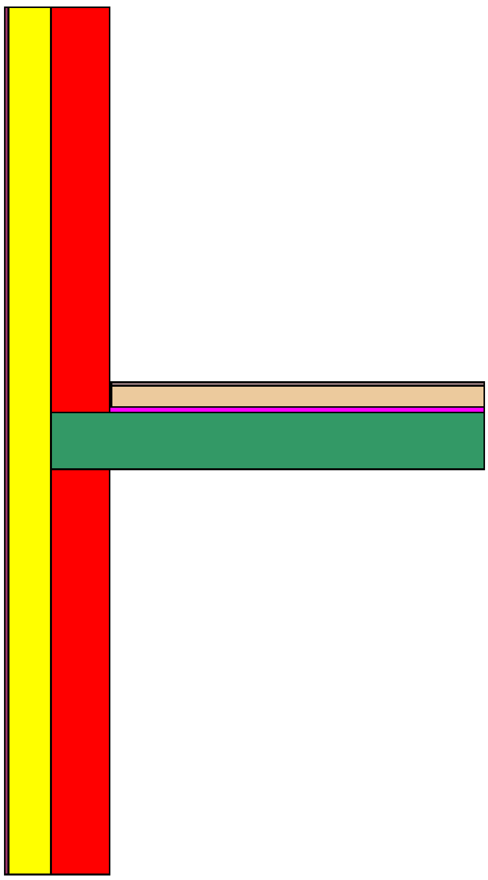
⊟ Content

Entering Model Geometry
Defining Materials
Defining Boundary Conditions
Running a Calculation and Displaying the Results in a Report

Start flixo if you have not already done so, and begin by defining the Program and Document Settings.

3.1.1 Entering model geometry

The physical model comprises individual domains, which together are delineated by a border. The construction in our tutorial is exclusively comprised of rectangular domains. Generally, however the domains can take on any form. Next, you will enter the geometry of the construction step by step. The final result of the input can be seen in the figure below, which indicates the names of the layers as well as the materials used in the individual layers.



Wall Construction:

- Exterior stucco (Plaster, lime, sand L=0.800)

- Insulation (Insulation L=0.040)

- Interior wall (Brick L=0.440)

Floor Construction:

- Wood flooring (Timber 700 kg/m3 L=0.180)

- Cement screed (L=1.400)

- Wall sound insulation stripping (Insulation L=0.040)

- Footfall sound insulation (Insulation L=0.040)

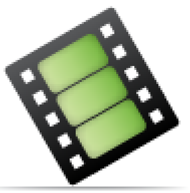
- Reinforced concrete ceiling (Reinforced concrete (with 1% steel) L=2.300)

The geometry of the physical model is entered with both the mouse and the keyboard.

Hints

- In dialog windows, you can skip to the next entry field by using the **Tab** key. If the **Shift** and the **Tab** key are used simultaneously, then you can skip back a field.
- While entering the model geometry, you can undo an action if needed (like other actions as well) by clicking on the icon in the standard toolbar, by using the menu command **Edit.Undo** or by using the shortcut **Ctrl+Z**

Click on the arrows in the graphic below to navigate in tutorial 1.



Summary

- While entering a Rectangular Domain with the keyboard, the x/y coordinates correspond to the **reference point**. The position of the reference point is defined with the reference point button (position of the red square).
- Objects can also be copied with the mouse by activating the Select, Move, Scale tool and by holding the **Ctrl key** down while moving the object. First release the mouse button and then the Ctrl key.
- While **moving** or **copying** objects with the mouse, the **reference point** is used to "Snap to object", "Snap to Grid", "Snap to Guidelines". While selecting, the reference point symbolizes the closest corner and is shown as a small circle.
- While **adjusting the size** of an object with the keyboard (see Select, Move, Scale tool), the reference point determines the point of the circumscribing rectangle, which is not moved in the adjustment.
- The order and the **visibility** of the objects are taken into account in the calculation. The order of the selected domain can be adjusted with the commands in the **Arrange** menu.
- A rectangular domain is defined with two mouse clicks of the Rectangular Domain tool . The corner points are captured, if the snap functions (Grid , Guidelines and Object , see menu Arrange) are activated.
- The **origin of the coordinate system** can be adjusted by clicking on the icon in the upper left hand corner of both rulers and by moving the mouse in direction of the new origin and then releasing.

To continue tutorial 1 you can click here or use the navigation buttons on the top.

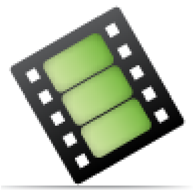
3.1.2 Defining materials

Next, we are going to assign materials to the individual domains of the construction. The materials of the wall/floor junction are listed in the table below (for simplification purposes only one type of sound insulation will be used):

Material	Category	Subcategory	Lambda value [W/mK]
Plaster, lime, sand	EN ISO 10456	Plasters and renders	0.800
Brick	---	---	0.440
Insulation	---	---	0.040
Timber 700 kg/m3	EN ISO 10456	Timber	0.180
Cement screed	---	---	1.400
Reinforced concrete (with 1% steel)	EN ISO 10456	Concrete	2.300

In the subsequent film, materials from other standards have been partly used. Also, the colors of the materials used can differ from those in your version of the program.

Click on the arrows in the graphic below to navigate in tutorial 1.



Summary

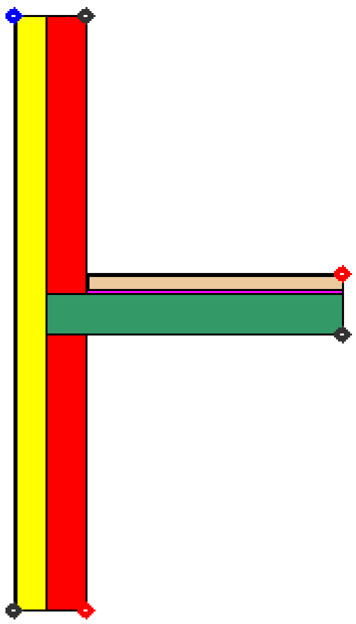
- The materials are arranged in the Materials flyout. The materials from the database can be found in the lower part, and the materials present in the document in the upper part of the flyout.
- You can assign materials with either the **Drag&Drop** function or with the Assign Properties tool.

- With the Assign Properties tool, the ***selected material from the upper materials list*** is assigned to the construction. Independently from the active tool, you can assign a material from either the materials database or from the list of materials present in the document with the **Drag&Drop** function.
- In the material database you can also search for materials (see Materials flyout toolbar). You can choose to either have the database filtered or not filtered (see Materials flyout toolbar).
- New materials can either be added to the upper materials list or to the materials database, if the database isn't write protected (see Materials flyout toolbar).

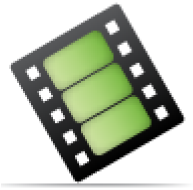
To continue tutorial 1 you can click here or use the navigation buttons on the top.

3.1.3 Defining boundary conditions

When assigning boundary conditions, you only define the starting point of the boundary condition. This boundary condition is valid until the starting point of the next boundary condition. On an exterior surface boundary conditions run **counterclockwise** (for ***interior boundaries***, boundary starting points run clockwise, e.g. hollow objects like chimney, floor heating systems, etc.). In the next step, you will assign 6 boundary conditions. In the figure below, you can see the assigned boundary conditions:



Click on the arrows in the graphic below to navigate in tutorial 1.



Summary

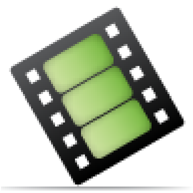
- Boundary conditions are arranged in the Boundary Condition flyout . In the lower part of the flyout is the boundary condition database, while in the upper part, the boundary conditions present in the document can be found.
- New boundary condition objects are generated with the Boundary Condition tool . Only the **starting point** of a BC is defined. On an exterior surface these BC run **counterclockwise** to the next BC starting point.
- The active boundary condition can either be chosen from the list containing all present boundary conditions in the document or from the boundary condition tool properties.
- With the option **Only Vertices with change of direction** in the boundary condition tool properties you can limit the possible start points to surface vertices with change of direction.
- If the according option is selected in the Option dialog window, then the polylines of the set boundary condition are shown. The display occurs time-delayed, depending on the complexity of the construction and on the used hardware. You still can continue your work without any restriction.

To continue tutorial 1 you can click here or use the navigation buttons on the top.

3.1.4 Starting the calculation and displaying the results in a report

Before you can display the results in a report and print them, the construction needs to be calculated.

Click on the arrows in the graphic below to navigate in tutorial 1.



Summary

- The calculation can be started with the command **Calculate** from the menu **File** or by clicking on the icon in the standard toolbar. Depending on the selected options, the first report page is activated after a successful calculation.
- The reports are based on a template (see tutorial 6). Both **General results** (such as Isotherms, Legends) and **object specific** results exist. General results for the selected

result object can be predefined in a template with a menu command (see Results), with the result object context menu or with the Properties flyout . Object specific results are prompted with specialized tools.

- Results are prompted and displayed on a result object (see Result Object tool). Result objects can also be cropped (see Crop tool) .
- Report pages can be inserted and deleted.
- The appearance properties of the results are defined with styles, which are arranged in the Styles flyout . Styles can be assigned with the **Drag&Drop** function. Styles can be adjusted as well.

3.2 Advanced Techniques

In the first introductory tutorial, Basic Techniques, you have learned how to create models with rectangular domains. In this tutorial, you will learn a few new techniques: how to create polygon material domains and components, as well as how to set guidelines.

In figure 1, a window frame is illustrated. This construction will be generated step by step.

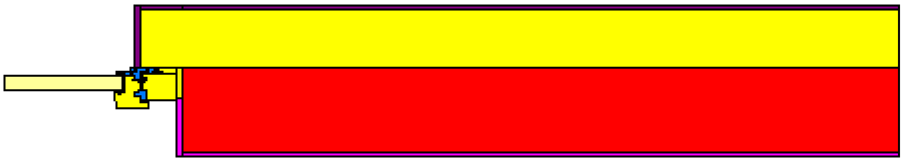


Figure 1

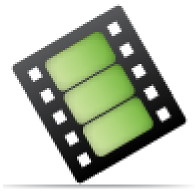
Content

Using components, rotating and mirroring
Extending construction
Grouping, creating own components
Calculating variants, customizing reports, detailed analysis

3.2.1 Using components, rotating and mirroring

The construction of this example consists mainly of 2 parts which are interconnected: window and wall. It's possible to save often used elements as components in flixo and to reuse them.

Click on the arrows in the graphic below to navigate in tutorial 2.



Summary

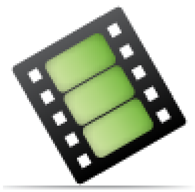
- Components, with the ***Drag&Drop*** function, can be moved and inserted into the preview of the Components flyout as well as into the application window.
- Selected elements can be rotated 90° (respectively)or mirrored(respectively) by clicking on the according icon in the **Arrange** toolbar or by selecting the according commands in the Arrange menu.

To continue the tutorial you can click here or use the navigation buttons on the top.

3.2.2 Extending construction

The construction can be completed and adjusted in different ways. You can position elements exactly, enlarge regains and create missing objects.

Click on the arrows in the graphic below to navigate in tutorial 2.



Summary

- The **origin of the coordinate system** can be adjusted by clicking on the icon in the upper left hand corner of both rulers, by dragging the mouse in direction of the new origin and by then releasing.
- While adjusting the position of an object with the keyboard (cf. Select, Move, Scale tool) the x- and y- coordinates indicate the position of the reference point .
- The order and visibility of objects are taken into account when calculated. The order of the selected domain can be adjusted in the menu Arrange or by clicking on the corresponding icon (, , respectively) in the Arrange toolbar .
- A rectangular domain is defined with two mouse clicks of the Rectangular Domain tool. The corner points are captured with active snap functions (Grid , Guidelines and Objects , cf. menu Arrange). The inputs can be entered partially with the mouse and partially with the keyboard.
- The tool flyout is activated by first pressing the **TAB key**. You can switch the focus step by step to the next control field of the tool flyout by further pressing the TAB key.

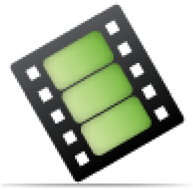
- You can activate the Select, Move, Scale tool temporarily by held down **Alt key**. After releasing the Alt key the previous tool is active again.

To continue the tutorial you can click here or use the navigation buttons on the top.

3.2.3 Grouping, creating own components

You can create a new component from an existing file or from selected elements.

Click on the arrows in the graphic below to navigate in tutorial 2.



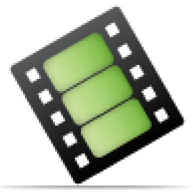
Summary

- Selected objects are grouped with the menu command Arrange.Group .
- You can create a component from the selected objects by clicking within the selected elements and by moving them with the ***Drag&Drop*** function into the preview of the Components flyout .
- Name and file path of the new components can be defined in a special dialog window. Please consider that you need ***write permissions*** on the corresponding directory for saving the components.

3.2.4 Calculating variants, customizing reports, detailed analysis

In this episode we will create and calculate variants. We will customize the layout of the report and finally demonstrate the application of all result tools.

Click on the arrows in the graphic below to navigate in tutorial 2.



Summary

- The reports are based on a template (see tutorial 6). Both **General results** (such as Isotherms, Legends) and **object specific** results exist. General results for the selected result object can be predefined in a template with a menu command (see Results), with the result object context menu or with the Properties flyout . Object specific results are prompted with specialized tools.

- Results are prompted and displayed on a result object (see Result Object tool). Result objects can also be cropped (see Crop tool) .
- Report pages can be inserted and deleted.
- The appearance properties of the results are defined with styles, which are arranged in the Styles flyout . Styles can be assigned with the **Drag&Drop** function. Styles can be adjusted as well.

3.3 DXF Import

energy plus, frame, pro

The method by which existing geometry data from CAD programs (e.g. AutoCAD) can be used again is explained in this tutorial.

The goal of this tutorial is to import a cross section of a window construction or to import a cross section of a roof-wall junction that has been created with AutoCAD.

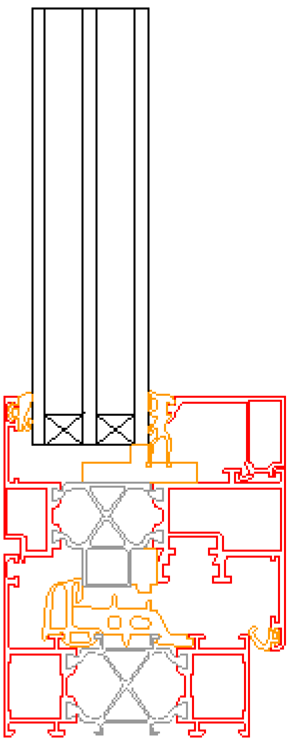


Figure 1: Window frame

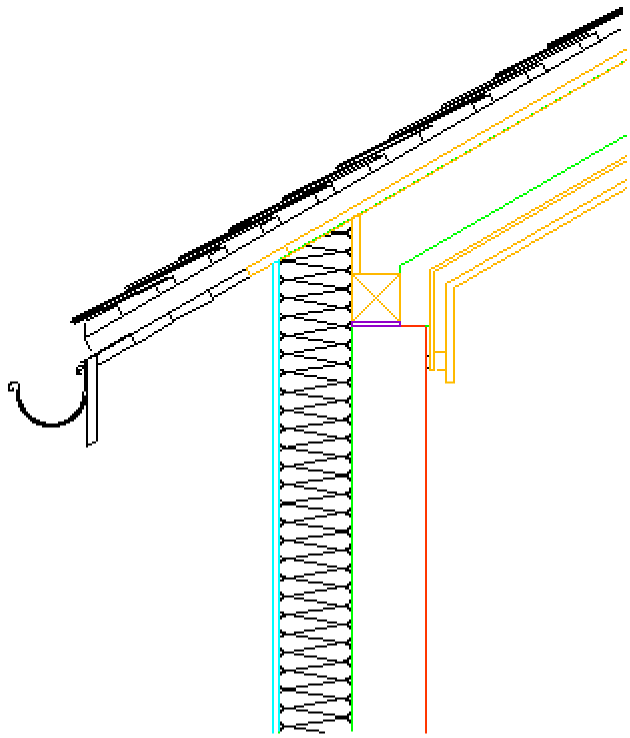


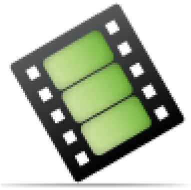
Figure 2: Roof-wall junction

Content

Adjusting the import settings and opening a DXF file
Selection of to be imported layers, Material assignment
Deleting unused edges and closing boundary lines
Converting DXF files and correcting "problem spots"

3.3.1 Adjusting the import settings and opening a DXF file

Before importing data, you should double check important flixo settings and then import a DXF file.



Summary

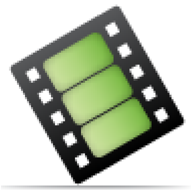
- The Import dialog window , in which the imported file is selected, can either be activated with the menu command File.Import... or by clicking on the icon.
- The **DXF import properties** are determined in a special Options dialog window that in certain cases have to be adjusted **before** the importing process begins. They will be used for the transformation of corner coordinates and the approximation of circles and arcs with polylines. The dialog window can either be activated directly from the Import dialog window or with the menu command Tools.Options....

To continue the tutorial you can click here or use the navigation buttons on the top.

3.3.2 Selection of to be imported layers, material assignment

Often, not all of the data in a DXF file is needed for the analysis (e.g. hatch, axis). In flixo, you can choose to maintain certain elements while importing, which are found on different layers. The layer visibility determines which elements on certain layers are to be imported.

pro: Optionally you can assign a material to the domains, based on the layer information of the imported polylines.



Summary

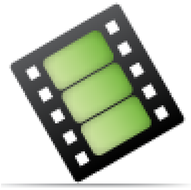
- Layers which are not to be used in the import can be hidden in the Layers flyout by deactivating the visibility option in the column.
- To facilitate the legibility, the layer color can optionally be changed.
- pro: Optionally you can assign automatically materials to the domains, based on the layer of the domains. The layer material assignment can be imported either from existing files or defined manually.

- You can open the layer material assignment dialog window by clicking the Options button in the DXF Import message line or by selecting the layer context menu item **Materials....**

To continue the tutorial you can click here or use the navigation buttons on the top.

3.3.3 Deleting unused edges and closing boundary lines

If individual edges and elements on the layer should be deleted, then this is done with the usual techniques and commands. Open line segments can be closed with special tools.



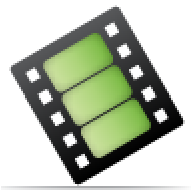
Summary

- Not needed lines can be selected and deleted with the Select, Move, Scale tool .
- Open line segments can be closed with the Line tool and the Polyline tool .
- It's only possible to draw on visible layers.

To continue the tutorial you can click here or use the navigation buttons on the top.

3.3.4 Converting DXF files and correcting "problem spots"

The main step of the import process is the conversion of all DXF data, which can be found on the visible layers. Often, not all line segments can be transformed to closed regions, "problem spots" or conflicts need to be corrected manually.



Summary

- Imported DXF-data are **converted** to flixo domains by clicking on the DXF Conversion button in the DXF Import message bar or by chosing the menu command Tool.DXF Conversion respectively by clicking on the according icon in the standard toolbar. All closed regions are automatically inserted into the model page.

- By clicking on the Zoom In button in the DXF Import message bar , the **next problem spot** is enlarged. There are two types of problem spots : Edges that are not used and line segments which are not closed.
- Edges which are not used can be **deleted** or ignored, and line segments which are not closed can be closed with the Select, Move, Scale tool .
- After the conversion and as soon as all problem spots are solved the program skips **automatically onto the report page**. Even if problem spots still exist, you can open the model page, in which all existing domains are shown.

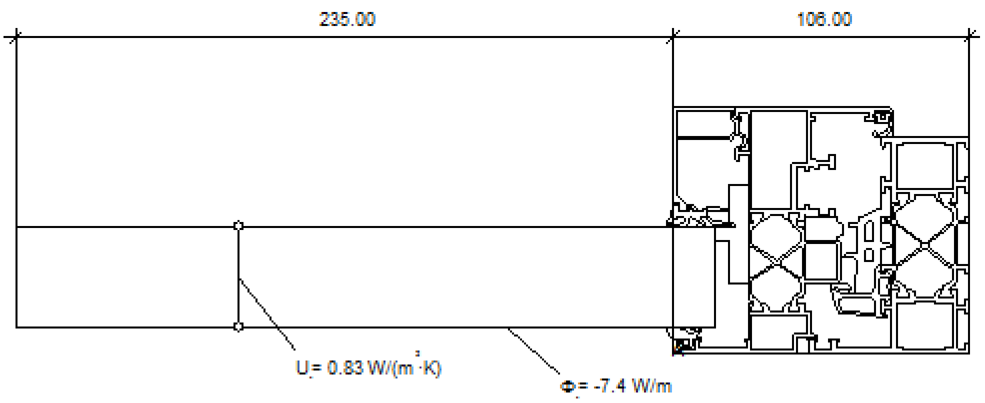
3.4 Window Frame Analysis and Wizards

frame , pro

In this tutorial, wizards and the modeling and calculation of window frame U-values with flixo is explained.

The goal of this tutorial is to make the window frame, which was imported in tutorial 3 for the analysis of the window frame U-value, conform to European Standards. In addition, we will examine correct boundary condition assignment, materials assignment, and the automatic calculation of frame U-value with flixo.

As in the other tutorials, we will go step by step.



$$U_f = \frac{\frac{\Phi}{\Delta T} - U_p \cdot b_p}{b_f} = \frac{\frac{7.418}{20.000} - 0.834 \cdot 0.235}{0.106} = 1.65\ W/(m^2 \cdot K)$$

⊟ Content

3.4.1 Adapting the construction

Either use your version of the construction from tutorial 3 or load the saved version ***WI77_EN.flx*** from the tutorial subcategory in the flixo program directory.

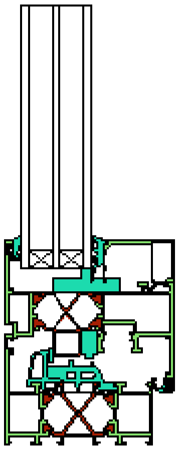


Figure 1

For the calculation of the frame U-value according to the European standard EN ISO 10077-2, the glass unit has to be replaced by a panel (cf. figure 2) with the following characteristics:

- The thickness of the panel should correspond to the thickness of the glass unit.
- The thickness of the air cavity between the panel and the frame (cf. b1) should at least be 5mm thick.
- The visible part of the panel (cf. bp) should at least be 190 mm.
- According to EN ISO 10077-1, the measurements b2 and bp should correspond to the larger of the two projected widths of the frame section, without taking into account the stripping between the frame and glass unit or panel (cf. figure 2).
- The thermal conductivity of the panel should be 0.035 W/(mK).

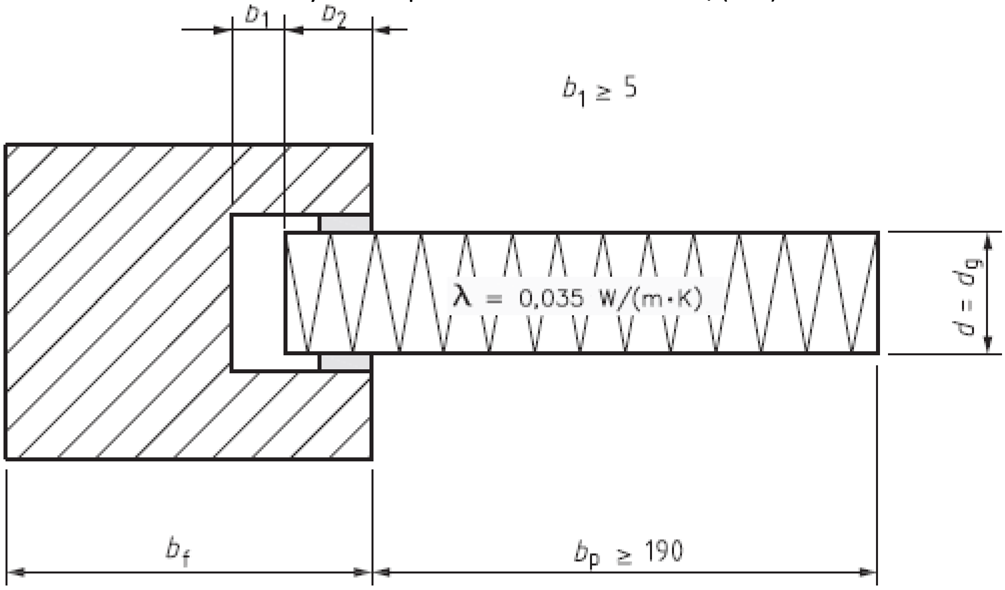
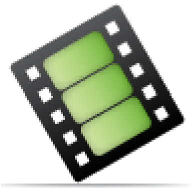


Figure 2

We will create the panel and an air cavity in the next steps. Click on the arrows in the graphic below to navigate in tutorial 4.



Summary

- A rectangular domain is created with the Rectangular Domain tool . The corner points are captured with active snap functions (Grid , Guidelines and Objects , see menu Arrange).
- The **origin of the coordinate system** can be adjusted by clicking on the icon in the upper left hand corner of the rulers and moving the mouse in the direction of the new origin and releasing at the desired spot.
- When adjusting the **position** of an object with the keyboard (cf. Select, Move, Scale tool), the **x-** and **y-**coordinates represent the position of the reference point .
- When adjusting the **size** of an object with the keyboard (cf. Select, Move, Scale tool), the reference point determines the point of the circumscribing rectangle, which is not moved during the adjustment.
- The **order** of the objects and their visibility is taken into account during the calculation. You can adjust the order of the selected domain with the Arrange menu commands or by clicking on the corresponding icon (, , respectively) in the arrange toolbar.
- Selected elements can be **rotated** 90° (respectively)or **mirrored** (respectively) by either clinking on the according icons in the arrange toolbar or by selecting the corresponding commands in the Arrange menu.

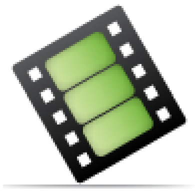
To continue the tutorial you can click here or use the navigation buttons on the top.

3.4.2 Assigning materials and cavity wizard

Assigning materials is done like in tutorial 1 and tutorial 2 with the ***Drag&Drop*** function. For the analysis of the window frame, we will select materials from the EN ISO 10077-2 Standard category.

In this cross-section of the window frame, all air cavities are assigned the material **Unventilated Air Cavities**. According to EN ISO 10077-2, all air cavities are considered unventilated. They are either completely enclosed by a material or have a connection to the exterior, which is no more than 2 mm large. The program will automatically and iteratively calculate the equivalent thermal conductivity of the individual air chambers.

Click on the arrows in the graphic below to navigate in tutorial 4.



Summary

- The materials are listed in the Materials flyout . In the lower part the materials from the database are listed, in the upper part materials present in the document are listed.
- New materials can either be added to the upper materials list or to the materials database (see Materials flyout toolbar).
- Materials can either be assigned with the **Drag&Drop** function or with the Assign Properties tool .
- The Assign Properties tool always assigns the **selected material from the upper list** by clicking on the desired element. With the Drag&Drop function you can, independently from the active tool, assign a material from the database list or from the document material list.
- **Air cavities** in the window frame are filled with the Air Cavity EN ISO 10077-2 tool . The user defines the domain, in which all cavities and open domains are automatically filled with special air cavity material according to the standard.

To continue the tutorial you can click here or use the navigation buttons on the top.

3.4.3 Assigning boundary conditions and radiation surface properties

When assigning boundary conditions, you are defining the start point for the boundary condition (cf. tutorial 1). The boundary condition is valid from this point to the next boundary condition, which runs **counterclockwise** (boundary conditions run clockwise for interior cavities, e.g. chimneys, or floor heating).

In the interior corners of the frame special boundary conditions for domains with reduced radiation and convection can be placed, according to EN ISO 10077-2 (cf. figure 3).

The boundary condition for reduced radiation and convection can be used on the interior side of the domain, where the width b is normally as big as the depression d, although should not exceed 30 mm.

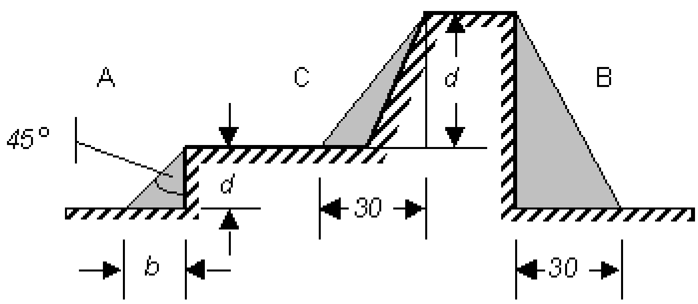


Figure 3

In our example, we can adjust the boundary conditions in two places on the interior side of the frame. With flixo this can be done in two ways:

- By setting a **special boundary condition**, which automatically recognizes the corner domains and sets the appropriate boundary conditions when preparing the calculation.
- **Manual assignment** of the boundary conditions in the proper places.

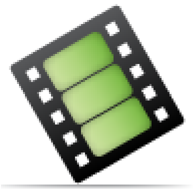
In this tutorial, we will examine the automatic version.

Surfaces of cavities can be treated specially, so the energy transport within air cavities is reduced by radiation. In flixo there are 2 ways to consider radiation properties for air cavities which differ from standard properties:

- By setting **special radiation properties.** flixo calculates automatically the resulting emissivities in the main heat flux direction which are necessary for the calculation of equivalent conductivities.
- By explicit **definition of the resulting emissivities** in heat flux direction in the cavity materials.

In this tutorial the radiation properties will be defined with special boundary conditions.

Click on the arrows in the graphic below to navigate in tutorial 4.



Summary

- The boundary conditions are listed in the Boundary Conditions flyout. In the lower part, the boundary conditions contained in the database are listed, whereas the upper list displays all boundary conditions present in the current document.
- New boundary condition objects are created with the Boundary Condition tool. Only the **start point** of a boundary condition is defined, which for exterior edges runs **counterclockwise** to the starting point of the next boundary condition.
- You can either choose the active boundary condition from the upper list displaying all boundary conditions present in the current document or from the boundary condition tool properties.
- Boundary conditions of the type frame recognize corners according to EN ISO 10077-2 and automatically set the correct h-value (heat transfer coefficient) respectively the correct R-value (thermal resistance), according to the standard.
- With the option **Only Vertices with change of direction** in the boundary condition tool properties you can limit the start points to surface vertices with change of direction.
- If the corresponding option is selected in the Option dialog window, then the polylines of the defined boundary condition are shown. The visualisation occurs time-delayed, depending on the complexity of the construction and on the used hardware. You still can continue your work without any restriction.

- Radiation properties in heat flux direction are either assigned directly to the cavity material or calculated by flixo. You can define the method in the Option dialog window.
- If the radiation properties are calculated by flixo then you can assign radiation properties to the surfaces with the Boundary Condition tool. You can also create them automatically with a material radiation properties assigning table using the Cavitiy EN ISO 10077-2 tool .
- If you define the resulting radiation properties in heat flux direction by yourself then you have to define them directly in the Material dialog window.

To continue the tutorial you can click here or use the navigation buttons on the top.

3.4.4 Material wizard

According to EN ISO 10077-2, air cavities, which have reductions smaller than 2 mm, are allowed to be divided into smaller air cavities. This segmentation - which can greatly influence the results - is done completely automatically by flixo, if the option is selected in the Cavities Options dialog window. This must be done before the calculation (see figure 4). The options dialog window can be activated with the menu command Tools.Options... or by clicking on the icon on the standard toolbar.

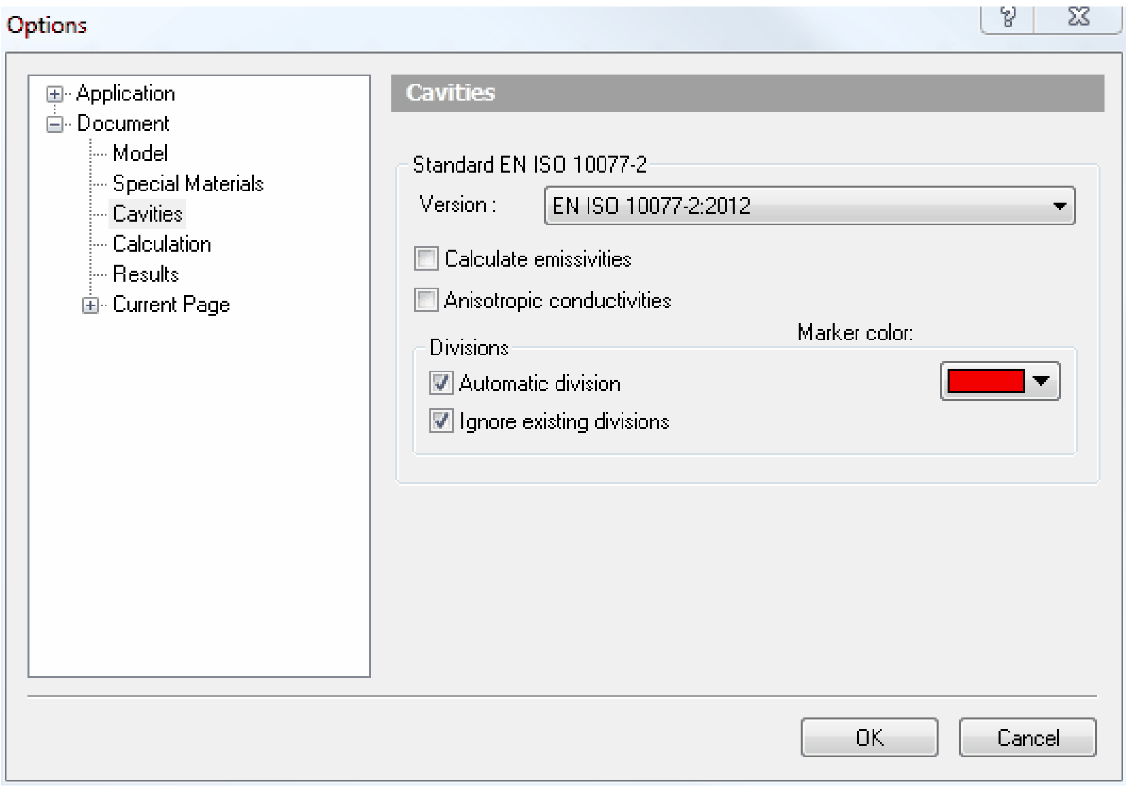


Figure 4

As soon as you are working with smaller domains in a construction, you may run the risk of missing smaller domains, which will not have any materials assigned and therefore could falsify the results of the calculation. flixo has a few wizards, which help with the assigning of materials.

Before starting a calculation, you can have the program search for domains containing special materials, automatically have materials assigned to small domains according to user-defined rules, or have particular domains highlighted. These settings can be determined in the Special Materials Options dialog window (see figure 5). All material domains with the material "Undefined Material" will be sought. If you are working with our example, all domains with a surface area smaller than 0.1 mm2 will be combined with the largest adjacent domain. If a larger domain is defined with "Undefined material", then a warning appears and the calculation can be interrupted.

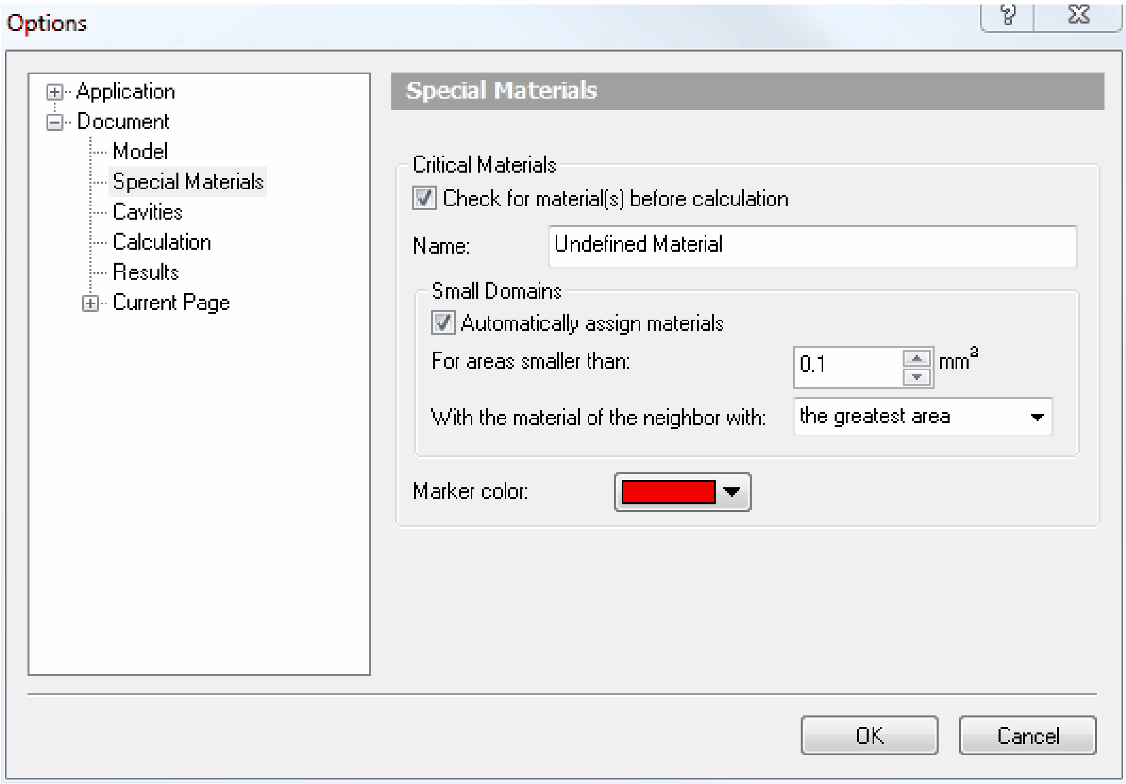
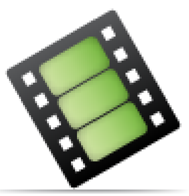


Figure 5

The examination begins after the start of the calculation, which can be activated with the menu command File.Calculate or by clicking on the icon on the standard toolbar.

Click on the arrows in the graphic below to navigate in tutorial 4.



Summary

- The calculation can be activated with the command **Calculate** in the menu **File** or by clicking on the icon on the standard toolbar. After a successful calculation, the program automatically skips to the first report page (depending on the chosen Options).

- If the construction does not fulfill certain conditions, a warning appears and the calculation can be interrupted. By clicking on the Zoom In button in the DXF Import message line, the **next problem spot** is magnified and the problem can be fixed.

To continue the tutorial you can click here or use the navigation buttons on the top.

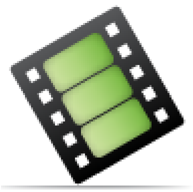
3.4.5 Calculation of frame U-values

Aside from other results, you can have the U-value for a window frame automatically be defined. This option is only available for constructions, which according to EN ISO 1007-2 are recognized as window frame U-value calculations:

- The thermal conductivity of the panel is 0.035 W/(mK).
- The visible part of the panel is at least 190 mm.
- There are exactly two air temperatures on the boundary conditions.
- There are exactly 2 adiabatic boundary conditions ("Symmetry/Model Section") each one on both model sections.

The automatic frame U-value calculation is also available for transoms and mullions (not mentioned in the standard). In which case, both glass units need to be replaced by an appropriate panel.

Click on the arrows in the graphic below to navigate in tutorial 4.



Summary

- The Uf-value can be calculated for constructions that fulfill all the conditions of the standard.
- The **calculation of the Uf-value** is started for the **selected** result object by choosing the menu command Results.U-Wert->Frame Uf-Value... or by activating the context menu command of the specific result object or by selecting the result object with the **U-Value tool Frame U-Value**. In the Uf-Value dialog window that appears, all frame materials (without stripping's) are determined.
- The frame materials and other properties which influence the calculation can be adjusted in the Properties flyout.

3.5 Psi-value and U-value Calculations

In this tutorial the calculations of Psi-values and U-values of some typical flixo applications are explained.

The goal of these examples is to illustrate the Psi-Value tool and the U-Value tool and to show typical cases where these tools are used in a meaningful way.

A Psi-value is mainly a ***value of adjustment*** during the calculation of energy loss of building surfaces. It quantifies the additional energy loss per degree Kelvin and linear meters which wasn't taken into account with the simplified approach of the energy loss calculation using U-values and areas. The calculation of the Psi-value therefore depends strongly on the calculation method of the simplified approach. These are defined in international and national standards. Therefore the same construction can have ***different Psi-Values*** depending on the national standard for the calculation of transmission losses!

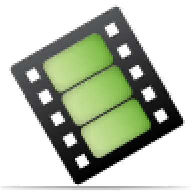
The calculation of ***Frame U-Values*** according to EN ISO 10077-2 is explained in tutorial 4.

Content

3.5.1 Psi-value 2 Constructions

energy, energy plus, pro

Psi-Values with 2 Components will be usefully calculated for constructions where 2 components are connected (e.g. wall ceiling junction, roof wall junction) or where a construction is interrupted by a single component (e.g. mullion of a curtain wall).



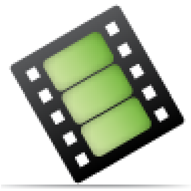
Summary

- A Psi-value with 2 components is defined using the Psi-Value tool 2 Components by applying 3 mouse clicks. The first two points define the surface line, through which the effective heat flow should be calculated. The third point (reference point) divides the surface into 2 parts with different U-values. The points are captured if the snap functions are active (Grid, Guidelines and Objects, cf. menu Arrange).
- You can adjust the properties of the Psi-value calculation in the Properties flyout.

Click here or on the navigation buttons in the title in order to continue with the tutorial.

3.5.2 Psi-value 3 Constructions

Psi-Values with 3 Components are usefully calculated for constructions where 3 components are connected sequentially. Either the same junction construction is used twice (e.g. 2 identical thermal bridges between a mullion and the glazing) or one junction between 2 components is "thermal bridge free" (e.g. window with panel instead of glazing).



Summary

- A Psi-value with 3 components is defined using the Psi-Value tool 3 Components by applying 4 mouse clicks. The first two points define the surface line, through which the effective heat flow should be calculated. The third and fourth point (reference points) divide the surface into 3 parts with different U-values. The reference points has to be defined ***counter clockwise***. The points are captured if the snap functions are active (Grid, Guidelines and Objects, cf. menu Arrange).
- The type of the Psi-value ***has to*** be adjusted in the Properties flyout.The Psi-value type is either twice the same junction construction (this means both has the same Psi-value) or 1 junction with Psi-value and the second junction without Psi-value.
- The U-value of the middle component ***has to*** be entered in the Properties flyout. You can enter the U-value either directly using the keyboard or using the tool by clicking on the U-value of another file.
- You can adjust the other properties of the Psi-value calculation in the Properties flyout.

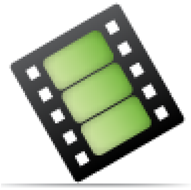
Click here or on the navigation buttons in the title in order to continue with the tutorial.

3.5.3 Edge Psi-value

pro

Edge Psi-Values according to EN ISO 10077-2 are calculated for the thermal evaluation of spacers and glazing edges.

Click on the arrows in the graphic below to navigate in tutorial 5.



Summary

- An edge Psi-value according to EN ISO 10077-2 is calculated with the Psi-Value tool Edge/Spacer. This happens in 2 steps: First select the result object for which you want to calculate the edge Psi-value, second select the file with the frame U-value calculation.
- The path of the file with the frame U-value calculation can be adjusted in the Properties Flyout

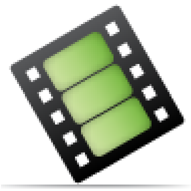
Click here or on the navigation buttons in the title in order to continue with the tutorial.

3.5.4 Equivalent U-value

energy, energy plus, pro

Equivalent U-Values are usefully calculated for constructions with periodically appearing thermal bridges (e.g. ceiling construction with rafter). The equivalent U-value considers already the influence of these thermal bridges.

Click on the arrows in the graphic below to navigate in tutorial 5.



Summary

- An equivalent U-value is defined using the U-Value tool Equivalent U-Value by applying with 3 mouse clicks. The first two points define the surface line, through which the effective heat flow should be calculated. The third point defines the position of the dimension line. The points are captured if the snap functions are active (Grid, Guidelines and Objects, cf. menu Arrange).
- You can adjust the properties of the U-value calculaion in the Properties flyout.

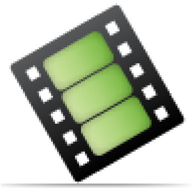
Click here or on the navigation buttons in the title in order to continue with the tutorial.

3.5.5 Joint U-value

pro

Joint U-Values according to EN ISO 12631 are calculated for mullion and transom constructions. In contrast to the frame U-value according to EN ISO 10077-2 the joint U-value already contains the thermal influence of the spacer or edge.

Click on the arrows in the graphic below to navigate in tutorial 5.



Summary

- The calculation of the ***Joint Uf-Value*** for the ***selected*** result object is performed either choosing the menu command Results.U-Value>Joint Uf-Value... or selecting the corresponding result object context menu command or using the U-Value tool kind Frame U-Value. All frame materials (without frame sealing) has to been marked in the opening Frame Material dialog window.
- The frame materials and other properties which influence the calculation can be adjusted in the Properties flyout

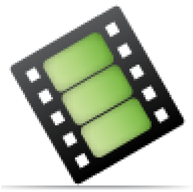
Click here or on the navigation buttons in the title in order to continue with the tutorial.

3.5.6 Roller shutterbox U-value

pro

Roller shutter box U-Value according to EN ISO 10077-2 are calculated for roller shutter boxes.

Click on the arrows in the graphic below to navigate in tutorial 5.



Summary

- The calculation of the ***Roller Shutter box Usb-Value*** for the ***selected*** result object is performed either choosing the menu command Results.U-Value>Shutter box Usb Value... or selecting the corresponding result object context menu command or using the U-Value tool kind Shutter box U-Value.

Click here or on the navigation buttons in the title in order to continue with the tutorial.

3.6 Document Templates

This tutorial explains the presentation of document templates. Document templates allow you to define the appearance and characteristics of the document generated from a template. For example, if you need your logo to appear on the report page, it would be best to create a template.

In this tutorial, you will create a report page template similar to the one illustrated below.

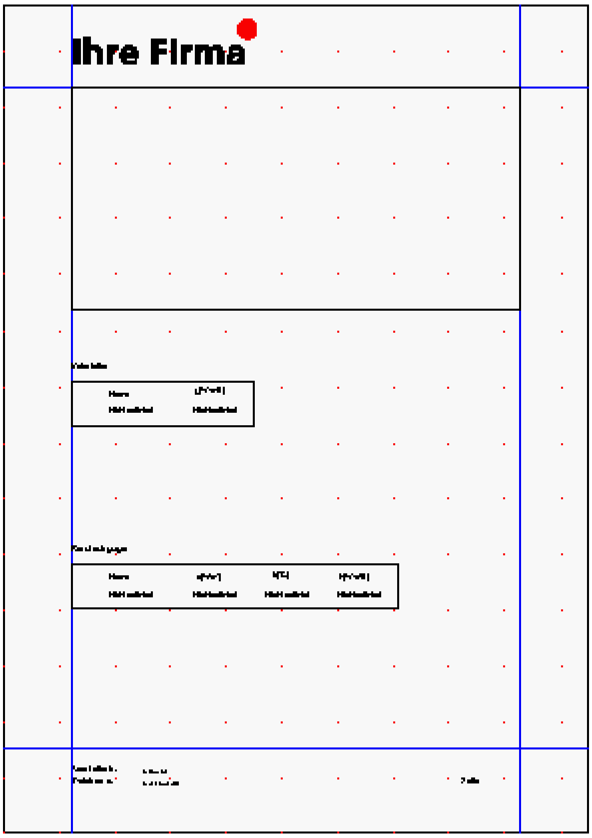


Figure 1: End result of the template format

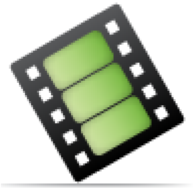
Content

- Master-Report page
- Creating a Standard Report Page
- Setting Standard Properties
- Saving the Document Template
- Creating Template-Based Documents

3.6.1 Master-Report page

All elements, which are present on a report page (e.g. logo, author, guidelines, page number, file name, etc.), will be added to the ***Master-Report page***. These elements can be hidden on any given page by clicking on the page and by editing the entry ("**Ref:**") on the reference layer (see Layers tab).

Click on the arrows in the graphic below to navigate in tutorial 6.



Summary

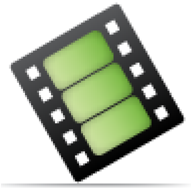
- A **new document template** is generated by creating a new document (cf. File.New) and then saving this document as a template (cf. Saving the document template).
- Elements, which should appear on all report pages are inserted into the **Master-Report page** and can **only** be edited in this page.
- **Guidelines** can be created in the Guidelines dialog window or by clicking within the rulers encompassing the application window. Marked guidelines can be adjusted either in the tool properties of the Select, Move, Scale tool or using the keyboard in the Properties flyout.
- A **new layer** can be created in the Layers flyout. New elements are always added to the Focus layer.
- You can insert pictures and field functions. Field functions are place holders, which display results after they have been generated. The type and properties of the field functions can be determined in a dialog window.
- Objects can proportionately be reduced or enlarged with the Select, Move, Scale tool, while the **Shift key is held down**.
- Selected objects can be moved with the **arrow keys**.
- New fonts can be created in the Styles flyout.

To continue the tutorial you can click here or use the navigation buttons on the top.

3.6.2 Creating a standard report page

The presentation of the results output (such as isotherms, streamlines, used materials), which influence the whole object , can be determined with the help of predefined result object properties (defined in the different report pages of the template).

Click on the arrows in the graphic below to navigate in tutorial 6.



Summary

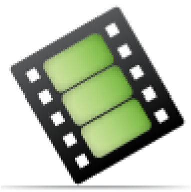
- **Report pages** can be **renamed** by double clicking on the corresponding tab at the bottom of the application window or by selecting the corresponding command from the tab context menu. New report pages are created with the context menu command **Insert New Page** and existing report pages are deleted with the context menu command **Delete Page**.
- The results are displayed with help of result objects. With 2 clicks of the Result Object tool, a rectangular place holder is created, which is replaced by the actual construction after the calculation. The corner points are captured, if the snap functions are active (Grid, Guidelines and Object, see menu Arrange).
- For every result object, different **properties** can be determined, which should be displayed. The Result Properties dialog window can either be activated with the context menu command **Properties,** with the menu command Edit.Properties... or by double clicking on the result object. Individual properties can also be activated or deactivated directly in the result object context menu or in the Results menu of the selected result object. Alternatively you can adjust easily all result object properties in the Properties flyout.

To continue the tutorial you can click here or use the navigation buttons on the top.

3.6.3 Setting standard properties

All settings, which are defined in the Options dialog window of the category document, are saved in the document template and are therefore automatically assumed by the document based on this template.

Click on the arrows in the graphic below to navigate in tutorial 6.



Summary

- **General document settings** , which are normally used should be defined in the template. Settings defined in the Options dialog window of the category document are automatically assumed when a new document is created.

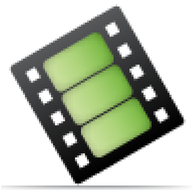
- **Materials and boundary conditions**, which are often used can be copied with the Drag&Drop function from the database to the upper list containing all materials or boundary conditions present in the document.
- It is easiest to define the number of decimal places and other result properties in the Styles flyout.

To continue the tutorial you can click here or use the navigation buttons on the top.

3.6.4 Saving the document template

The document template should be saved in a particular directory, so that they are displayed in the Dialog window New File, if a new document is generated with the menu command File.New... By default, this directory can be found in the templates subcategory of the flixo program directory. In the Save Options dialog window you can determine which directory should be accessed when a template is activated.

Click on the arrows in the graphic below to navigate in tutorial 6.



Summary

- Document templates should be saved in the **template directory**. The template directory can be found, and if necessary, adjusted in the Save Options dialog window.
- A template is saved with the command File.Save As... and also has to be saved as the **file type "flixo Templates (*.flt)"**.

To continue the tutorial you can click here or use the navigation buttons on the top.

3.6.5 Creating template-based documents

Generally, all flixo documents are based on a template. If, while generating a new document, you do not specify which template the document should be based on, then the document is automatically based on the template **Normal_DE.flt**. You can only select a template, if you generate the new document with the menu command File.New.... If you use the icon to create a new document, then it will automatically be based on the **Normal_DE.flt** template.

Click on the arrows in the graphic below to navigate in tutorial 6.

Summary

- A new document based on a specific template can be generated with the command File.New. The template can be selected from the dialog window which then appears.
- **Retroactively,** the layout of a report can be adjusted with the command File.Template...Because the adjustment cannot be reversed, the document should be saved before any changes are made.
  When changing a template, the layout and all Result Object properties, as well as the visible attributes and legends of the new template are assumed, while all results generated with tools are deleted.

3.7 Parametric Objects

energy, energy plus, pro

In this tutorial, the application and the generation of a parametric object in flixo is explained with the example of a glass unit as well as a ceiling-wall junction.

Parametric objects can be useful in two situations. First, they allow the generation of generic construction elements and components (e.g. spacer, generic walls), which can have other dimensions in a concrete situation. Second, they can rapidly generate versions of an existing result, whereby a specific dimension is changed and all layers and elements are moved correspondingly.

Content

3.7.1 Applying parametric objects

At the time of shipment, several components in flixo are parametric objects (generic walls, spacers, etc.), which can be inserted into a document with the Drag&Drop function. Glass units and layer objects, which can be generated with help of a dialog window, are also parametric objects.

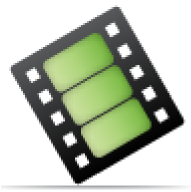
Summary

- **Glass units**, with a known U-value, are defined with the menu command Insert.Glass Unit... in the Glass Unit dialog window and are then inserted.
- **Components** from the Components flyout preview window are inserted into the document with the ***Drag&Drop*** function.
- The **length of parametric objects** can be adjusted with the Select, Move, Scale tool. Either the mouse or the keyboard can be used.

To continue the tutorial you can click here or use the navigation buttons on the top.

3.7.2 Defining a parametric object

energy, energy plus, pro



Summary

- **Parametric objects** can be generated, edited, and dissolved with the commands from the Edit.Parameter menu point.
- The definition of **parametric edges** can be done in two steps with the mouse:
  - Defining the edge, whose length can be changed and
  - Defining all vertexes, which are also to be moved when the edge length is changed.
- pro: With the menu command Tools.Protection>Protect the object..., parametric objects can be **protected** and other actions prevented.

3.8 Screw according to EN ISO 12631

pro

In this tutorial the definition of a periodically existing screw as an equivalent 3-dimensional object is explained and critical points are emphasized. The equivalent lambda values for equivalent 3-dimensional objects and bordering air cavities are calculated directly by flixo according to EN ISO 12631 depending on distances, dimensions and hidden materials.

In the next step you will define a screw as an equivalent 3-dimensional object of a mullion construction. Following the calculation you will check the critical material key values of the cavities.

Click on the arrows in the graphic below to navigate in tutorial 8.

Summary

- Select all objects of the screw with the Select, Move, Scale tool. By pressing the ***Shift key*** the selection is extended continuously with every further mouse click.
- ***Periodically appearing screws*** or ***objects with rectangular shaped cross section*** can be defined with the command Edit.3D equivalent Object > Define. You can adjust the material, the kind (screw or rectangular cross section) and geometry materials (radius and distance respectively depth and distance) which are necessary for the calculation of the equivalent conductivities in the opening dialog window.
- The material of the background regions can also be assigned afterwards with the Drag&Drop function, the material of the equivalent 3-dimensional object can be adapted in the Properties flyout or in the corresponding dialog window.
- ***Hidden lines*** subdivide the screws (respectively the object with rectangular shaped cross section) into regions. The cavities which border to the screw form one single air cavity for which the equivalent key values are calculated according to EN ISO 10077-2. The subdivision of the screw into several parts has an ***essential*** influence on the results. In our example the joint U-value enlarges for 0.1 W/m^2K if the screw is not subdivided into several parts. The subdivision of the equivalent 3-dimensional objects occurs where the object penetrates ***punctually*** another region.

Part IV

4 Reference

4.1 Tools

General

To create documents in flixo, you work with various tools. Each tool is used for a certain type of work process. For example, a tool can create objects, another tool can rotate or move objects. In this chapter, you will find an overview of the functions of the tools and descriptions of their use.

At the beginning of each tool description, you see a row of symbols, which symbolize the tools. An explanation of these symbols can be found on the conventions page.

Activation

In principle, you can activate every tool from the Tools, Insert, or Results menus or also by using the corresponding symbol in the Tools flyout.

Multiple Use and single Use

Tools which create new objects such as e.g. heat flux objects could either be set up in multiple use mode or in single use mode:

- In the ***multiple use*** mode the present tool stays active until another tool explicitly is activated. By holding the **Alt key** down the Select, Move, Scale tool can temporarily be activated. After releasing the Alt key the previous tool is active again.
- In the ***single use*** mode the Select, Move, Scale tool is activated automatically after creating an object or finishing an action. Tools with a single use mode setup can temporarily be put into the multiple use mode by double clicking on the corresponding symbol in the Tool flyout, until another tool explicitly is activated.

The standard mode of a tool can be configured in the tool options, compare the General Tools Options dialog window.

Tools overview

Tools in this chapter:

- Select, Move, Scale tool
- Rotate tool
- Assign Properties tool
- Cut tool
- Delete tool
- Measure Distance tool
- Zoom tool
- Rectangular Domain tool
- Elliptical Domain tool
- Polygon Domain tool
- Air Cavity EN ISO 10077-2

- Boundary Condition tool
- Heat Source tool
- Crop tool
- Result Object tool
- Temperature tool
- Min./Max. Temperature tool
- Heat Flux tool
- Heat Flow tool
- U-Value tool
- Psi-Value tool
- Dimension tool
- Information tool
- Line tool
- Polyline tool
- Rectangle tool
- Ellipse tool
- Text tool

4.1.1 Select, Move, Scale Tool

Use

Description

Generally, it is necessary to select one or several objects before an action can be carried out. To this end, there is the ***Select, Move, Scale*** tool. Furthermore, this tool can also move and scale objects.

If the ***Select, Move, Scale*** tool is merged with the ***Edit*** tool (cf. Options dialog Tools General), this tool lets you ***move*** corners, ***add*** new corners or ***remove*** existing ones as well the ***edge lengths*** of parametric objects can get adjusted by this tool too.

How do I activate it?

The ***Select, Move, Scale*** tool can be activated with the command **Select, Move, Scale** from the menu **Tools** or by clicking on the icon in the Toolbox flyout. When the tool is active, the mouse takes the shape of an arrow or a symbol of an action, which potentially could be executed.

How do I use it?

Mouse

The use of this tool is similar to the mouse control in Windows Explorer. To ***select*** an object, just click on it. The outlines of the selected object are highlighted with a particular line (cf.

the corresponding option in the Options dialog window category **Application>User Interface**). This means that the object is active.

To mark several objects, hold down the Shift key then click on the next object. It is also possible to select several objects at the same time by clicking on a point (which does not belong to an object) or by holding down the Alt key and then drawing a rectangle. All objects, which are entirely in this designated area, will be marked. In the expanded selection mode (cf. Options dialog window, category **Application>Tools>General**), whereby if the start point is to the right of the endpoint, all objects will be marked that are only partly contained within the rectangle.

If you would like to mark all the objects, which are present on a report page, you can do this with the menu command **Edit.Select All** out of the menu or with the keyboard combination **Ctrl+A**

To ***move*** objects, you must first mark them. Now you can click on one of the selected objects and move it to a desired position with the mouse (the mouse button must be kept down while moving it). With the **Shift** and **Ctrl** keys, you can influence the behavior of the object while you move it. If you hold down **Ctrl** while moving the object, then a copy of the object will be moved, the original object will remain untouched. If you hold down the **Shift** key, then the object can only be moved vertically and horizontally. It is also possible to combine both options at once.

Objects (or copied objects) will snap to a ***reference point*** when they are being moved if the snap option is set. The position of the reference point corresponds to that of the corner point, which is closest to the mouse click. If you want the upper left corner to be the snap reference for that object, then you must click near the upper left corner of the object.

The reference point is marked with a little circle when the object is being moved, and after moving it, it becomes a filled circle .

To ***scale*** an object, you must also select it first. Unlike moving an object, you must click on one of the highlighted points and then move it (you should keep the mouse button held down). You can then see how the object gets bigger or smaller. If you hold down the **Shift** key while scaling, then the scaling is done proportionally: that is the vertical increase (or decrease) matches that of the horizontal.

Corners, Edge length

If the ***Select, Move, Scale*** tool is merged with the ***Edit*** tool (cf. Options dialog Tools General), you can modify also corner points and edge lengths, otherwise this has to be done using the Edit tool.

Click on the corner point that you would like to ***move***. The point, which is now highlighted in black (see figure 1), can be moved by clicking on it and moving the point.

To ***delete*** a corner point, choose the command **Remove Vertex** from the context menu as you right click on the particular vertex.

To ***add*** a new corner point, click on the spot on the line where you want to insert the vertex. A small circle now marks the spot (see figure 2). Now choose the command **Split Edge** from the context menu (right click) in order to add the new corner point.

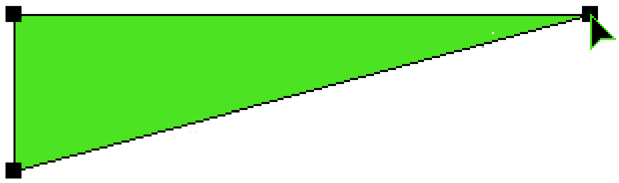


Figure 1

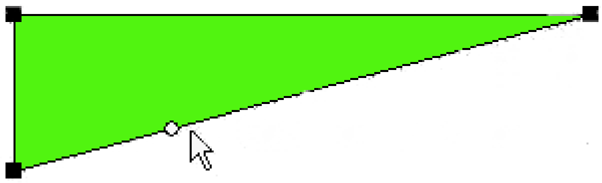
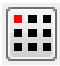


Figure 2

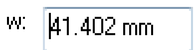
Keyboard

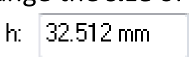
You can also numerically adjust the ***position***, the ***size*** and the ***scale*** of one or more than one object.

Make sure that the object is selected and the tool properties are visible (see figure 3). If this is not the case, you activate the Toolbox flyout and the Select, Move, Scale tool. You can now activate the properties with the Tab key or by clicking on a field. You can activate the individual fields by repeatedly hitting the Tab key or by using the Tab key while the Shift key is held down.

The x: and y: fields are relevant for the position of the object. These fields are the coordinates of the reference point. The red point on the reference point control  indicates, which point on the object (corner, midpoint, or object center) is to take the coordinates. You can change the reference point (after activating it with Tab) with the arrow keys or with the mouse. You can finish the input by either clicking on the **Apply** button or by activating it with the Tab key and then hitting Return.

You can also move an object a little at a time vertically and horizontally with the arrow keys.

You can change the ***size*** of the selected object by entering the width 

and the height 

of the bounding rectangle. You can finish the input by either clicking on the **Apply** button or by activating it with the Tab key and then hitting Return.

You can change the ***scale*** of the selected object, by entering a value into the field marked with a %. Within the upper % field, you change the width by the corresponding magnitude, within the lower % field, you change the height. If the scaling should be done proportionally, you can activate the option **Lock aspect**. You can then finish the input by either clicking on the **Apply** button or by activating it with the Tab key and then hitting Return.

If the option **Create Copy** is activated, a duplicate will automatically be created.

All coordinates correspond to the current coordinate system. The origin of the coordinate system can easily be adjusted.

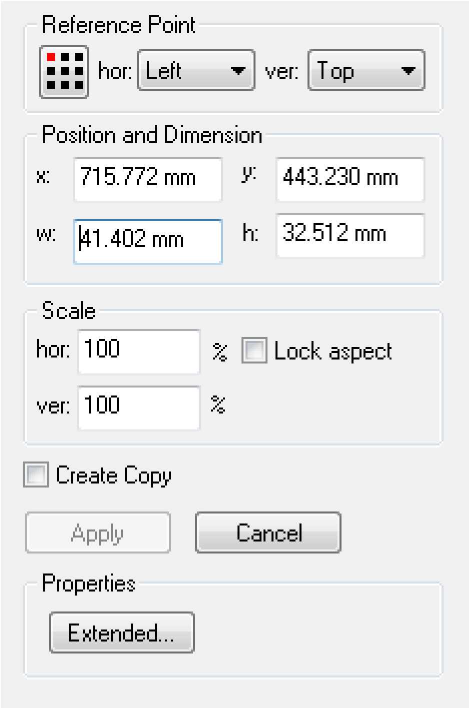


Figure 3: Select, Move, Scale tool properties

4.1.2 Edit Tool

Use Limitations

Description

The ***Edit*** tool lets you ***move*** corner points of existing objects, ***add*** new corner points, or ***remove*** corner points. If the object in question is a parametric object, then the Edit tool can adjust the ***edge length*** of this object.

How do I activate it?

The Edit tool can be activated with the command **Edit** from the menu **Tools** or by clicking on the symbol on the Toolbox flyout. When this tool is active, the mouse becomes an arrow.

How do I use it?

Mouse

After you have activated the Edit tool, click on the domain that you would like to change to select it (you can also select it with the Select, Move, Scale Tool). Now click on the corner point that you would like to ***move***. The point, which is now highlighted in black (see Figure 1), can be moved by clicking on it and moving the point.

To ***delete*** a corner point, choose the command **Remove Vertex** from the context menu as you right click on the particular vertex.
Alternatively you can delete a corner point with a mouse click, when pressing down **ALT**-key.

To ***add*** a new corner point, click on the spot on the line where you want to insert the vertex. A small circle now marks the spot (see Figure 2). Now choose the command Split Edge from the context menu (right click) in order to add the new corner point.
Alternatively you can insert a corner point with a mouse click, when pressing down **Ctrl**-key.

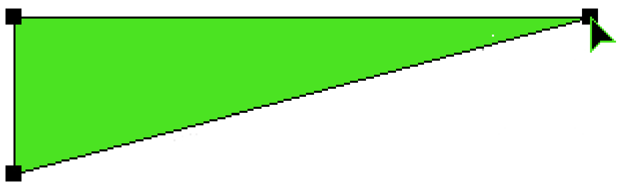


Figure 1

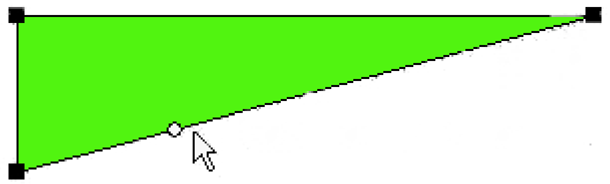


Figure 2

The marked objects can be **stretched** as follows:

- selection of objects with the Select, Move, Scale Tool

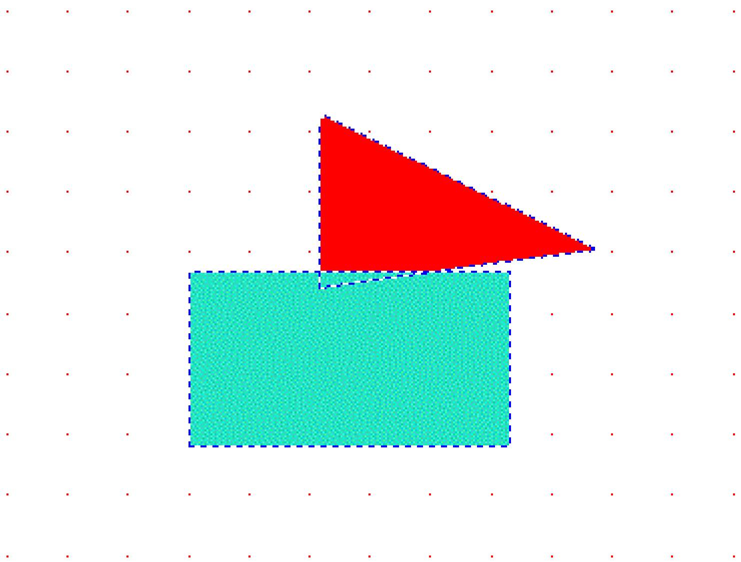


Figure 3: selection of objects

- selection of corner points that are to be moved with the Edit Tool by drawing a rectangle

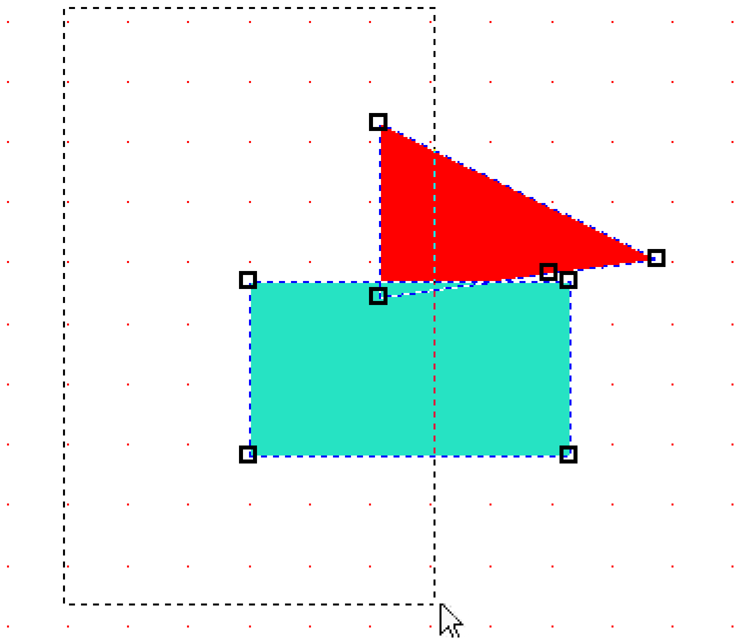


Figure 4: selection of corner points

- moving of selected corner points by moving one of the selected corner points

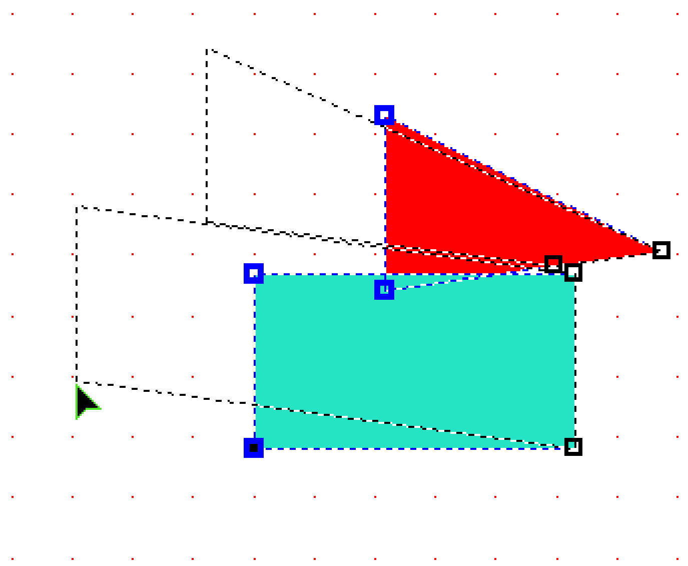


Figure 5: moving of corner points

Keyboard

You can also change the position of a corner point with the keyboard:

Make sure that the tool properties are visible (see Figure 3). If this is not the case then you can activate the Toolbox flyout and the ***Edit*** tool.

Mark the endpoint you wish to ***change*** with the keyboard. The corner point will be highlighted as a small white square. Now, activate the tool properties by hitting the Tab key or by clicking on one of the fields. You can activate the individual fields by repeatingly hitting the Tab key or by using the Tab key while the Shift key is held down.

For numerical input you can choose between different systems:

- **absolute Cartesian:** you input the coordinates x and y.
- **absolute polar:** you input the angle (relative to the x-axis) in field **w:** and the distance (of the zero point) in field **r:**.
- **relative Cartesian:** you input the horizontal **dx:** and the vertical distance **dy:** relative to the starting point.
- **relative polar:** you input the angle (to the x-axis) and the distance to the starting point.

By clicking the **modify** button you will now move the corner point or stretch the objects.

For the coordinates, the fields marked with **x:** and **y:** are relevant. Enter the desired coordinates, and finish the input entry by hitting the Return key.

All coordinates correspond to the current coordinate system. The origin of the coordinate system can easily be adjusted.

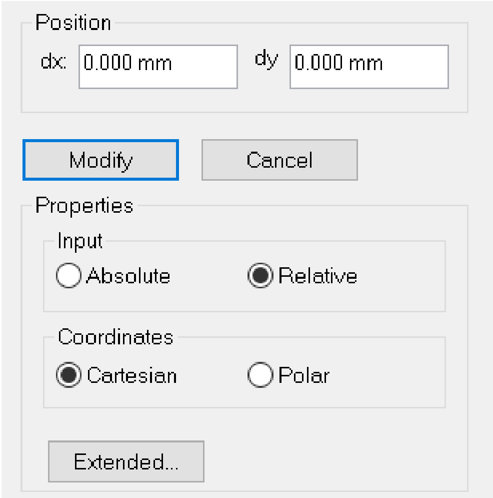


Figure 6: Edit tool properties

Limitations

- It is not possible to delete a corner point, when there are only 3 corner points in the domain.
- The ***Edit*** tool can be selected only if it is not merged with the ***Select, Move, Scale*** tool (cf. Options dialog Tools General).
- The stretching of objects is only possible, if no groups are marked and the edges of the objects don't intersect.

4.1.3 Rotate Tool

Use

Description

The ***Rotate*** tool allows you to rotate objects.

How do I activate it?

The Rotate tool can be activated with the command **Rotate** from the menu **Tools** or by clicking on the icon in the Toolbox flyout. When this tool is active, the mouse turns into a round arrow.

How do I use it?

Click on the object to be rotated, if the object is not yet selected. The corners of the object will be shown as white dots. There is a further black dot in the center: the center of rotation (see figure 1).

Mouse

When you move the mouse over a white dot, it will take on the following form . When you click the mouse button, you can move the point and thus rotate the object until you release the mouse button again.

You can also move the center of rotation before rotating the object. You can do this by clicking and then holding the mouse button down and moving the center point.

Is the **Shift** key is held down while rotating an object then the rotation angle will be limited to a multiple of a given element angle. First release the mouse button and then the Shift key. The element angle is defined in the Options dialog window (category **Application>Tools>General**).

Keyboard

If you wish to rotate an object by a certain number of degrees, you can define the rotation numerically:

Make sure that the object you would like to rotate is selected and that the Rotate tool properties are visible (see figure 2). If this is not the case, activate the Toolbox flyout and the Rotate tool (see menu command **View.Tools**). Activate the tool properties by hitting Tab or by clicking on one of the fields. You can activate the individual fields by repeatedly hitting the Tab key or by using the Shift + Tab keys simultaneously).

For the rotation, the field marked **Angle:** is relevant. Here the desired rotation angle is entered. If you need to adjust the center of rotation you can do so in the subsequent two fields **x:** and **y:** and finish the input by hitting the Return key.

If the option **Create Copy** is activated, a duplicate will automatically be created.

All coordinates correspond to the current coordinate system. The origin of the coordinate system can easily be adjusted.

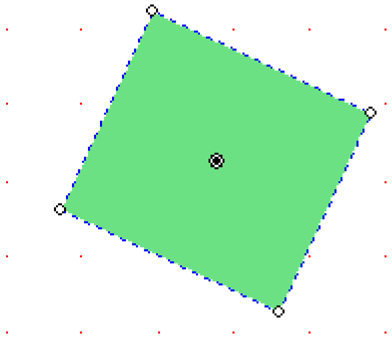


Figure 1

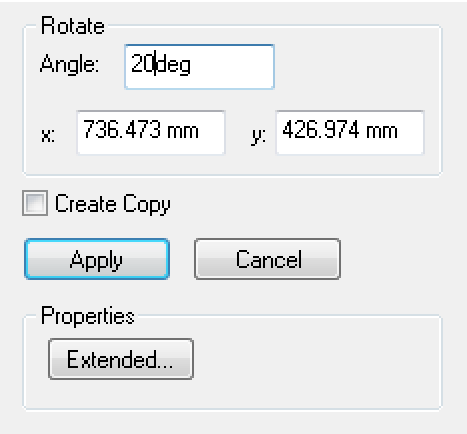


Figure 2: Rotate tool properties

4.1.4 Assign Properties Tool

Use Limitations

Description

Retroactively other materials from the Materials flyout, other boundary conditions from the Boundary Condition flyout or other font styles from the Styles flyout can be assigned to material domains, boundary condition objects, and other arrangement elements.This can be done with the ***Assign Properties*** tool.

The Assign Properties tool encompasses two functions:

- The ***Assign*** function to assign properties
- The ***Pipette*** function to collect properties

How do I activate it?

The tool can be activated with the command **Assign Properties** in the menu **Tools** or by clicking on the icon on the Toolbox flyout. When the tool is active the symbol becomes a and respectively a .

How do I use it?

When the Assign Properties tool is active you can click on the corresponding domain and that in the ***flyouts*** selected and highlighted ***material***, ***boundary conditions*** or ***font styles*** will be assigned.

By hitting the **Alt** key the Assign Properties tool becomes a pipette - correspondingly the mouse changes its symbol to a - and the information of other objects (material, boundary conditions, or styles) can be assumed as the current markings and subsequently can be assigned with the Assign Properties tool.

Limitations

- Only materials or boundary conditions which have already been used in the document can be used with the Assign Properties tool. In other words, entries from the materials or boundary conditions database ***can't*** be used by the Assign Properties tool.

4.1.5 Delete Tool

Description

The Delete tool lets you delete objects

How do I activate it?

The Delete tool can be activated with the command **Delete** from the menu **Tools** or by clicking on the symbol on the Toolbox flyout. When this tool is active, the symbol becomes a and the object is highlighted.

How do I use it?

Click on the Object that you wish to delete.

4.1.6 Cut Tool

Use Limitations

Description

The ***Cut*** tool lets you cut domains along a straight line or a closed polygon.

How do I activate it?

The ***Cut*** tool can be activated with the command **Cut** in the menu **Tools** or by clicking on the icon in the Toolbox flyout. When the tool is active, the mouse becomes a .

How do I use it?

Cutting is done in two steps:
- Defining of a cutting line
- Actual cutting

Areas can either be cut along a straight ***line*** or along a ***closed polygon***. Figure 1, for example, demonstrates the cutting of a corner of a rectangle.

The form of the cutting line can be defined in the Properties toolbar (see **Input**, figure 2). If the option **Only selected objects** (cf. figure 2) is checked, only the selected objects will be considered else all objects.

Mouse

The entry of a ***closed polygon*** (see Input: **Polygon**) as a cutting line with the mouse is done in a similar way to using the Polygon Domain tool. To remove the previous point, choose the command Remove Previous from the context menu that appears with a right click. To cancel the entire process you can either choose the command Cancel from the context menu (right click), or you can hit the Esc key.
You can then cut the domain using the **Cut** command from the context menu (right click).

The entry of a line (see Input: **Line**) as a cutting line is done by determining two reference points on the line. With the first mouse click you define the benchmark, with the second mouse click you define the direction of the line and cut the domains.

If the **Shift** key is pressed while determining the cutting line, then the angle will be limited to the multiple of the element angle. First release the mouse button and then the Shift key. The element angle can be defined in the Options dialog window (**Application>Tools>General**).

Keyboard

The entry of a ***closed polygon*** (see Input: **Polygon**) as a cutting line with the keyboard is done in a similar way to using the Polygon Domain tool.

The entry of a line (see Input: **Line**) as a cutting line is done by determining two reference points on the line (Coordinates: **Cartesian**) or accordingly by determining one point and the direction of the line (Coordinates: **Polar**).

The cutting is carried out by in that the **Cut** key is activated by repeatedly hitting the Tab key and by then hitting the Return key. Alternatively, you can just click on the **Cut** key.

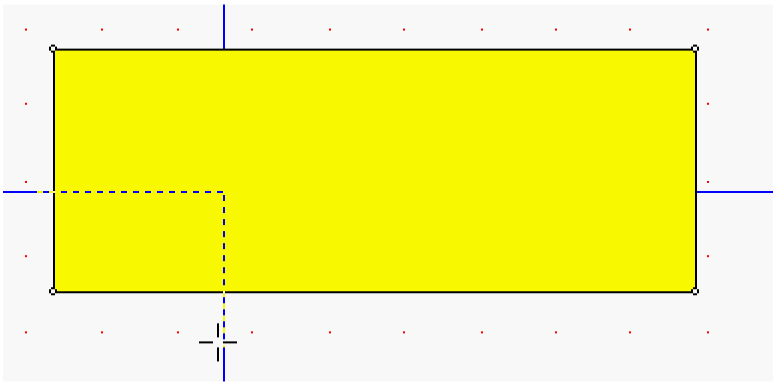


Figure 1: Example of a cutting path

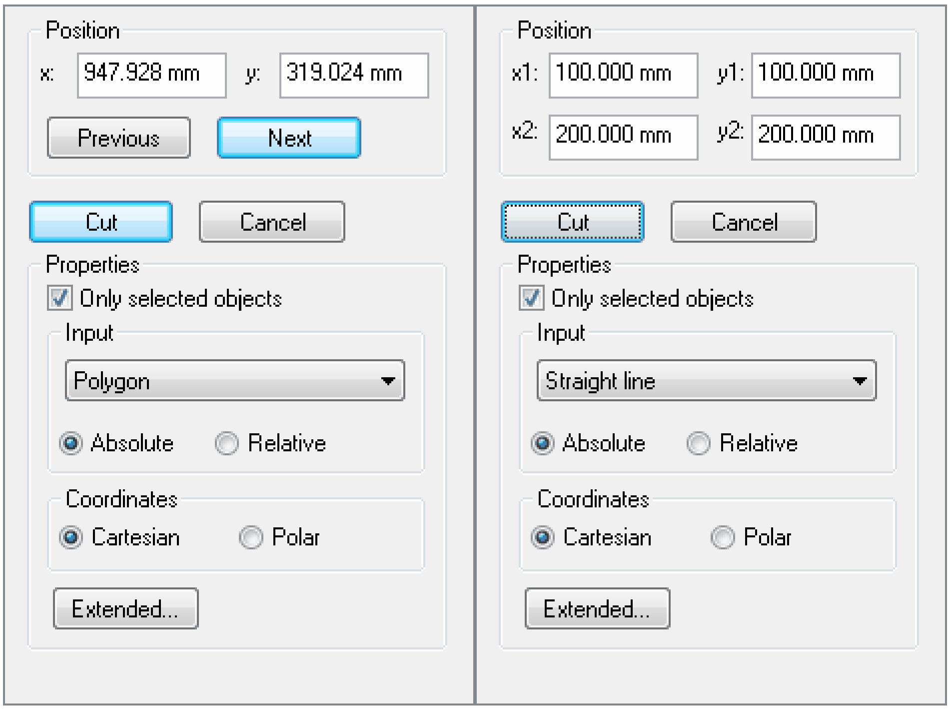


Figure 2: Cut tool properties

Limitations

- Only those domains, which are separated into at least two parts, can be cut. Depending on the setting of **Only selected objects** (cf. figure 2), all domains or only the selected ones will be cut.

4.1.7 Measure Distance

Use

Description

The ***Measure Distance*** tool allows you to measure the distance between two points.

How do I activate it?

The measure distance tool can be activated with the command **Measure Distance** in the menu **Tools** or by clicking on the icon in the Toolbox flyout. When the tool is active, the mouse becomes a .

How do I use it?

After entering the coordinates of two points the distance as well as the horizontal and vertical projection will appear in the measure distance tool properties (see figure 1).

Mouse

Click on two points in the construction you wish to measure. After the second click the distance will be measured. If you want to cancel the entry before the endpoint is defined, choose the command **Cancel** from the context menu (right click) or hit the **ESC** key.

Keyboard

Enter the coordinates of the two points in the following fields **x1:, y1:** as well as **x2:, y2:** (see figure 1). You can jump to the next entry fields by hitting the Tab key.

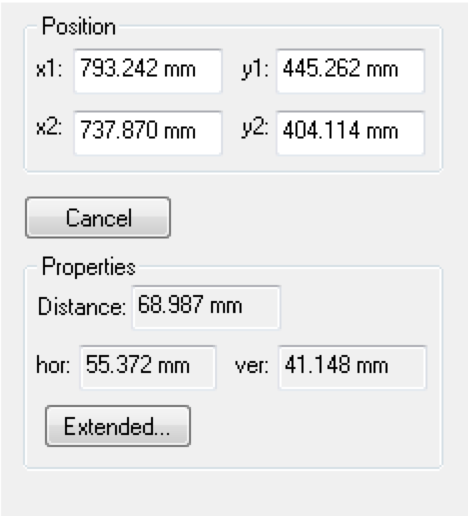


Figure 1: Measure Distance tool properties

4.1.8 Zoom

Use

Description

In documents, there are often several different objects that you would like to work on. With the ***Zoom*** tool you can ***zoom in*** or ***zoom out*** to enlarge or reduce a visible area of an active document. Besides the Zoom tool, there is a whole list of commands, which also function as zoom functions.

If the option **Activate mouse wheel zoom** in the Options dialog window (see **Application>Tools>General**) is activated, the application window view can be changed by turning the mouse wheel, independently of the active instrument.

How do I activate it?

The Zoom tool can be activated by clicking on the icon in the Toolbox flyout. For predefined Zoom operations, there is the **Zoom** toolbar. You can find the Zoom commands in the **View** menu or by using the commands from the context menus (right click).

How do I use it?

Zoom In : The command can be found under **View.Zoom** or can be activated by clicking on the icon on the Zoom toolbar (cf. figure 1). The content of the window will be magnified by a factor of 2.

Zoom out: The command can be found under **View.Zoom** or can be activated by clicking on the icon on the Zoom toolbar (cf. figure 1). The content of the window will be halved.

Zoom: The tool can be activated by clicking on the icon in the Toolbox flyout. It allows you to ***freely magnify*** and ***freely reduce*** an area of your choice. When the tool is active, the mouse becomes a . You can enlarge a specific spot within the drawing area by clicking on that spot; that spot now becomes the focus of the work area. If you move the mouse before releasing the mouse button, you can define a rectangular area to be enlarged. The content of this window will fill the application window.

Hold down the Ctrl key, when you click on an area and the mouse will change to a and the area will be reduced. In the context menu (right click) you will find other zoom commands that change the size of application window.

Zoom Percent: In the menu **View.Zoom.Percent**, there is a list of different sizes; these can also be found in the context menu (right click). The desired zoom factor can be defined in the number field on the Standard toolbar (cf. figure 1), alternatively, the slider at the bottom of the application window can be moved (cf. figure 2).

Zoom to Selection: The command can be found under **View.Zoom** or can be activated by clicking on the icon on the Zoom toolbar (cf.figure 1) respectively at the bottom of the application window (cf. figure 2). The selected object will be magnified to fill the entire application window.

Zoom to all Objects:The command can be activated under **View.Zoom** or can be activated by clicking on the icon on the Zoom toolbar (cf. figure 1) respectively at the bottom of the application window (cf. figure 2). All objects will be displayed at maximum magnification. If the option **Activate mouse wheel zoom** in the Option dialog window (cf. **Application>Tools>General**) is activated, then the same command can be activated by double clicking the mouse wheel.

Zoom to Material: The command can be activated under **View.Zoom** or by clicking on the icon located on the Zoom toolbar respectively at the bottom of the application window (cf. figure 2). The next object defined with material selected from the Materials flyout will be magnified to fill the application window.

Zoom to Conflict:The command can be activated under **View.Zoom** or by clicking on the icon on the Zoom toolbar (cf.figure 1). The function is available in case of problems arised during DXF import or in the preparation steps for the calculation. After a DXF conversion, it allows you to find unclosed edges, and after the start of a calculation, it allows you to find critical materials (see tutorial 3 and tutorial 5).

Alternatively, in case of problems a message bar is shown. Within this message bar you can click on the Zoom-button for enlarging the conflict area (e.g. if not all boundary conditions are defined, cf. figure 3).

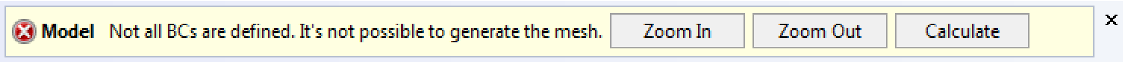


Figure 3: Message Bar

Zoom Page: This command can be activated by clicking on the icon on the Zoom toolbar (cf.figure 1) respectively at the bottom of the application window (cf. figure 2). The entire page of the report will be shown at maximum magnification (only available on report pages).

Zoom Page Width: This command can be activated by clicking on the icon on the Zoom toolbar (cf.figure 1) respectively at the bottom of the application window (cf. figure 2). The entire width of the report will be shown at maximum magnification (only available on report pages).

Zoom Page Height:This command can be activated by clicking on the icon on the Zoom toolbar (cf. figure 1) respectively at the bottom of the application window (cf. figure 2). The entire height of the report will be shown at maximum magnification (only available on report pages).

Analog to other commands, zoom commands can also be undone or redone by clicking on the and icons.

Figure 1: Zoom toolbar

Figure 2: Zoom possibilities at the bottom of the application window

4.1.9 Rectangular Domain Tool

Use

Description

The ***Rectangular Domain*** tool allows you to create rectangular and square material domains.

How do I activate it?

The Rectangular Domain tool can be activated with the command **Rectangular Domain** in the menu **Insert** or by clicking on the icon in the Toolbox flyout. When the tool is active, the mouse becomes a .

How do I use it?

Mouse

After you have activated the Rectangular Domain tool, you can define the rectangle with two points, which lie diagonal from one another. Click on the place where you would like to set the first point. When you move the mouse, you will see a dashed preview of the domain, which constantly changes as you move the mouse. When you click again you will determine the second point defining the domain.

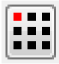
If you hold down **Ctrl** when you draw the domain, the first point is not a corner point of the domain, but rather the ***center*** point of the domain. If you hold down the **Shift** key when you draw the domain, then a square will be created. The combination of both keys then allows you to create a square that is defined by its center point.

You can interrupt the input of a domain by right clicking and choosing **Cancel** from the context menu or by hitting the **ESC** key.

Keyboard

After you have activated the Rectangular Domain tool, make sure that the Rectangular Domain tool properties are visible. If this is not the case, then you can make them visible with the menu command **View.Toolbox** (see figure 1). Hit the Tab key or click on the first field of the tool properties . You can activate the individual fields by repeatedly hitting the Tab key or by using the Tab key while the Shift key is held down.

You can enter the coordinates, the dimensions, and the reference of the domain. The coordinates of the reference point can be entered in the fields marked with **x:** and **y:**, the width can be entered in the field marked **b:** and the height can be entered in the **h:** field.

The Reference Point Control, symbolized by  determines the position that defines the reference point. The reference point can be a corner, midpoint or the center of an object. The selected point is marked in red, and can be changed with the mouse or the arrow keys. When all the parameters have been defined then you can finish the process by clicking on the **Create** key.

Defining Materials

A new domain will always be created with the material, which is selected in the Materials table. You can change the material of domains you want to create either by changing the selection in the Materials table or by choosing the material from the drop down list 

.

You can also change the material of the domain afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see also tutorial 1).

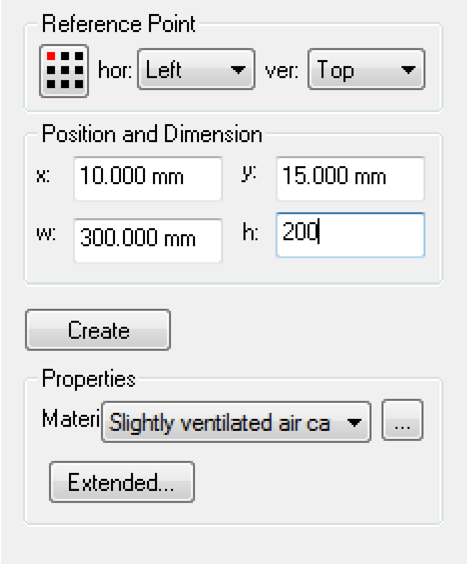


Figure 1: Rectangular Domain properties

4.1.10 Elliptical Domain Tool

Use Limitations

Description

The ***Elliptical Domain*** tool allows you to create elliptical shapes and circular approximations. Because these domains have to be broken down for the computational process, neither true ellipsis nor true circles are created, flixo generates a polygon approximation. The quality of these approximations, i.e. the number of points on the polygon, can be determined.

How do I activate it?

The Elliptical Domain tool can be activated with the command **Elliptical Domain** in the menu **Insert** or by clicking on the icon in the Toolbox flyout. When the tool is active, the mouse becomes a .

How do I use it?

The use of the Elliptical Domain tool is similar to that of the Rectangular Domain tool. The only difference is that the indicated coordinates and dimensions do not define a rectangle, but rather the dimensions of the bound ellipse. Also here, the holding down of the **Shift** key means that the dimensions in vertical and horizontal direction are identical, i.e. a circle is created.

Number of Reference Points on the Polygon

The number of reference points on the polygon determines how precise the elliptical (circular) approximation is. The higher the number of points, the better the approximation.

However, you should take into account that as the number of reference points increases, the calculation performance will also be prolonged.

The number of reference points on the polygon can be determined in the **No of Segments** field in the Ellipse Domain tool properties (see figure 1).

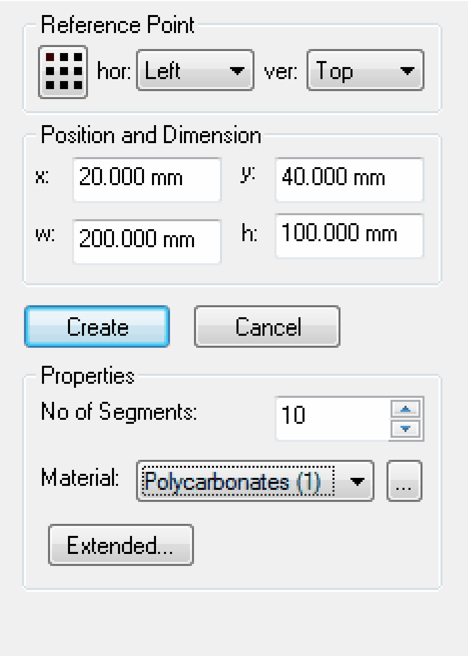


Figure 1

Limitations

- The maximum number of reference points for the polygon approximation is 99.

4.1.11 Polygon Domain Tool

Use   Limitations

Description

The ***Polygon Domain*** tool is the most flexible tool for creating material domains. It allows you to define a domain as non-regular polygons with any number of corner points.

How do I activate it?

The Polygon Domain tool can be activated with command **Polygon Domain** in the menu **Insert** or by clicking on the icon in the Toolbox flyout. When this tool is active, the mouse becomes a .

How do I use it?

For both the mouse and the keyboard, domains are created by entering the corner points one at a time. It is also possible to switch between mouse input and keyboard input while you are entering the domain.

Mouse

Activate the Polygon Domain tool and click on the spot where you would like the first corner point of the domain to be. Then click on spot where you would like the second corner point to be until you have created the domain in mind. To finish the input process and thus to close the domain, there are two possibilities:

- Click on the point where you began (the starting point of the visible line). When you reach this point with the mouse, the cursor should become a .
- Choose the command **Close** from the context menu (right click).

If you want to remove a point you have just defined, choose the command **Remove Previous** from the context menu (right click).

To interrupt the entire domain entry process, you can either choose **Cancel** from the context menu (right click) or you can hit the **ESC** key.

If, while you are drawing, you additionally press the **Shift** key, then the entry will be limited to the multiple of an element angle. First release the mouse button and then the Shift key. The element angle can be defined in the Option dialog window (**Application>Tools>General**).

Keyboard

After you have activated the Polygon Domain tool, make sure that the Polygon Domain tool properties are visible. If they are not visible you can activate them with menu command **View.Toolbox**. Hit the **Tab** key or click on the first field of the tool properties. You can activate the individual fields by repeatedly hitting the **Tab** key or by using the **Tab** key while the **Shift** key is held down.

Now you can enter the numerical coordinates of the first and subsequent points. For the numerical input, you can choose several different types of coordinates:

- **absolute cartesian:** You enter the x- and y- coordinates of the point.
- **absolute polar:** You enter the angle (to the x-axis) in the field denoted by a **w:** and the distance (radius) from the origin in the field marked with an **r:** .

Click on the **Next** button in order to enter the coordinates of the next point.

The subsequent points can also be entered with relative coordinates:

- **relative cartesian:** You enter the horizontal **dx:** and vertical **dy:** distance from the previous point.
- **relative polar:** You enter the angle (to the x-axis) and the distance (radius) from the previous point.

The **Create** button closes the domain, while the **Previous** button deletes the previous point.

Defining Materials

A new domain will always be created with the material, which is currently selected in the Materials table or the material drop down list, respectively. You can change the material of a domain you would like to create by either changing the material in the Materials table or by choosing the material from the materials drop down list 

in the tool properties.

You can also change the material of the domain afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

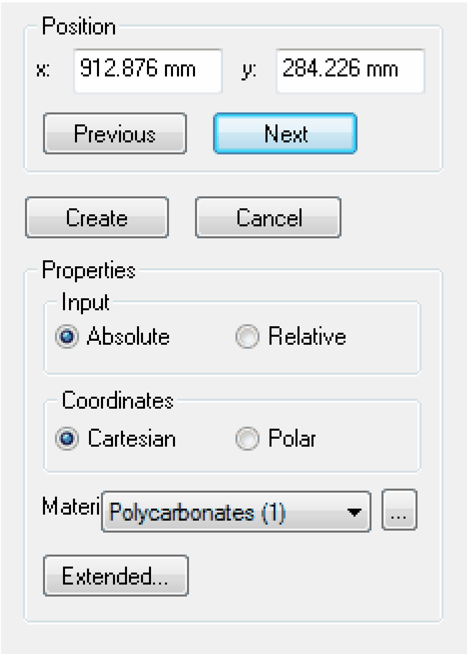


Figure 1

Limitations

- The domain must have at least three corner points.
- It is not possible to create edges where one edge of a domain cuts across another edge of the same domain.

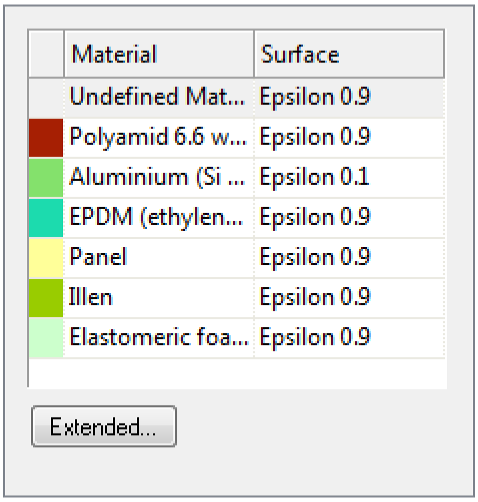
4.1.12 Air Cavity EN ISO 10077-2 Tool

frame , pro

Use Limitations

Description

With the ***Air Cavity EN ISO 10077-2*** tool you can create air cavity material domains according to the EN ISO 10077-2 standard. You can also assign emissivities (long-wave radiation properties) at the surface of cavities according a material surface properties table.



	Material	Surface
	Undefined Mat...	Epsilon 0.9
	Polyamid 6.6 w...	Epsilon 0.9
	Aluminium (Si ...	Epsilon 0.1
	EPDM (ethylen...	Epsilon 0.9
	Panel	Epsilon 0.9
	Illen	Epsilon 0.9
	Elastomeric foa...	Epsilon 0.9

Figure 1: Material Surface Properties table

The assignment of the surface emissivities is only possible, if the emissivities of the air cavities are calculated by flixo (cf. Cavity Options dialog window) and are not directly defined by the air cavities materials (cf. Material dialog window).

How do I activate it?

The Cavity Wizard can be activated with the command **Air Cavity** in the menu **Insert** or by clicking on the icon in the Toolbox flyout . When the tool is active the mouse cursor becomes a .

How do I use it?

Generating Cavities

With the Cavity Wizard, you define the material domains in which air cavities should automatically be created according to EN ISO 10077-2. The EN ISO 10077-2 differentiates between two categories of air cavity material domains:

- ***Unventilated*** air cavities
- ***Slightly ventilated*** cavities and grooves

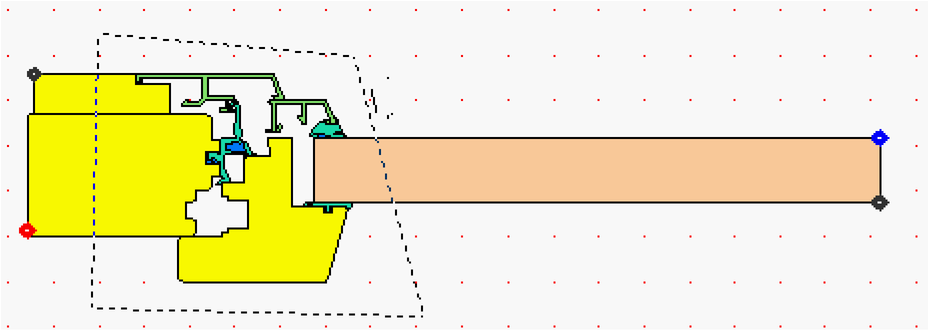


Figure 2: Domain definition

Activate the Cavity Wizard and click on the spot where you want to position the first corner point of the domain. Then click on the spot where you want the second corner point to be until you have created the domain in mind. To finish the input process and thus to close the domain, there are two possibilities:

- Click on the point where you began (the starting point of the visible line). When you reach this point with the mouse, the cursor should become a .
- Choose the command Close from the context menu (right click). The Cavity Wizard will automatically create the corresponding material domains.

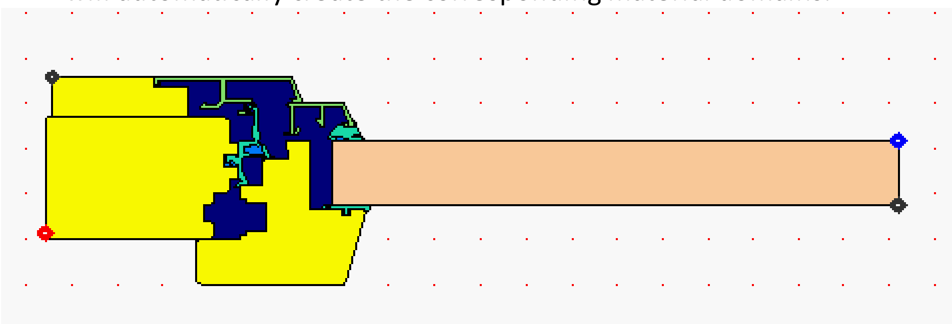


Figure 3: Air Cavities

If you want to remove a point you have just defined, choose the command **Remove Previous** from the context menu (right click).

To interrupt the entire domain entry process, you can either choose **Cancel** from the context menu (right click) or you can hit the **ESC** key.

If you additionally press the **Shift** key while you are drawing, then the entry will be limited to the multiple of an element angle. First release the mouse button and then the Shift key. The element angle can be defined in the Option dialog window (**Application>Tools>General**).

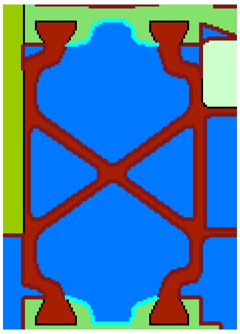
Defining Materials

The Cavity Wizard needs two air cavity materials - one for the ***unventilated*** and one for the ***slightly ventilated*** air cavities. The names of these materials are defined in the Options dialog window (**Application>Tools>Cavity Wizard**). If these materials do not exist in the document or the material type does not correspond, while defining the start point, you will be prompted with an Air Cavity Material dialog window to either choose existing air cavity material(s) from the Document Materials table or to create new air cavity material(s).

Optionally, you have the possibility to define small domains, which often arise as a consequence of drawing impressions, as "critical material" (see Special Materials Options dialog window) instead of being defined as "cavity material". These layers will then automatically be assigned with the material of a particular neighbor.

Assigning Surface Properties

Activate the Cavity Wizard and click into the cavity to whose surfaces you want to assign automatically the emissivities, based on the neighboring materials and the material surface properties table. The assignment takes place according to the present settings in the material surface properties table (cf. figure 1).



Picture 4: Air Cavity with automatically assigned surface properties (4 radiation boundary condition points)

Limitations

- The wizard polygon domain must have at least three corner points.
- It is not possible to create edges where one edge of a domain cuts across another edge of the same domain.
- There must be at least one unventilated air cavity material and one slightly ventilated air cavity material present in the document.
- It is only possible to assign surface emissivities, if the cavity emissivities are calculated by flixo (cf. Cavity Options dialog window) .

4.1.13 Boundary Condition Tool

Use

Description

The ***Boundary Condition*** tool allows you to define the boundary conditions and optionally (cf. Cavity Options dialog window) the radiation properties of the surfaces, which are needed for a complete definition of a physical problem.

How do I activate it?

The Boundary Condition tool can be activated with the menu command **Boundary Condition** in the menu **Insert** or by clicking on the icon in the Toolbox flyout. When the tool is active, the mouse cursor becomes a .

How do I use it?

When you have activated the tool, choose the desired boundary condition from the Boundary Condition flyout or from the Boundary Condition drop down list in the tool properties (see figure 3). Set the mouse on the spot on the edge of a domain from which the boundary condition should be valid, and click on the spot. If the option ***Only vertices with change of direction*** is checked, the boundary condition will be assigned to the nearest corner with change of direction. The boundary condition is now represented as a point, which has the color of the corresponding boundary condition (see Boundary Condition flyout). Only the start points of the boundary conditions are defined (see figure 1). The boundary conditions are shown deferred as polygons, if the corresponding option is selected in options dialog window (cf.User Interface Options dialog window). When you calculate the construction, the validity of the construction is also computed. After doing so, the boundary conditions will be shown with the corresponding color around them (see figure 2).

Following rotational direction rules apply for boundary conditions:

- General: The boundary condition of the starting point is valid for all edges, where there is material to the left of the boundary condition up to the next starting point (see figure 1 and figure 2).
- Exterior Boundaries: for all edges, the boundary condition runs ***counterclockwise*** from the starting point to the next starting point.
- Interior Boundaries: for all edges, the boundary condition runs ***clockwise*** from the starting point to the next starting point.

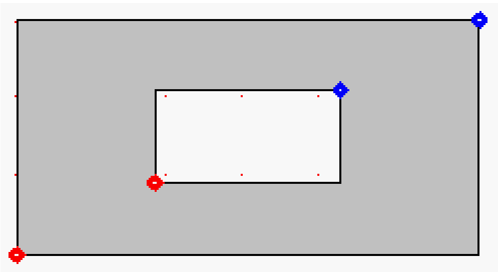


Figure 1

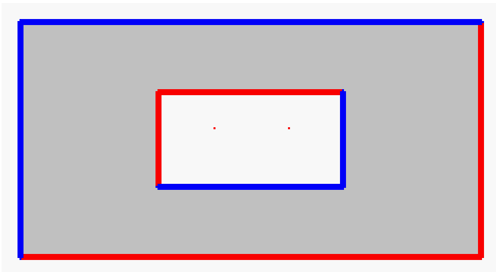


Figure 2

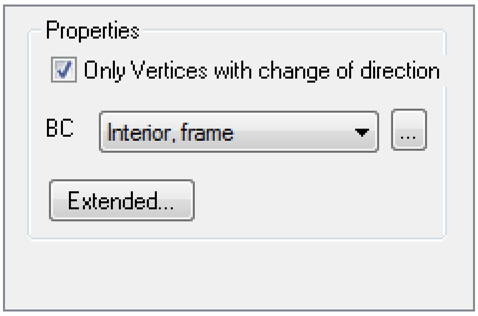


Figure 3

Defining Boundary Conditions

A new boundary condition is always created by the indicated boundary condition from the boundary condition drop down list (see figure 3). You can either change the selection in the Boundary Condition flyout or you can choose the respective boundary condition from the boundary condition drop down list in the tool properties.

The boundary condition can also be changed afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

4.1.14 Heat Source Tool

pro

Use   Limitations

Description

The ***Heat Source*** tool allows to define heat sources and heat sinks in a construction.

How do I activate it?

The Heat Source tool can be activated with the command **Heat Source** in the menu **Insert** or by clicking on the icon in the Toolbox flyout. When the tool is active, the mouse becomes a .

How do I use it?

Mouse

To create a heat source **two steps** are required: Click on the spot where the heat source should begin. The source is defined as soon as you click on the desired endpoint. You can interrupt the entry process by right clicking and selecting **Cancel** from the context menu or by hitting **Esc**.

Keyboard

For a keyboard input, make sure that the Properties toolbar is visible. If the toolbar is not visible you can activate it with the command **Toolbox** in the menu **View**. Activate the toolbar by hitting the **Tab** key or by clicking on one of its fields. You can activate the individual fields by repeatedly hitting the **Tab** key (or by hitting the **Tab** key while the **Shift** key is held down).

Enter the x and y coordinates of the start point into the appropriate fields, and then enter the coordinates of the endpoint (or the direction and length of the line). The coordinates can either be defined as Cartesian or polar coordinates. To end the entry click on the **Create** button.

Defining Styles

The power of a heat source can be defined in the Style list in the Heat Source tool properties. The list shows all available source powers that have previously been determined in the Boundary Conditions flyout.

A new source or sink will always be created with the power currently marked in the Style drop down list (see figure 1). You can either change the selection of the power in the Boundary Conditions flyout or in the Style drop down list in the tool properties.

The power of the source can also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

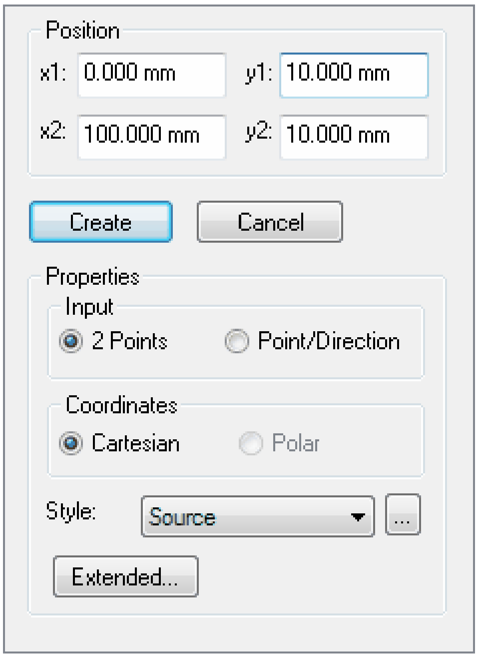


Figure 1: Tool properties

Limitiations

- Start- and endpoint have to be inside the same material domain.

4.1.15 Crop Tool

Use   Limitations

Description

The ***Crop*** tool allows you to crop the visible part of a result object, and hide parts of that same object. It is possible, for example, to display only a part of a construction in a report. After magnifying such a section, details will become more apparent.

The ***Crop*** tool also allows you to select a specific range of an imported DXF sketch which should be converted.

How do I activate it?

The Crop tool can be activated with the command **Crop** in the menu **Tools** or by clicking on the icon in the Toolbox flyout. When the tool is active the mouse cursor becomes a .

How do I use it?

Cropping (Mouse)

- Select the object that you would like to crop with either the Select, Move, Scale tool or with the Crop tool.
- Activate the Crop tool.
- Is the mouse cursor positioned over a small square at a corner of a result object, then the mouse cursor changes its symbol. By clicking and moving this corner with the mouse button held down, you can crop the object. When you release the mouse button, the object will be cropped.
- Alternatively, it is also possible to click on a point within the result object and create a rectangle. Consequently, only the domain within this area will be visible.
- After an object has been cropped, the visible domain can be altered by clicking on a point within this domain and by moving it with the mouse button held down.

Cropping (Keyboard)

Enter the following edges in the Crop tool properties: **Left:, Top:, Right**: and **Bottom:** (see figure 3) and then click on the **Crop** button or activate this key and hit Return.

The size of the edges correspond to the sizes in the model and not to the current drawing scale of the result object.

Undo Cropping

To undo a crop, choose either the menu command **Undo cropping** from the context menu (right click on the cropped result object) or by clicking on the **Remove** button in the Crop tool properties (see figure 3).

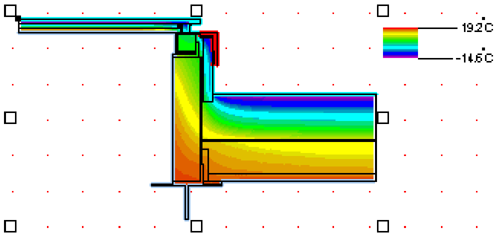


Figure 1

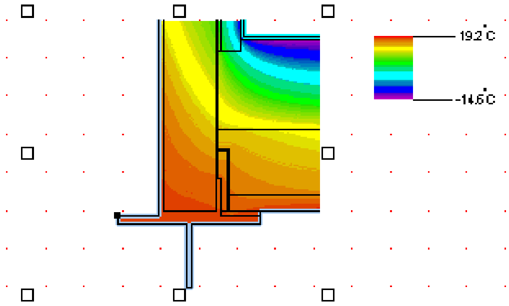


Figure 2

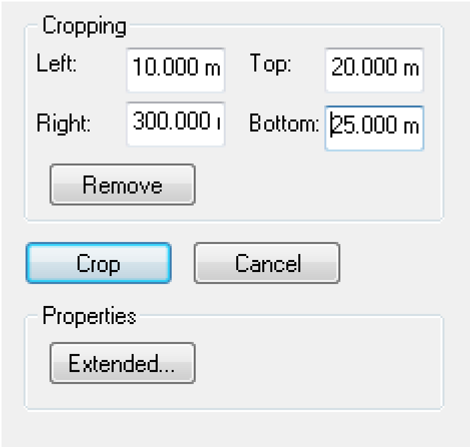


Figure 3

Limitations

- Only Result objects can be cropped.

4.1.16 Result Object Tool

Use

Description

With the ***Result Object*** tool you can create a result object on report pages and on the Master-Reports, displaying different results such as temperatures or isotherms.

How do I activate it?

The Result Object tool can be activated with command **Result Object** in the menu **Results**, or by clicking on the icon in the Toolbox flyout. When the tool is active, the mouse becomes a .

How do I use it?

After you have activated the Result Object tool, you can define a result object through two corner points of a rectangle that lie diagonally to each other. Click on a point, where you would like the first corner to be. When you now move the mouse, an outline of that result object appears and will constantly be adjusted until you click again, now defining the second corner point.

You can stop the entry process, by either right clicking and choosing the command **Cancel** or by hitting the **ESC** key.

If a model has not yet been calculated, the result object is simply a rectangular place holder. After the calculation, the place holder will automatically be filled with the result object according to the indicated Result Object styles properties.

If there are multiple models in the document, the model on which the result object refers can be selected in the **Model** list (cf. figure 1 and cf. figure 2)

Attributes that should be shown by default when a result object is created in flixo, can be defined in the corresponding Options dialog window (**Application>Tools>Result Object Tool**).

If additional legends, attributes or global results like isotherms need to be shown, you can adjust the object in the Result Object Properties dialog window. You can access the Result Object Properties dialog window by right clicking and choosing context menu command **Properties**, with the menu command **Edit.Properties**, or by hitting **Alt + Return**. Alternatively you can adjust the properties directly in the Properties flyout (cf. figure 1).

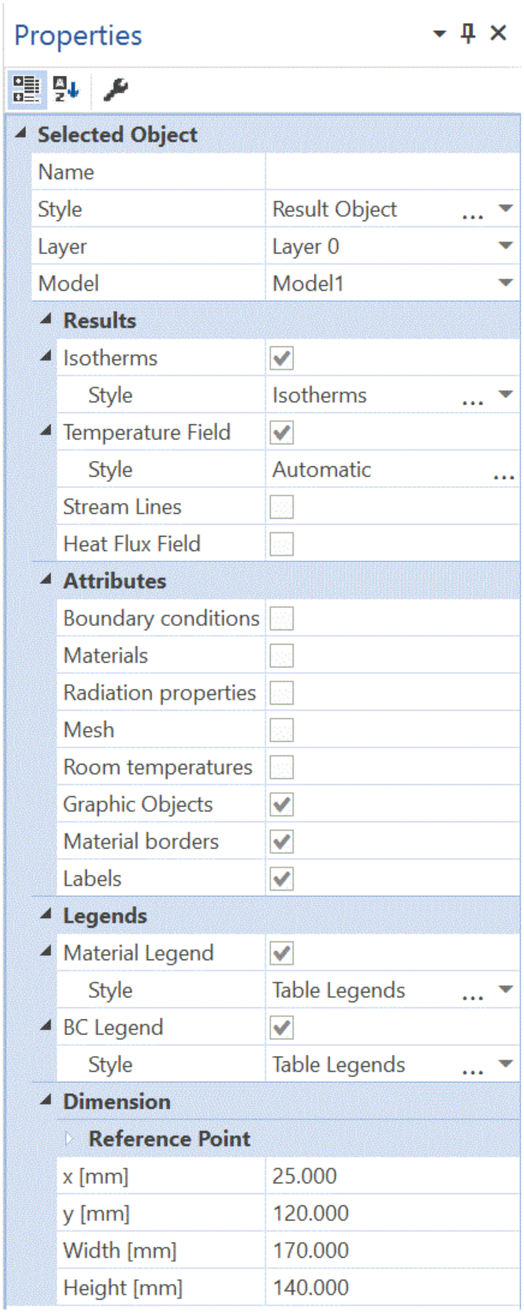


Figure 1: Properties flyout of a result object

Other properties of the result object, (e.g. display rulers, optional border around the object, line properties, position within the place holder), can be changed in the corresponding Style in the Styles flyout.

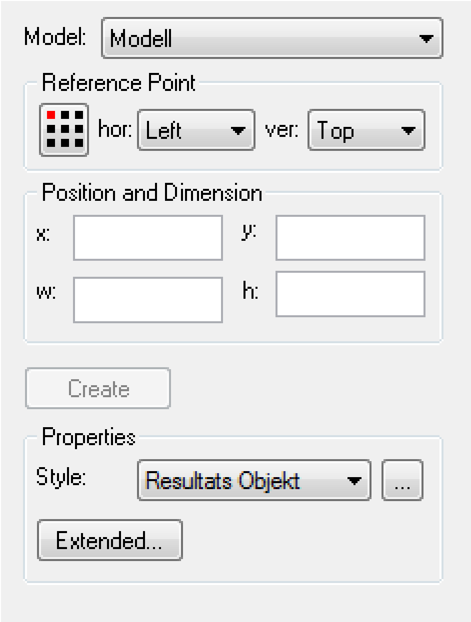


Figure 2

Defining Styles

The presentation of a result object can be defined in the Style list of the Result Object tool properties. The list shows all styles that have previously been determined in the Styles flyout.

A new result object will always be created with the styles currently marked in the Style drop down list (see figure 2). You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the Result Object tool properties.

The styles can be also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

4.1.17 Temperature Tool

Use Physics Limitations

Description

The ***Temperature*** tool lets you retrieve the local temperature at any given point on a calculated model. It also lets you label these local temperatures. The critical humidity (point at which surface condensation forms, and/or at which the predefined surface humidity is reached along with the temperature factor), the fRsi factor, and the surface humidity at a given humidity, are optionally calculated for points on the interior surface.

How do I activate it?

The temperature tool can be activated with the menu command **Temperature** in the menu **Results** or by clicking on the icon in the Toolbox flyout.

How do I use it?

Moving the mouse cursor over a result object allows you to see the local temperature. To add this local temperature to a report, click on the desired location. This will generate a label, which you can move to a desired location by not releasing the mouse button (see figure 1).

If you would like to interrupt the process before a label is created, you can do this with the **ESC** key.

The type of label (e.g. number of decimal places), the graphic properties of label lines, as well as the optional presentation of additional values of surface points can be changed with the corresponding Style in the Styles flyout.

With the Select, Move, Scale tool and Edit tool respectively, you can change the position of the label and its reference point.

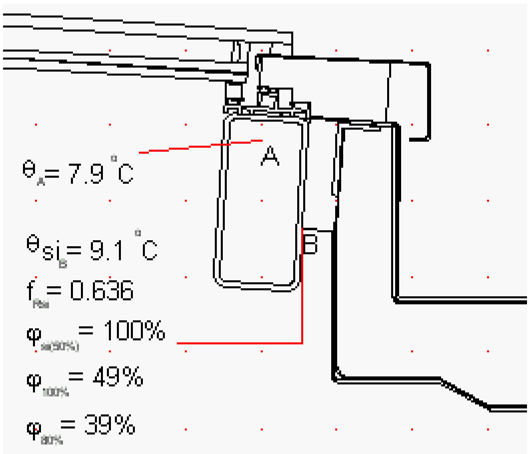


Figure 1: Label of a temperature

You can adjust the critical surface humidity and the humidity of the neighboring rooms, which influences the optional results, in the Properties flyout (cf. figure 2).

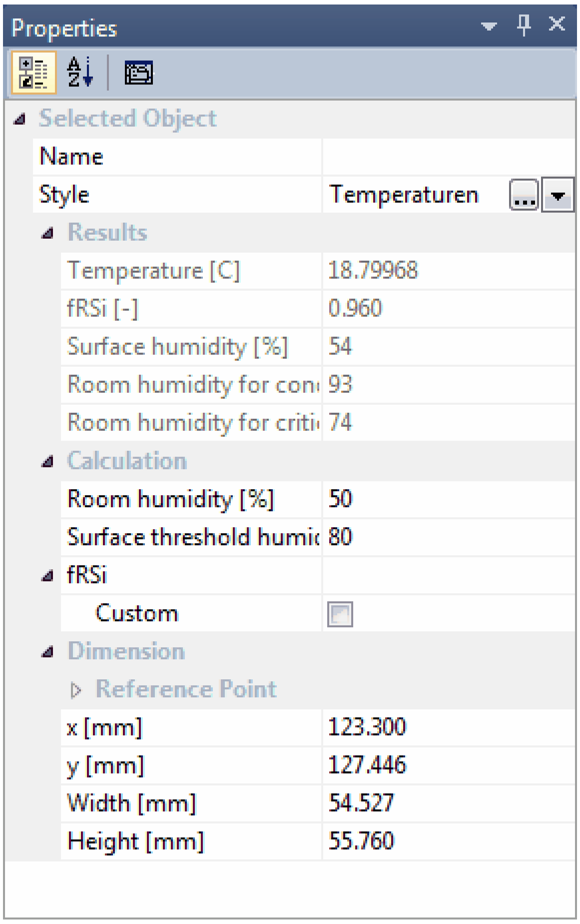


Figure 2: Properties flyout of a temperature object lying on the surface

Defining Styles

The presentation of an object can be defined in the Style list of the Temperature tool properties. The list shows all styles for temperature objects that have previously been determined in the Styles flyout.

A new temperature object will always be created with the styles currently marked in the Style drop down list (see figure 3). You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the Temperature tool properties.

The styles can also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

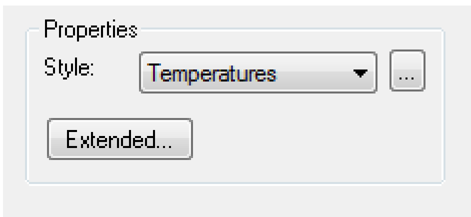


Figure 3: Temperature tool properties

Physical Explanation

For temperature points on the interior surface an additional ***temperature factor,*** the ***surface humidity*** and two ***critical room humidities*** can be displayed (see figure 1):

Temperature factor fRsi is calculated according to the following formula:

$$f_{Rsi} = \frac{\theta_{si\,\min} - \theta_e}{\theta_i - \theta_e}$$

And the temperature factor fRsi* for problems with more than 2 boundary condition temperatures is calculated according to the following formula:

$$f_{Rsi^*} = \frac{\theta_{si\,\min} - \theta_e}{\theta_i - \theta_{e\,\min}}$$

where:
θsi min: surface temperature [°C]
θi: interior boundary condition temperature [°C]
θe: exterior boundary condition temperature [°C]
θe min: lowest exterior boundary condition temperature [°C]

The external temperature, on which the calculation of the temperature factor fRsi is based, can be adjusted in the Properties Flyout (cf. Figure 2).

Surface humidity φsi(*%): This is the humidity on the surface of the temperature point for the room humidity given in the parentheses. The room humidity of the bordering room is defined in the Properties flyout. The surface humidity depends on the selected interior temperature and room humidity. Thus, the surface humidity must be respectively calculated for different interior temperatures and room humidities.

Room humidity φ100%, is shown where condensation forms. If the room humidity is equal or higher than the calculated room humidity φ100% (as in the example above 49%), then condensation forms at this spot. This value depends on the selected interior temperature and therefore the room humidity must be respectively calculated for different interior temperatures.

Room humidity φ80%, is shown at the surface temperature spot where the surface humidity is 80%. In the example above with a 39% room humidity there would be an 80% humidity at that minimum surface temperature spot. This value depends on the selected interior temperature and therefore the room humidity must be respectively calculated for different interior temperatures.

The critical surface humidity (e.g. to analyse a potential mildew problem) as well as the room humidity can be adjusted in the Properties flyout.

Limitations

- The temperature tool can only be used on report pages, and only if a calculated model result object is present.

- The critical room humidities and the surface humidity are only shown, if this option is selected in the corresponding Styles, if the point is on a boundary segment with a temperature/h-value boundary condition, and the temperature is lower than the adjacent room temperature.
- The temperature factor fRsi is only shown if there are exactly two boundary conditions in the model, and the temperature on the surface is lower than the adjacent room temperature.
- The temperature factor fRsi* is shown if there are more than two boundary conditions in the model, and the temperature on the surface is lower than the adjacent room temperature. In general, the temperature factor fRsi* cannot be used for other boundary condition temperatures.

4.1.18 Min. / Max. Temperature Tool

Use Physics Limitations

Description

The ***Min. / Max. Temperature*** tool lets you calculate and display the minimum, maximum and average ***surface temperatures,*** and the ***condensation zone*** along a surface line. For the minimum surface temperature spot, the ***fRsi factor,*** the ***surface humidity*** at a given humidity, and the ***critical humidity*** (point at which surface condensation forms, and/or at which predefined surface humidity is reached along with the temperature factor) are additionally calculated.

How do I activate it?

The Min. / Max. Temperature tool can be activate with the menu command **Min./Max. Temperature** in the menu **Results** or by clicking on the icon in the Toolbox flyout. When the tool is active, the mouse becomes a in areas of possible start and endpoints.

How do I use it?

The desired surface line is defined by entering a **start** and an **endpoint:**

Click on the boundary point, where the surface line should begin. The boundaries of the construction, which come into question for the definition of the surface line, will appear as a dashed line. Move the mouse to the desired endpoint. The dashed line now highlights those parts, for which the minimum, average, and maximum surface temperature and the fRsi factor, the condensation zone, the surface temperature, and critical room humidities for the minimum temperature is calculated and shown. The surface line is defined as soon as you click on the endpoint.

If the **SHIFT** key is pressed while you define the surface line, then the first click sets the **start** and **endpoints** automatically at the adjacent adiabatic edges (e.g. construction sections).

If the start and endpoints lie on the exterior boundary of a construction, then flixo calculates the minimum and, optionally, the maximum surface temperature (cf. Min. / Max. Temperature style dialog window on the **Extended** tab) along with the critical room humidity for the line segment, which leads counterclockwise from the start to the endpoint; if the start and endpoint lie on an interior boundary (e.g. the interior boundary of an entire chimney), then the calculation for the line segment is a carried out clockwise.

The values of the minimum and maximum surface temperatures can be displayed at the corresponding positions, in addition with the minimum temperature spot, the fRsi factor, the surface humidity as well as the critical room humidity. The start and endpoint of the surface line segment will be labeled as well.

If you would like to interrupt the input process before defining the endpoint, then use the context menu command (right click) Cancel or hit the ESC key.

With the Select, Move, Scale tool or Edit tool respectively, you can change the position of the label along with the start and endpoints.

The style of the label (e.g. the number of post decimal place holders, display of maximum temperatures, display of average temperatures), the optional display of condensation zones humidity, as well as the graphical properties of the label lines can all be changed through the corresponding Style in the Styles flyout.

You can adjust the critical surface humidity and the humidity of the neighboring rooms, the consideration of the open air cavities in the calculation and the calculation style of the fRSi-Value, which influence the optional results, in the Properties flyout (cf. figure 1).

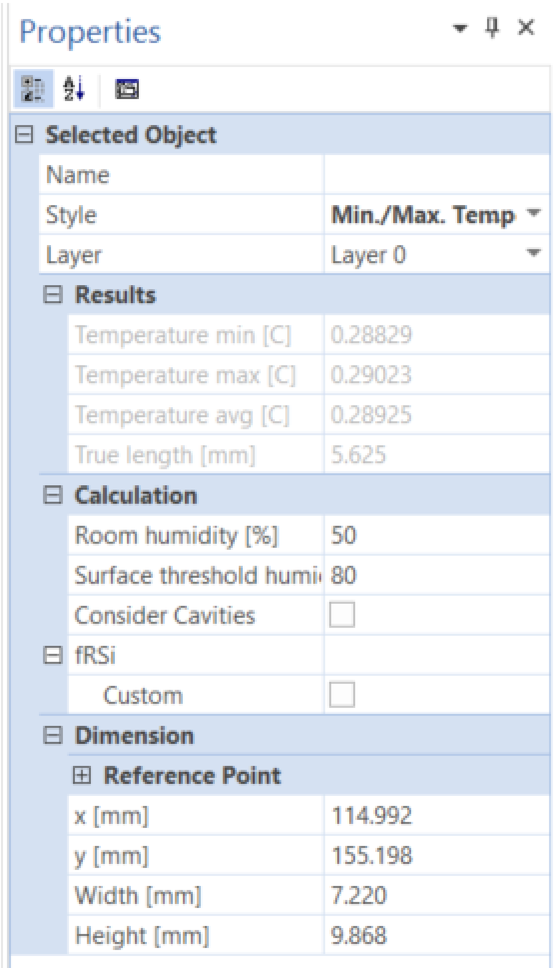


Figure 1: Properties flyout of a Min. / Max. Temperature object

Defining Styles

The presentation of a Min./Max. Temperature object can be defined in the Style list of the Min./Max Temperature tool properties (see figure 2). The list shows all styles for Min./Max. Temperature objects that have previously been determined in the Styles flyout.

A new Min./Max. Temperature object will always be created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or in the Style drop down list from the Min./Max. Temperature tool properties.

The styles can be also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

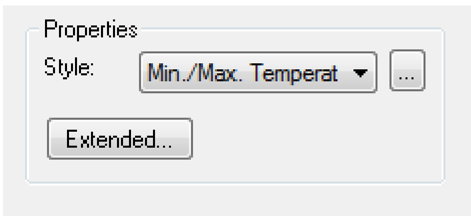


Figure 2: Min./Max. Temperature tool properties

Physical Explanation

In addition to the minimum temperature, the temperature factor, the surface humidity and the two critical room humidities can be displayed (cf. figure 3):

Temperature factor fRsi is calculated according to the following formula:

$$f_{Rsi} = \frac{\theta_{si\,\min} - \theta_e}{\theta_i - \theta_e}$$

And the temperature factor fRsi* for problems with more than 2 boundary condition temperatures is calculated according to the following formula:

$$f_{Rsi*} = \frac{\theta_{si\,\min} - \theta_e}{\theta_i - \theta_{e\,\min}}$$

where:
θsi min: calculated minimum surface temperature according to the defined surface line segment [°C]
θi: interior boundary condition temperature [°C]
θe: exterior boundary condition temperature [°C]
θe min: lowest exterior boundary condition temperature [°C]

The external temperature, on which the calculation of the temperature factor fRsi is based, can be adjusted in the Properties Flyout (cf. Figure 1).

Surface humidity φsi(*%): This is the humidity on the surface of the minimum temperature point for the humidity given in the parentheses. The room humidity of the bordering room is defined in the Properties flyout. The surface humidity depends on the selected interior temperature and room humidity. Thus, the surface humidity must be respectively calculated for different interior temperatures and room humidities.

Room humidity φ100%, is shown where condensation forms. If the room humidity is equal or higher than the calculated room humidity φ100% (as in the example above 49%), then condensation forms at this spot. This value depends on the selected interior temperature and therefore the room humidity must be respectively calculated for different interior temperatures.

Room humidity φ80%, is shown at the surface temperature spot where the surface humidity is 80%. In the example above with a 39% room humidity, there would be an 80% humidity at that minimum surface temperature spot. This value depends on the selected interior temperature and therefore the room humidity must be respectively calculated for different interior temperatures.

The critical surface humidity (e.g. to analyse a potential mildew problem) as well as the room humidity can be adjusted in the Properties flyout.

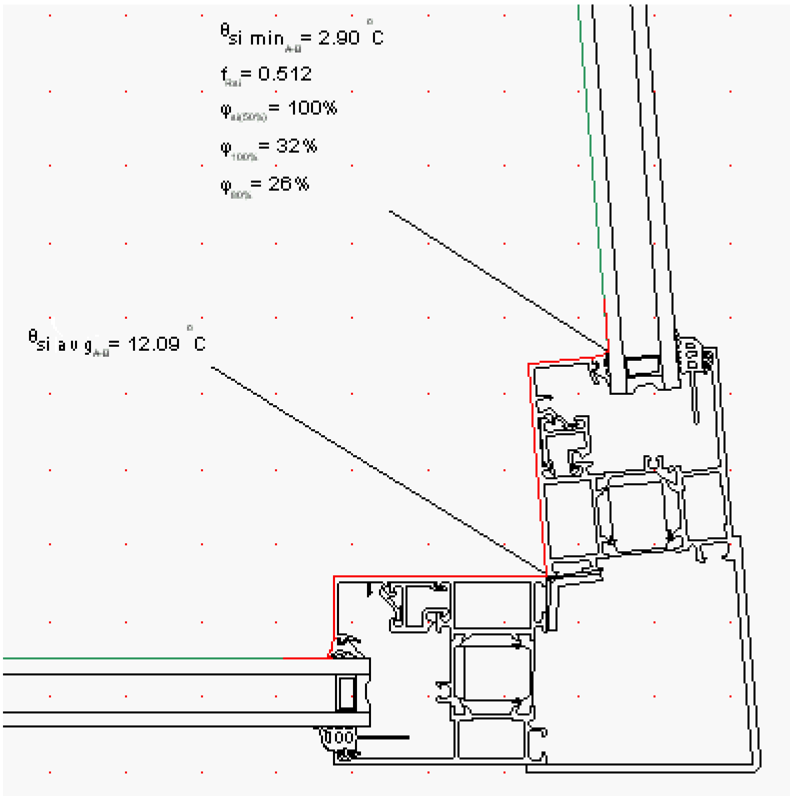


Figure 3

Limitations

- The Min./Max. Temperature tool can only be used on report pages, and only if a calculated model result object is present.
- The start and endpoints of the border line or the surface must be on the surface of the construction.
- The start and endpoints of the border line or the surface line must be able to be joined by a line.
- The critical room humidities and the surface humidity are only shown if the minimum temperature point is on a boundary segment with a temperature/h-value boundary condition, and the minimum temperature is lower than the adjacent room temperature.
- The temperature factor fRsi is only shown if there are exactly two boundary conditions in the model, and the minimum temperature is lower than the adjacent room temperature.
- The temperature factor fRsi* is shown if there are more than two boundary conditions in the model, and the minimum temperature is lower than the adjacent room temperature. In general, the temperature factor fRsi*cannot be used for other boundary condition temperatures.

4.1.19 Heat Flux Tool

Use Limitations

Description

The ***Heat Flux*** tool allows you to display the value and the direction of the heat flux at any given point on a result object.

How do I activate it?

The Heat Flux tool can be activated with the command **Heat Flux** in the menu **Results** or by clicking on the icon in the Toolbox flyout.

How do I use it?

The use of the Heat Flux tool is comparable with that of the Temperature tool. As soon as you move the mouse over the result object the heat flux of a given spot is displayed. To lock this value, click on the desired spot with the mouse button. This will create a label and an arrow indicating the direction of the heat flux (see figure 1).

With the Select, Move, Scale tool or Edit tool respectively, you can change the position of the label as well as the position of the start and endpoints.

The type of label (e.g. the number of post decimal placeholders) as well as the visual characteristics of the arrow can be changed by using the corresponding Style in the Styles flyout.

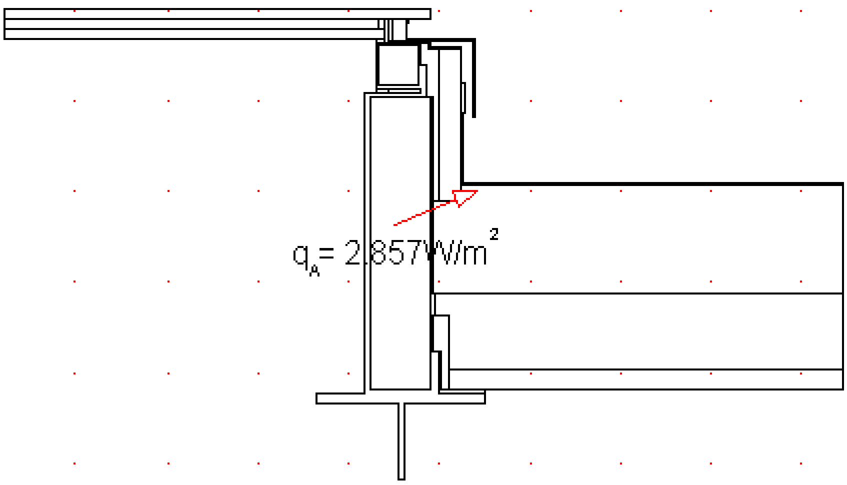


Figure 1: Heat Flux tool

The horizontal and vertical rate of the local heat flux are additionally displayed in Properties flyout (cf. figure 2).

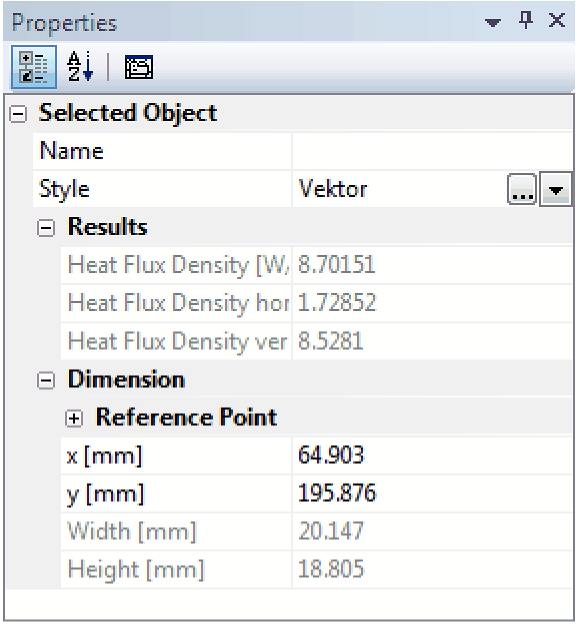


Figure 2: Properties Flyout of heat flux object

Defining Styles

The presentation of a heat flux object can be defined in the Style list in the Heat Flux tool properties (see figure 3). The list shows all styles for heat flux objects that have previously been determined in the Styles flyout.

A new heat flux object will always be created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the Heat Flux tool properties .

The styles can be also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

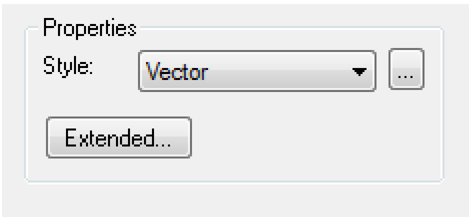


Figure 2: Heat Flux tool properties

Limitations

- This tool can only be used on the report page and only if result objects of a previously calculated model are present.

4.1.20 Heat Flow Tool

Use   Limitations

Description

The ***Heat Flow*** tool allows you to calculate and display the heat flow that flows through a particular surface line.

How do I activate it?

The Heat Flow tool can only be activated on report pages if a result object has been generated. The Heat Flow tool can be activated with the command **Heat Flow** in the menu **Results** or by clicking on the icon in the Toolbox flyout. When the tool is active the mouse becomes a in areas of possible start and endpoints.

How do I use it?

The surface line is defined through the input of the **start** and **endpoints:**

Click on one of the surface boundary points from which the surface line should begin. The borders of the construction, which will represent the surface line are then highlighted with a dashed line. Move the mouse to the desired endpoint. The dashed line now shows the section in which the heat flow will be calculated. The line is defined as soon as you click on the desired endpoint.

If the **SHIFT** key is pressed while you define the surface line, then the first click sets the **start** and **endpoints** automatically at the adjacent adiabatic edges (e.g. construction sections).

If the start and endpoints are on the exterior boundary of a construction, then <%APP% > calculates the heat flow from the start point **counterclockwise** ; if the surface line is on an internal boundary (e.g. the inside of a chimney), then the calculation is carried out **clockwise**.

If you wish to interrupt the entry before defining an endpoint, then you can use the command **Cancel** from the context menu (right click) or you can hit the **ESC** key.

The value of the heat flow calculation is displayed and the start and endpoints of the surface line are labeled. With the Select, Move, Scale tool or Edit tool respectively, you can change the position of the label as well as the position of the start and endpoints. The heat flow value is **positive** if it flows into the observed system, and **negative** if it flows out of the system. The unit [W/m] indicates Watts per linear meter constructional element perpendicular to the section.

The style of the label (e.g. the number of post-decimal place holders, critical surface humidity) as well as the graphical properties of the surface line can be changed by adjusting the corresponding Style in the Styles flyout.

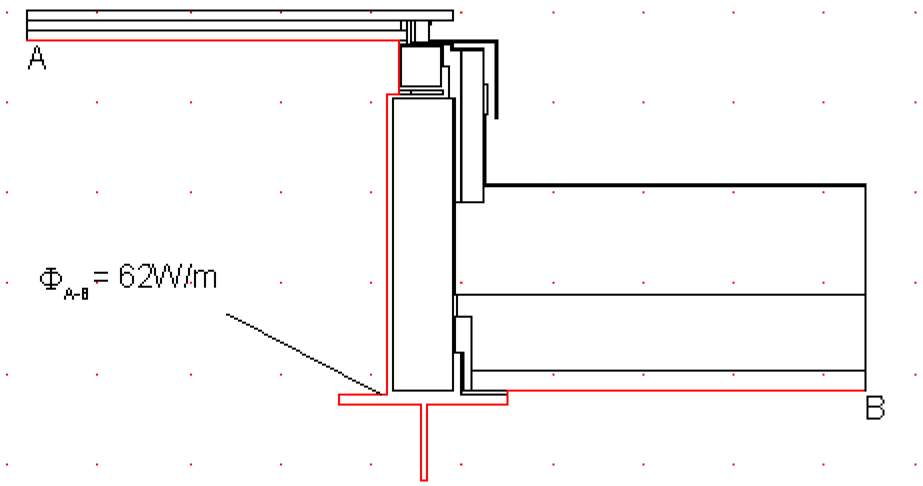


Figure 1

Beside the resulting heat flow through the surface polyline you find additionally the true length of the polyline in the Properties flyout (cf. figure 2).

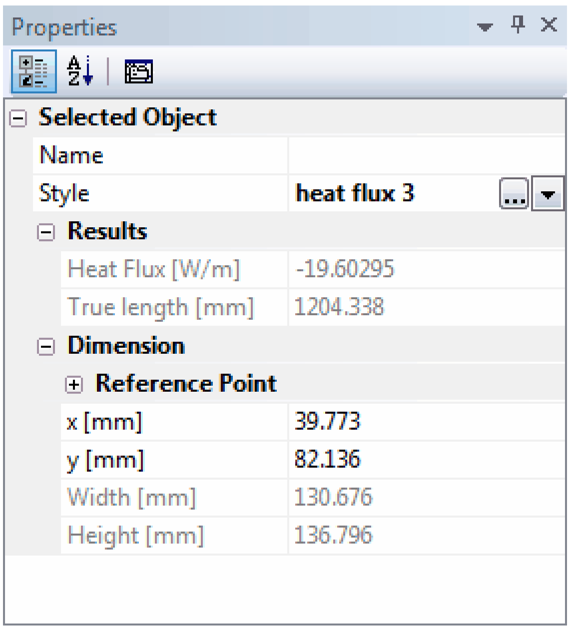


Figure 2: Properties flyout of a heat flow object

Defining Styles

The presentation of a heat flow object can be defined in the Style list in the Heat Flow tool properties (see figure 3). The list shows all styles for heat flow objects that have previously been determined in the Styles flyout.

A new heat flow object will always be created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the Heat Flow tool properties.

The styles can be also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

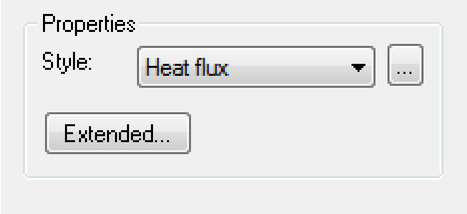


Figure 3: Heat Flow tool properties

Limitations

- This tool can only be used on the report page and only if result objects of a previously calculated model are present.
- Start and endpoints can only be mesh points. The surface line will automatically snap to the mesh point when it is drawn.
- Start and endpoints of the border line or of the surface line must be on the surface of the construction.
- The start and endpoints of the border line or the surface line must be able to be joined by a line.

4.1.21 U-Value Tool

Use

Description

The ***U-Value*** tool allows you:

- to prompt the ***U-Value*** for any given construction section
- to calculate ***Equivalent U-Values*** of constructions with periodic thermal bridges
- to calculate ***Frame U-Values*** according to EN ISO 10077-2 standard
- to calculate ***Joint U-Values*** of frame constructions according to EN ISO 12631

and to label them in the reports.

How do I activate it?

The ***U-Value*** tool can be activated with the command **U-Value** in the menu **Results** or by clicking on the icon in the Toolbox flyout. Then you have to define the U-value kind in the tool properties (cf. figure 1):

- Parallel Layers
- Equivalent U-Value
- Frame Uf-Value
- Joint UTJ-Value

How do I use it?

The use, meaningful application, physical explanations and limitations of the calculation kinds are described in different chapters:

- U-Value Tool Parallel layers
- U-Value Tool Equivalent U-Value
- U-Value Tool Frame Uf-Value
- U-Value Tool Joint UTJ-Value
- U-Value Tool Roller Shutter Box Usb-Value

Defining Styles

The presentation of an U-value object can be defined in the Style list in the U-Value tool properties (see figure 1). The list shows all styles for U-value objects that have previously been determined in the Styles flyout.

A new U-value object will always be created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the U-Value tool properties.

The styles can also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

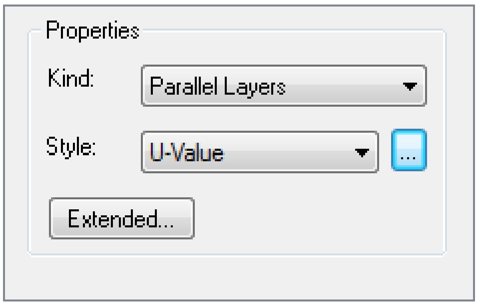


Figure 1: U-Value tool properties

Limitations

- This tool can only be used on the report page and only if result objects of a previously calculated model are present.

4.1.21.1 Parallel Layers

Use   Physics   Limitations

Description

The ***U-Value*** tool kind ***Parallel Layers*** allows you to prompt the U-value for any given construction section and creates a label of that value in the diagram.

How do I activate it?

The U-Value tool can be activated with the command **U-Value** in the menu **Results** or by clicking on the icon in the Toolbox flyout. Then select ***Parallel Layers*** as kind in the tool properties (cf. figure 2). When the tool is active, the mouse becomes a in the area of the construction.

How do I use it?

With a **first click** the point where the cross section line should be is defined. The direction of the cross section line should be perpendicular to the nearest edge. If the mouse button is held down a dashed preview appears as well as the label, which can be placed accordingly. You can define the ultimate position of the label by releasing the mouse button (see figure 1).

If you would like to interrupt the entry, then hit the ESC key.

With the Select, Move, Scale tool or Edit tool respectively, you can change the position of the label along with the position of the cross section line for which the U-value should be calculated.

The style of the label (e.g. the number of post decimal place holders) along with the graphical properties of the cross section line can be adjusted with the corresponding styles in the Styles flyout.

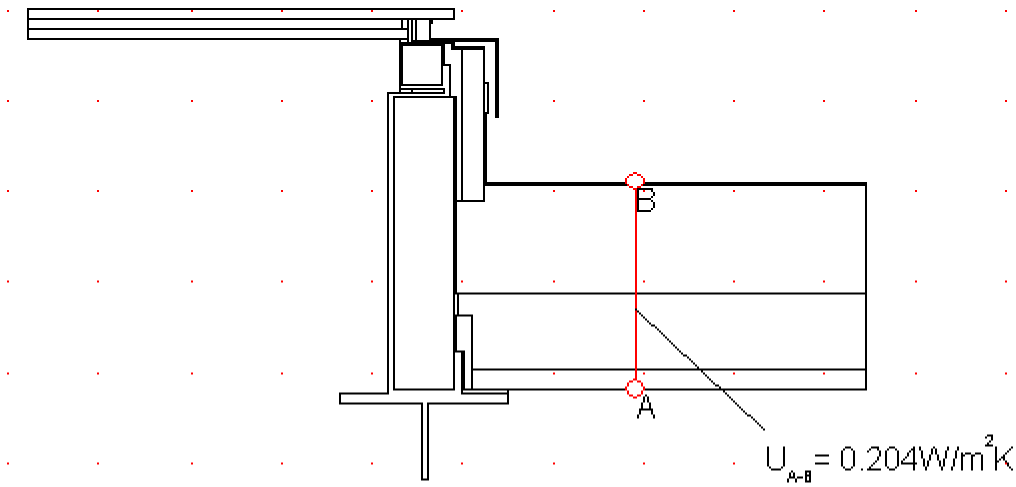


Figure 1

If the U-value should be calculated for other heat transfer resistances respectively - coefficients than the ones given in the model, you can adjust them in the Properties flyout (cf. figure 2).

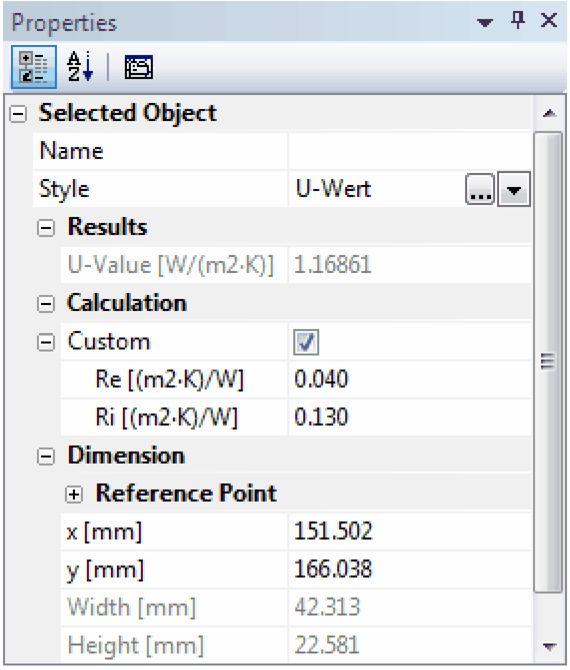


Figure 2: Properties flyout of an U-value object

Defining Styles

The presentation of an U-value object can be defined in the Style list in the U-Value tool properties (see figure 3). The list shows all styles for U-value objects that have previously been determined in the Styles flyout.

A new U-value object will always be created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the U-Value tool properties.

The styles can also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

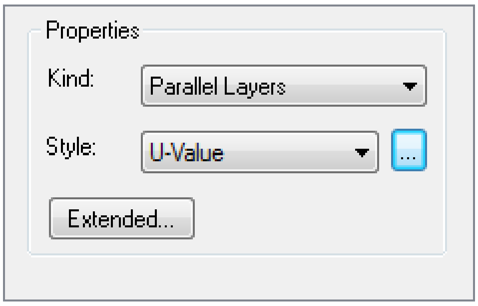


Figure 3: U-Value tool properties

Significant Results

To receive significant results the following conditions have to be fulfilled:

- U-values should only be calculated from examples with parallel layers (e.g. like in figure 1).

Physical Explanation

The **U-value** of a construction cross section of a layered component is calculated according to the following formula:

$$U = \frac{1}{\frac{1}{h_i} + \sum_j \frac{d_j}{\lambda_j} + \frac{1}{h_e}}$$

where:

U: U-Value [W/m2K]

h_i: interior heat transfer coefficient [W/m2K]

d_j: thickness of layer j [m]

λ_j: thermal conductivity of the layer j [W/mK]

h_e: exterior heat transfer coefficient [W/m2K]

Comments

- The measurement dj is based on the cross section line.
- A requirement is that all the layer boundaries are parallel and perpendicular to the cross section line as in figure 4. The calculation of the U-value, therefore, only makes sense in the boundary areas and in the middle of this construction.

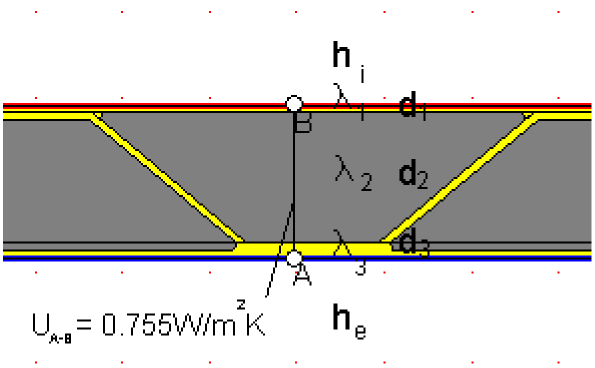


Figure 4

Limitations

- This tool can only be used on the report page and only if result objects of a previously calculated model are present.

4.1.21.2 Equivalent U-Value

energy, energy plus, pro

Use Physics Limitations

Description

The ***U-Value*** tool ***Equivalent U-Value*** allows you to calculate and display ***equivalent U-values***. The equivalent U-value takes periodic thermal bridge occurrences into account (e.g. the joist in figure 2).

How do I activate it?

The ***U-Value*** tool can be activated with the command **U-Value** in the menu **Results** or by clicking on the icon in the Toolbox flyout. Then select ***Equivalent U-Value*** as kind in the tool properties (cf. figure 1). When the tool is active, the mouse becomes a ⌖.

How do I use it?

To calculate the equivalent U-value over a surface edge, **three steps** are required:

With the first two clicks (at points A and B in the example at hand) you set the cross section line, thus defining the surface edge over which the equivalent U-value is to be calculated. After you have moved the mouse outside of the construction area, and a dashed preview appears, define with a third click the position of the measurement lines.

If the **SHIFT** key is pressed while you define the surface line, then the first click sets **start** and **endpoints** automatically at the adjacent adiabatic edges (e.g. construction sections).

The first click should be set near the boundary where the bordering surface line should begin. With the second click you define the parallel cross section and confine the bordering surface line.

Defining Styles

The presentation of an equivalent U-value object can be defined in the Style list in the Equivalent U-Value tool properties (see figure 1). The list shows all styles for equivalent U-value objects that have previously been determined in the Styles flyout.

A new equivalent U-value object will always be created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the Equivalent U-Value tool properties.

The styles can also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

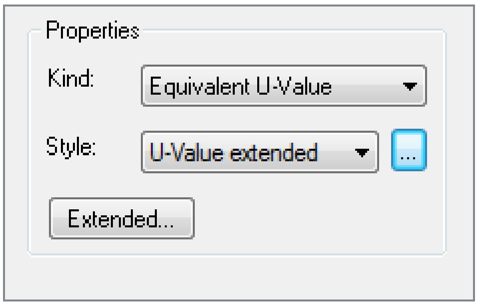


Figure 1: Equivalent U-Value tool properties

Accurate Results

To receive accurate results, the following conditions must be fulfilled:

- equivalent U-values should only be calculated for constructions where thermal bridges appear periodically (cf. figure 2).
- the cross section line should be defined within sections, through which no heat flow flows (e.g. symmetrical axes or on edges where there the boundary condition: heat flux q=0.0 W/m2K is set).
- If you choose an exterior reference system, start and end points should lie on the exterior boundary of the construction. If you choose an interior reference system, start and end points should lie on the interior boundary of the construction. Make note of the order of the inputs: also here endpoint inputs are entered counterclockwise.
- Also consider the Comments section in the chapter Physical Explanation .

If you would like to interrupt the input before defining the 3rd point, you can use context menu command (right click) **Cancel** or you can hit the **ESC** key.

With the Select, Move, Scale toolor Edit tool respectively, you can change the position of the label as well as the position of the section.

The type of label (e.g. number of post-decimal place holders, whether the heat flow should be calculated through the section surface) as well as the graphical properties of the cross section line can be adjusted with the corresponding styles in the Styles flyout.

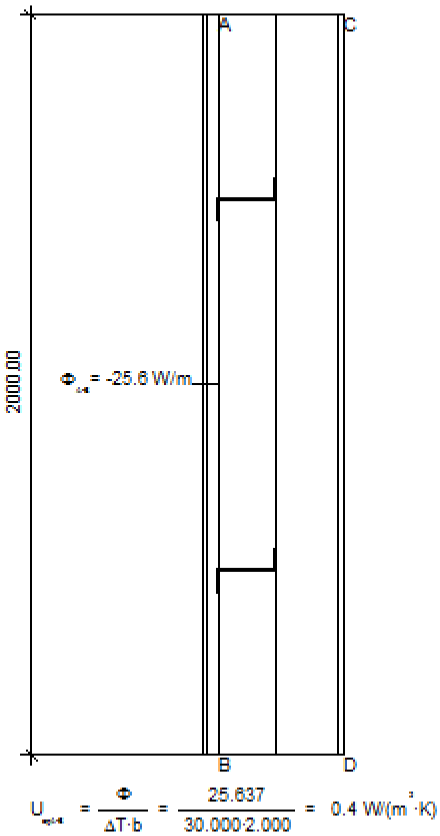


Figure 2

Physical Explanation

The equivalent U-value (previously equivalent K-value) is calculated as follows:

$$U_{eqA-B} = \frac{\Phi_{A-B}}{b \cdot \Delta T}$$

and calculated as follows if the heat flux through the section should be taken into account (cf. figure 4, Properties flyout)

$$U_{eqA-B(C-A,B-D)} = \frac{\Phi_{A-B} + \Phi_{C-A} + \Phi_{B-D}}{b \cdot \Delta T}$$

where:

U_{eq}: equivalent U-value [W/m2K]
Φ_{A-B}: heat flow from A to B (counterclockwise) [W/m]
Φ_{C-A}: heat flow from C to A (counterclockwise) [W/m]
Φ_{B-D}: heat flow from B to D (counterclockwise) [W/m]
b: length of the projection of the distance line AB on the perpendicular to the section line AC [m]

ΔT: temperature difference TA-TC of the boundary condition temperatures TA und TC by the points A and C [K]

Comments

- In general, the equivalent U-values are dependent on the side of a construction (interior or exterior) for which the value is being calculated. The following relations are valid:

Ueq A-B, exterior= Ueq A-B (C-A,B-D), interior   and   Ueq A-B, interior = Ueq A-B (C-A,B-D), exterior

- In case the heat flow through the section surface exceeds a critical, relatively high value, then an appropriate warning will be shown. You can adjust these borders in the Properties flyout (cf. figure 4)
- The heat flows will be calculated counterclockwise from start to endpoint.
- To calculate the heat flow, all reference points will automatically be shown at the nearest mesh point on the surface.
- The sign of the heat flow is taken into account: heat flows into the system are positive; heat flows out of the system are negative.

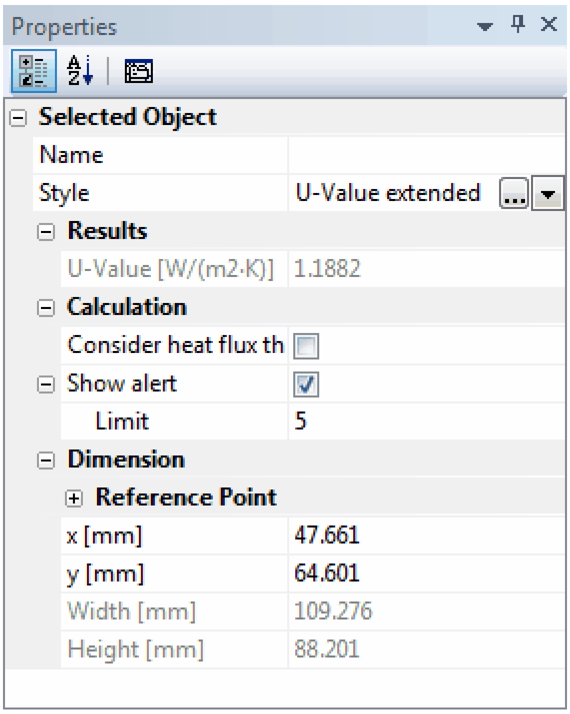


Figure 4: Properties flyout of a equivalent U-value object

Limitations

- This tool can only be used on the report page and only if result objects of a previously calculated model are present.
- The **temperature differences** must have the **same values** for both sections.

- Only **2 boundary condition temperatures** can be present in the model.
- Neither **interior borders** nor **heat sources** can be present.

4.1.21.3 Frame U-Value

frame , pro

Use Physics Limitations

Description

With the ***U-Value*** tool ***Frame U-Value*** you can calculate and display ***Frame Uf-Values*** according to EN ISO 10077-2.

How do I activate it?

The ***U-Value*** tool can be activated with the command **U-Value** in the menu **Results** or by clicking on the U icon in the Toolbox flyout. Then select ***Frame Uf-Value*** as kind in the tool properties (cf. figure 3).

How do I use it?

Click with the tool on the result object for which the Uf-value should be calculated. If flixo recognizes the construction as a frame according to EN ISO 10077-2 then a dialog window opens.

For the calculation of the window frame U-value according to the European standard EN ISO 10077-2, the glass unit has to be replaced by a panel (cf. figure 1) with the following characteristics:

- The thickness of the panel should correspond to the thickness of the glass unit.
- The thickness of the air cavity between the panel and the frame (cf. b1) should at least be 5mm thick.
- The glass unit rebate (cf. b2) of the panel should be 15 mm at the most.
- The visible part of the panel (cf. bp) should at least be 190 mm.
- According to EN ISO 10077-1, the measurements b2 and bp should correspond to the larger of the two projected widths of the frame section, without taking into account the stripping between the frame and glass unit or panel (cf. figure 1).
- The thermal conductivity of the panel should be 0.035 W/(mK).

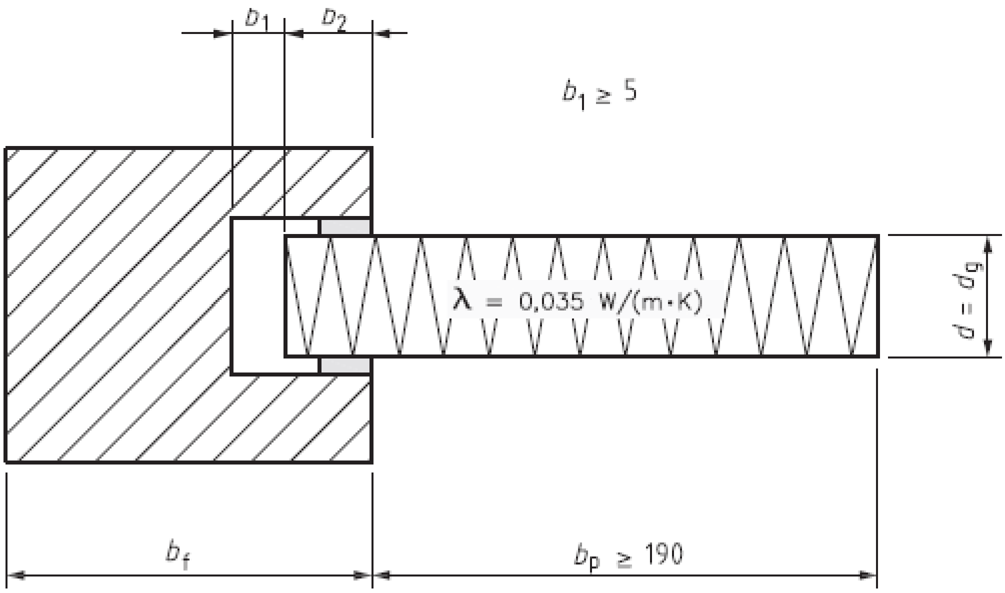


Figure 1: Preconditions for a Uf-value calculation

You have to define the frame materials which determine the frame measurements in the opening dialog window. Alternatively the kind and orientation of the window can be set in the dialog window.

The type to determine the frame width (**Automatic, 2 Constructions or 3 Constructions**) can be customized later in the Properties flyout (cf. figure 2). If the frame width is determined automatically based on the selected frame materials, the method to calculate the frame width (**Max. of projections, Cold side projection, Warm side projection**), as well as the frame materials themselves, can be customized later in the Properties flyout (cf. figure 2) too.

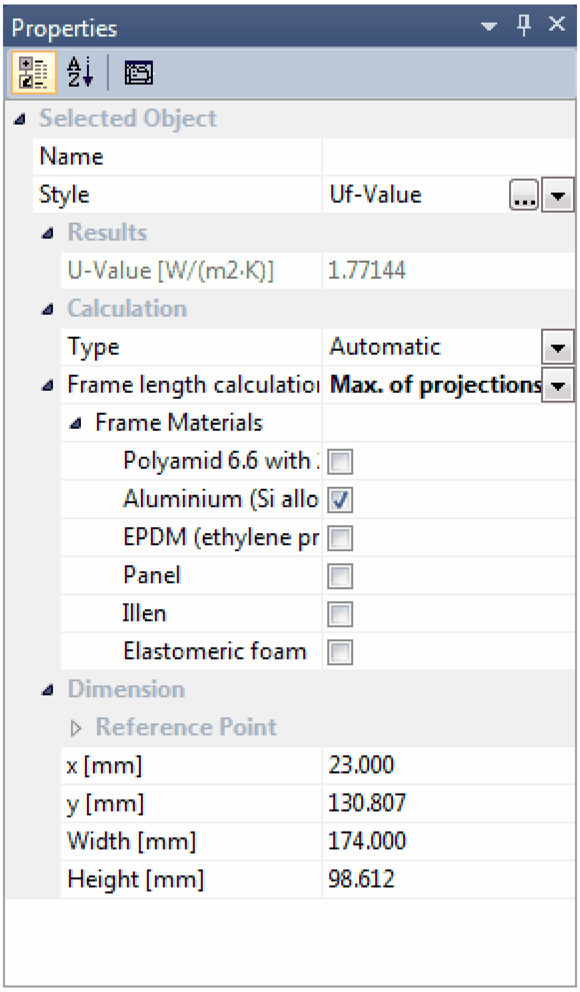


Figure 2: Properties flyout for an Uf-value object

The calculation of a frame U-value is explained in detail in tutorial 4.

In special cases, if the frame width can't be exactly defined with the materials (e.g. "Z-Profiles"), you can also adapt the dimension manually by moving the position of the start points of the dimension objects using the Select, Move, Scale tool or the Edit tool respectively. You can also change the text positions of the help objects, of the U-values of the base construction and the dimension lines with the same tool.

Defining Styles

The presentation of an Uf-value object can be selected in the Style list in the tool properties (cf. figure 3). The list shows all styles for Uf-value objects that have previously been determined in the Styles flyout.

A new Uf-value object is always created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or select the according style in the Style drop down list of the tool properties.

The styles can also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (cf. tutorial 1).

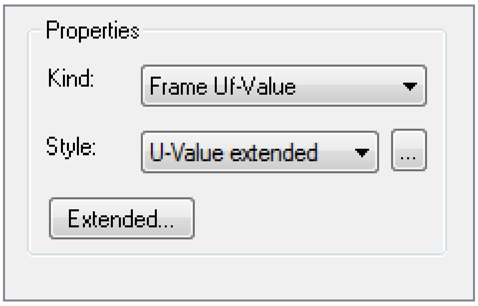


Figure 3: Tool Properties

Physical Explanations

The frame U-value is calculated according to EN ISO 10077-2 as follows:

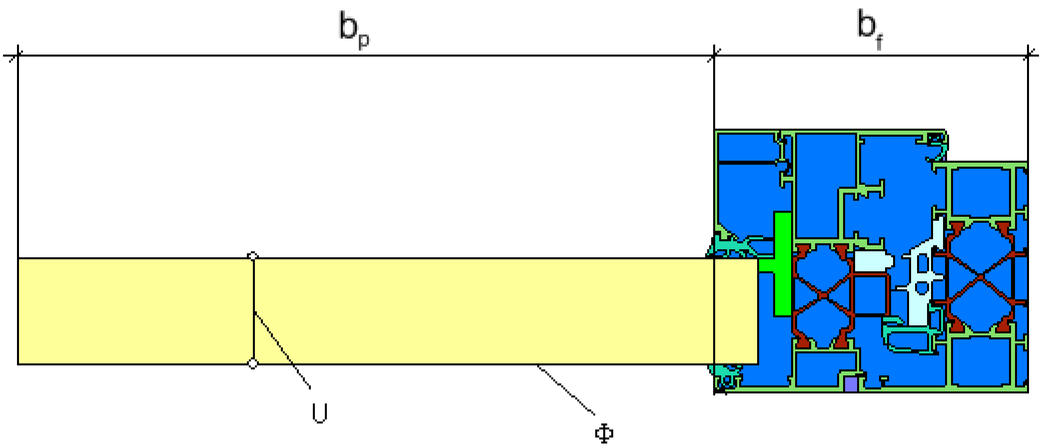


Figure 4: Calculation of Uf-values

$$U_m = \frac{\frac{\Phi}{\Delta T} - U_p \cdot b_p}{b_f}$$

with:

- Φ: Total heat flow in to or out of the model [W/m]
- ΔT: Temperature difference [K]
- U_p: U-value of the panel [W/m²K]
- b_p: Width of the panel [m]
- b_f: Width of the frame [m]

If the calculated construction is a mullion or a transom, than 2 panels are used instead of the glazings, the formula is as follows:

$$U_{f,B} = \frac{\frac{\Phi}{\Delta T} - U_{p1} \cdot b_{p1} - U_{p2} \cdot b_{p2}}{b_f}$$

Limitations

This option is only available for constructions, which are recognized as window frame according to EN ISO 10077-2:

- The thermal conductivity of the panel is **0.035 W/(mK)**.
- The visible part of the **panel is at least 190 mm**.
- There are exactly **2 boundary condition temperatures**.
- There are exactly **2 adiabatic boundary conditions** ("Symmetry/Model Section") each one on both model sections.
- Either it's a **horizontal** or a **vertical** window

4.1.21.4 Joint U-Value

pro
Use Physics Limitations

Description

With the ***U-Value*** tool ***Joint U-Value*** you can calculate and display ***Joint U-Values*** according to EN ISO 12631.

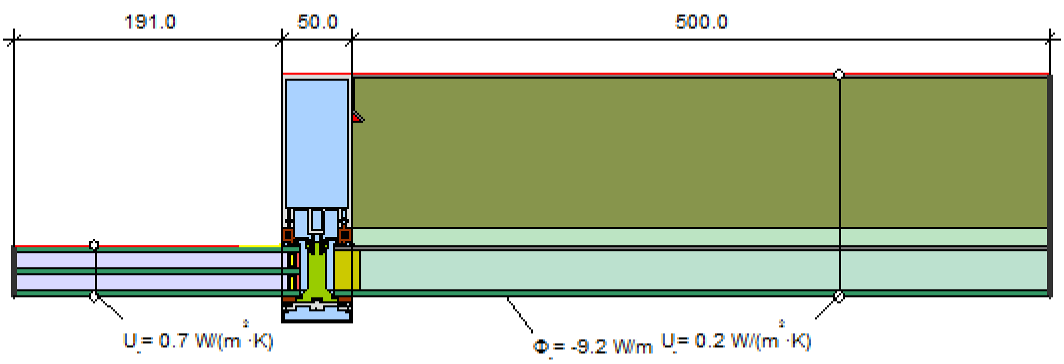
How do I activate it?

The ***U-Value*** tool can be activated with the command **U-Value** of the menu **Results** or by clicking on the U icon in the Toolbox flyout. Then select ***Joint UTJ-Value*** as kind in the tool properties (cf. figure 3).

How do I use it?

Click with the tool on the result object for which the Joint U-value should be calculated. Unlike to the Uf-value the given construction including glazing and spacer is analysed.

Then the frame materials which define the frame measurements have to be determined in the opening dialog window. Consider that these materials should only be used ***exclusively*** in the frame. If necessary, you have to create duplicates of materials (e.g. aluminium frame and aluminium panel) and you have to assign them to the according domains on the model page.



$$U_{TJAS} = \frac{\frac{\Phi}{\Delta T} - U_{g1} \cdot b_{g1} - U_{g2} \cdot b_{g2}}{b_f} = \frac{\frac{9.242}{20.000} - 0.700 \cdot 0.191 - 0.238 \cdot 0.500}{0.050} = 4.19\ W/(m^2 \cdot K)$$

Figure 1: Example of a Joint U-Value object

You have to define the frame materials which determine the frame measurements in the opening dialog window. Alternatively the kind and orientation of the frame can be set in the dialog window.

The type to determine the frame width (A**utomatic, 2 Constructions or 3 Constructions**) can be customized later in the Properties flyout (cf. figure 2). If the frame width is determined automatically based on the selected frame materials, the method to calculate the frame width (**Max of projections, Cold side projection, Warm side projection**), as well as the frame materials themselves, can be customized later in the Properties flyout (cf. figure 2) too.

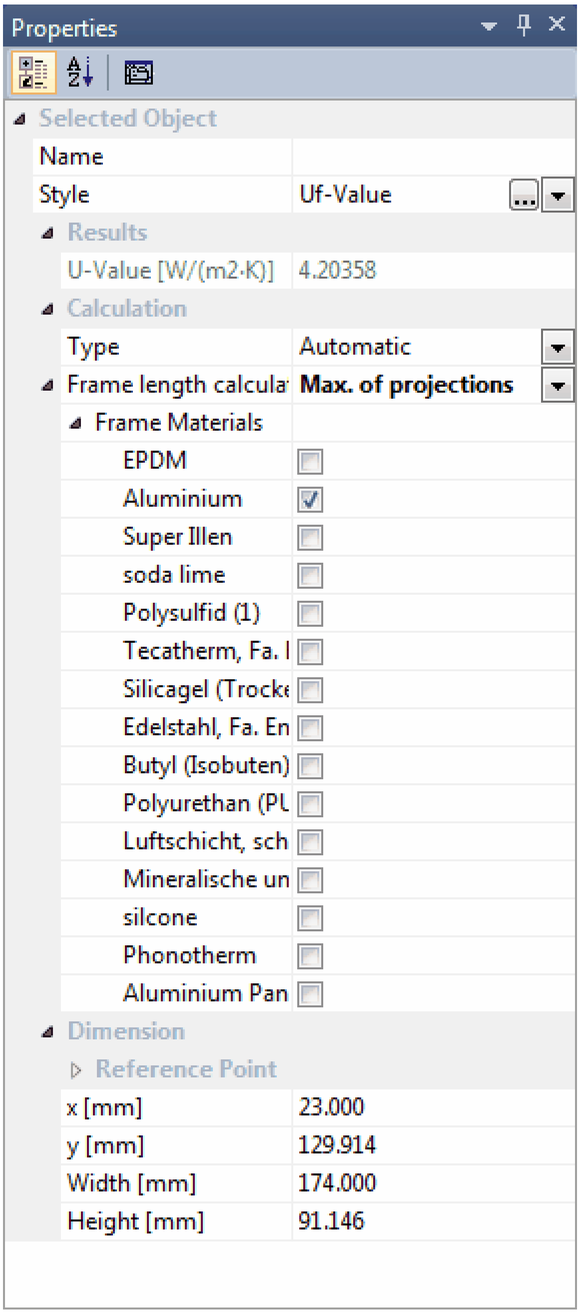


Figure 2: Properties flyout for the joint U-value object

The definition of a joint U-value is explained in detail in tutorial 5.

In special cases, if the frame width can't be exactly defined using the frame materials (e.g. "Z-Profiles"), you can also adapt the dimension manually by moving the position of the start points of the dimension objects using the Select, Move, Scale tool or the Edit tool respectively. You can also change the text positions of the help objects, of the U-values of the base construction and the dimension lines with the same tool.

Defining Styles

The presentation of a joint U-Value object can be selected in the Style list in the tool properties (cf. figure 3). The list shows all styles for joint U-value objects that have previously been determined in the Styles flyout.

A new joint U-value object is always created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or select the according style in the Style drop down list of the tool properties.

The styles can also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (cf. tutorial 1).

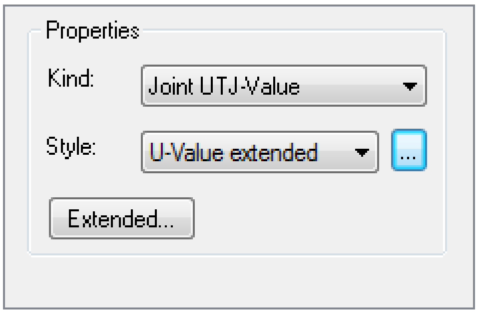


Figure 3: Tool Properties

Physical Explanations

The joint U-value is calculated according to EN ISO 12631 as follows:

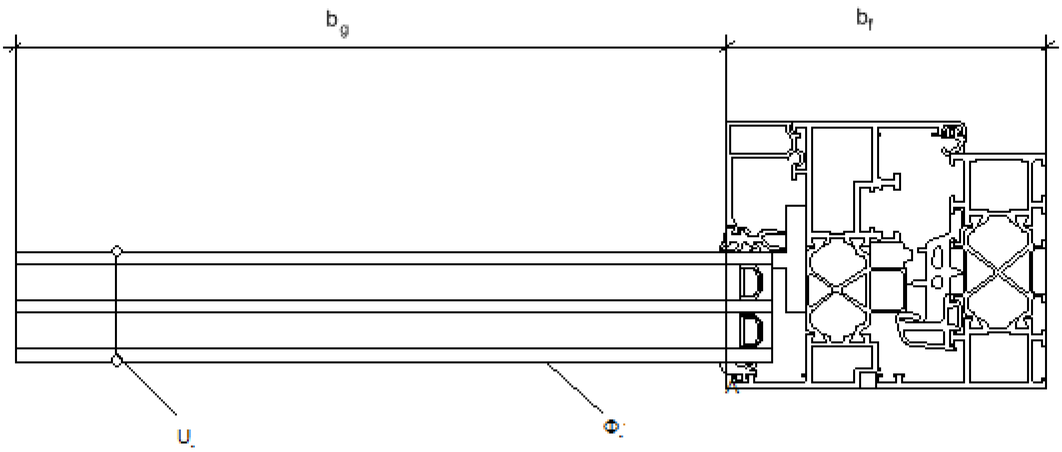


Figure 4: Calculation of a joint U-value

$$U_{TJ} = \frac{\frac{\Phi}{\Delta T} - U_g \cdot b_g}{b_f}$$

with:

Φ: Total heat flow in to or out of the model [W/m]

ΔT: Temperature difference [K]

U_g: U-Value of the panel or glazing [W/m²K]

b_g: Width of the glazing or the panel [m]

b_f: Width of the frame [m]

If the calculated construction is a mullion or a transom, than 2 glazing or panels are used, the formula is as follows:

$$U_{TJA,B} = \frac{\frac{\Phi}{\Delta T} - U_{g1} \cdot b_{g1} - U_{g2} \cdot b_{g2}}{b_f}$$

Limitations

Joint U-values can only be calculated for constructions, which are recognized as window frames according to EN ISO 12631:

- Materials which defines the frame dimension can occur exclusively in the frame (if necessary, you have to create duplicates of materials).
- There are exactly **2 boundary condition temperatures**.
- There are exactly **2 adiabatic boundary conditions** ("Symmetry/Model Section") each one on both model sections.
- Either it's a **horizontal** or a **vertical** window

4.1.21.5 Roller Shutter Box U-value

pro

Use Physics Limitations

Description

The ***U-Value*** tool ***Roller Shutter Box U-Value*** allows you to calculate and display ***roller shutter box U-Values*** according to standard EN ISO 10077-2.

How do I activate it?

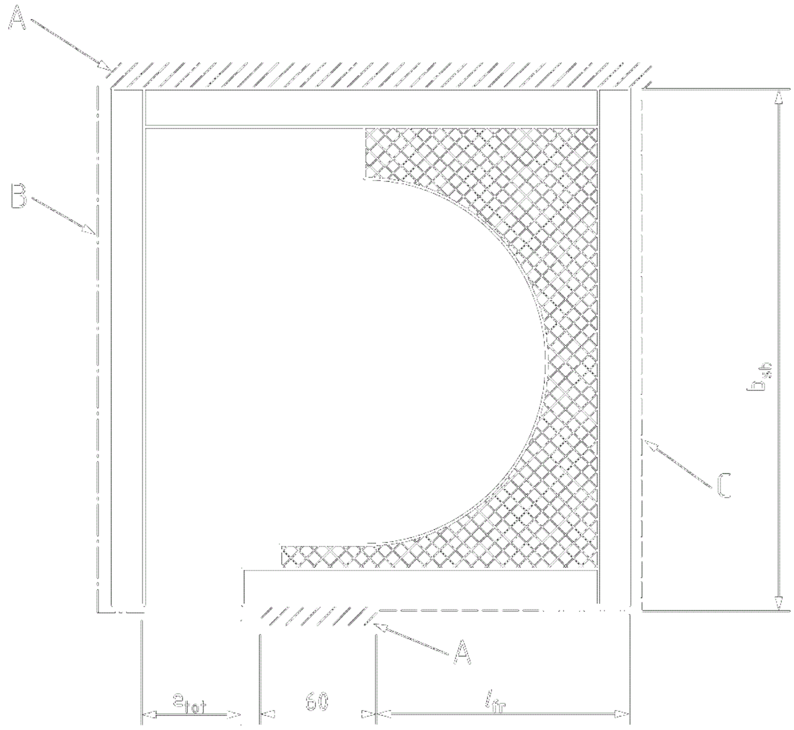
The ***U-Value*** tool can be activated with the command **U-Value** in the menu **Results** or by clicking on the icon in the Toolbox flyout. Then select ***Shutter Box Usb-Value*** as kind in the tool properties (cf. figure 2).

How do I use it?

Click with the tool on the result object for which the Usb-Value should be calculated. If flixo recognizes the construction as a roller shutter box according to EN ISO 10077-2, then the Usb-Value will be calculated and displayed, otherwise a corresponding error message will be displayed.

For the calculation of the roller shutter box U-value according to the European standard EN ISO 10077-2, following preconditions have to be fulfilled:

- Top of roller shutter box: adiabatic.
- At the bottom of the roller shutter box where it adjoins the window frame: adiabatic for a distance of 60 mm.
- Surface adjacent to the internal environment: Surface resistance of 0.13 m²K/W.
- Surface adjacent to the external environment: Surface resistance of 0.04 m²K/W.



The method to calculate the frame width (**Max. of projections, Cold side projection, Warm side projection**) can be customized later in the Properties flyout (cf. figure 1).

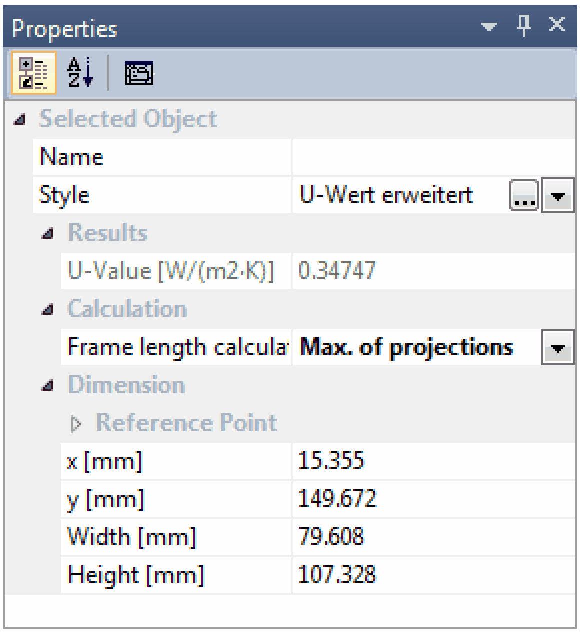


Figure 1: Properties Flyout for a Usb-Value Object

Defining Styles

The presentation of a Roller Shutter Box U-Value object can be selected in the Style list in the tool properties (cf. figure 2). The list shows all styles for Uf-value objects that have previously been determined in the Styles flyout.

A new Roller Shutter Box U-Value object is always created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or select the according style in the Style drop down list of the tool properties.

The styles can also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (cf. tutorial 1).

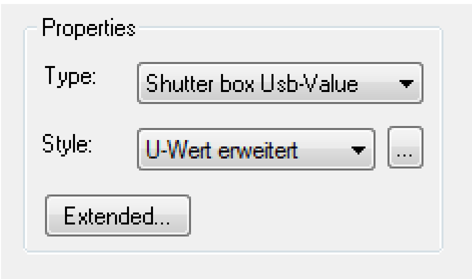
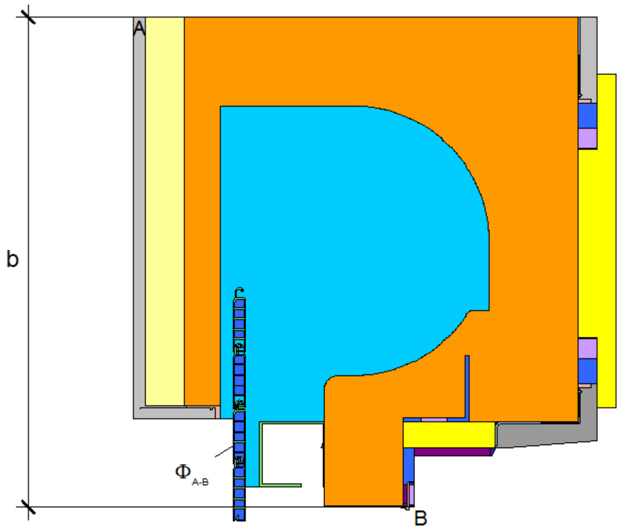


Figure 2: Tool Properties

Physical Explanations

The Roller Shutter Box U-Value is calculated according to EN ISO 10077-2 as follows:



$$U_{sb\ A-B} = \frac{\Phi_{A-B}}{b \cdot \Delta T}$$

with:

Usb: Roller Shutter Box U-Value [W/m²K]

Φ_{A-B}: Heat flow from A to B (counter-clockwise) [W/m]

b: length of the projection of the roller shutter box AB [m]

ΔT: Temperature difference Interior-Exterior [K]

Limitations

- This tool can only be used on the report page and only if result objects of a previously calculated model are present.
- Only **2 boundary condition temperatures** can be present in the model.
- Neither **interior borders** nor **heat sources** can be present.
- There are exactly **2 adiabatic boundaries**; the distance of the adiabatic boundary at the bottom of the roller shutter box where it adjoins the window frame is 60 mm.
- The **temperature differences** must have the **same values** for both sections.
- Resistance of surface adjacent to the internal environment is **0.13 m²K/W**.
- Resistance of surface adjacent to the external environment is **0.04 m²K/W**.
- **Vertical orientation** of the roller shutter box (the width of the roller shutter box will be always calculated as vertical projection)

4.1.22 Psi-Value Tool

Use

Description

The ***Psi-Value*** tool allows you to calculate the Ψ-values (previously linear k-value). The Ψ-value, which quantifies additional energy, is a correction value that is caused by linear shaped thermal bridges, and is not taken into account in 1 dimensional energy calculations with U-values.

The magnitude of the Ψ-value depends on several values: the quality of the construction, the quality of the dimensions, as well as the U-value which calculates the loss of heat of the undisturbed component. Hence, it is possible that a poorly insulating flat roof protection has a better Ψ-value than a well insulating window-wall junction, because in the case of the former, the calculation of the Ψ-value is done with exterior measurements through which part of the energy is already covered by thermal bridge energy losses by flat roof and wall measurements. Ψ-values can also be ***negative***.

The calculation of Ψ-values is explained in detail in tutorial 5.

How do I activate it?

The Psi-Value tool can be activated with the menu command **Psi-Value** in the menu **Results** or by clicking on the Ψ icon in the Toolbox flyout. Then you have to define the kind of Ψ-value in the tool properties (cf. figure 1):

- 1 Construction
- 2 Constructions
- 3 Constructions
- Edge/Spacer

How do I use it?

The use, meaningful applications, physical explanations and limitations of the calculation kinds are explained in following chapters:

- 1 Construction
- 2 Constructions
- 3 Constructions
- Edge/Spacer

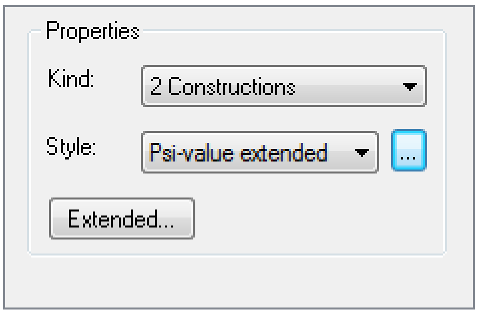


Figure 1: Psi-Value tool properties

4.1.22.1 1 Construction

energy, energy plus, pro

Use Physics Limitations

Description

With the ***Psi-Value*** tool kind 1 ***Construction*** you can calculate Ψ-values of constructions, in which one component with a single U-value has a thermal bridge.

The ***Psi-Value*** tool allows you to calculate the Ψ-values (previously linear k-value). The Ψ-value, which quantifies additional energy, is a correction value that is caused by linear shaped thermal bridges, and is not taken into account in 1 dimensional energy calculations with U-values.

The magnitude of the Ψ-value depends on several values: the quality of the construction, the quality of the dimensions, as well as the U-value which calculates the loss of heat of the undisturbed component. Hence, it is possible that a poorly insulating flat roof protection has a better Ψ-value than a well insulating window-wall junction, because in the case of the former, the calculation of the Ψ-value is done with exterior measurements through which part of the energy is already covered by thermal bridge energy losses by flat roof and wall measurements. Ψ-values can also be ***negative***.

The calculation of Ψ-values is explained in detail in tutorial 5.

How do I activate it?

The Psi-Value tool can be activated with the menu command **Psi-Value** in the menu **Results** or by clicking on the Ψ icon in the Toolbox flyout. Then select the kind 1 ***Construction*** in the tool properties (cf. figure 2). When the tool is active, the mouse becomes a .

How do I use it?

To calculate the Psi-value 1 Construction two clicks are necessary: By defining starting- and endpoint the surface line is determined, with which the actual heat flow is calculated.

The surface line is defined by the input of the start and endpoints:

First click on the spot where the surface line should begin. The boundaries of the construction, which come into question for the surface line, will be highlighted as a dashed line. Move your mouse **counterclockwise** to the desired endpoint. The dashed line now shows the segment through which the heat flow will be calculated. As soon as you click on the endpoint the line will be defined.

If the **SHIFT** key is pressed while you define the surface line, then the first click sets **start** and **endpoints** automatically at the adjacent adiabatic edges (e.g. construction sections).

The Psi-value will be displayed, and the start and end points of the surface line will be labeled.

If you want to interrupt the line input process, then you can use context menu (right click) command **Cancel** or you can hit the **ESC** key.

You can change the reference lengths of the base constructions using the Scale, Move, Select tool or Edit tool respectively, by moving the start-, end and reference point. You can also change the text positions of the help objects, of the U-Values of the base construction and the dimension lines using the same tool.

The base values of the Ψ-value calculation, which have a significant influence on the result (U-values of the constructions, length of the components, if the heat flow through the sections at the start- and endpoint should be considered and the temperature difference), can be adjusted afterwards in the Properties flyout (cf. figure 1).

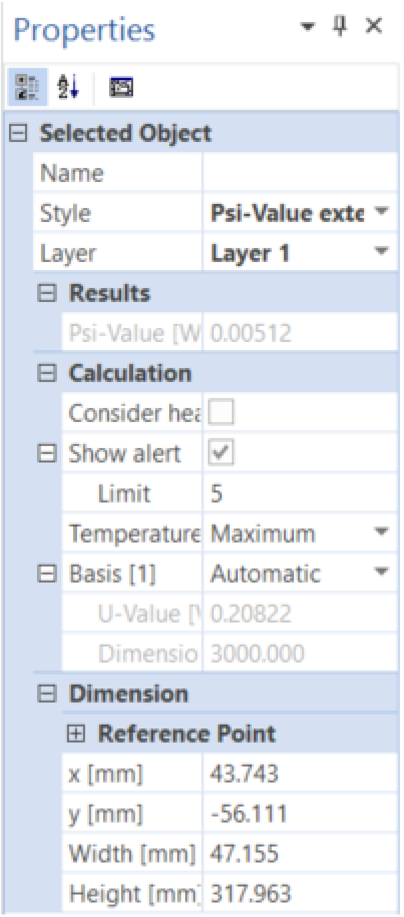


Figure 1: Properties flyout of a Ψ-value object based on 1 construction

Defining Styles

The presentation of a Psi-value object can be defined in the Style list in the Psi-Value tool properties (see figure 2). The list shows all styles for Psi-value objects that have previously been determined in the Styles flyout.

A new Psi-value object will always be created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the Psi-Value tool properties.

The styles can also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

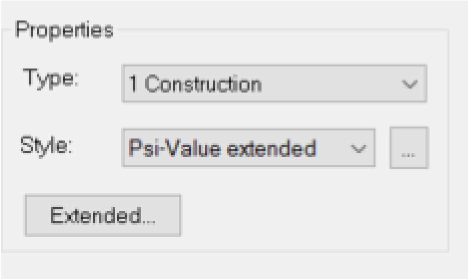


Figure 2: Psi-Value tool properties

Accurate Results

To receive accurate results, the following conditions must be met:

- the cross section line should be defined within sections, through which no heat flow flows (e.g. symmetrical axes or on edges where there the boundary condition is set at: heat flux q=0.0 W/m2K).
- If you choose an exterior reference system, start and endpoints should lie on the exterior boundary of the construction. If you choose an interior reference system, start and endpoints should lie on the interior boundary of the construction. Make note of the order of the inputs: also here endpoint inputs are entered counterclockwise.
- Also consider the Comments section in the chapter Physical Explanation.

The type of label (e.g. number of post-decimal place holders) as well as the graphical properties of the cross section line can be adjusted with the corresponding styles in the Styles flyout.

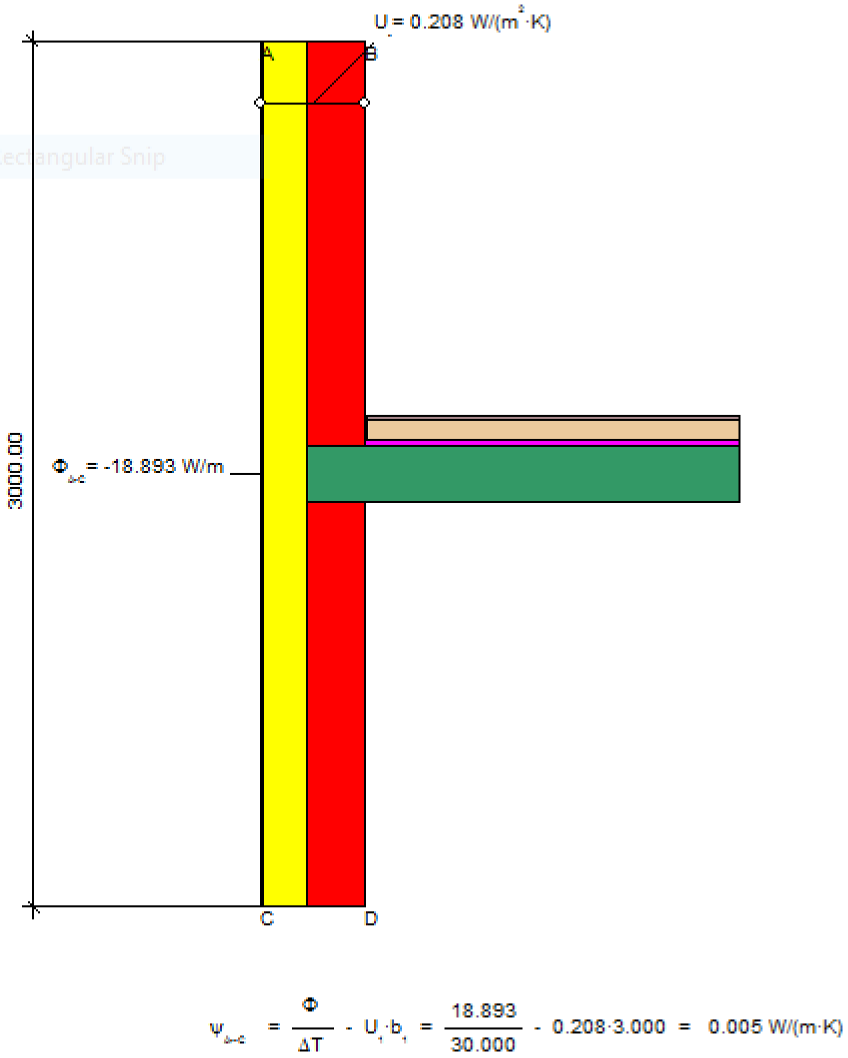


Figure 3

Physical Explanation

The Ψ-value is calculated as follows:

$$\Psi_{A-C}=\frac{\Phi_{A-C}}{\Delta T}-U_{A-B}*l_{A-C}$$

and calculated as follows if the heat flux through the section should be taken into account (cf. Properties flyout):

$$\Psi_{A-C}=\frac{\Phi_{A-C}+\Phi_{B-A}+\Phi_{C-D}}{\Delta T}-U_{A-B}*l_{A-C}$$

where:

Φ_{A-C}: heat flow from A to C (counterclockwise) [W/m]
Φ_{B-A}: heat flow from B to A (counterclockwise) [W/m]
Φ_{C-D}: heat flow from C to D (counterclockwise) [W/m]
ΔT: if there are only two boundary condition temperatures: Temperature difference TA - TB of the boundary condition temperatures TA and TB at the points A and B [K]
ΔT: if there are more than 2 boundary condition temperatures, then the temperature difference will be calculated according to the rule defined in the Properties flyout.
U_{A-B}: U-value for the whole construction from A to E [W/m2K]
l_{A-E}: length of the projection of the line segment AC on the perpendicular to the section line AB [m]

Comments

- In general, the Ψ-values are dependent on the side of a construction (interior or exterior) for which the value is being calculated. In particular, the relational lines (lA-E and lC-E) can be different.
- If the heat flow exceeds a critical, relatively high value, then the corresponding warning will be shown. You can adjust this value in the Properties flyout.
- All heat flows will be calculated counterclockwise from start to endpoint.
- To calculate the heat flow, all reference points will automatically be shown at the nearest mesh point on the surface.
- The sign of the heat flow is taken into account: heat flows into the system are positive; heat flows out of the system are negative.
- If there are more than 2 boundary condition temperatures in the model, then the Ψ-value depends on the temperature relations, and is not valid for other boundary condition temperatures. The rule for the calculation of the needed temperature difference must be defined in the Properties flyout.

Limitations

- This tool can only be used on the report page and only if result objects of a previously calculated model are present.
- Neither interior edges nor heat sources can be present.

4.1.22.2 2 Constructions

energy, energy plus, pro

Use Physics Limitations

Description

With the ***Psi-Value*** tool kind ***2 Constructions*** you can calculate Ψ-values of constructions, which have 2 connected constructions. You can also calculate joint Ψ-values (ΨTJ-Values) according to EN ISO 12631 using this tool.

The ***Psi-Value*** tool allows you to calculate the Ψ-values (previously linear k-value). The Ψ-value, which quantifies additional energy, is a correction value that is caused by linear shaped thermal bridges, and is not taken into account in 1 dimensional energy calculations with U-values.

The magnitude of the Ψ-value depends on several values: the quality of the construction, the quality of the dimensions, as well as the U-value which calculates the loss of heat of the undisturbed component. Hence, it is possible that a poorly insulating flat roof protection has a better Ψ-value than a well insulating window-wall junction, because in the case of the former, the calculation of the Ψ-value is done with exterior measurements through which part of the energy is already covered by thermal bridge energy losses by flat roof and wall measurements. Ψ-values can also be ***negative***.

The calculation of Ψ-values is explained in detail in tutorial 5.

How do I activate it?

The Psi-Value tool can be activated with the menu command **Psi-Value** in the menu **Results** or by clicking on the Ψ icon in the Toolbox flyout. Then select the kind ***2 Constructions*** in the tool properties (cf. figure 2). When the tool is active, the mouse becomes a .

How do I use it?

To calculate the Psi-value **three points** are required: the first two points define the surface line, through which the effective heat flow should be calculated. You then have to define a third reference point, which divides the surface line in two parts with different U-values.

The surface line is defined by the input of the start and endpoints:

First click on the spot where the surface line should begin. The boundaries of the construction, which come into question for the surface line, will be highlighted as a dashed line. Move your mouse **counterclockwise** to the desired endpoint. The dashed line now shows the segment through which the heat flow will be calculated. As soon as you click on the endpoint the line will be defined.

If the **SHIFT** key is pressed while you define the surface line, then the first click sets **start** and **endpoints** automatically at the adjacent adiabatic edges (e.g. construction sections).

The Psi-value will be displayed, and the start and end points of the surface line will be labeled.

If you want to interrupt the line input process, then you can use context menu (right click) command **Cancel** or you can hit the **ESC** key.

You can change the reference lengths of the base constructions using the Scale, Move, Select tool or Edit tool respectively, by moving the start-, end and reference point. You can also change the text positions of the help objects, of the U-Values of the base construction and the dimension lines using the same tool.

The base values of the Ψ-value calculation, which have a significant influence on the result (U-values of the constructions, length of the components, if the heat flow through the sections at the start- and endpoint should be considered and the temperature difference), can be adjusted afterwards in the Properties flyout (cf. figure 1).

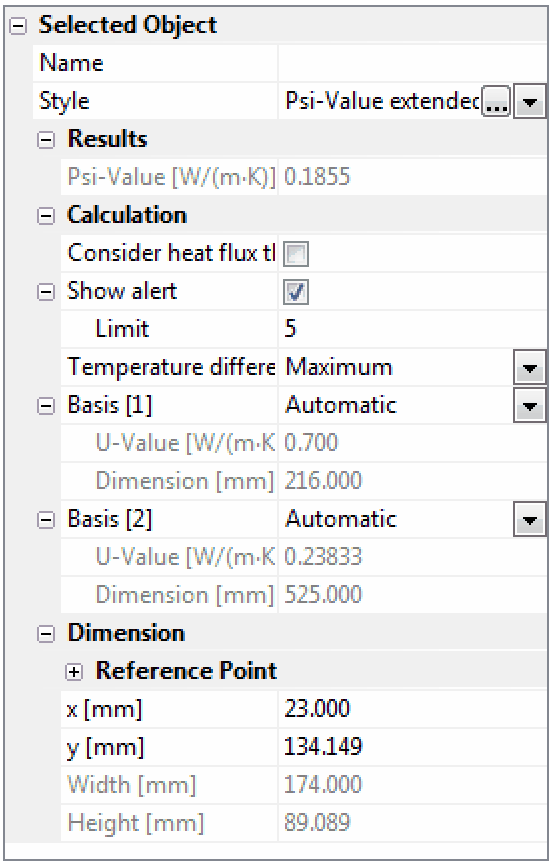


Figure 1: Properties flyout of a Ψ-value object based on 2 constructions

Defining Styles

The presentation of a Psi-value object can be defined in the Style list in the Psi-Value tool properties (see figure 2). The list shows all styles for Psi-value objects that have previously been determined in the Styles flyout.

A new Psi-value object will always be created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the Psi-Value tool properties.

The styles can also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

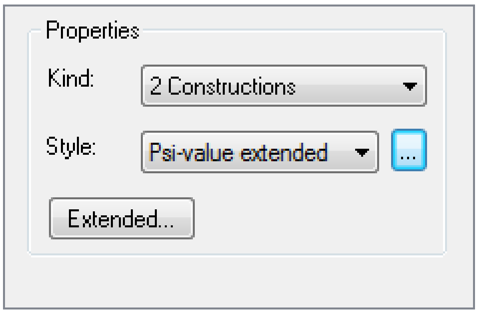


Figure 2: Psi-Value tool properties

Accurate Results

To receive accurate results, the following conditions must be met:

- the cross section line should be defined within sections, through which no heat flow flows (e.g. symmetrical axes or on edges where there the boundary condition is set at: heat flux q=0.0 W/m2K).
- If you choose an exterior reference system, start and endpoints should lie on the exterior boundary of the construction. If you choose an interior reference system, start and endpoints should lie on the interior boundary of the construction. Make note of the order of the inputs: also here endpoint inputs are entered counterclockwise.
- Also consider the Comments section in the chapter Physical Explanation.

The type of label (e.g. number of post-decimal place holders) as well as the graphical properties of the cross section line can be adjusted with the corresponding styles in the Styles flyout.

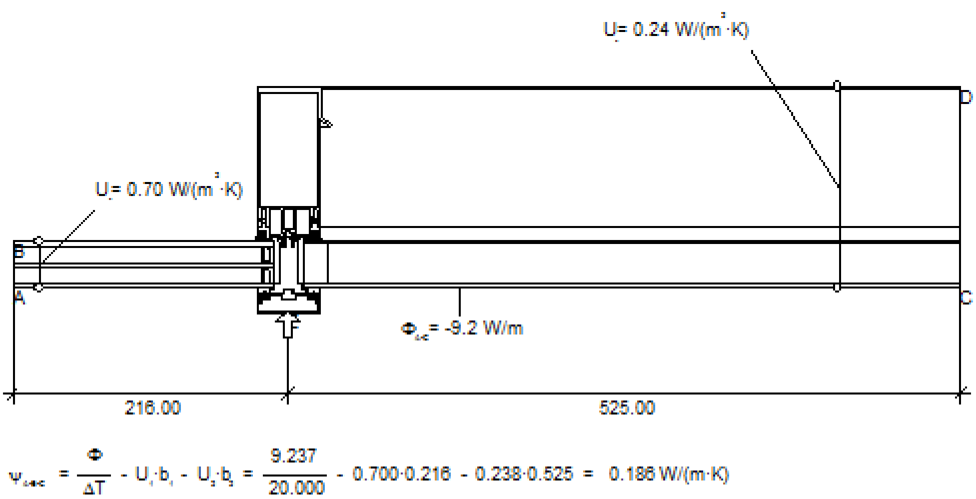


Figure 3

Physical Explanation

The Ψ-value is calculated as follows:

$$\Psi_{A-E-C} = \frac{\Phi_{A-C}}{\Delta T} - \left(U_{B-A} \cdot l_{A-E} + U_{C-D} \cdot l_{C-E}\right)$$

and calculated as follows if the heat flux through the section should be taken into account (cf. Properties flyout):

$$\Psi_{A-E-C(B-A,C-D)} = \frac{\Phi_{A-C} + \Phi_{B-A} + \Phi_{C-D}}{\Delta T} - \left(U_{B-A} \cdot l_{A-E} + U_{C-D} \cdot l_{C-E}\right)$$

where:

Φ_{A-C}: heat flow from A to C (counterclockwise) [W/m]
Φ_{B-A}: heat flow from B to A (counterclockwise) [W/m]
Φ_{C-D}: heat flow from C to D (counterclockwise) [W/m]
ΔT: if there are only two boundary condition temperatures: Temperature difference TA - TB of the boundary condition temperatures TA and TB at the points A and B [K]
ΔT: if there are more than 2 boundary condition temperatures, then the temperature difference will be calculated according to the rule defined in the Properties flyout.
U_{B-A}: U-value for the whole construction from A to E [W/m2K]
U_{C-D}: U-value for the whole construction from E to C [W/m2K]
l_{A-E}: length of the projection of the line segment AE on the perpendicular to the section line AB [m]
l_{C-E}: length of the projection of the line segment CE on the perpendicular to the section line CD [m]

Comments

- In general, the Ψ-values are dependent on the side of a construction (interior or exterior) for which the value is being calculated. In particular, the relational lines (lA-E and lC-E) can be different.
- If the heat flow exceeds a critical, relatively high value, then the corresponding warning will be shown. You can adjust this value in the Properties flyout.
- All heat flows will be calculated counterclockwise from start to endpoint.
- To calculate the heat flow, all reference points will automatically be shown at the nearest mesh point on the surface.
- The sign of the heat flow is taken into account: heat flows into the system are positive; heat flows out of the system are negative.
- If there are more than 2 boundary condition temperatures in the model, then the Ψ-value depends on the temperature relations, and is not valid for other boundary condition temperatures. The rule for the calculation of the needed temperature difference must be defined in the Properties flyout.

Limitations

- This tool can only be used on the report page and only if result objects of a previously calculated model are present.
- Neither interior edges nor heat sources can be present.

4.1.22.3 3 Constructions

energy, energy plus, pro

Use Physics Limitations

Description

With the ***Psi-Value*** tool kind ***3 Constructions*** you can calculate Ψ-values of constructions, which have 3 connected constructions. The 2 thermal junctions are either identical (both Ψ-values are equal) or one connection is "free of thermal bridges" (one Ψ-value of one connection is zero). Typical applications are spacer Ψ-values for mullion constructions or the Ψ-value of a window-wall junction, if the glazing of the window is replaced by a panel according to EN ISO 10077-22.

The ***Psi-Value*** tool allows you to calculate the Ψ-values (previously linear k-value). The Ψ-value, which quantifies additional energy, is a correction value that is caused by linear shaped thermal bridges, and is not taken into account in 1 dimensional energy calculations with U-values.

The magnitude of the Ψ-value depends on several values: the quality of the construction, the quality of the dimensions, as well as the U-value which calculates the loss of heat of the undisturbed component. Hence, it is possible that a poorly insulating flat roof protection has a better Ψ-value than a well insulating window-wall junction, because in the case of the former, the calculation of the Ψ-value is done with exterior measurements through which part of the energy is already covered by thermal bridge energy losses by flat roof and wall measurements. Ψ-values can also be ***negative***.

The calculation of Ψ-values is explained in detail in tutorial 5.

How do I activate it?

The Psi-Value tool can be activated with the menu command **Psi-Value** in the menu **Results** or by clicking on the Ψ icon in the Toolbox flyout. Then select the kind ***3 Constructions*** in the tool properties (cf. figure 2). When the tool is active, the mouse becomes a .

How do I use it?

To calculate the Psi-value **four points** are required: the first two points define the surface line, through which the effective heat flow should be calculated. The third and the fourth point (***reference points***) divide the surface line in 3 constructions with different U-values. The reference points have to been defined ***counter clockwise***.

Then you have to determine the kind and position of the reference point in the Properties flyout (cf. figure 1): **First** or **Second** if one connection is "thermal bridge free" or **Both** if both thermal junctions have the same Ψ-value. You can enter the U-value of the middle construction either directly with the keyboard or fetch it with the tool (cf. figure 1) by clicking on an U-value object in another file.

The surface line is defined by the input of the start and end points:

First click on the spot where the surface line should begin. The boundaries of the construction, which come into question for the surface line, will be highlighted as a dashed line. Move your mouse **counterclockwise** to the desired endpoint. The dashed line now shows the segment through which the heat flow will be calculated. As soon as you click on the endpoint the line will be defined.

If the **SHIFT** key is pressed while you define the surface line, then the first click sets the **start** and **endpoints** automatically at the adjacent adiabatic edges (e.g. construction sections).

The Psi-value will be displayed, and the start and end points of the surface line will be labeled.

If you want to interrupt the line input process, then you can use context menu (right click) command **Cancel** or you can hit the **ESC** key.

You can change the reference lengths of the base constructions using the Scale, Move, Select tool or Edit tool respectively, by moving the start-, end and reference point. You can also change the text positions of the help objects, of the U-Values of the base construction and the dimension lines using the same tool.

The base values of the Ψ-value calculation, which have a significant influence on the result (U-values of the constructions, length of the components, if the heat flow through the sections at the start- and endpoint should be considered and the temperature difference), can be adjusted afterwards in the Properties flyout (cf. figure 1).

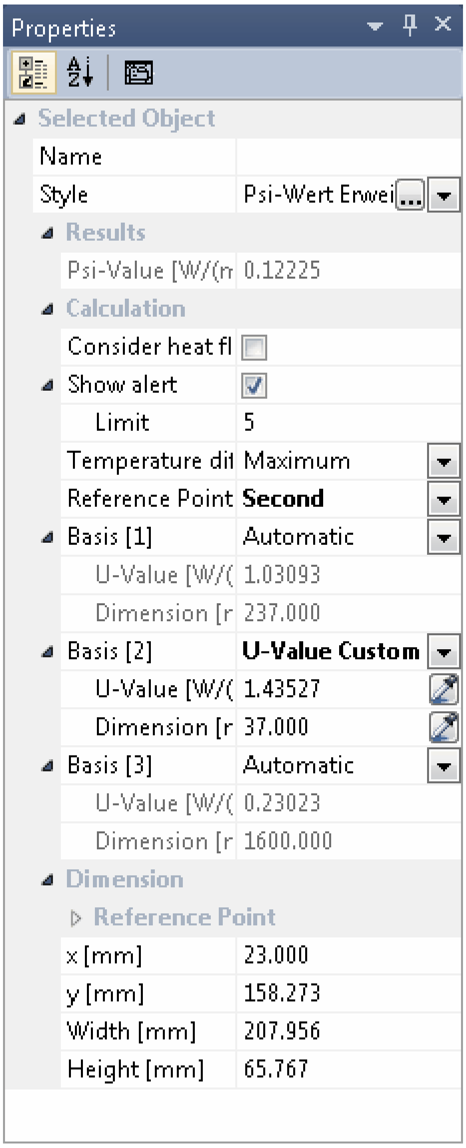


Figure 1: Properties flyout for a Ψ-value object with 3 basis constructions

Defining Styles

The presentation of a Psi-value object can be defined in the Style list in the Psi-Value tool properties (see figure 2). The list shows all styles for Psi-value objects that have previously been determined in the Styles flyout.

A new Psi-value object will always be created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the Psi-Value tool properties.

The styles can be also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

The type of label (e.g. places after decimal point, display of formula) as well as the graphical properties of the cross section line can be adjusted with the corresponding styles in the Styles flyout.

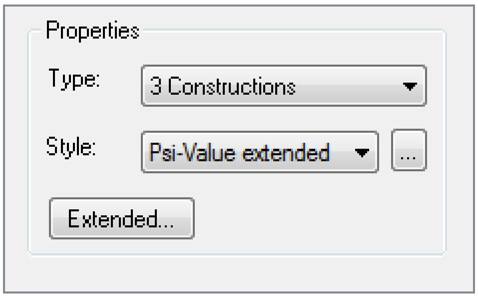


Figure 2: Tool properties

Accurate Results

To receive accurate results, the following conditions must be met:

- the cross section line should be defined within sections, through which no heat flow flows (e.g. symmetrical axes or on edges where there the boundary condition is set at: heat flux q=0.0 W/m2K).
- If you choose an exterior reference system, start and end points should lie on the exterior boundary of the construction. If you choose an interior reference system, start and end points should lie on the interior boundary of the construction. Make note of the order of the inputs: also here endpoint inputs are entered counterclockwise.
- This Psi-value kind should only be calculated for cases where two connections of the 3 components are identical or where one junction is "thermal bridge free".
- Also consider the Comments section in the chapter Physical Explanation.

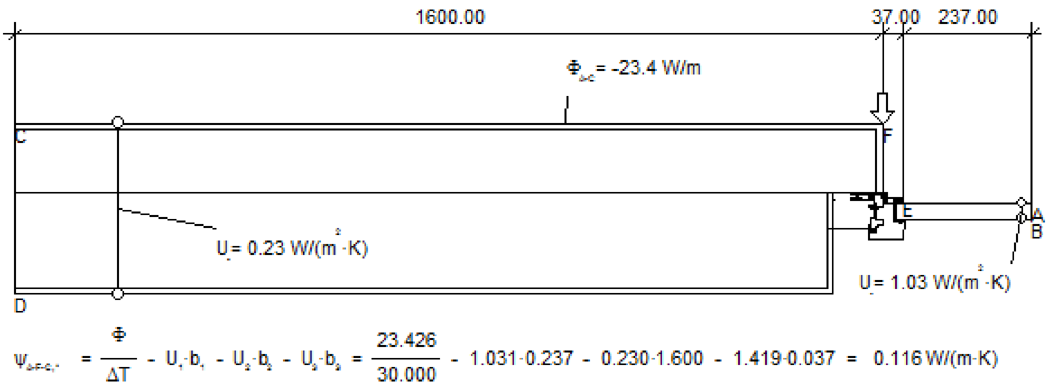


Figure 3

Physical Explanation

The Ψ-value with 3 components is calculated as follows, if both junctions constructions are identical:

$$\Psi = \frac{\Phi_{2dim} - \Phi_{1dim}}{\Delta T \cdot 2}$$

respectively if the Ψ-value of one connection is zero (e.g. in figure 3) the other Ψ-value is calculated as follows:

$$\Psi = \frac{\Phi_{2dim} - \Phi_{1dim}}{\Delta T}$$

The total heat flow per linear meter consists of following partial fluxes:

$$\Phi_{2dim} = \Phi_{A-C}$$

respectively if the heat flows through the cross sections are taken into account (cf. Properties flyout):

$$\Phi_{2dim} = \Phi_{A-C} + \Phi_{B-A} + \Phi_{C-D}$$

The heat flow which is already taken into account through the 1-dimensional point of view is calculated as follows:

$$\Phi_{1dim} = U_{A-E} \cdot l_{A-E} + U_{F-C} \cdot l_{F-C} + U_{F-E} \cdot l_{F-E}$$

where:

Φ_{A-C}: heat flow from A to C (counterclockwise) [W/m]
Φ_{B-A}: heat flow from B to A (counterclockwise) [W/m]
Φ_{C-D}: heat flow from C to D (counterclockwise) [W/m]
ΔT: if there are only two boundary condition temperatures: Temperature difference TA - TB of the boundary condition temperatures TA and TB at the points A and B [K]
ΔT: if there are more than 2 boundary condition temperatures, then the temperature difference will be calculated according to the rule defined in the Properties flyout.
U_{A-E}: U-value for the construction from A to E [W/m2K]
U_{F-C}: U-value for the construction from F to C [W/m2K]
U_{F-E}: U-value for the construction from F to E [W/m2K]
l_{A-E}: length of the projection of the line segment AE on the perpendicular to the section line AB [m]
l_{F-C}: length of the projection of the line segment FC on the perpendicular to the section line CD [m]
l_{F-E}: length of the projection of the line segment FE [m]

Comments

- In general, the Ψ-values are dependent on the side of a construction (interior or exterior) for which the value is being calculated. In particular, the relational lines (lA-E and lC-E) can be different.
- If the heat flow exceeds a critical, relatively high value, then the corresponding warning will be shown. You can adjust this value in the Properties flyout.
- All heat flows will be calculated counterclockwise from start to end point.

- To calculate the heat flow, all reference points will automatically be shown at the nearest mesh point on the surface.
- The sign of the heat flow is taken into account: heat flows into the system are positive; heat flows out of the system are negative.
- If there are more than 2 boundary condition temperatures in the model, then the Ψ-value depends on the temperature relations, and is not valid for other boundary condition temperatures. The rule for the calculation of the needed temperature difference must be defined in the Properties flyout.

Limitations

- This tool can only be used on the report page and only if result objects of a previously calculated model are present.
- Neither interior edges nor heat sources can be present.

4.1.22.4 Edge/Spacer

pro

Use Physics Limitations

Description

With the ***Psi-Value*** tool kind ***Edge/Spacer*** you can calculate the Ψ-values of glazing spacers or edge influences of frames according to EN ISO 10077-2.

The Ψ-value, which quantifies additional energy, is a correction value that is caused by linear shaped thermal bridges, and is not taken into account in 1 dimensional energy calculations with U-values.

The magnitude of the Ψ-value depends on several values: the quality of the construction, the quality of the measurements, as well as the U-value which calculates the loss of heat of the undisturbed component. Hence, it is possible that a poorly insulating flat roof protection has a better Ψ-value than a well insulating window-wall junction, because in the case of the former, the calculation of the Ψ-value is done with exterior measurements through which part of the energy is already covered by thermal bridge energy losses by flat roof and wall measurements. Ψ-values can also be ***negative***.

The calculation of Ψ-values is explained in detail in tutorial 5.

How do I activate it?

The Psi-Value tool can be activated with the menu command **Psi-Value** in the menu **Results** or by clicking on the Ψ icon in the Toolbox flyout. Then select the kind ***Edge/Spacer*** in the tool properties (cf. figure 2).

How do i use it?

First select the result object for which the spacer Psi-value should be calculated. Second you have to define in the dialog window the file with the Uf-value of the same frame.

If you want to calculate the spacer Psi-value according to EN ISO 10077-2 using this tool, following preconditions have to be fulfilled (see Limitations):

- The Uf-value of teh frame is already calculated and saved in a file.
- The frame matches exactly with the one of the Uf-value calculation.
- The orientation of the frame matches exactly with the one of the Uf-value calculation.
- Materials which define the frame dimensions on the Uf-value calculation are not used in other parts than the frame (e.g aluminium). If necessary, you have to create duplicates of materials (e.g. aluminium frame and aluminium panel).

If these conditions are not fulfilled, you can calculate the spacer Psi-value using the Psi-value tool kind 2 Constructions respectively 3 Constructions (for mullion or transom). Using these tools, you have to enter the frame U-value manually in the Properties flyout.

With the Select, Move, Scale tool or Edit tool respectively, you can change the text positions of the help objects, of the U-values of the base construction and the dimensions distance.

You can adjust afterwards the file with the Uf-value calculation in the Properties flyout (cf. file 1). If the Psi-value couldn't get calculated you can see the according error message in the Properties flyout (cf. figure 1 see Psi-value entry) as well.

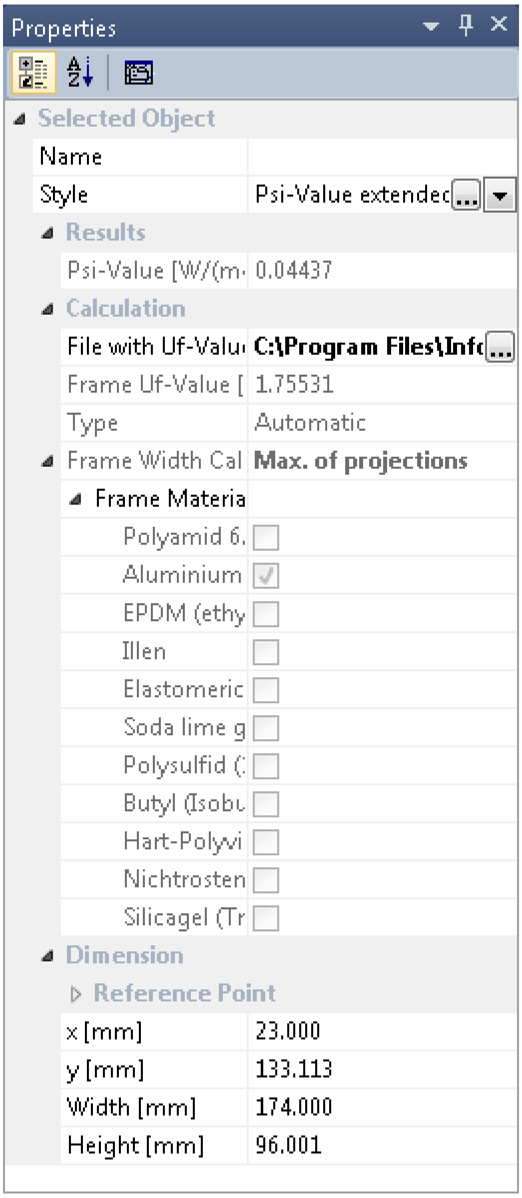


Figure 1: Properties flyout for the spacer Ψ-value object

Defining Styles

The presentation of a Psi-value object can be defined in the Style list in the Psi-Value tool properties (see figure 2). The list shows all styles for Psi-value objects that have previously been determined in the Styles flyout.

A new Psi-value object is always created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the Psi-Value tool properties.

The styles can also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

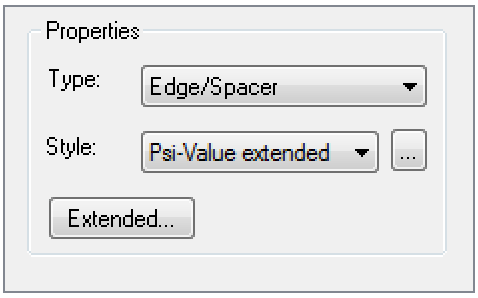


Figure 2: Tool Properties

Physical Explanation

The Ψ-value according to EN ISO 10077-2 is calculated as follows:

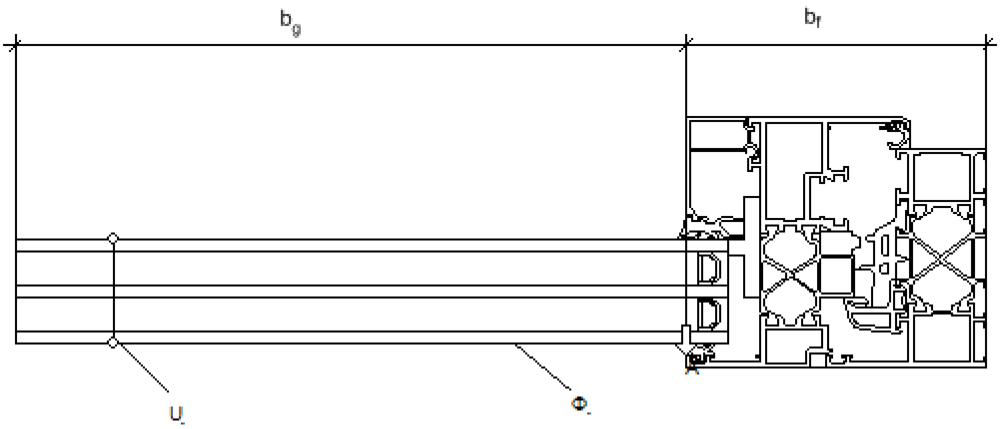


Figure 4: Calculation of spacer Psi-value

$$\psi_A = \frac{\Phi}{\Delta T} - U_g \cdot b_g - U_f \cdot b_f$$

if it is a mullion or a transom with 2 glazings and 2 spacers the formula is as follows:

$$\psi = \frac{\frac{\Phi}{\Delta T} - U_{g1} \cdot b_{g1} - U_f \cdot b_f - U_{g2} \cdot b_{g2}}{2}$$

with:

Φ: Total heat flow in to or out of the model [W/m]
ΔT: Temperature difference [K]
U_g: U-Value of the panel [W/m²K]
b_g: Width of the glazing or the panel [m]
b_f: Width of the frame [m]

Limitations

- This tool can only be used on the report page and only if result objects of a previously calculated model are present.
- The Uf-value of a frame according to EN ISO 10077-2 (cf. U-value tool kind Frame U-Value) must already be calculated using type "Automatic" and saved in a file.
- The orientation of the frame of the Psi-value calculation matches exactly with the one of the Uf-value calculation.
- Materials which define the frame dimensions on the Uf-value calculation are not used in other parts than the frame (e.g aluminium). If necessary, you have to create duplicates of materials (e.g. aluminium frame and aluminium panel).

4.1.23 Dimension Tool

Use Limitations

Description

The ***Dimension*** tool enables the display of measurements within Result objects. You can chose between orthogonal dimension and distance of 2 points.

How do I activate it?

The Dimension tool can be activated with the command **Dimension** in the menu **Insert** or by clicking on the icon in the Toolbox flyout. Then select the kind **Orthogonal** or **Coordinate** in the tool properties (cf. figure 1). When the tool is active the mouse becomes a in the area of the construction.

How do I use it?

Three steps are required to measure a distance: First click on the position where the measurement should begin and then on the spot where the measurement should end. Next, move the mouse outside of the construction; the dashed lines represent a preview of the dimension lines. With the third mouse click, you define the definitive position of the dimension lines.

If you want to interrupt the line input process, then you can use the context menu (right click) command **Cancel** or hit the **ESC** key.

With the Select, Move, Scale tool or Edit tool respectively, you can change the position of the dimension lines, the relative position of the label, and the start and endpoints.

The type of label (e.g. the number of post decimal placeholders) and the graphical properties of the dimension lines can be changed with the corresponding Styles in the Styles flyout.

Defining Styles

The presentation of a dimension line object can be defined in the Style list in the Dimension tool properties (see figure 1). The list shows all styles for dimension line objects that have previously been determined in the Styles flyout.

A new dimension line object will always be created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the Dimension tool properties.

The styles can also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

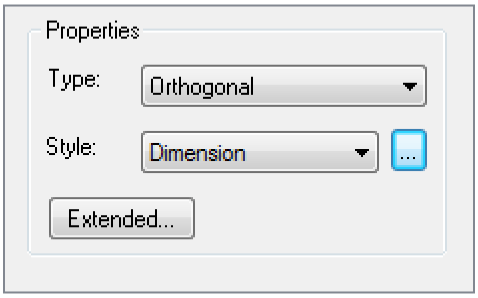


Figure 1: Dimension tool properties

Limitations

- This tool can only be used on the report page and only if result objects of a previously calculated model are present.

4.1.24 Information Tool

Use   Limitations

Description

The ***Information*** tool allows you to prompt information of material domains and boundary conditions of any given construction points and enables the creation of labels containing this information.

How do I activate it?

The tool can be activated with the command **Information** in the menu **Insert** or by clicking on the icon in the Toolbox flyout.

How do I use it?

As soon as you move the mouse over a result object, information about the material domain and the boundary condition edges will be displayed. To record this information along with more detailed information in the report click on the respective spot. This creates a label, which you can move, while keeping the mouse button pressed down (see figure 1).

If you would like to interrupt the labeling process, hit the **ESC** key.

The type of label (e.g. the number of post decimal placeholders, optional display of min. / max. temperatures, optional display of the measurements) along with the graphical properties of the label lines can be adjusted by using the corresponding Styles in the Styles flyout.

With the Select, Move, Scale tool or Edit tool respectively, you can change the position of a label as well as its reference point.

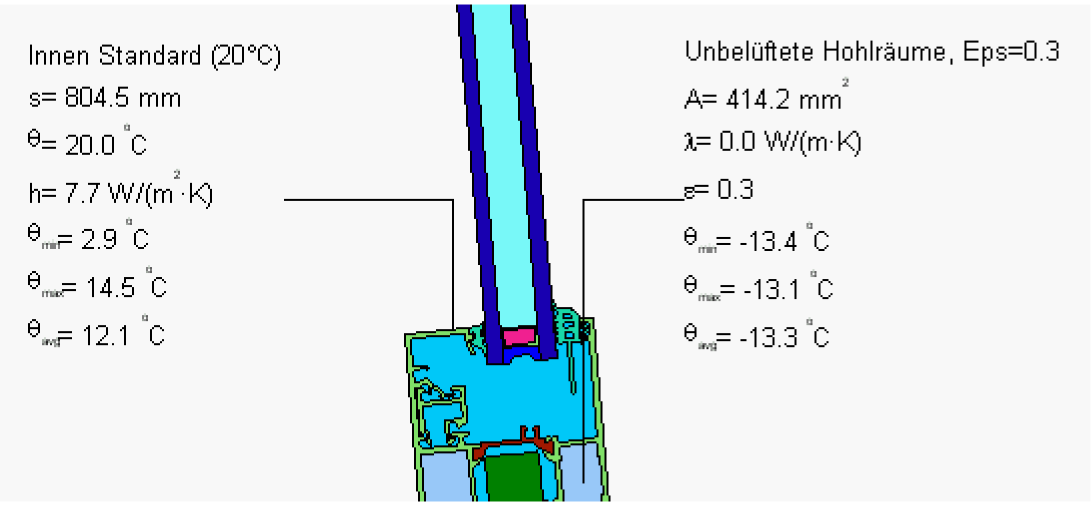


Figure 1: Labeling of a material domain and a boundary condition line

Defining Styles

The presentation of an information object can be defined in the Style list in the Information tool properties (see figure 2). The list shows all styles for information objects that have previously been determined in the Styles flyout.

A new information object will always be created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the Information tool properties.

The styles can be also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

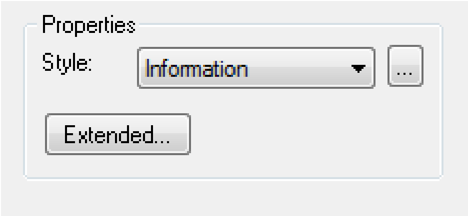


Figure 2: Information tool properties

Limitations

- This tool can only be used on the report page and only if result objects of a previously calculated model are present.

4.1.25 Line Tool

Use

Description

The ***Line*** tool allows you to create lines that help you arrange a report.

How do I activate it?

The Line tool can be activated with the command **Line** in the menu **Insert** or by clicking on the icon in the Toolbox flyout. When the tool is active, the mouse becomes a .

How do I use it?

Mouse

To create a line **two steps** are required: Click on the spot, where the line should begin. The line is defined as soon as you click on the desired endpoint. If you would like to interrupt the entry process then you can right click and choose **Cancel** from the context menu or you can hit **ESC**.

Keyboard

For keyboard input, make sure that the properties toolbar is visible. If the toolbar is not visible you can activate it with the command **Toolbox** in the menu **View**. Activate the toolbar by hitting the Tab key or by clicking on one of its fields. You can activate the individual fields by repeatedly hitting the **Tab** key (or by hitting the **Tab** key while the **Shift** key is held down).

Enter the x- and y-coordinates of the start point into the appropriate fields, and then enter the coordinates of the endpoint (or the direction and length of the line). The coordinates can either be defined as cartesian or polar coordinates. To end the entry click on the **Create** button.

Defining Styles

The presentation of a line can be defined in the Style list in the Line tool properties. The list shows all styles for lines that have previously been determined in the Styles flyout.

A new line will always be created with the styles currently marked in the Style drop down list (see figure 1). You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the Line tool properties.

The line styles can be also be adjusted afterwards by using Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

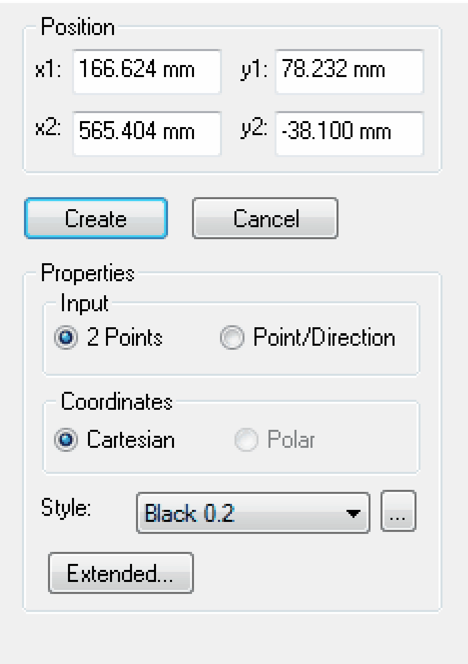


Figure 1

4.1.26 Polyline Tool

Use

Description

The ***Polyline*** tool lets you create open or closed shapes to structure a report.

How do I activate it?

The Polyline tool can be activated with the command **Polyline** in the menu **Insert** or by clicking on the icon in the Toolbox flyout. When the tool is active, the mouse becomes a .

How do I use it?

The use of the Polyline tool is related to that of the Polygon Domain tool. You can create a polyline by clicking on spots where the corners of the shape should be. The individual corners of the polyline can also be entered with coordinates in the Polyline tool properties (see figure 1).

Differences between the Polyline tool and the Polygon Domain tool:

- The Polyline tool does not create material domains, but is rather used to structure reports. The style of the lines can be adjusted in the Styles flyout.
- A polyline can either be closed or unclosed. Use the right click command **Finish** from the context menu to leave the polyline unclosed (you can also click on the Create button in the Polyline tool properties), or select **Close** from the context menu to close the line.

Defining Styles

The presentation of a polyline object can be defined in the Style list in the Polyline tool properties (see figure 1). The list shows all styles for dimension line objects that have previously been determined in the Styles flyout.

A new Polyline object will always be created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the Polyline tool properties.

The styles can also be adjusted afterwards by using the Assign Properties tool or by using the ***Drag&Drop*** function (see tutorial 1).

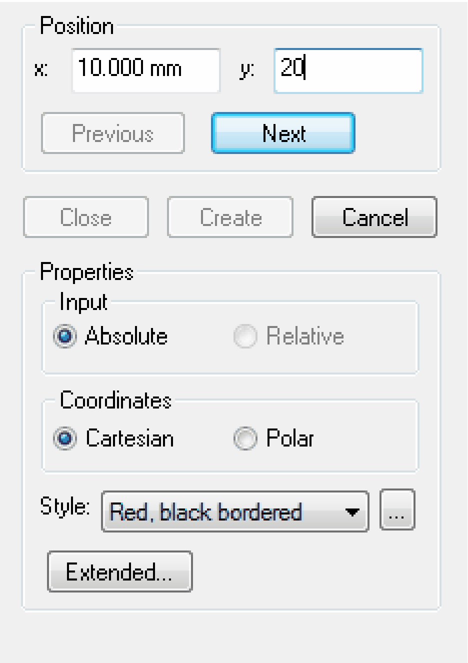


Figure 1

4.1.27 Rectangle Tool

Use

Description

With the ***Rectangle*** tool, you can create rectangles on report pages.

How do I activate it?

The Rectangle tool can be activated with the command **Rectangle** in the menu **Insert** or by clicking on the icon in the Toolbox flyout. When this tool is activated, the mouse becomes a .

How do I use it?

The rectangle tool is similar to the Rectangular Domain tool, with a few exceptions:

- The Rectangle tool does not create material domains.
- The style of the drawn rectangle can be defined with the corresponding Styles in the Styles flyout.

Defining Styles

The presentation of a rectangle object can be defined in the Style list of the Rectangle tool properties (see figure 1). The list shows all styles for rectangle objects that have previously been determined in the Styles flyout .

A new rectangle object will always be created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles Flyout or in the Style drop down list in the Rectangle tool properties.

The styles can be also be adjusted afterwards by using the Assign properties tool or by using the ***Drag&Drop*** function (see Tutorial 1).

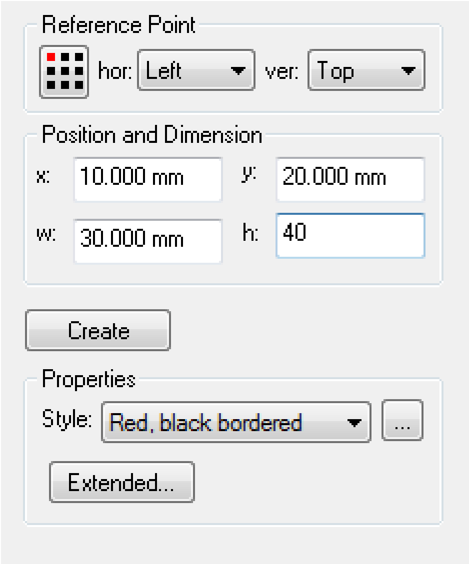


Figure 1:

4.1.28 Ellipse Tool

Use

Description

The ***Ellipse*** tool lets you create ellipses and circles on report pages.

How do I activate it?

The Ellipse tool can be activated with the command **Ellipse** in the menu **Insert** or by clicking on the  icon in the Toolbox flyout. When the tool is active, the mouse becomes a .

How do I use it?

The Ellipse tool is used like the Rectangle tool, aside from the fact that instead of a rectangle, the tool creates an ellipse.

4.1.29 Text Tool

Use   Limitations

Description

The ***Text*** tool lets you add comments, title, and explanations to a report page. Furthermore, you can choose the style (bold, italic, font, etc.) of the text.

How do I activate it?

The Text tool can be activated with the command **Text** in the menu **Insert** or by clicking on the icon in the Toolbox flyout. When this tool is active the mouse becomes a I.

How do I use it?

Click on the spot where the text box should be placed, and enter the text (see figure 1). The cursor shows the current position of the text. To finish entering a text, click outside the text box or choose the next command.

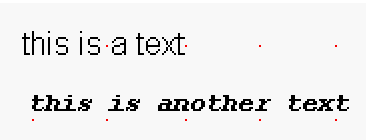


Figure 1

Defining Styles

A new text object will always be created with the styles currently marked in the Style drop down list. You can either change the selection of the styles in the Styles flyout or in the Style drop down list in the Text tool properties.

The text styles can also be adjusted afterwards by using the Assign Properties tool or by using the Drag&Drop function (see tutorial 1).

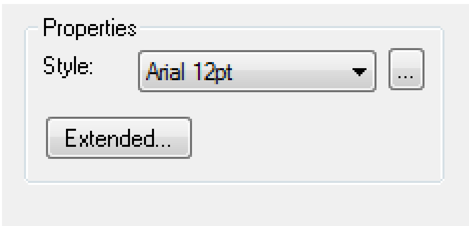


Figure 2: Text tool properties

Limitations

- Only one text style can be applied to a text object.

4.2 Menu Commands

In this chapter, you will find a complete list of the menu commands and their explanations.

The menu structure in flixo follows the general conventions. The usual commands, like Undo, are located in the same spots as in other Windows programs.

Functions that are used often are found on the ***toolbars***, on the active dialogs, or can be activated through the context menu commands (right click) on objects. You can hide or unhide a symbol toolbar by right clicking on an active toolbar and by (de)selecting the desired toolbar or by using the menu command **Toolbars** in the menu **View**.

You can customize your toolbars with the menu command Customize in the Toolbars menu.

Subjects in this chapter:
- File Processes: "File" Menu
- General Editing: "Edit" Menu
- Display Settings: "View" Menu
- Inserting Objects: "Insert" Menu
- Arranging Objects: "Arrange" Menu
- Selecting Tools: "Tools" Menu
- Setting up a Report: "Results" Menu
- Window Management: "Window" Menu
- Help: "Help" Menu

4.2.1 "File" Menu

New...	Creates a new flixo document based on a document template. The document template can be chosen from a dialog window. The creation of a new document template is explained in Tutorial 5.
Open...	Opens a flixo document from a volume.
Recent Files	Displays a list of the recently used files. Select the desired file, and it will open in a new window.
Calulate	A submenu with commands to calculate the temperature distribution of either all not calculated constructions, all constructions or a selected construction.
pro **Batch Calculations...**	Allows the calculation of several constructions at the same time. The definition of the files to be calculated, along with the activation of the calculation is done through a special dialog window.
Remove Calculation	Deletes the mesh and calculation data of the current construction.
Import...	Imports files of various formats. **DXF** files, **SVG** files, and **ISO2** files can be imported. The file type can be changed in the corresponding dialog window from the list "**File Type:**". From the list "**Template**" the file template can be defined as the basis for the document to be imported. DXF file imports are explained in Tutorial 3.
Export...	Exports the current report page as a graphics or a SVG file. You can also choose to export only the selected objects.

	The type of file can be changed in the corresponding dialog window from the list "**File Type**". Here, you can also define the resolution of the picture for the graphics file formats.
Export to	Exports the temperature landscape of the current result object to the called program. This command is only active when the file was directly imported from another program.
Close	Closes the active document.
Save	Saves the active document. If the file is being saved for the first time, then a file dialog window appears where the file name and directory path can be defined.
Save As...	Save as saves the file in a new location and under a new name.
Save All	Saves all the active files.
Print	Sends the current document to the default printer.
Print Preview...	Displays the print preview screen.
Page Setup...	Opens the Page Setup dialog window where the page and print settings are displayed and can be adjusted.
Send...	Saves the current document, and starts the default email program, and adds the data as an attachment.
pro **Template...**	Changes the document template. All reports and settings will be replaced by those selected in the document template. The document template can be selected in a dialog window . The creation of document templates is explained in Tutorial 5.
Translations...	Opens the Translations dialog window, where the translations for all names, descriptions, labels etc. for the multiple language support can be adjusted for the active document and support of a new language can be added.
Properties...	Opens the Properties dialog window, where the document properties are displayed and can be adjusted.
Exit	Exits the active file.

4.2.1.1 File Menu, Submenu "Calculate"

All Non-Calculated Models	Generates a mesh and calculates the temperature distribution of ***all non-calculated constructions***.

Optionally, you can pause the program after the mesh has been generated (see corresponding Options dialog window, which can be activated with the command **Options...** in the menu Tools). The progress of the calculation will be displayed in a corresponding dialog window.

All Models

Generates a mesh and calculates the temperature distribution of ***all constructions present in the file***. Optionally, you can pause the program after the mesh has been generated (see corresponding Options dialog window, which can be activated with the command **Options...** in the menu Tools). The progress of the calculation will be displayed in a corresponding dialog window.

List of Model Names

Generates a mesh and calculates the temperature distribution of ***a construction selected from a list***. Optionally, you can pause the program after the mesh has been generated (see corresponding Options dialog window, which can be activated with the command **Options...** in the menu Tools). The progress of the calculation will be displayed in a corresponding dialog window.

4.2.2 "Edit" Menu

Undo

Undoes the last action. The type of action will be added to this menu entry. For example, the entry says "Undo Insert" after an insert operation.

Redo

Repeats a command, which has been undone. The type of action will be added to this menu entry. For example, the entry says "Redo Insert" after an Undo insert command.

Cut

Cuts the currently selected object, and puts it onto the clip board.

Copy

Copies the currently selected object and puts the copy onto the clip board.

Paste

Inserts contents of the clipboard into the active document.

Paste Special

Inserts the contents of the clip board either as an ***embedded*** or ***linked*** object. This can be done in the corresponding dialog window.

By **embedded** objects, the data in the flixo document is not changed, when you change the original data.

Embedded objects become a part of the flixo document after they are added. Double click on the embedded object, to work with it in its original program.

By **linked** objects, the data will only be updated when the corresponding original file (e.g. spreadsheet) is modified. Linked data is saved in the original file, flixo only saves the file path of the original file and displays the linked data.

Delete	Deletes everything that is currently selected.
Select	A submenu with commands to select different object groups.
Adjust	A submenu with commands to adjust the size, materials, and boundary conditions in the current document.
Parameter	A submenu with commands to define or change a ***parametric*** object. The creation and use of parametric objects is explained in Tutorial 6.
Layer Object	A submenu with commands to define or change a ***layered parametric*** object.
3D equivalent Object	A submenu with commands to define or change an ***equivalent 3 dimensional element with regular distances*** as approximation (e.g. screw) according to prEN ISO 12631.
Graphic Object	A submenu with commands to define or change a ***graphic*** object.
Properties	Displays the Properties dialog window for the selected object.
Links...	

4.2.2.1 Edit Menu, Submenu "Select"

All Objects	Selects all objects in the current window.
Guidelines	Selects all (not grouped) guidelines in the current window.
Groups	Selects all groups in the current window.
Invert	Inverts the selection: all unselected objects are selected, all selected objects are unselected.
Hide	Hides the selected objects.

Display	Displays all hidden objects again.
Material Domains	Selects all (not grouped) material domains in the current window.
Selected Material	Selects all objects that have the property that is selected in the material list.
Boundary Conditions	Selects all (not grouped) boundary condition objects in the current window.
frame , pro **Radiation Properties**	Selects all (not grouped) radiation property objects in the current window.
energy, energy plus, pro **Parametric Objects**	Selects all (not grouped) parametric objects, layered objects, and glass units in the current window.
Results	Selects all (not grouped) results like Temperature objects, Heat flow objects etc. in the current window.
Dimensions	Selects all (not grouped) dimension objects in the current window.
Text	Selects all (not grouped) text objects in the current window.
Graphical Objects	Selects all (not grouped) graphical objects like lines, pictures, field functions etc. in the current window.

4.2.2.2 Edit Menu, Submenu "Adjust"

Size	Changes the size of the entire construction depending on the modified size of a selected object. The new size of the selected elements can be entered in the corresponding dialog window.
Material	Replaces the material properties of one defined material with that of another. The two materials can be defined in the corresponding dialog window.
Boundary Condition	Replaces the boundary condition properties of for example all B.C. start points with that of other boundary condition properties. The two boundary conditions can be defined in the corresponding dialog window.

4.2.2.3 Edit Menu, Submenu "Parametric Object"

energy, energy plus, pro

Define or Modify	Defines a new parametric object, and lets you modify existing parametric objects.
Dissolve Object	Dissolves an existing parametric object, and deletes all the parameter information.
New Parameter	Saves the properties of the edge, which was last adjusted, and creates a new parametric edge after this edge has been selected.
Edit Parameter	Saves the properties of the edge, which was last adjusted, and either creates a new parametric edge or changes an existing parametric edge after this edge has been selected.
Delete Parameter	Removes the properties of the currently selected parametric edge.
Delete All Parameters	Removes all the parameters from an object.
Finish	Ends the definition of parameters. This command is only active when the current parametric edge is correctly defined, i.e. exactly one corner of the edge must display a cross.
Cancel	Ends the definition of parameters. Cancels the parameter entry process for an object; parameters entered for that section of the object are lost.
Check Markers	Marks all the corners of the current domain. If you adjust the length of the currently selected parametric object, the entire object is adjusted.
Toggle Markers	Marks all unmarked corners and removes the marks from marked corners of the current domain. Marked corners will be adjusted when the length is modified, unmarked corners will stay in place.
Remove Markers	Removes all markers of all the corners of the current object. When the object is adjusted, it simply holds its position.
Check all Markers	Checks all the corners of all domains. If you adjust the length of the currently edited parametric edge, all edges will be adjusted.
Toggle All Markers	Marks all unselected corners and removes the marks of the selected corners of all domains. Selected corners will be adjusted if the length of the parametric object is modified, unmarked edges will stay in their position.
Remove All Markers	Removes the marks in all corners of all domains. When the object is adjusted, then all objects retain their position.

4.2.2.4 Edit Menu, Submenu "Layer Object"

energy, energy plus, pro

Edit Edits layered parametric objects. In the opened dialog window you can modify the number of layers, the layer's thickness, the orientation, and the materials.

Transform Transforms a layered parametric object into a normal parametric object. After this action, it is no longer possible to edit the layer properties in a special dialog window.

Dissolve Object Dissolves an existing layered parametric object and deletes all the parametric information.

4.2.2.5 Edit Menu, Submenu "3D equivalent Object"

energy, energy plus, pro

Define Creates an equivalent 3D object from the selected objects either area weighted or according to prEN ISO 12631. The material, kind and dimensions can be defined in the opening dialog window.

The use of 3D equivalent objects is explained in tutorial 8.

Edit Opens a dialog window where the material, kind and dimensions of a 3D equivalent object can be adjusted.

Dissolve Object Dissolves an existing 3D equivalent object.

4.2.2.6 Edit Menu, Submenu "Graphic Object"

Define Defines a ***graphic object***. Graphic objects are not taken into account in the calculation, they facilitate the legibility of the illustration.

Dissolve Object Dissolves a graphic object and deletes all parametric information.

4.2.3 "View" Menu

Draft Shows all objects in draft mode. Only the outline of objects in the color of the layer on which they are located is shown. Works together with Proof command.

Proof Shows all objects with all their graphical properties. Works together with Draft command.

Zoom	A submenu with commands to magnify and reduce the object on the screen.
Toolbox	Shows the Toolbox flyout.
Materials	Shows the Materials flyout.
Boundary Conditions	Shows the Boundary Conditions flyout.
Layers	Shows the Layers flyout.
Styles	Shows the Styles flyout.
Components	Shows the Components flyout.
Documents	Shows the Documents flyout.
Properties	Shows the Properties flyout.
Start Page	Shows the page that appears by default when you start. The default behavior can be customized in the options.
Context Help	Shows the Context Help flyout.
Toolbars	Submenu to show or hide the toolbars.
Message Bar	In case of problems during a calculation or a DXF conversion, a message bar is shown automatically above the document window. Shows or hides the message bar.
Status Bar	Shows or hides the status bar.
Rulers	Shows or hides the rulers.
Guidelines	Shows or hides the guidelines.
Grid	Shows or hides the grid.

4.2.3.1 View Menu, Submenu "Zoom"

Zoom Undo	Undoes the last zoom or scroll action.
Zoom Redo	Repeats the last undone zoom or scroll action.
Zoom Out	Reduces the object on the screen.
Zoom In	Magnifies the object on the screen.
Zoom to Selection	Magnifies the selected objects to fill the work area.

Zoom to All Objects	Magnifies all objects to fill the work area.
Zoom to Material	Searches for a domain with a specific material and then magnifies to it to fill the work area. The use of this function is also discussed in Tutorial 4.
Zoom to Conflict	Displays problem areas while importing DXF files and during the calculation. The use of this function is explained in Tutorial 3 and in Tutorial 4.
Percent	Submenu with predefined zoom factors.

4.2.4 "Insert" menu

Polygon Domain	Activates the Polygon Domain tool.
Elliptical Domain	Activates the Elliptical Domain tool.
Rectangular Domain	Activates the Rectangular Domain tool.
Boundary Condition	Activates the Boundary Condition tool.
pro **Heat Source**	Activates the Heat Source tool.
frame, pro **Air Cavity EN ISO 10077-2**	Activates the Air Cavity EN ISO 10077-2 Wizard.
Dimension	Activates the Dimension tool.
Information	Activates the Information tool.
Line	Activates the Line tool.
Polyline	Activates the Polyline tool.
Rectangle	Activates the Rectangle tool.
Ellipse	Activates the Ellipse tool.

Text	Activates the Text tool.
Picture...	Inserts a saved picture into the current document. The file name and the data type must be entered into the corresponding File dialog window. The position of the picture is determined by the upper left hand corner of the picture; the position of this corner can be defined with a mouse click.
Field...	Inserts a field object into the current document. The type of field and its options must be defined in the Field dialog window that appears. The position of the upper left corner can be determined with a mouse click.
energy plus, pro **Layer Object...**	Inserts a layered parametric object into the current document. In the corresponding Layer Object dialog window you have to define the number of layers, the layer's thickness, the orientation, and the materials. You can also adapt the dimensions, orientation and materials directly in the document using the usual methods and tools.
energy plus, frame, pro **Glas Unit...**	Inserts a glass unit object into the current document. In the corresponding Glas Unit dialog window you have to define the glass unit type (double or tripple glassings) and other properties. You can also adapt the dimensions, orientation and material of the glass in the document directly by using the usual methods and tools.
Object...	Inserts a new ***embedded*** or a ***linked*** object. The type of object and its options must be defined in the corresponding dialog window. **Embedded** objects do not change in a flixo document, when you edit the original file data. After the insertion, embedded objects become a part of the flixo document. Double click on the object to edit it in its source file. **Linked** objects will only change when the source file (e.g. spreadsheet) is modified with the appropriate program. The data of a linked object is stored in the source file; flixoonly saves the file path, and displays a representation of the linked file.

4.2.5 "Arrange" Menu

Order	A submenu with commands to change the order and therefore also the visibility of objects.
Align	A submenu with commands to change the position of an object relative to another object.
Same Dimension	A submenu with commands to change the dimensions of an object.
Group	Groups the selected objects; grouped objects behave as if they were a single object.

Ungroup	Dissolves the grouping(s).
Unite	Unites the selected objects (see also Unification Operation).
Intersect	Intersects the selected objects (see also Intersection Operation).
Subtract	Subtracts the last marked object from the previously selected object(s) (see also Subtraction Operation).
Hor. Mirroring	Mirrors the selected object horizontally.
Ver. Mirroring	Mirrors the selected object vertically.
Rotate 90° CW	Rotates the selected object(s) 90° clockwise.
Rotate 90° CCW	Rotates the selected object(s) 90° counterclockwise.
Snap to Grid	Toggles the grid alignment of objects. If the **Snap to Grid** function is active, then the mouse can only define points which are on the grid. This function, however, can also be used in conjunction with other the functions ("Snap to Guidelines" or "Snap to Objects"). The grid refinement can be changed in the Options dialog window, which can be activated with the menu command Tools - Options.
Object capturing	Opens the object capturing dialog window, where different helps for capturing objects can be set.

4.2.5.1 Arrange Menu, Submenu "Order"

Bring to Front	Positions the selected object to the very front of the same layer.
Send to Back	Positions the selected object to the very back of the same layer.
To Front one Step	Moves the selected object one position ahead in the same layer.
To Back one Step	Moves the selected object one position back in the same layer.

Notify that the order of the objects is also influenced by the order of the layers.

4.2.5.2 Arrange Menu, Submenu "Align"

Left	Positions all objects in such a manner that all the encompassing rectangles of the selected objects align themselves left of the first marked object.
Right	Positions all objects in such a manner that all the encompassing rectangles of the selected objects align themselves right of the first marked object.
Top	Positions all objects in such a manner that all the encompassing rectangles of the selected objects align themselves to the top of the first marked object.
Bottom	Positions all objects in such a manner that all the encompassing rectangles of the selected objects align themselves to the bottom of the first marked object.

4.2.5.3 Arrange Menu, Submenu "Same Dimension"

Width	Changes the width of all objects to the width of the first marked object.
Height	Changes the height of all objects to the height of the first marked object.
Width and Height	Changes the height and the width of all objects to that of the first marked object.

4.2.6 "Tools" Menu

Select, Move, Scale	Activates the Select, Move, Scale tool.
Rotate	Activates the Rotate tool.
Assign Properties	Activates the Assign Properties tool.
Cut	Activates the Cut tool.
Crop	Activates the Crop tool.
Measure Distance	Activates the Measure Distance tool.
Protection	A submenu with commands to protect objects, reports and current documents from changes.

DXF-Conversion	Converts a DXF file into a flixo file. More detailed information can be found in tutorial 3.
Customize...	Shows the Customize dialog window. You can hide and unhide toolbars, create new toolbars, add commands to toolbars with the Drag&Drop function, define keyboard shortcuts, change the appearance of the menu toolbar, and change the drop down menus.
Options...	Displays the Options dialog window.

4.2.6.1 Tools Menu, Submenu "Protection"

pro

Protect Object...	Protects a parametric object selectively against certain changes. In the appearing dialog window, you can determine which changes to protect the object against, and define an optional password. You can also remove the object protection. If necessary the password will be requested in a dialog window.
Protect Report...	Protects a report selectively against certain changes. In the appearing dialog window, you can determine which changes to protect the report against, and define an optional password. You can also remove the report protection. If necessary the password will be requested in a dialog window.
Protect Document...	Protects a document selectively against certain changes. In the appearing dialog window, you can determine which changes to protect the document against, and define an optional password. You can also remove the document protection. If necessary the password will be requested in a dialog window.

4.2.7 "Results" Menu

Result Object	Activates the Result Object tool.

Material Legend	Inserts a material legend into the active result object. The graphical properties of material legends can be set in the corresponding dialog window. The position of the legend can be adjusted with the Select, Move, Scale tool.
BC Legend	Inserts a boundary condition legend into the active result object. The graphical properties of the boundary condition legends can be set in the corresponding dialog window. The position of the legend can be adjusted with either the Select, Move, Scale tool or with the Edit tool.
Temperature	Activates the Temperature tool.
Min./Max. Temperature	Activates the Min./Max. Temperature tool.
Heat Flux	Activates the Heat Flux tool.
Heat Flow	Activates the Heat Flow tool.
U-Value	Activates the corresponding U-Value tool.
energy, energy plus, pro **Psi-Value**	Activates the corresponding Psi-Value tool.
Isotherms	Hides or unhides the isotherms of the selected result object. The properties of the isotherms, such as equidistance, color and labeling, can be defined in a corresponding dialog window.
Temperature Field	Hides or unhides the temperature color fields (false coloring) of the currently selected object. The properties of the temperature color fields can be changed in the corresponding Options dialog window, which can be activated with the menu command Options in the Tools menu.
energy plus, pro **Stream Lines**	Hides or unhides stream lines of the selected result object. The properties of the stream lines, such as equidistance and line properties can be defined in the corresponding dialog window.
Heat Flux Field	Hides or unhides the heat flux color fields (false coloring) of the heat flux distribution for the selected result object. The properties of the color fields can be defined in the Options dialog window, which can be activated with the menu command Options in the Tools menu.
Refresh	Explicitly refreshes all results. Should the result objects be automatically refreshed after every change, then the corresponding option can be set in the Options dialog window,

which can be activated with the menu command Options in the Tools menu.

Limitations

- Some of these commands can only be used after the model has been calculated (Calculate).
- The display of isotherms, temperature fields, stream lines for stream density fields, Uf-values, and legends is only possible when a result object is selected.

4.2.7.1 Results Menu, Submenu "U-Value"

Parallel Layers	Activates the **U-Value Tool, Parallel Layers**.
Equivalent U-Value	Activates the**U-Value Tool, Equivalent U-Value**.
Frame U-Value	Activates the **U-Value Tool, Frame U-Value**.
Joint UTJ-Value	Activates the **U-Value Tool, Joint U-Value**.
Roller Shutter Box Usb-Value	Activates the **U-Value, Roller Shutter Box**.

4.2.7.2 Results Menu, Submenu "Psi-Value"

1 Construction	Activates the **Psi-Value Tool, 1 Construction**.
2 Constructions	Activates the **Psi-Value Tool, 2 Constructions**.
3 Constructions	Activates the **Psi-Value Tool, 3 Constructions**.
Edge/Spacer	Activates the **Psi-Value Tool, Edge/Spacer.**

4.2.8 "Window" Menu

New Window	Opens another window for the current document.

Auto Hide All	Closes all Flyout domains
New Vertical Tab Group	Arranges the active window in a new vertical tab group. This menu command is only visible if at least 2 document windows are open.
New Horizontal Tab Group	Arranges the active window in a new horizontal tab group. This menu command is only visible if at least 2 document windows are open.
Close all documents	Closes all active windows. If you did not save a document, you will be prompted to do so.
Window List	Shows a list with all the active documents.
More Windows...	Displays a dialog window with all the active documents.

4.2.9 "Help" Menu

Contents	Opens the flixo manual.
flixo online...	Opens the flixo website with additional help functions.
Tutorials	Displays the list of all tutorials.
Check for Updates...	Checks if the installed version of flixo is the most recent version or if there are any updates or service releases to be found. Depending on the settings these updates and service releases will be downloaded from the internet and installed. You need access to the Internet to check for updates. You need a **write permission** for the download folder in order to check for updates.
About flixo...	Displays the dialog window About flixo.

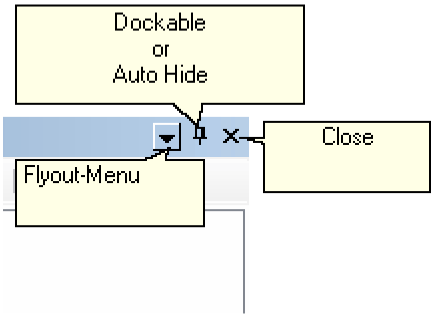
4.3 Flyout

Flyouts are windows, which you can position, arrange, and compile as you wish.

A Flyout can take on the four following properties:

- **Floating**
- **Dockable**
- **Auto Hide**
- **Hide**

You can choose the properties in the Flyout menu located at the top of the window:



Floating flyout windows can be placed anywhere within an application window. In the default setting of flixo (see User Interface), there are no floating flyouts.

Dockable flyout windows can be grouped together, as for example the materials, boundary conditions, components, styles, and layers flyouts are in the flixo default setting (see User Interface) and they can also be docked onto anywhere on the application window sides.
You can move flyouts by moving the top title section and with help of the appearing dock icon. You can dissolve a grouped flyout from the group by moving the corresponding tab at the bottom of the flyout window and by docking it into a new position.

Auto Hide In an open state, flyouts are only visible for a short moment after which they close automatically. In a closed state, flyouts are represented by tabs placed on the sides of the corresponding application windows.
You can open automatically closing flyouts by moving the mouse cursor over the corresponding tab or by actually clicking on the tab. In the default setting of flixo (see User Interface), all Help flyouts close automatically.

Hide closed flyouts can be opened with the menu command View or by clicking on the corresponding icon on the standard toolbar.

List of the different Flyouts:

- Layers
- Materials
- Boundary Conditions
- Styles
- Components
- Documents
- Toolbox
- Properties
- Context Help

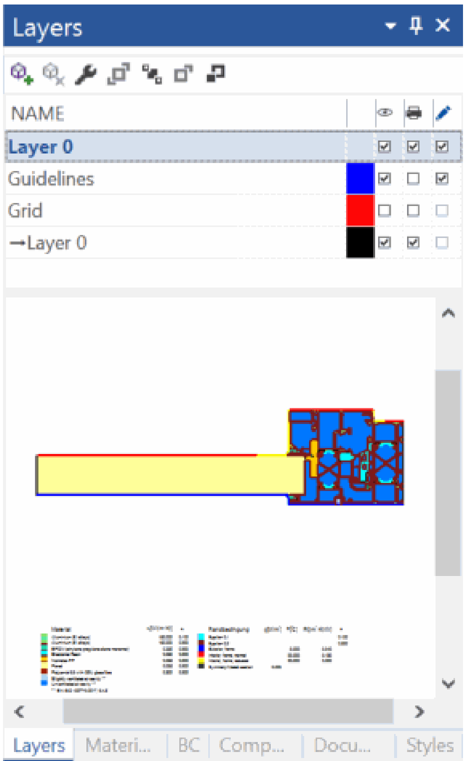
4.3.1 Layers Flyout

Content

Description

The ***Layers flyout*** lets you manipulate the layers. Here, you can add layers, change their properties and delete them. The list shows all layers of the **current page** with a preview of the active layer (***focus layer***).

Material domains and other objects can be placed on the ***different layers***, and they can selectively be made **visible**, **printable**, or **editable.** For each layer you can define the properties, for example the visibility of the layer.
Some layers are predefined for every document (the so-called ***system layers***); these layers have special properties (layers for guidelines and grid).



Focus Layer

Many operations, which you can carry out in the individual model and report pages work in conjunction with the active layers or ***focus layers***. The focus layer is marked in the list and its name is highlighted in bold. In the preview window all elements of the focus layer are displayed - either in an expanded or draft mode.

Properties and their Symbols

The symbols for the properties of the layers are located on top of the list. A check mark next to the layer name means that this property is activated :

Displays the visibility of the layer.

Shows whether the layer will be taken into account while printing.

Shows whether the layer can be edited.

Determines the color of the objects when the draft mode is active.

To **activate** or **deactivate** a property, you can either click on the corresponding icon, use the commands from the **context menu** (right click) of the layers: Visible, Printable, Editable or open the Layer dialog window by selecting the layer and clicking on the icon in the flyout. Alternatively, you can use the menu command **Edit**. Not only can the properties of a layer be changed, but also the name and the color of a layer can be adjusted. Choose the context menu command Rename, or repeatedly click on the layer name as you would to rename a file under Windows Explorer.

System layers cannot be renamed.

The order of the layers in the list determines the **visibility** of the objects. Objects, which are positioned on the upper layers cover the objects located on the lower layers. The order of the layers can be changed by dragging the layers into the desired order. Thus, it is possible to always position the guidelines as the first layer so that they will never be obscured by other objects. The order of the layers can also be defined with the order icons located in the flyout.

System Layers

On each page of a document there are two layers, which cannot be renamed or deleted. These two layers are the **Guidelines** and the **Grid** layers. Naturally, the guidelines can be found on the guidelines layer and the grid is located on the grid layer. The grid layer cannot be modified.

The Model Page has an extra layer, the **Model** layer. Here, you will find the physical information that flixo uses for the calculation and analysis. This layer cannot be modified.

Each report page contains an extra layer, the **Reference** layer for the individual layers of the Master Page. The reference layer can be hid and unhidden, printed or not printed. The reference layer, abbreviated Ref:, shows all the corresponding layers on the Master Page. Changes to the original objects or renaming of the layers must take place on the Master Page itself. Typical elements, which are located on the Master Page, are company logos and other stationary type elements. These elements appear automatically on all the pages of the report. Additional information can be found in tutorial 6.

Creating a New Layer

To create a new layer, click on the icon in the flyout or choose the context menu command (right click) **New Layer** on either one of the layers already on the page or on the page category (name of the page). The new layer will be added to the list.

Deleting a Layer

To delete a layer, click on the icon, or choose the context menu command (right click) **Delete** of the corresponding layer you wish to remove. Please note that there must be at least one user layer present in the page because editing and model entries can only be done on this user layer.

System layers cannot be deleted.

4.3.2 Materials Flyout

Content

Description

The ***Material flyout*** allows the manipulation of materials in an active document. Materials can be added or deleted and their properties can be changed.

The **top tree view** displays the materials that are present in the active document. The **bottom tree view** displays materials from a material databank, which can be used for all documents.

The material database is divided into two parts: the first part is read-only and is maintained by infomind or third-party, the second part includes all user-defined materials. The material data maintained by infomind or third-party can be updated either using the menu command **Update...** or in the Options dialog window Database/Templates.

The Properties flyout shows the properties of the selected material.

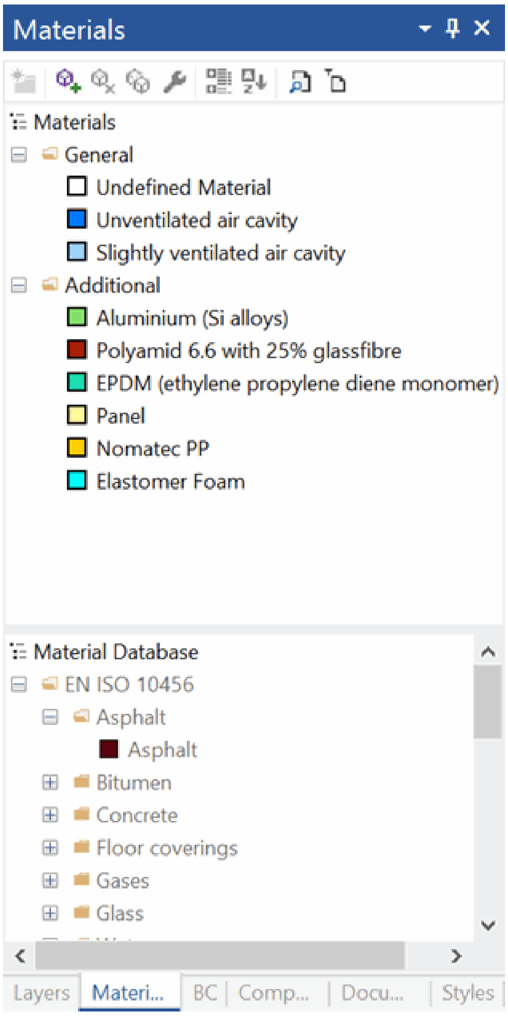


Figure 1: Material flyout

Creation, Modification, Duplication and Deletion of Materials

The following description is valid for both the top materials list (**current document**) and for the lower materials list (**database**).

To create a new material, click on the icon in the flyout. Alternatively, you can use the context menu command (right click) **New Material**. The new material can be defined in the Material dialog window. After you have defined the properties, click on **OK** and the new material appears in the current category.

If you would like to modify the **properties** of an existing material, select it in the list, and click either on the icon or use the context menu command (right click) **Edit** or double click on the material. This opens the same dialog window that is used to define a new material. You can now modify the properties of the material.

If you wish to **rename** a material you can do so by choosing the context menu command (right click) **Rename**. You can also rename the material by selecting it and then clicking on its name just as you would rename a file under Windows Explorer.

To **duplicate** a material entry with an unique name, click on the icon in the flyout. Alternatively, you can create a duplicate by using the corresponding command in the context menu (right click on the source entry) **Duplicate**. All properties of the copy are identical with the ones of the source, if needed the properties can be adapted in the Material dialog window.

You can **delete** a material by selecting it and either clicking on the icon or by using the context menu command (right click) **Delete**.

Grouping Materials into Categories

If you are using many materials, it makes sense to organize these into **Groups** or **Categories**.

Categories are comparable to file folders and directories; this is why they have the same symbol. Upon creation, every document generates a ***General*** category. You can create further categories by clicking on the icon in the flyout or by right clicking on an existing category (or material for that matter) and using the context menu command **New Category**. The category is created within the selected category. You can also define a name for this category.

To **delete** a category select the category and click on the icon, or use the context menu command (right click) **Delete**. You can rename a category by selecting it and then clicking on the icon, or by using either the context menu command (right click) **Edit** or **Rename**. Alternatively, you can rename the category by selecting it and then clicking on it again to rename it just as you would rename a file under Windows Explorer.

Only empty categories can be deleted.

To move a material from one category to another you can simply drag the material with the mouse into the desired category.

Copying Materials

You can copy materials from the database to the materials list of the current document (or vice versa) by dragging the material to the desired location.

Searching the Material Database

If you are looking for a specific material or you would like to get an overview of a specific material group, then you can use the filter function in the flyout.

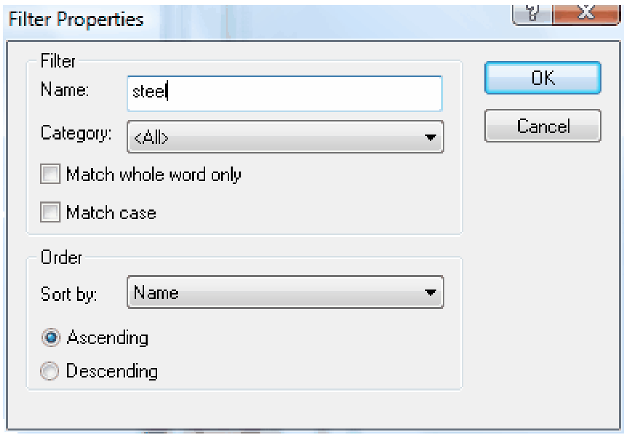


Figure 2: Dialog window - Filter properties

Figure 3: Top part of Material flyout

To look for a material or a material group in the material database click on the icon in the flyout, or use context menu command (right click) **Define Filter...**. The Filter properties dialog window will appear.

Under **Name** you can enter the name or part of the material name that you are searching for. Additionally, you can define the **Category** for which you would like to limit your search to. After you have defined the filter criteria, click on **OK**, and the filter results will be displayed in the materials database window of the flyout.

You can the switch between the **Filter Criteria** and the **Filtered View** settings by clicking on the icon in the flyout or by using the context menu command (right click) **Filtered View**.

Exporting and Importing Materials

The Export command from the context menu (right click) for the category, materials, or database allows you to export the corresponding user-defined material(s) to a ***comma separated*** (CSV) file. The selection of the exported materials is defined by the mouse cursor position at the time the context menu is prompted. Only the user-defined materials can be exported.

Besides loading an existing material database or creating a new database (cf. Options), you can complement the existing flixo material database with data from specially structured text files. To do this, use the context menu command (right click) **Import**. The data will either be inserted into the database or into a category. The position for the insertion of the date depends on whether you right click on the database icon or on the category icon.

The structure of the ***comma separated*** text file corresponds to that of the materials export file. To create a ***comma separated*** text file with the correct structure, it is easiest to export an existing database and then edit that file with a spreadsheet program (e.g. Excel) making sure to save the file as a ***CSV*** text file.

4.3.3 Boundary Conditions Flyout

Content

Description

The ***Boundary Conditions flyout*** allows the manipulation of boundary conditions in an active document. Boundary conditions can be added or deleted and their properties changed.

The **top tree view** displays the boundary conditions that are present in the active document. The **bottom tree view** displays boundary conditions from a boundary conditions databank, which can be used for all documents.

The BC database is divided into two parts: the first part is read-only and is maintained by infomind or third-party, the second part includes all user-defined materials. The boundary conditions maintained by infomind or third-party can be updated either using the menu command **Update...** or in the Options dialog window Database/Templates.

The Properties flyout shows the properties of the selected boundary conditions.

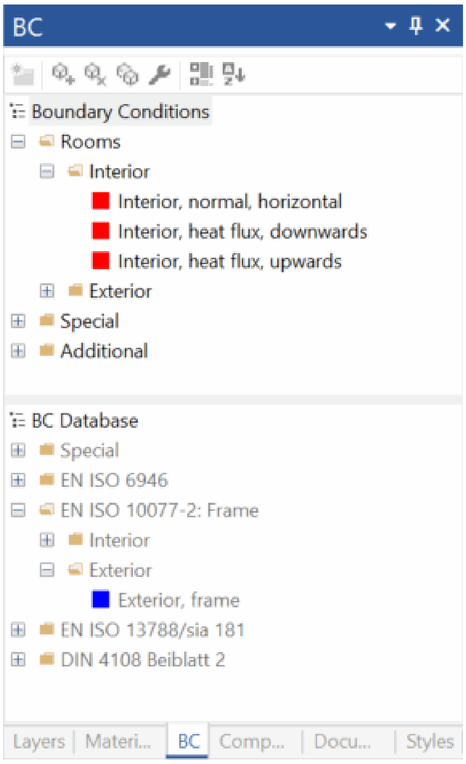


Figure 1: Boundary Conditions flyout

Creation, Modification, Duplication and Deletion of Boundary Conditions

The following description is valid for both the top boundary conditions list (current document) and for the lower list (database).

To create a new boundary condition, click on the icon in the flyout. Alternatively, you can use the context menu command (right click) **New Boundary Condition**. The new material can be defined in the Boundary Condition dialog window. After you have defined the properties, click on **OK** and the new boundary condition appears in the active category.

If you would like to modify the properties of an existing boundary condition, select it in the list, and click either on the icon, or use the context menu command (right click) **Edit**, or double click on the boundary condition you wish to edit. This opens the same dialog window that is used to define a new boundary condition. You can now modify the properties of the boundary condition.

If you wish to rename a material you can do so by choosing the context menu command (right click) **Rename** . You can also rename the boundary condition by selecting it and then clicking on its name just as you would rename a file under Windows Explorer.

To **duplicate** a boundary condition entry with an unique name click on the icon in the flyout. Alternatively, you can create a duplicate by using the corresponding command in the context menu (right click) **Duplicate**. All properties of the copy are identical with the ones of the source, if needed the properties can be adapted in the Boundary Condition dialog window .

Grouping Boundary Conditions into Categories

If you are using many boundary conditions, it makes sense to organize these into groups or categories .

Categories are comparable to file folders and directories; this is why they have the same symbol. Upon creation, every document generates a General category. You can create further categories by clicking on the icon in the flyout or by right clicking on an existing category and using the context menu command New Category. The category is created within the selected category. You can also define a name for this category.

To delete a category select the category and click on the icon, or use the context menu command (right click) Delete. You can rename a category by selecting it and then clicking on the or by using either the context menu command (right click) Edit or Rename. Alternatively, you can rename the category by selecting it and then clicking on it again to rename it just as you would rename a file under Windows Explorer.

Only empty categories can be deleted.

To move a boundary condition from one category to another you can simply drag the boundary condition with the mouse into the desired category.

Copying Boundary Conditions

You can copy boundary conditions from the database to the boundary conditions list of the current document (or vice versa) by dragging the boundary condition to the desired location.

Exporting and Importing Boundary Conditions

The **Export** command from the context menu (right click) for the database, category, or boundary conditions allows you to export the corresponding user-defined boundary conditions to a ***comma separated*** (CSV) file. The selection of the exported boundary conditions is defined by the mouse cursor position at the time the context menu is prompted.

Besides loading an existing boundary condition database or creating a new database (cf. Options), you can complement the existing flixo boundary condition database with data from specially structured text files. To do this, use the context menu command (right click) **Import**. The data will either be inserted into the database or into a category. The position for the insertion of the data depends on whether you right click on the database icon or on the category icon.

The structure of the ***comma separated*** text file corresponds to that of the boundary condition export file. To create a ***comma separated*** text file with the correct structure, it is easiest to export an existing database and then edit that file with a spreadsheet program (e.g. Excel) making sure to save the file as a ***CSV*** text file.

4.3.4 Styles Flyout

Content

Description

The ***Styles flyout*** allows the manipulation of styles in an active document. Styles can be added or deleted and their properties changed.

Styles allow you to define how data, such as isotherms, Psi-values, Heat flows, U-values, lines, rectangles, text, etc., are displayed. All objects will be created according to the style, which is active in the flyout. You can also define new styles or change existing styles.

Styles can be assigned to objects by dragging them onto the object.

The list of styles is organized into various categories. Each category contains a type of style. More detailed information to the styles can be found in the corresponding Styles dialog window.

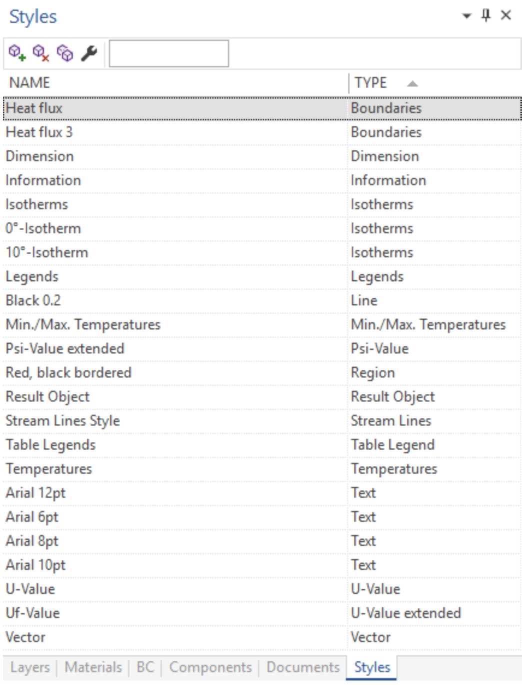


Figure 1: Styles flyout

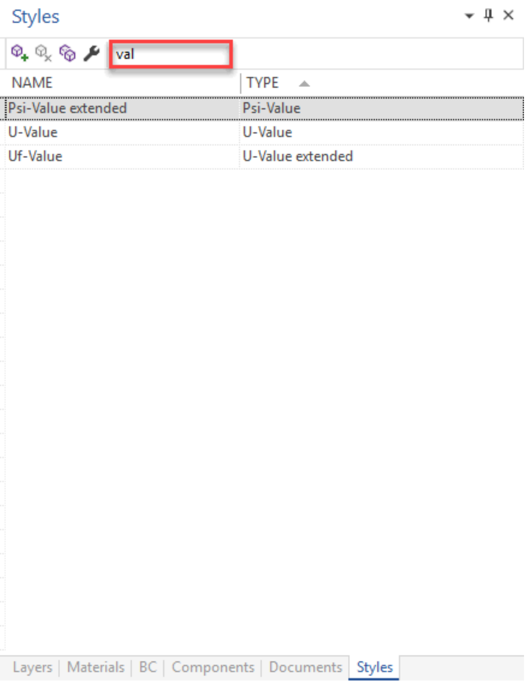


Figure 2: View after filtered for "val"

Creating and Duplicating a Style

You can **create a new style** by clicking on the icon (the corresponding category name or another style of the corresponding category must be marked), or by right clicking on a category (or a style within the category) where you would like to create a new style and use the command **New Style**. In the appearing dialog window, you can define the properties of the new style.

To **duplicate a style** entry with an unique name, click on the icon in the flyout. Alternatively, you can create a duplicate by using the corresponding command in the context menu (right click on the source entry) **Duplicate**. All properties of the copy are identical with the ones of the source, if needed the properties can be adapted in the corresponding style dialog window.

Changing and Deleting a Style

To change a style, select it and click on the icon in the top part of the flyout, or choose the context menu command (right click) **Edit**.

When you want to **rename** a style, then select the style and choose the command **Rename** from the context menu (right click). You can also rename a style by selecting it and then clicking on its name just as you would rename a file under Windows Explorer.

You can **delete a style** by selecting it and clicking on the icon, or by choosing the context menu command (right click) **Delete**.
Note that at least one style per category must be present.

Search Styles, Sort Views

You can also **search** the list for particular styles by entering the desired term in the ***filter field*** in the top part of the flyout. In figure 2 for example, all styles containing "val" in their names are selected.

The list view can be **sorted** by clicking on the titles "name" or "type". In figure 1 for example, the list of styles/types is sorted in a descending manner.

4.3.5 Components Flyout

Content

Description

The ***Components flyout*** allows the management of components. Components are construction elements such as window frames or wall constructions that are regularly used in the various documents.

In the upper half of the flyout, the list of the components at your disposal is displayed. In the lower half, a preview of the selected component is visible (figure 1). New components can be **created**, existing ones **changed** or **deleted**. Components can also be grouped together in categories, just like material and boundary conditions can be organized in such a manner (see Materials chapter for further details). Working with components is explained in detail in tutorial 2.

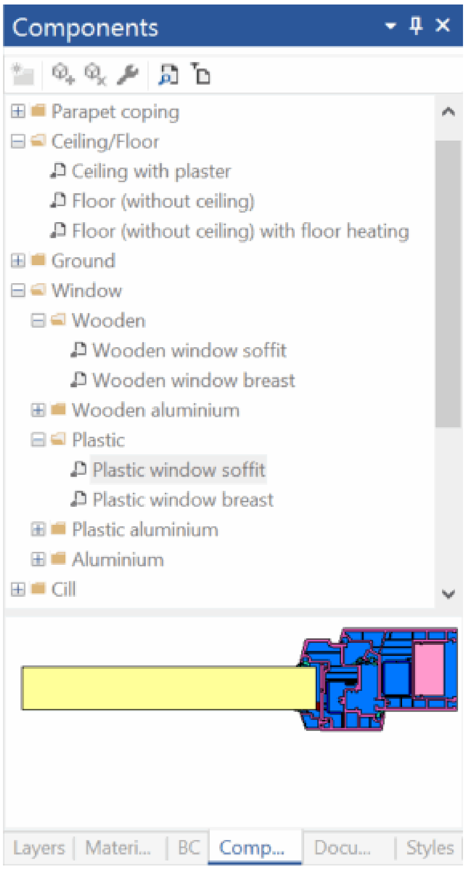


Figure 1: Components flyout

Creating a New Component

Creating a component can be done in two different ways: Either a part of a document can be turned into a component (**selection**), or the component can be created from an **existing file**.

From an Existing File

Components created by flixo are saved with the **fcp** file extension. For example, the components "Window" is normally saved in the Frame.fcp. To integrate an existing component file, proceed as follows:

- Switch to the Component tab and either click on the icon or select the command **New Component** from the context menu (right click) of the category.
- Change the option to **From File** (if you have a selected object in the current document, the option "From selection" is marked by default) in the appearing dialog window.
- In the file dialog window choose the respective component.
- Confirm by clicking **OK**.

After this process, you will find the new component in the components list.

From Selection

To create a new component from a document, proceed as follows:

- First select the object(s) within the document, which will form the component (use the Select, Move, Scale tool).
- With the ***Drag&Drop*** function move the marked objects into the **preview window** of the components flyout. Alternatively, you can also switch to the Component tab and either click on the icon or use the context menu command (right click) **New Component** of the corresponding category.
- In the appearing dialog window enter the name of the component. Additionally, you can also determine in which directory the data should be saved by clicking on the "**...**" button.

 As long as there is not a file with the same name, you can finish by clicking the **OK** button. If there is a file with the same name, there are two remedies: 1. you import the file as described above; 2. you choose From Selection in the "New Components" dialog window and click on the **OK** button. In this case, the existing component will be overwritten by the new component that contains the elements, which you selected.

Changing and Deleting Components

To **change** a component, you must open the corresponding file. You can do this by clicking on the icon or by using context menu command (right click) **Edit** or by simply double clicking on the name of the component. The component file is opened in flixo and can be edited and saved like other documents.

To **delete** a component from the component list, select the component and then click on the icon or the context menu command (right click) **Delete** to remove the component. Note that the component file itself will not be deleted, rather only the entry will be deleted from the components list.

Searching Component Database

If you are looking for a specific component in the component database or if you would like to get an overview of a specific component group, then you can use the filter function in the flyout.

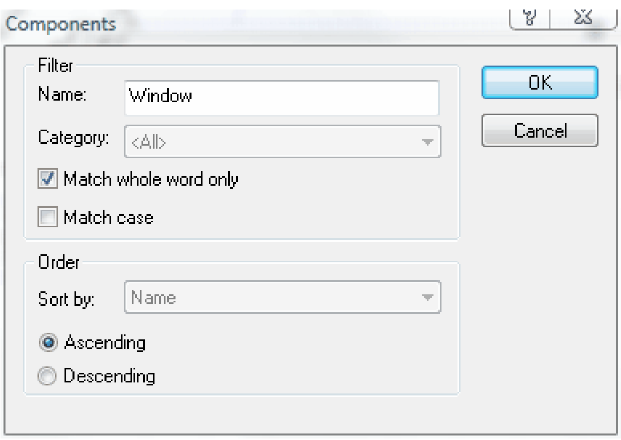


Figure 2: Components Search dialog window

Figure 3: Toolbar

To look for a component or a component group in the component database click on the icon in the Flyout toolbar, or use the context menu command (right click) **Define Filter...**. The Components Search dialog window appears.

Under **Name** you can enter the name or part of the component name that you are looking for. After you have defined the filter criteria, click on **OK**, and the filter results will be displayed in the component database window of the flyout.

You can the switch between the **Filter Criteria** and the **Filtered View** settings by clicking on the icon in the Flyout toolbar or by using the context menu command (right click) **Filtered View**.

Inserting a Component into a Document

To insert a component into a document proceed as follows:

- Open the document into which you would like to insert the component.
- Activate the Component flyout by clicking on the respective tab or by selecting the menu command **View.Components**.
- Select the components that you would like to insert.
- Drag the component from either the list or from the preview window into the document.

Exporting and Importing Component File Lists

With the **Export** command from the context menu (right click) of the component database, you can export a list of user-defined components as a ***comma separated*** value (CSV) file including all components as zipped file. You can export the user-defined components database, a category, or a component by right clicking on corresponding entry, and using the Export command. The selection of the exported components is defined by the mouse cursor position at the time the context menu is prompted.

Besides loading an existing boundary condition database or creating a new database (cf. Options), you can complement the existing flixo components database with data from specially structured text files. To do this, use the context menu command (right click) **Import**. The data will either be inserted into the database or into a not write protected category. The position for the insertion of the data depends on whether you right click on the database icon or on the category icon.

4.3.6 Documents Flyout

Content

Defining

The ***Documents flyout*** allows the management, searching and filtering of documents in a file directory.

In the upper part of the selected directory the list of the flixo documents is visible, in the lower part the preview of the selected documents (cf. file 1). The preview is only possible for files from the most actual flixo.

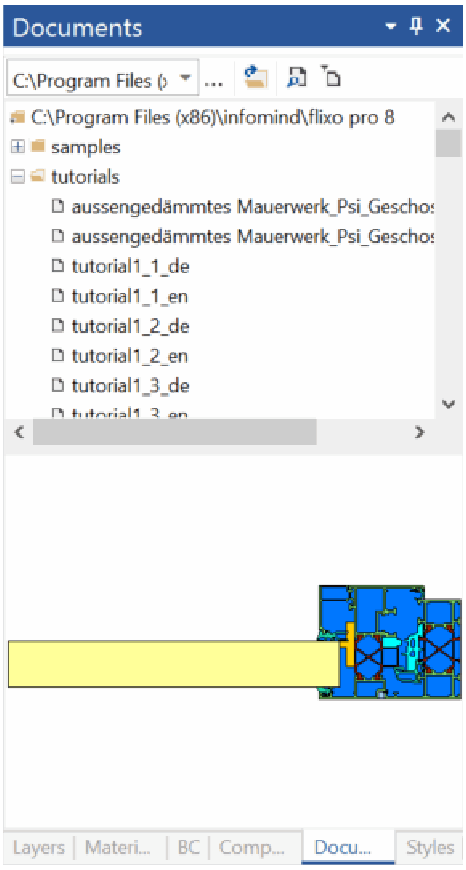


Figure 1: Documents flyout

Searching Files

If you are searching for specific documents in the selected directory including subdirectories you can use the filter function.

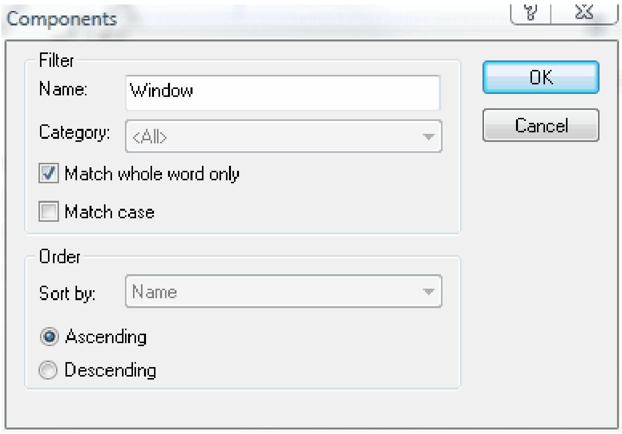


Figure 2: Documents Search dialog window

Figure 3: Toolbar

To search for a document in the file list of the selected directory click on the icon in the Flyout toolbar, or use the context menu command (right click) **Search...**. The Documents Search dialog window appears.

Under **Name** you can enter the component name or part of the component name that you are looking for. After you have defined the filter criteria, click on **OK**, and the filter results will be displayed in the component database window of the flyout.

You can switch in the component database window between the **Filter Criteria** and the **Filtered View** settings by clicking on the icon in the Flyout toolbar or by using the context menu command (right click) **Filtered View**.

Defining Start Directory

You can adjust the Start Directory for displaying flixo documents. Enter the directory path into the control field in
the Flyout toolbar or click on the button "**...**" in the toolbar (cf. figure 3). You can select the directory in the dialog window.

4.3.7 Toolbox Flyout

Description

In the **top part** of the ***Toolbox flyout*** you can select a tool, while in the **bottom part** you can enter respective tool values via keyboard and adjust the tool properties.

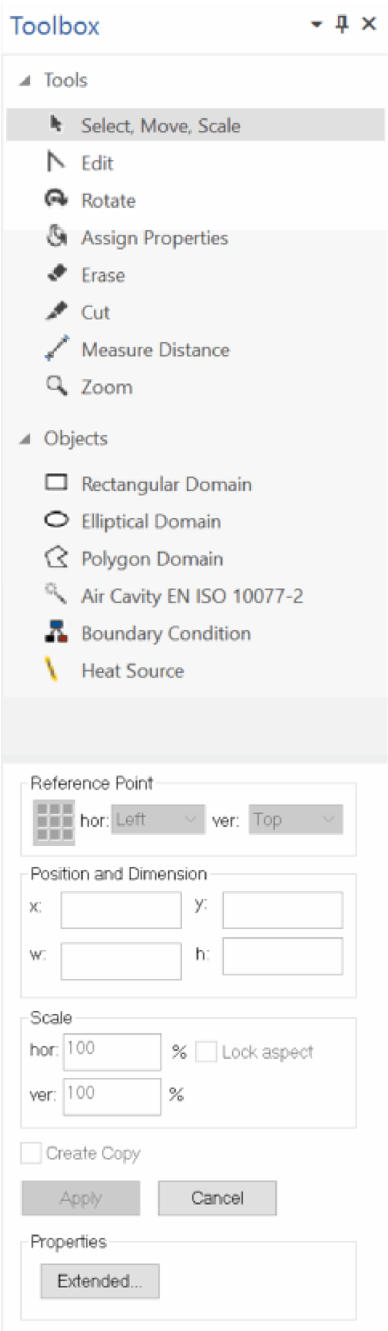


Figure 1: Toolbox flyout

The use of the individual tools is described in detail in the corresponding chapters.

4.3.8 Properties Flyout

Description

In the ***Properties flyout***, the properties of the currently selected object, document, material, boundary condition or component are displayed.

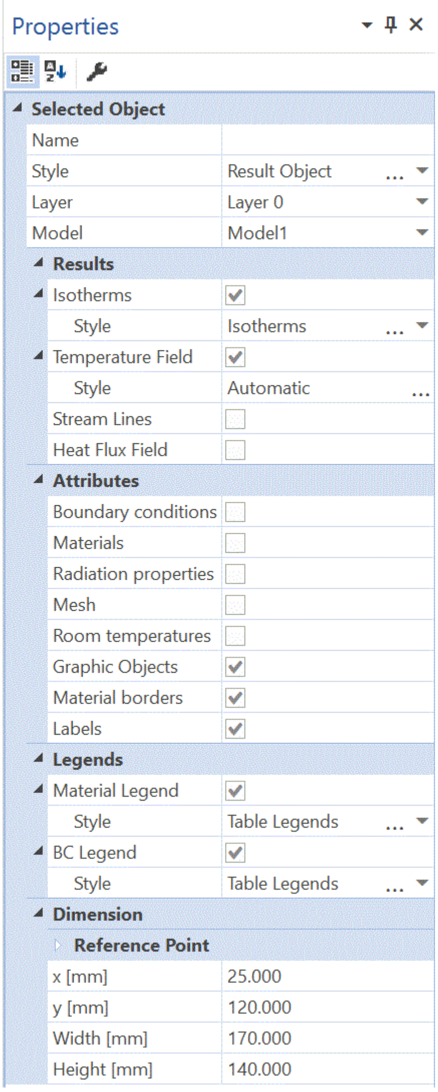


Figure 1: Properties flyout

The use of the Materials, Boundary Conditions, Document and Components flyouts are described in individual flyout chapters.

You can adjust most of the **properties** of the displayed entries directly in the flyout or in the corresponding property dialog windows. Click on the icon in the toolbar of the Properties flyout to activate the property dialog window.

4.3.9 Context Help Flyout

Description

The ***Context Help flyout*** displays the explanation of the active tool.

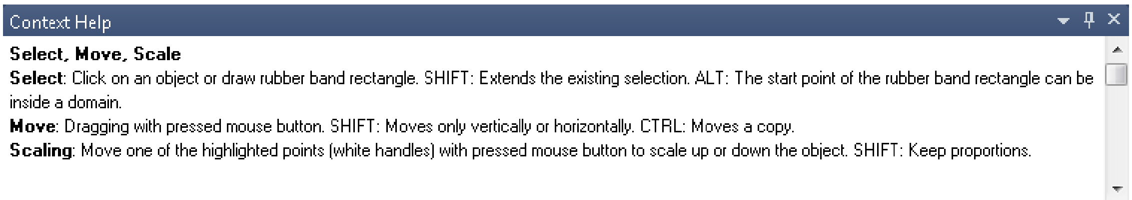


Figure 1: Context Help flyout

If the Context Help flyout is not visible, it can be opened again by the corresponding command in the View menu.

4.4 Dialog Windows

4.4.1 Styles Dialog Window

In this chapter, the style dialog windows are described in detail. These styles are used to change the style of an object in flixo. These dialog windows can be accessed from the Styles flyout. Most dialog windows have a preview of the style on an example.

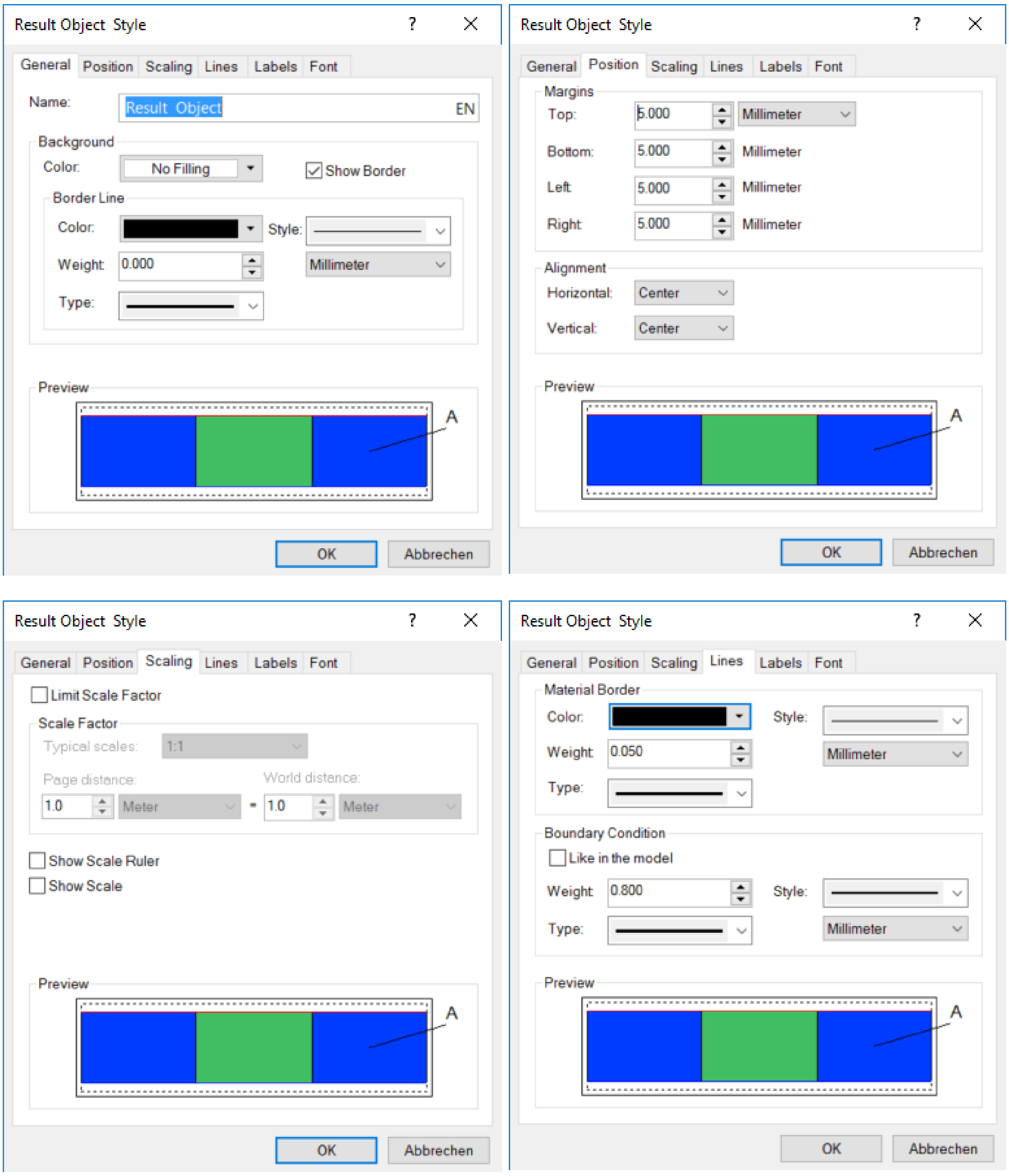
The following style dialog windows are explained in further detail:

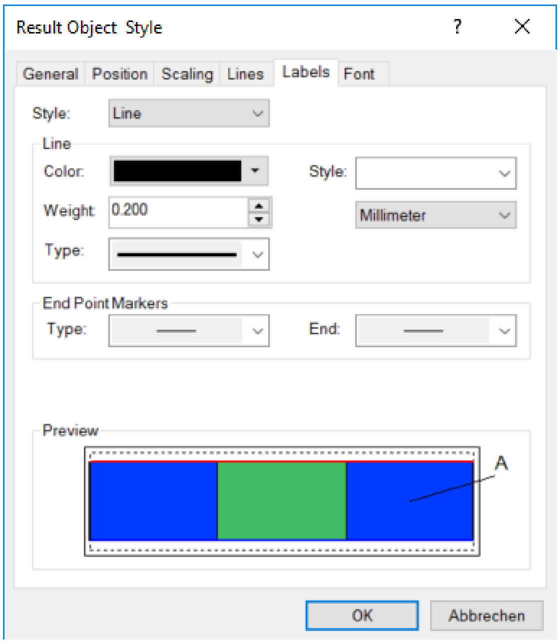
- Result Object
- Temperature
- Min./Max. Temperature
- Isotherms
- Vectors
- Heat Flow
- Stream Lines
- U-Value
- U-Value extended
- Psi-Value
- Table Legends
- Legends
- Dimension
- Information
- Line
- Regions
- Text
- Font Tab

4.4.1.1 Result Object Style

Description

This style is used when you are working with the Result Object tool. The result object style dialog window encompasses properties of the place holder backgrounds, the position of the result within the place holder, the display scale, and, line, label and font properties.


Result Object Style
General
Position
Scaling
Lines
Labels
Font
Name:
Result Object
EN
Background
Color:
No Filling
Show Border
Border Line
Color:
Style:
Weight:
0.000
Millimeter
Type:
Preview
A
OK
Abbrechen
Result Object Style
Margins
Top:
5.000
Millimeter
Bottom:
5.000
Millimeter
Left:
5.000
Millimeter
Right:
5.000
Millimeter
Alignment
Horizontal:
Center
Vertical:
Center
Preview
OK
Abbrechen
Result Object Style
Limit Scale Factor
Scale Factor
Typical scales:
1:1
Page distance:
World distance:
1.0
Meter
=
1.0
Meter
Show Scale Ruler
Show Scale
Preview
OK
Abbrechen
Result Object Style
Material Border
Color:
Style:
Weight:
0.050
Millimeter
Type:
Boundary Condition
Like in the model
Weight:
0.800
Style:
Type:
Millimeter
Preview
OK
Abbrechen



Fields

General Tab

Name

Here, you define the name of the style. You cannot leave this field empty; the chosen name must be unique, i.e. the name cannot be identical to another existing name. Aside from these limitations, you are free to choose any name.

Click on the language abbreviation in this field to set the name in all supported languages.

Background

This group contains the settings for the background color of the result object area. It allows you to hide, unhide, and adjust the border lines.

Border

This setting group determines the characteristics of the line, which circumscribes the result object (color, style, size and type).

Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.

Position Tab

Margins

In this group, you can adjust the size of the borders circumscribing the results.

Alignment	In this group, you can define the horizontal and vertical alignment of the results within the result object area.

Scaling Tab

Scale Factor	In this group you define the maximum scale for the results display. If it is not possible to display the result with this scale in the result object area, then the scale will be automatically adjusted.
Show Scale Ruler	Draws a scale ruler below the result object
Show Scale	Labels the result object with the current scale

Lines Tab

Material Border	This setting group determines the properties of the line, which surrounds the material borders (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.
Boundary Condition	This setting group determines the properties of boundary condition lines (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.

Labels Tab

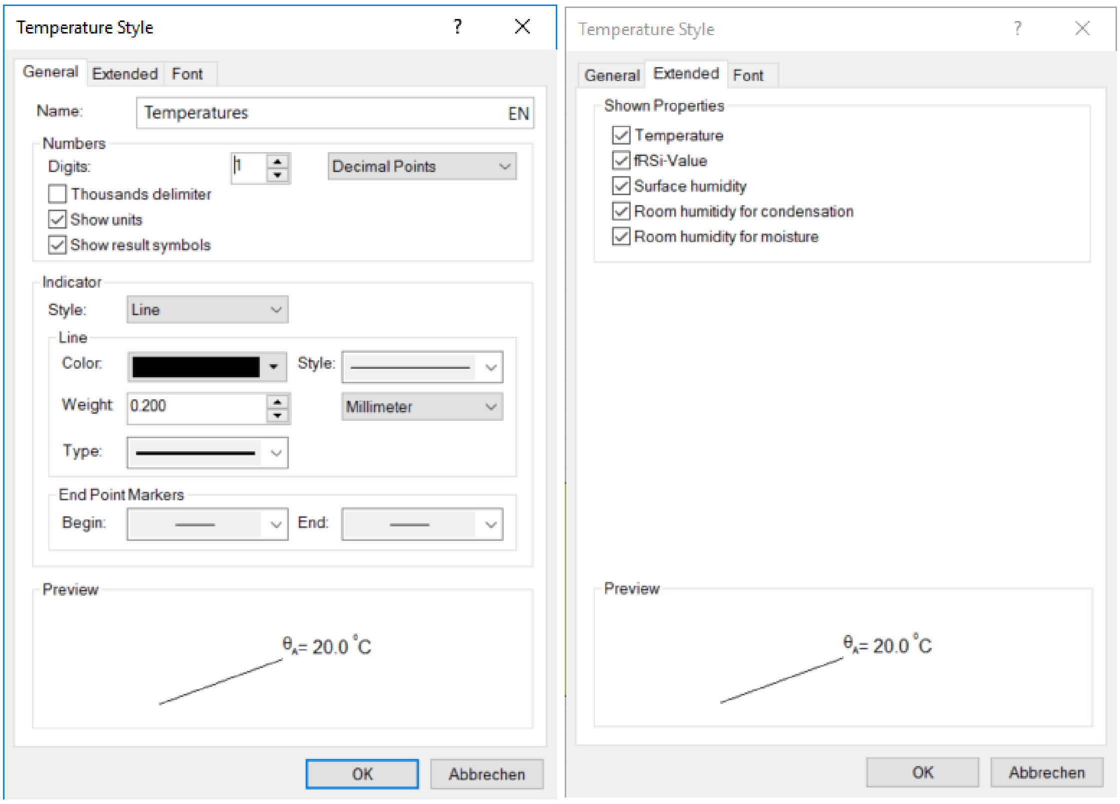
Line	This setting group determines the characteristics of label lines (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.
End Point Markers	Here, you can set how the end of label lines look. There are two drop down lists that show the different possibilities.

Font Tab

4.4.1.2 Temperature Style

Description

This style is used when you are working with the Temperature tool. The temperature style dialog window consists of a label line, a text field, and additional properties for surface points.



Fields

General Tab

Name

Here, you define the name of the temperature style. You cannot leave this field empty; the chosen name must be unique, i.e. the name cannot be identical to another existing name. Aside from these limitations, you are free to choose any name.

Click on the language abbreviation in this field to set the name in all supported languages.

Digits

Number of decimal places or significant digits, which are to be displayed. The kind - **Decimal Points** or

	Significant - you can select using the adjacent drop down-list.
Thousands delimiter	The digits left of the decimal point will be grouped into thousands.
Show units	The appropriate units will be shown after each numerical value. The units can be adjusted in the Units tab of the Options dialog window.
Show result symbols	The appropriate symbol (dependent on the type of value) will be shown before each numerical value.
Style	Here, you determine whether the label line should be a simple or an orthogonal (right-angle bent) line.
Line	This setting group determines the characteristics of label lines (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.
End Point Markers	Here, you can determine how the ends of both label lines should look like. There are two drop down lists that show the different possibilities.

Extended Tab

Temperature	If this option is activated, the temperature of the reference point is displayed. At least one option of the group "Shown Properties" has to be selected. For points within the construction the temperature is always displayed.
fRSi-Value	If this option is activated, the fRSi-factor **for surface points** is shown additionally. At least one option of the group "Shown Properties" has to be selected.
Surface humidity	If this option is activated, the surface humidity **for surface points** is shown additionally. At least one option of the group "Shown Properties" has to be selected.
Room humidity for condensation	If this option is activated, the room humidity for condensation **for surface points** is shown additionally. At least one option of the group "Shown Properties" has to be selected.
Room humidity for moisture	If this option is activated, the room humidity for moisture **for surface points** is shown additionally. At

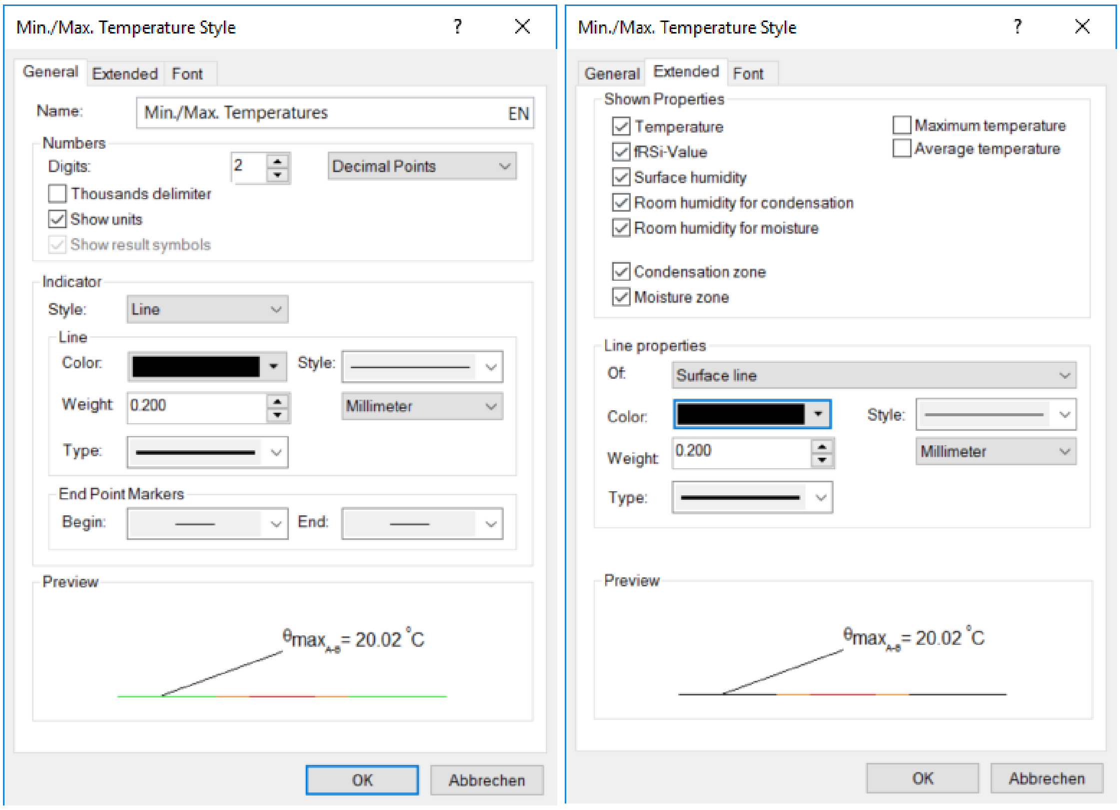
least one option of the group "Shown Properties" has to be selected.

Font Tab

4.4.1.3 Min./Max. Temperature Style

Description

This style is used when you are working with the Min./Max. Temperature tool.



Fields

General Tab

Name

Here, you define the name of the style. You cannot leave this field empty; the chosen name must be unique, i.e. not identical to another existing name. Aside from these limitations, you are free to choose any name.

	Click on the language abbreviation in this field to set the name in all supported languages.
Digits	Number of decimal places or significant digits, which are to be displayed. The kind - **Decimal Points** or **Significant** - you can select using the adjacent drop down-list.
Thousands delimiter	The digits left of the decimal point will be grouped into thousands.
Show units	The appropriate units will be shown after each numerical value. The units can be adjusted in the Units tab of the Options dialog window.
Show result symbol	The appropriate symbols (dependent on the type of value) will be shown before each numerical value.
Style	Here, you determine whether the label line should be a simple or an orthogonal (right-angle bent) line.
Line	This setting group determines the properties of label lines (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.
End Point Markers	Here, you can determine how the end of both label lines should look like. There are two drop down lists that show the different possibilities.

Extended Tab

Temperature	If this option is activated, the minimal surface temperature is displayed. At least one option of the group "Shown Properties" has to be selected.
fRSi-Value	If this option is activated, the fRSi-factor is shown additionally to the minimal surface temperature. At least one option of the group "Shown Properties" has to be selected.
Surface humidity	If this option is activated, the surface humidity is shown additionally to the minimal surface temperature. At least one option of the group "Shown Properties" has to be selected.
Room humidity for condensation	If this option is activated, the room humidity for condensation is shown additionally to the minimal

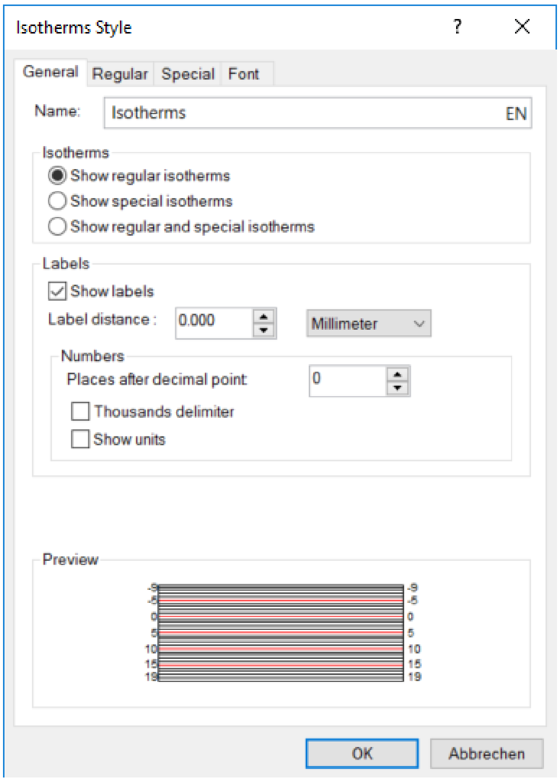
	surface temperature. At least one option of the group "Shown Properties" has to be selected.
Room humidity for moisture	If this option is activated, the room humidity for moisture is shown additionally to the minimal surface temperature. At least one Option of the group "Shown Properties" has to be selected.
Condensation zone	If this option is activated, a condensation zone will be displayed with a unique line style - only if a condensation zone is present. The properties of the polyline you can adapt in the setting group **line properties** (**Of: Condensation zone**).
Moisture zone	If this option is activated, zones where the surface humidity exceed the critical humidity will be displayed with a unique line style - only if this zone is present. The properties of the polyline you can adapt in the setting group **line properties** (**Of: Moisture zone**). The critical surface humidity you can define in the Properties flyout of the min./max. temperaturobject.
Maximum temperature	If this option is activated then the maximum temperature is shown next to the minimum temperature.
Average temperature	If this option is activated then the average temperature will be shown next to the minimum temperature.
Line properties	This setting group determines the properties of the **surface edge line**, the **condensation zone** and the **moisture zone** (color, style, size and type). The line kind you can select using the **Of:** drop down-list. Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.

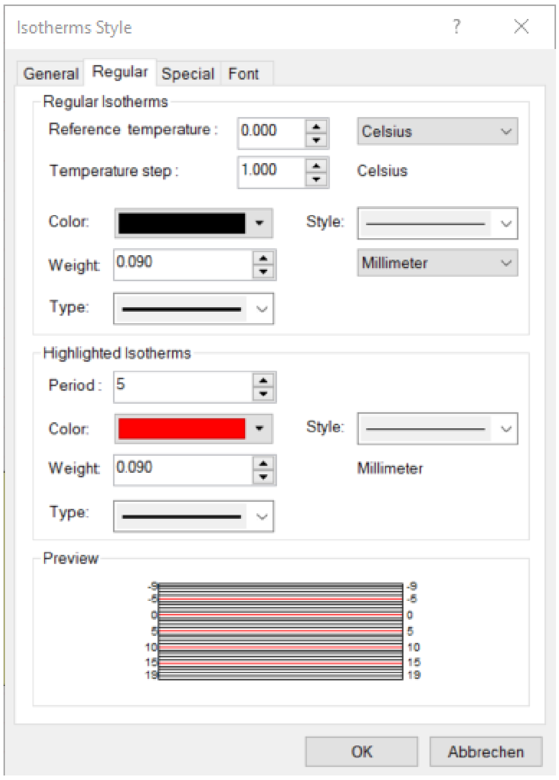
Font Tab

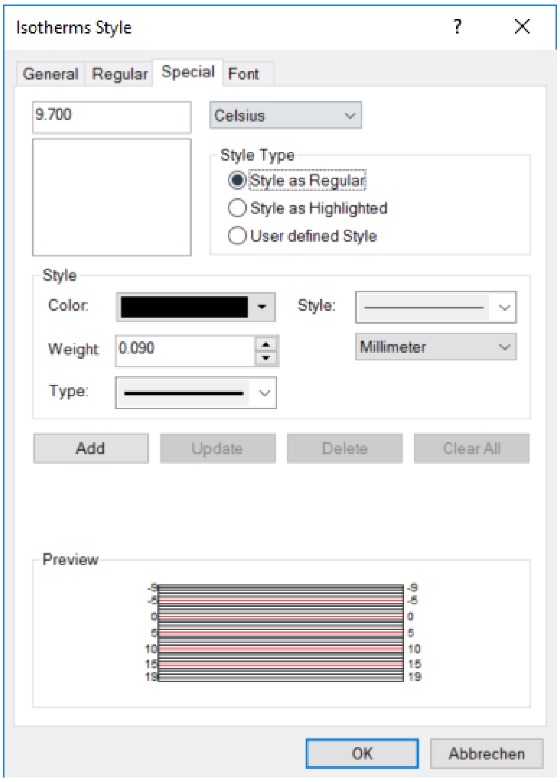
4.4.1.4 Isotherms Style

Description

This style is used when you have isotherms displayed. You can edit both the graphical aspects of the isotherms as well as the isotherm separation. You can also define isotherms with special temperatures.


Isotherms Style
?
General
Regular
Special
Font
Name:
Isotherms
EN
Isotherms
Show regular isotherms
Show special isotherms
Show regular and special isotherms
Labels
Show labels
Label distance :
0.000
Millimeter
Numbers
Places after decimal point
0
Thousands delimiter
Show units
Preview
OK
Abbrechen


Isotherms Style
?
General
Regular
Special
Font
Regular Isotherms
Reference temperature :
0.000
Celsius
Temperature step :
1.000
Celsius
Color:
Style:
Weight
0.090
Millimeter
Type:
Highlighted Isotherms
Period :
5
Color:
Style:
Weight
0.090
Millimeter
Type:
Preview
OK
Abbrechen


Isotherms Style
?
General
Regular
Special
Font
9.700
Celsius
Style Type
Style as Regular
Style as Highlighted
User defined Style
Style
Color:
Style:
Weight
0.090
Millimeter
Type:
Add
Update
Delete
Clear All
Preview
OK
Abbrechen

Fields

General Tab

Name	Here, you define the name of the style. You cannot leave this field empty; the chosen name must be unique, i.e. not identical to another existing name. Aside from these limitations, you are free to choose any name. Click on the language abbreviation in this field to set the name in all supported languages.
Isotherms	Here, you determine which isotherms will be displayed: **Show regular isotherms**: Isotherms with regular isotherm separation will be displayed. The properties of this type of isotherm can be set in the Regular tab. **Show special isotherms**: Isotherms with freely definable temperatures will be displayed. The temperatures and the properties of this type of isotherm can be set in the Special tab. **Show regular and special isotherms**: Both the aforementioned regular and special isotherm types will be displayed. The characteristics of these isotherms can be set in the corresponding tabs.
Isotherm labels	Here, you determine whether the isotherms should be labeled or not.
Label distance	Here, you can define an additional label separation distance.
Places after decimal point	Number of post decimal place holders in the isotherm label, which are to be shown.
Thousands delimiter	The digits left of the decimal point will be grouped into thousands.
Show units	The appropriate units will be shown after each numerical value. The units can be adjusted in the Units tab of the Options dialog window.

Regular Tab

First Temperature	The calculation of all other regular isotherms is based on this temperature. That means that all regular isotherms, along with the defined isotherm separation

distance, are defined based on this temperature. This temperature determines at what point flixo displays isotherms. The first temperature is displayed, and then all others are shown at the "Highlighted Line" period.

Temperature step

With this setting, you can define the isotherm temperature difference distance of two regular isotherms. Changing the isotherm separation distance affects all of the regular isotherms of the result object, which use this style.

Regular Line

This setting group determines the properties of regular isotherm lines (color, style, size and type).

Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.

Highlighted Isotherms

To make isotherm lines more legible, you can highlight every n-th isotherm by determining special properties. The specific settings of the **highlighted isotherms** (color, style, thickness) are set in the "Highlighted Line" frame.

Highlighted Line

This setting group determines the properties of highlighted isotherm lines (color, style, size and type).

Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.

Special Tab

Temperatures

Here, you can define the temperatures of special isotherms. All special temperatures are shown in the list. A temperature will only be added after you click on the **Add** button. A temperature will only be changed after clicking the **Update** button.

Art

In this setting group, you can define the graphical properties of special isotherms. You can either use the properties of the regular or highlighted isotherms (cf. Regular tab). Alternatively, a special user-defined style can be selected, which you can define in the Style group below.

The settings will only be applied after clicking on **Add** or **Update**.

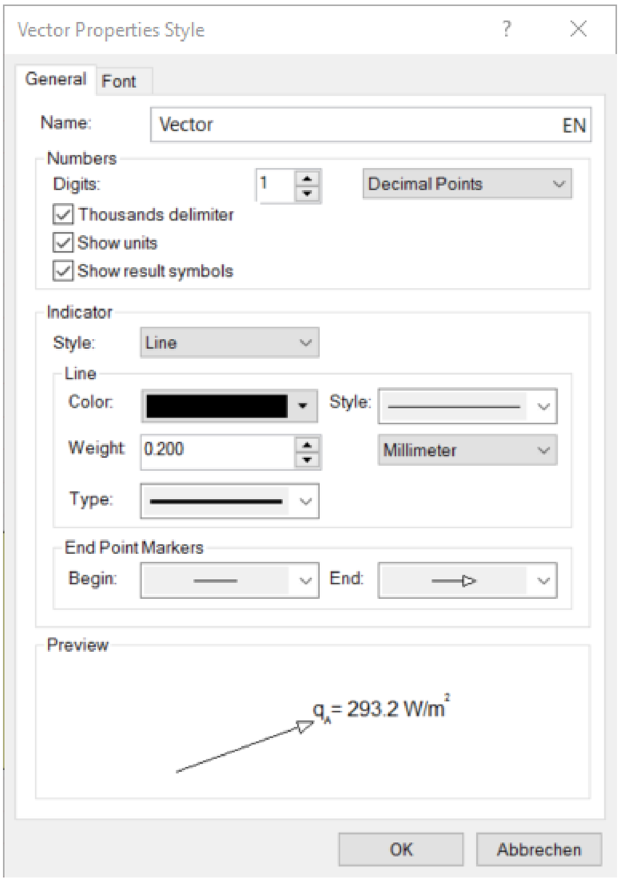
Style	This setting group determines the properties (color, thickness, type, style) of the special user-defined isotherms. The temperature corresponds to the appropriate edit field. The settings will only be applied after you click on **Add** or **Update**. Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.
Add	Adds a new special isotherm to the list. The temperature is taken from the edit field and the style, from the settings described above, is applied.
Update	Changes the settings of a special isotherm. The temperature is taken from the edit field and the style, from the settings described above, is applied.
Delete	Deletes the special isotherm selected in the edit field.
Clear All	Deletes the isotherm selected in the edit field.

Font Tab

4.4.1.5 Vector Style

Description

This style is used when working with the Heat Flux tool.



Fields

General Tab

Name

Here, you define the name of the style. You cannot leave this field empty; the chosen name must be unique, i.e. not identical to another existing name. Aside from these limitations, you are free to choose any name.

Click on the language abbreviation in this field to set the name in all supported languages.

Digits

Number of decimal places or significant digits, which are to be displayed. The kind - **Decimal Points** or **Significant** - you can select using the adjacent drop down-list.

Thousands delimiter

The digits left of the decimal point will be grouped into thousands.

Show units

The appropriate units will be shown after each numerical value. The units can be adjusted in the Units tab of the Options dialog window.

Show result symbols

The appropriate symbol (dependent on the type of value) will be shown before each numerical value.

Style

Here, you determine whether the label line should be a simple or an orthogonal (right-angle bent) line.

Line

This setting group determines the properties of label lines (color, style, size and type).

Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.

End Point Markers

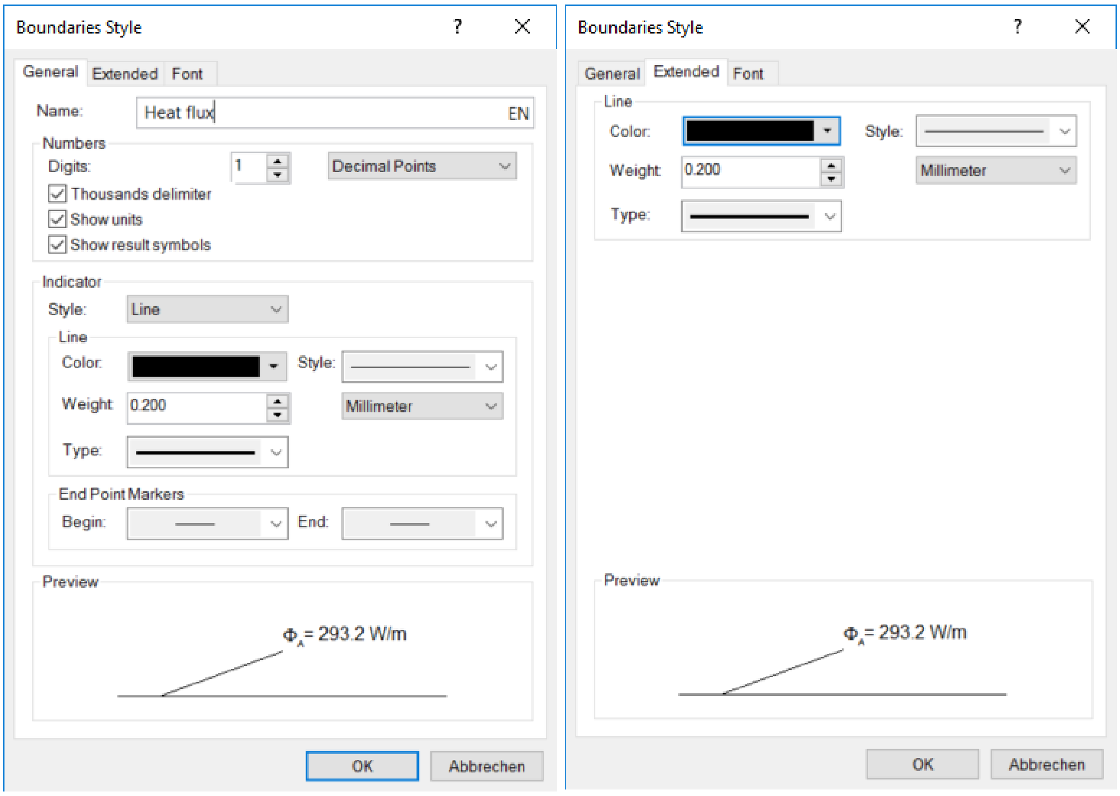
Here, you can determine how the end of both label lines should look like. There are two drop down lists that show the different possibilities.

Font Tab

4.4.1.6 Heat Flux Style

Description

This style is used when working with the Heat Flux tool. The Heat Flow style consists of an object edge line and a label line.



Fields

General Tab

Name	Here, you define the name of the style. You cannot leave this field empty; the chosen name must be unique, i.e. not identical to another existing name. Aside from these limitations, you are free to choose any name. Click on the language abbreviation in this field to set the name in all supported languages.
Digits	Number of decimal places or significant digits, which are to be displayed. The kind - **Decimal Points** or **Significant** - you can select using the adjacent drop down-list.
Thousands delimiter	The digits left of the decimal point will be grouped into thousands.
Show units	The appropriate units will be shown after each numerical value. The units can be adjusted in the Units tab of the Options dialog window.
Show result symbols	The appropriate symbol (dependent on the type of value) will be displayed before each numerical value.
Style	Here, you determine whether the label line should be a simple or an orthogonal (right angle bent) line.
Line	This setting group determines the properties of the label lines (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.
End Point Markers	Here, you can determine how both ends of the label line should look like. There are two drop down lists that show the different possibilities.

Extended Tab

Line	This setting group determines the characteristics of the object edge lines (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.

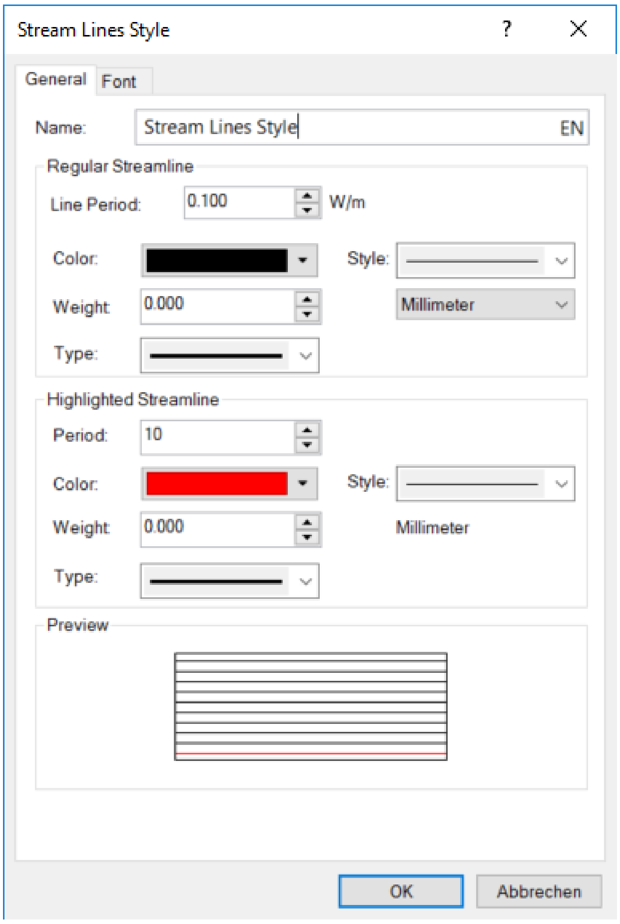
Font Tab

4.4.1.7 Streamline Style

energy plus, pro

Description

This style is used when streamlines are displayed. The graphic properties of the streamlines as well as the streamline period can be adjusted.



Fields

General Tab

Name	Here, you define the name of the style. You cannot leave this field empty; the chosen name must be unique, i.e. not identical to another existing name. Aside from these limitations, you are free to choose any name. Click on the language abbreviation in this field to set the name in all supported languages.
Line Period	In this setting, the number and density of the streamlines can be determined be defining the

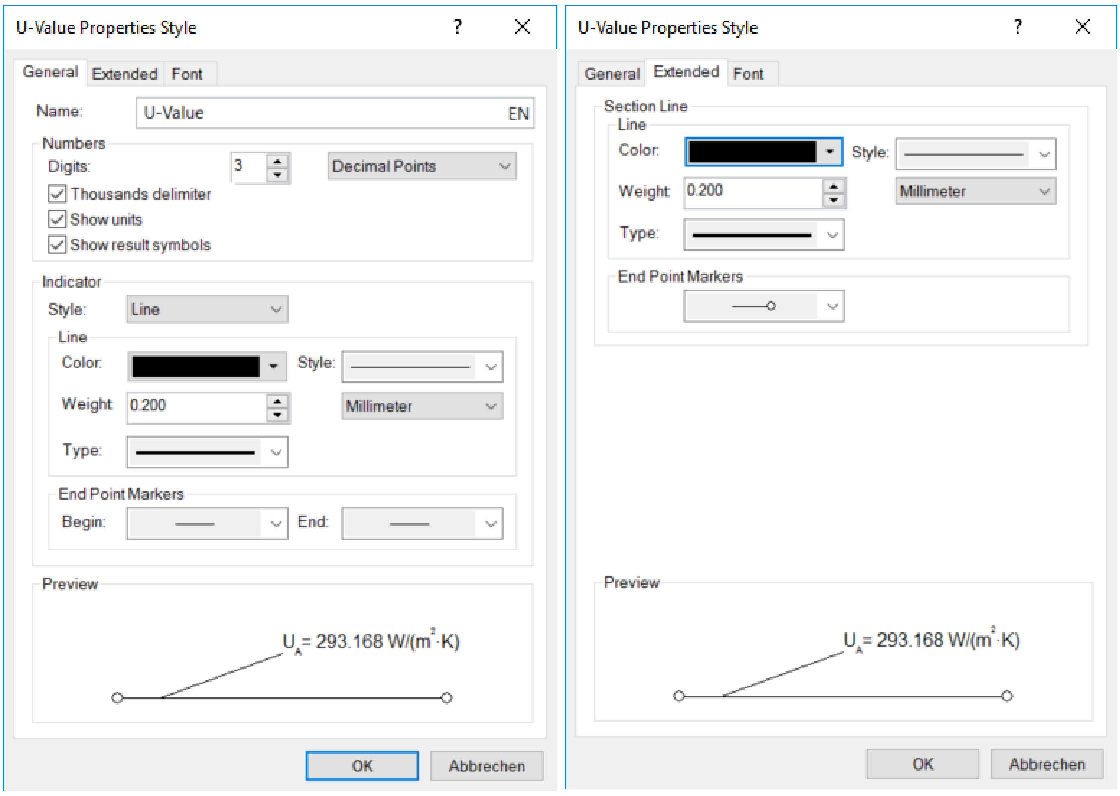
	heat flow, which flows between two neighboring streamlines.
Regular Line	This setting group determines the properties of regular streamlines (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.
Period	To make streamlines more legible, you can have every n-th highlighted with the properties you define here. The specific settings of the highlighted streamlines (color, style, thickness) are set in the **"Highlighted Line"** frame.
Highlighted Line	This setting group determines the properties of highlighted streamlines (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.

Font Tab

4.4.1.8 U-Value Style

Description

This style is used when you are working with the U-Value tool. The U-value style consists of a section line (line segment with two end point markers), and of a connected label line.



Fields

General Tab

Name

Here, you define the name of the style. You cannot leave this field empty; the chosen name must be unique, i.e. not identical to another existing name. Aside from these limitations, you are free to choose any name.

Click on the language abbreviation in this field to set the name in all supported languages.

Digits

Number of decimal places or significant digits, which are to be displayed. The kind - **Decimal Points** or **Significant** - you can select using the adjacent drop down-list.

Thousands delimiter

The digits left of the decimal point will be grouped into thousands.

Show Units

The appropriate units will be shown after each numerical value. The units can be adjusted in the Units tab of the Options dialog window.

Show result symbols

The appropriate symbol (dependent on the type of value) will be shown before each numerical value.

Style	Here, you determine whether the label line should be a simple or an orthogonal (right angle bent) line.
Line	This setting group determines the properties of label lines (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.
End Point Markers	Here, you can define how both ends of the label line should look like. There are two drop down lists that show the different possibilities.

Extended Tab

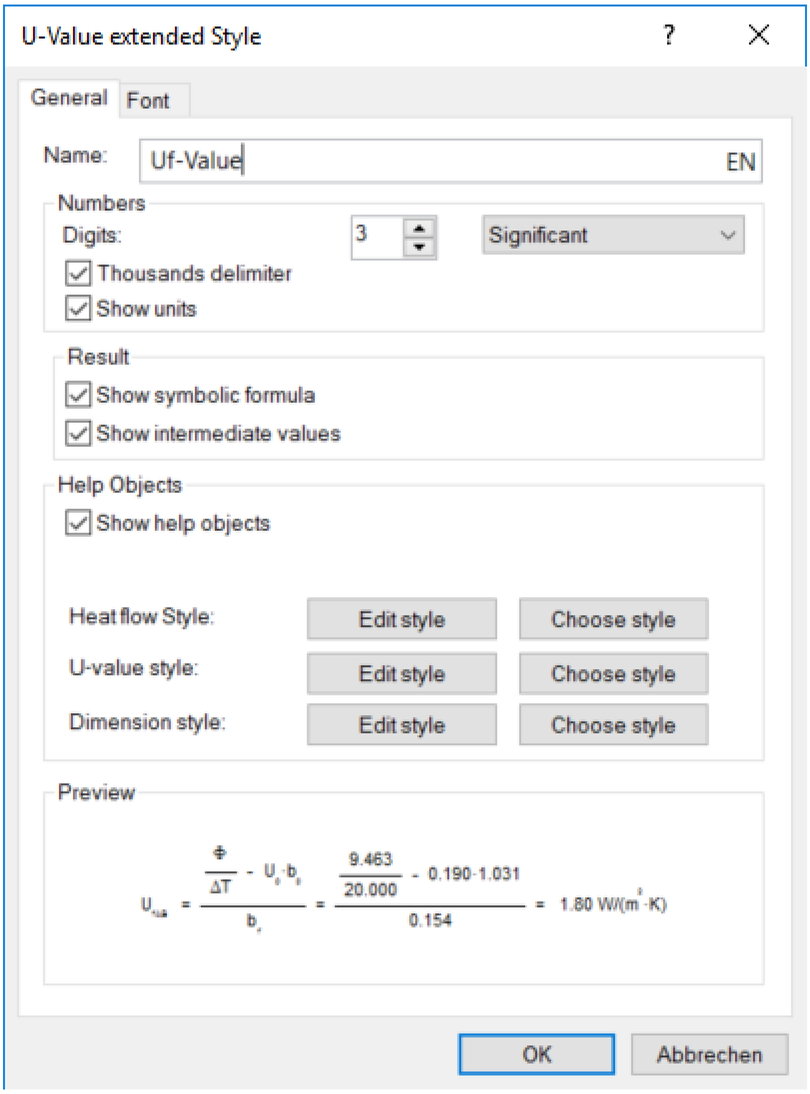
Line	This setting group determines the properties of the section lines (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.
End Point Markers	Here, you can define how the ends of the section line (segment lines) should look like. There are two drop down lists that show the different possibilities. The same style is used for both ends of the line segment.
Use custom boundary conditions	When this option is active, then the U-value calculation is done with the user defined boundary condition and not with the boundary condition defined in the object. The chosen boundary condition is shown on the U-value object.
he	Here, you define the exterior heat transfer coefficient for the calculation of the U-value, if you are not going to use the value already defined in the object.
hi	Here, you define the interior heat transfer coefficient for the calculation of the U-value, if you are not going to use the value already defined in the object.

Font Tab

4.4.1.9 U-Value extended Style

Description

This style is used when you let flixo calculate a frame U-value, a joint U-value, a roller shutter box U-value or an equivalent U-value and display the results. In tutorial 4, frame U-value calculation is described and in tutorial 5, joint U-value and equivalent are described.



Fields

General Tab

Name

Here, you define the name of the style. You cannot leave this field empty; the chosen name must be unique, i.e. not identical to another existing name. Aside from these limitations, you are free to choose any name.

Click on the language abbreviation in this field to set the name in all supported languages.

Digits

Number of decimal places or significant digits, which are to be displayed. The kind - **Decimal Points** or **Significant** - you can select using the adjacent drop down-list.

Thousands delimiter	The digits left of the decimal point will be grouped into thousands.
Show units	The appropriate units will be shown after each numerical value. The units can be adjusted in the Units tab of the Options dialog window.
Show symbolic formula	Here, you determine whether you want to display the symbolic formula for the result object.
Show intermediate values	Here you determine whether you want to display the intermediate formula used for the result object U-value calculation.
Show help objects	Here, you determine whether help result objects (dimension, U-value and heat flow objects) should be automatically added or not.
Heat flow style	By clicking on the "Edit Style" button, a dialog window will appear where you can adjust the style of the heat flow help object. By clicking on the "Choose Style" button, a dialog window will appear where you can choose and apply a heat flow style. The current settings of the selected style will be copied. If the selected style is modified later, it has no influence on the appearance of an U-value object.
U-value style	By clicking on the "Edit Style" button, a dialog window will appear where you can adjust the style of the U-value help object. By clicking on the "Choose Style" button, a dialog window will appear where you can choose and apply a U-value style. The current settings of the selected style will be copied. If the selected style is modified later, it has no influence on the appearance of an U-value object.
Dimension style	By clicking on the "Edit Style" button, a dialog window will appear where you can adjust the style of the dimension help object. A dialog window will appear where you can adjust the style of the dimension help object. By clicking on the "Choose Style" button, a dialog window will appear where you can choose and apply a dimension style. The current settings of the selected style will be copied. If the selected style is modified later, it has no influence on the appearance of an U-value object.

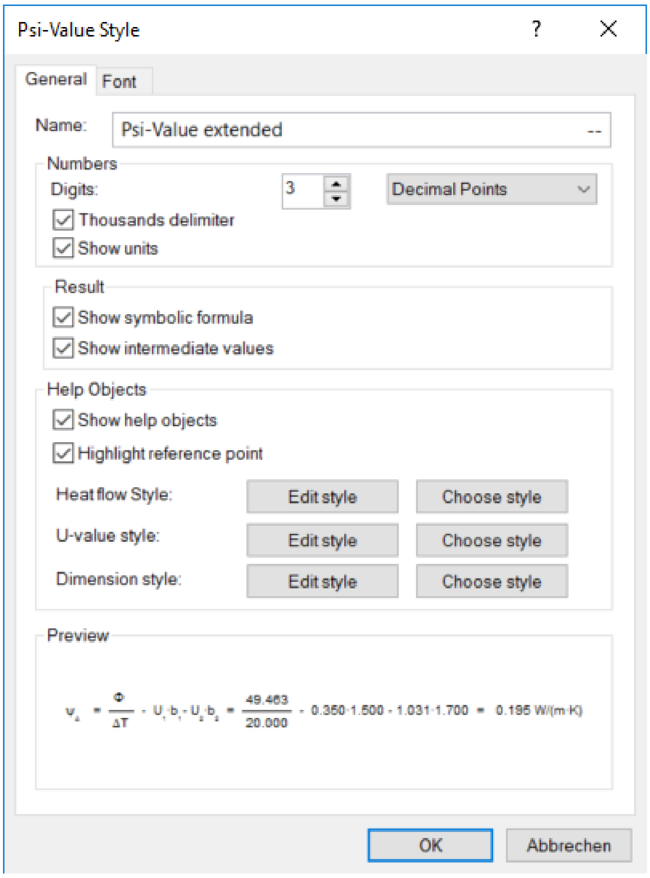
Font Tab

4.4.1.10 Psi-Value Style

energy, energy plus, pro

Description

This style is applied when you are working with the Psi-Value tool.



Fields

General Tab

Name	Here, you define the name of the style. You cannot leave this field empty; the chosen name must be unique, i.e. not identical to another existing name. Aside from these limitations, you are free to choose any name.

	Click on the language abbreviation in this field to set the name in all supported languages.
Digits	Number of decimal places or significant digits, which are to be displayed. The kind - **Decimal Points** or **Significant** - you can select using the adjacent drop down-list.
Thousands delimiter	The digits left of the decimal point will be grouped into thousands.
Show units	The appropriate units will be shown after each numerical value. The units can be adjusted in the Units tab of the Options dialog window.
Show symbolic formula	Here, you determine whether you want to display the symbolic formula for the result object.
Show intermediate values	Here, you determine whether you want to display the intermediate formula used for the result object Psi-value calculation.
Show help objects	Here, you determine whether help result objects (dimension, U-value and heat flow objects) should be automatically added or not.
Highlight reference point	If this option is activated, the position of the reference point is emphasized by an arrow.
Heat flow style	By clicking on the "Edit Style" button, a dialog window will appear where you can adjust the style of the heat flow help object. By clicking on the "Choose Style" button, a dialog window will appear where you can choose and apply a heat flow style. The current settings of the selected style will be copied. If the selected style is modified later, it has no influence on the appearance of a Psi-value object
U-value style	By clicking on the "Edit Style" button, a dialog window will appear where you can adjust the style of the U-value help object. By clicking on the "Choose Style" button, a dialog window will appear where you can choose and apply a U-value style. The current settings of the selected style will be copied. If the selected style is modified later, it has no influence on the appearance of a Psi-value object.

Dimension style

By clicking on the "Edit Style" button, a dialog window will appear where you can adjust the style of the dimension help object.

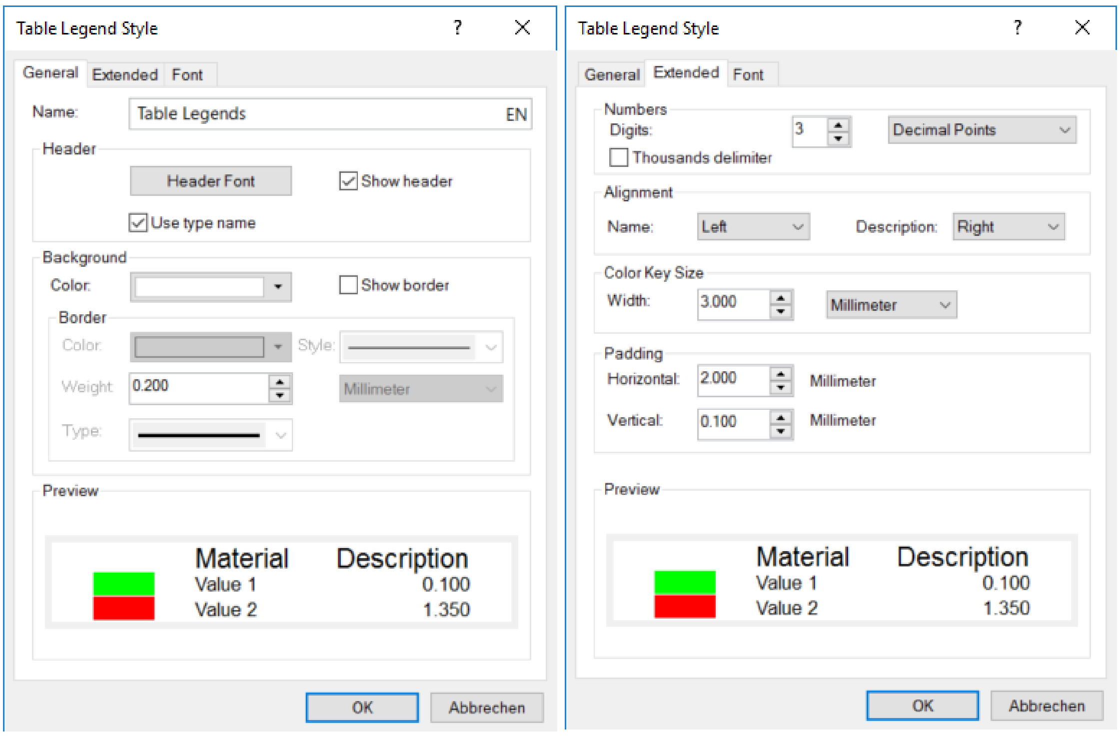
By clicking on the "Choose Style" button, a dialog window will appear where you can choose and apply a dimension style. The current settings of the selected style will be copied. If the selected style is modified later, it has no influence on the appearance of a Psi-value object.

Font Tab

4.4.1.11 Table Legend Style

Description

This style is used when you display a Material or Boundary Condition legend.



Fields

General Tab

Name	Here, you define the name of the style. You cannot leave this field empty; the chosen name must be unique, i.e. not identical to another existing name. Aside from these limitations, you are free to choose any name. Click on the language abbreviation in this field to set the name in all supported languages.
Header	This setting group contains the button Header Font, which activates a standard dialog window where the font style of the header can be adjusted.
Show header	If this option is active, then the header will be shown.
Use type name	If this option is active, then the type of the table ("Material" or "Boundary Condition") will be shown, instead of "Name".
Background	This group contains the settings for the background color of the legend. It also allows you to adjust the border lines as well as hide or unhide them.
Border	This setting group determines the properties of the legend border lines (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.

Extended Tab

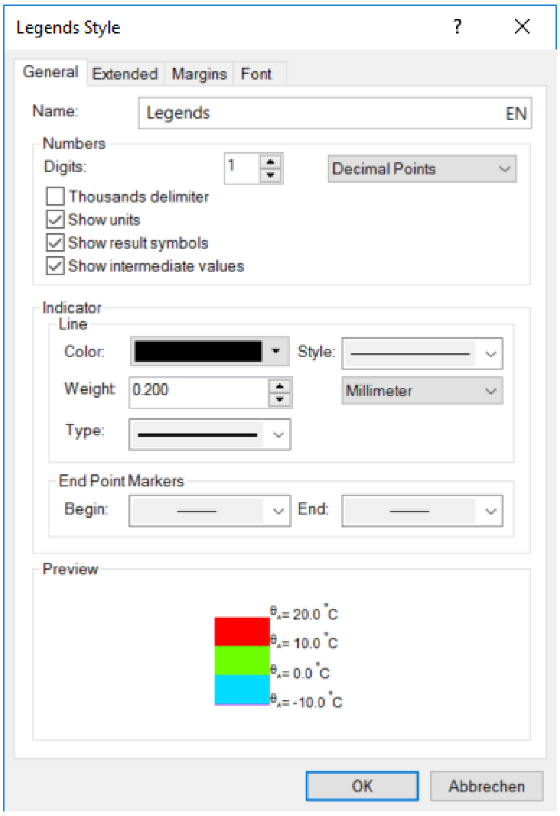
Digits	Number of decimal places or significant digits, which are to be displayed. The kind - **Decimal Points** or **Significant** - you can select using the adjacent drop down-list.
Thousands delimiter	The digits left of the decimal point will be grouped into thousands.
Alignment	This setting group lets you align the legend text. You can determine the alignment with the two drop down lists "Name" and "Description".
Key Color Size	Here, you can determine the width ("size") of the color keys in the legends. You can observe the change in size in the preview pane.
Padding	This setting adjusts the padding between horizontal and vertical legend columns.

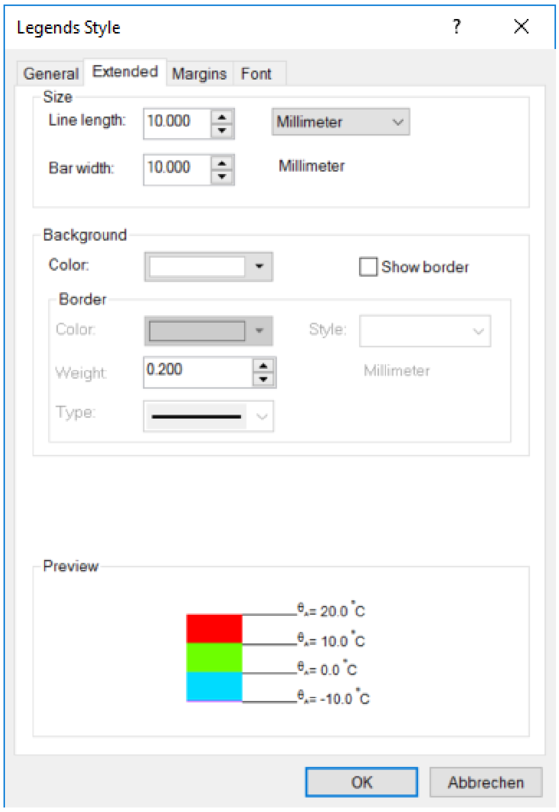
Font Tab

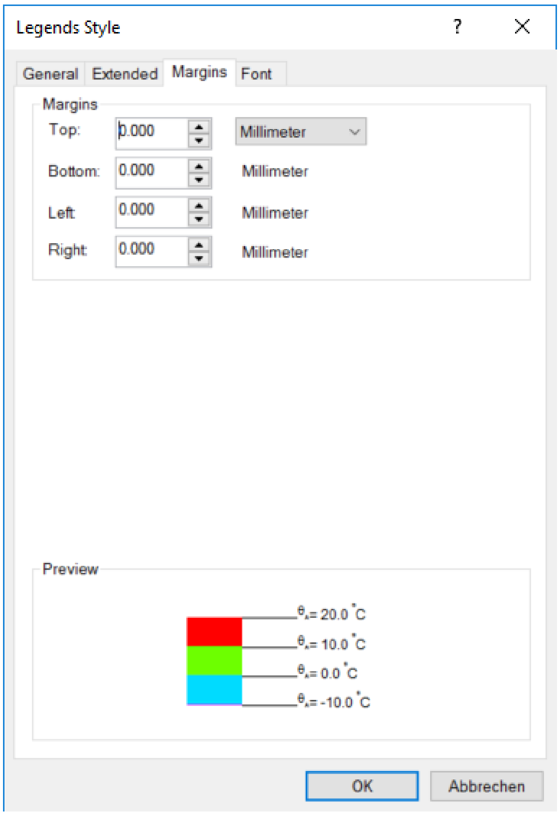
4.4.1.12 Legend Style

Description

This style is applied when you create Temperature and Heat Flux fields.







Fields

General Tab

Name

Here, you define the name of the style. You cannot leave this field empty; the chosen name must be unique, i.e. not identical to another existing name. Aside from these limitations, you are free to choose any name.

Click on the language abbreviation in this field to set the name in all supported languages.

Digits

Number of decimal places or significant digits, which are to be displayed. The kind - **Decimal Points** or **Significant** - you can select using the adjacent drop down-list.

Thousands delimiter

The digits left of the decimal point will be grouped into thousands.

Show units

The appropriate units will be shown after each numerical value. The units can be adjusted in the Units tab of the Options dialog window.

Show result symbols

The appropriate symbol (dependent on what type of value) will be displayed before each numerical value.

Show intermediate values	Is this option selected, then intermediate values of the legend are highlighted as well.
Line	This setting group determines the properties of the label lines (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.
End Point Markers	Here, you can determine how both ends of the lines should look like. There are two drop down lists that show the different possibilities.

Extended Tab

Size	Here, you can determine the line length as well as the bar width.
Background	This group contains the settings for the background color of the legends and the styles of the border lines. It also lets you hide and unhide the border lines.

Border Tab

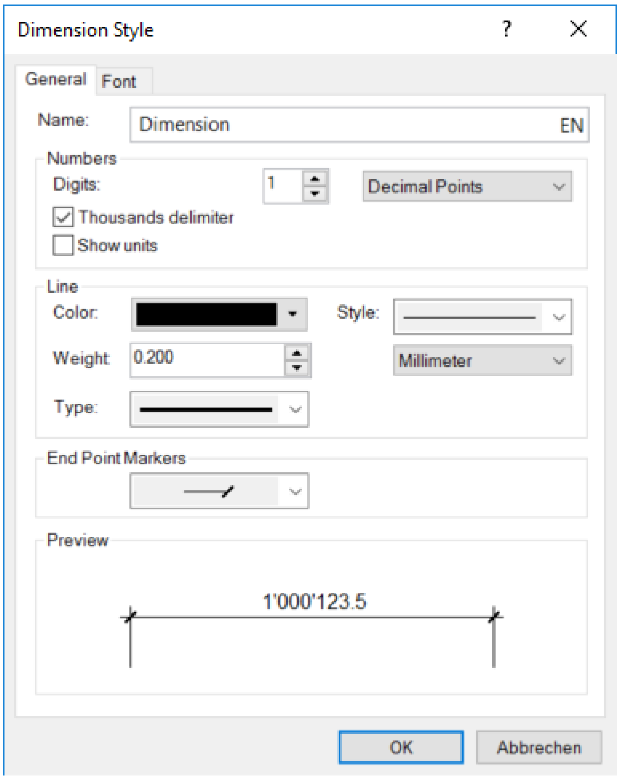
Margins	Here, you can define the size of the borders around the legend.

Font Tab

4.4.1.13 Dimension Style

Description

This style is applied when you work with the Dimension tool. The dimension style consists of an automatically generated line combination, which displays the distance between two points.



Fields

General Tab

Name

Here, you define the name of the style. You cannot leave this field empty; the chosen name must be unique, i.e. not identical to another existing name. Aside from these limitations, you are free to choose any name.

Click on the language abbreviation in this field to set the name in all supported languages.

Digits

Number of decimal places or significant digits, which are to be displayed. The kind - Decimal Points or Significant - you can select using the adjacent drop down-list.

Thousands delimiter

The digits left of the decimal point will be grouped into thousands.

Show units

The appropriate units will be shown after each numerical value. The units can be adjusted in the Units tab of the Options dialog window.

Show result symbols

The appropriate symbol (dependent on what type of value) will be displayed before each numerical value.

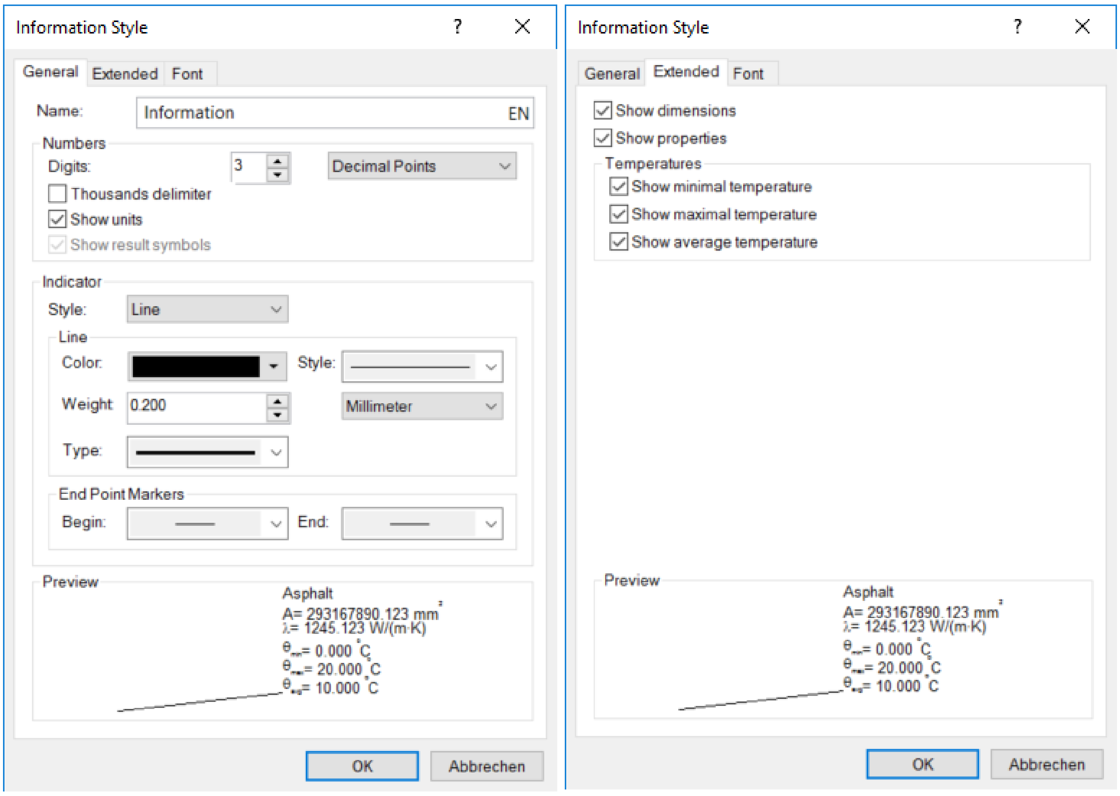
Line	This setting group determines the properties of the dimension lines (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.
End Point Markers	Here, you can determine how the ends of distance lines should look like. There is a drop down lists that shows the different possibilities. The same style is used for both ends of a distance line.

Font Tab

4.4.1.14 Information Style

Description

This style is applied when you are working with the Information tool . The Information style consists of a label line and a text box, which can contain various definable information.



Fields

General Tab

Name	Here, you define the name of the style. You cannot leave this field empty; the chosen name must be unique, i.e. not identical to another existing name. Aside from these limitations, you are free to choose any name. Click on the language abbreviation in this field to set the name in all supported languages.
Digits	Number of decimal places or significant digits, which are to be displayed. The kind - **Decimal Points** or **Significant** - you can select using the adjacent drop down-list.
Thousands delimiter	The digits left of the decimal point will be grouped into thousands.
Show units	The appropriate units will be shown after each numerical value. The units can be adjusted in the Units tab of the Options dialog window. In the Information style, the units are always shown.
Line	This setting group determines the characteristics of the label lines (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.
End Point Markers	Here, you can determine how the ends of a label line should look like. There are two drop down lists that show the different possibilities.

Extended Tab

Show dimension	If this option is activated, then the surface of the domain or the length of the edge will be displayed as well.
Show property	If this option is activated, then the properties of the domain (e.g. Lambda value) or the edges (e.g. boundary condition attributes) will be displayed as well.
Show minimum temperature	If this option is activated, then the minimum temperature of the domain or of its edges will be displayed as well.

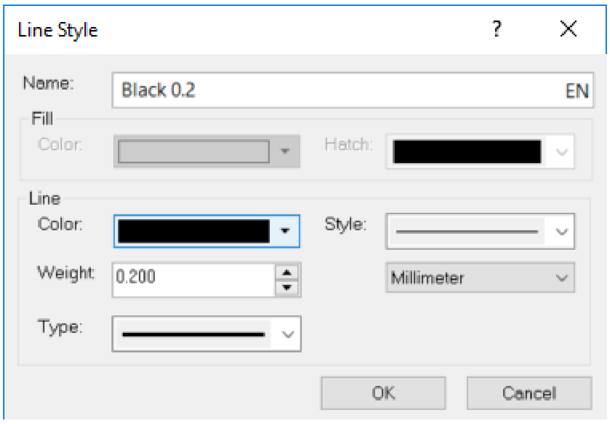
Show maximum temperature	If this option is activated, then the maximum temperature of the domain or of its edges will be displayed as well.
Show average temperature	If this option is activated, then the average temperature of the domain or of its edge will be shown as well.

Font Tab

4.4.1.15 Line Style

Description

This style is applied when you are working with the Line or Polyline tool. The style determines line properties such as color, style, thickness and type.



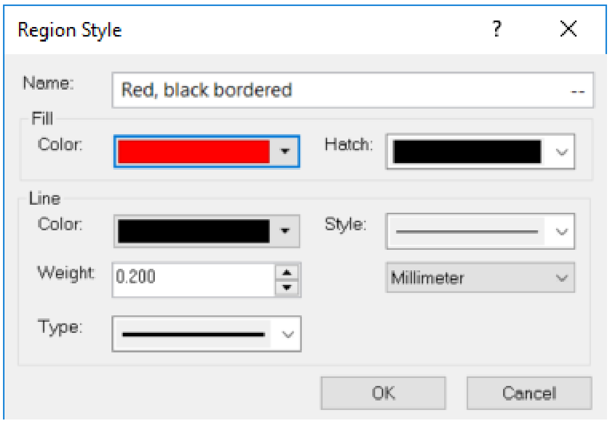
Fields

Name	Here, you define the name of the style. You cannot leave this field empty; the chosen name must be unique, i.e. not identical to another existing name. Aside from these limitations, you are free to choose any name. Click on the language abbreviation in this field to set the name in all supported languages.
Line	This setting group determines the characteristics of the line (color, style, size and type). Note that some of the lines cannot be shown when their thickness exceeds 0.394 mm.

4.4.1.16 Region Style

Description

This style is applied when you are working with the Rectangle or Ellipse tool.



Fields

Name Here, you define the name of the style. You cannot leave this field empty; the chosen name must be unique, i.e. not identical to another existing name. Aside from these limitations, you are free to choose any name.

Click on the language abbreviation in this field to set the name in all supported languages.

Fill Here, you define the filling color and the hatch for the object.

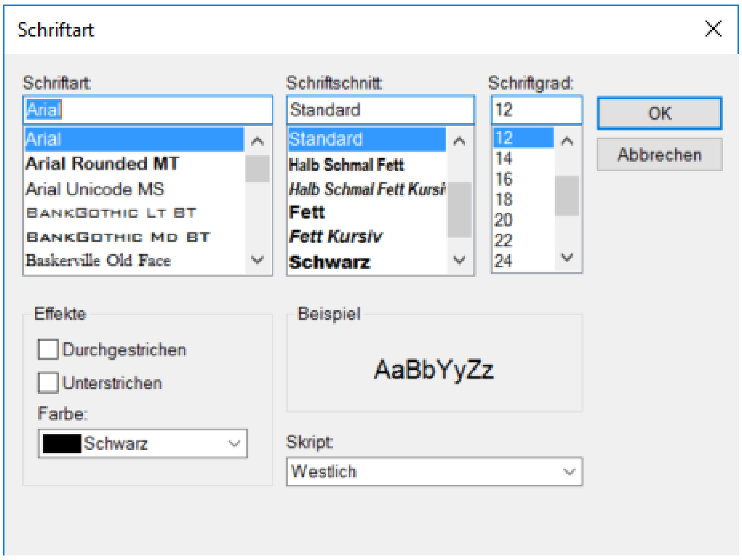
Line This setting group determines the properties of the line (color, style, size and type).

Note that some of the lines can't be shown when their thickness exceeds 0.394 mm.

4.4.1.17 Text Style

Description

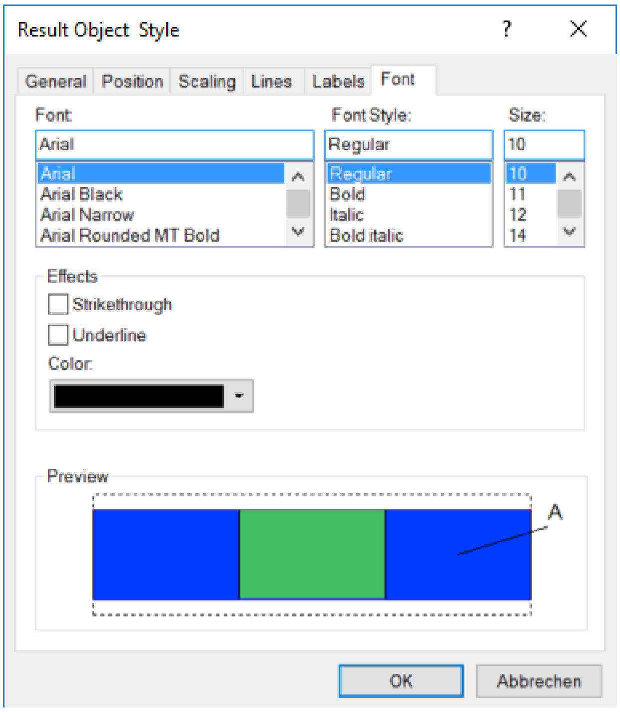
This style is used when you are working with the Text tool. In the Text style dialog window, a standard Windows "**Font**" dialog window is displayed.



4.4.1.18 Font Tab

Description

Most of the aforementioned style dialog windows have a Font tab, where you can edit the font properties.



Fields

Font On the font tab, you have a standard set of Windows font options. With these settings, you can edit the font settings. The font tab has a preview pane, where you can see how the settings will look in the document.

4.4.2 Options Dialog Window

In this section, we will examine the Options dialog windows, with which you can edit the various settings in flixo. These settings fall into three main categories:

- **Application wide settings**
- **Document settings**
- **Current Page** settings (Model, Master and Report pages; cf. the tabs on the lower edge of the work area)

The application settings are saved when you quit the program and are loaded again upon starting. The settings affecting the current document or page are saved in the current document. If you want the document specific settings to apply to all documents, then you want to create a document template with the desired settings (cf. tutorial 6).

Program or document options can be set in several dialog windows. In an Options dialog window you can adjust the parameters of inputs, views, and calculations. You can activate an Options dialog window with the menu command **Options**... in the menu **Tools** or by clicking on the ⚙ icon on the Standard toolbar. The Options dialog window has the following layout: On the left side, the individual parameter pages are presented in a tree view. On the right side, the information of the active page is displayed (figure 1). At the bottom, the standard commands: **OK** and **Cancel** are present as well.

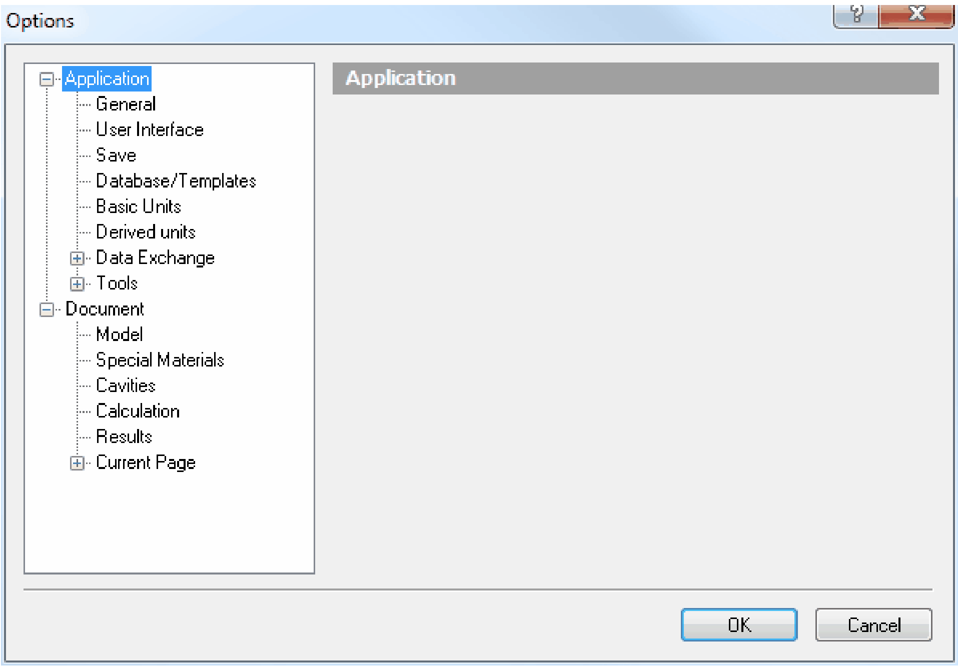


Figure 1: Options dialog window

Detailed information on the following Options dialog windows can be found in this chapter:

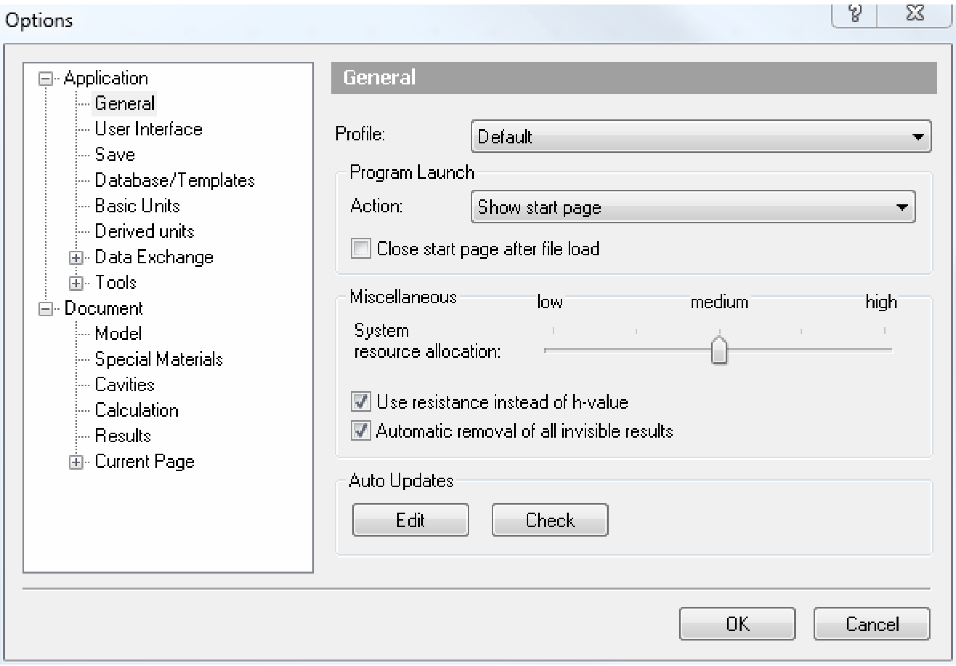
- General
- User Interface

- Save
- Database/Templates
- Basic and Derived Units
- DXF Import
- SVG Import/Export
- Tools General
- Object Capturing
- Result Object Tool
- Cavity Wizard
- Model
- Special Materials
- Cavities
- Calculation
- Results
- Grid and Guidelines
- Guidelines

4.4.2.1 General

Description

The general settings of flixo are defined in the following dialog window:



Fields

Profile Here you can select the current profile. The standard template and the tools that should be hidden can be set in a profile.

Program Launch

In this list, you can set the action that is performed when you launch flixo: **Create new default file**, **Open most recent used file** or **Show start page**. On the Start Page you can either open one of the recently used files, create a new default file or import a file.

If the option **Close start page after file load** is selected, any open Start Page closes automatically after loading a file.

System Resource Allocation

This controls the priority of the calculation process: the higher the priority, the more system resources will be allocated to the calculation process. Note that other applications may take a performance hit.

Use resistance instead of h-value

Determines whether boundary condition input and outputs should use resistance rather than h-values. The resistance is the reciprocal of the h-value.

Automatic removal of invisible results

Determines whether invisible results should automatically be deleted while loading.

Auto Updates: Edit

Adjusts the settings for the automatic control for existing updates and service releases. Here, the check frequency and downloads folder can be determined.

Auto Updates: Check

Checks if the installed version of flixo is the most recent version and if other updates and service releases exist. The frequency of the check can be adjusted in the options.

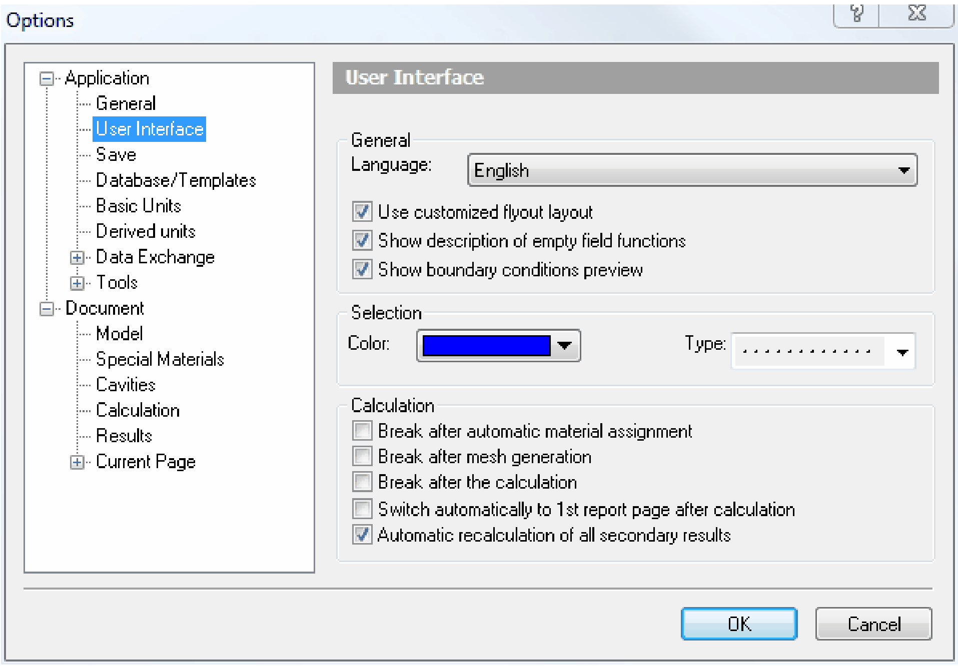
For Auto Updates you need access to the Internet.

For update checks you need **write permissions** for the downloads folder.

4.4.2.2 User Interface

Description

In the User Interface options dialog window you can determine the settings pertinent to the user interface.



Fields

Language

Here, the language of the user interface can be defined. The newly defined language will become active after flixo is restarted.

Use customized flyout layout

Determines whether the customized or the default positions of the flyouts are used.

Show description of empty field functions

Determines whether the function instead of the invisible description of field functions with empty or invisible results should be displayed.

Show boundary conditions preview

Determines whether the boundary condition lines at the surface are displayed after entering the boundary condition start points. The display takes place time-delayed, depending on the complexity of the construction and the used hardware. Nevertheless you can continue your work without any restrictions.

Selection

Here, the line properties (Color and Type) can be selected, with which the edges of the selected objects are drawn.

Break after automatic material assignment

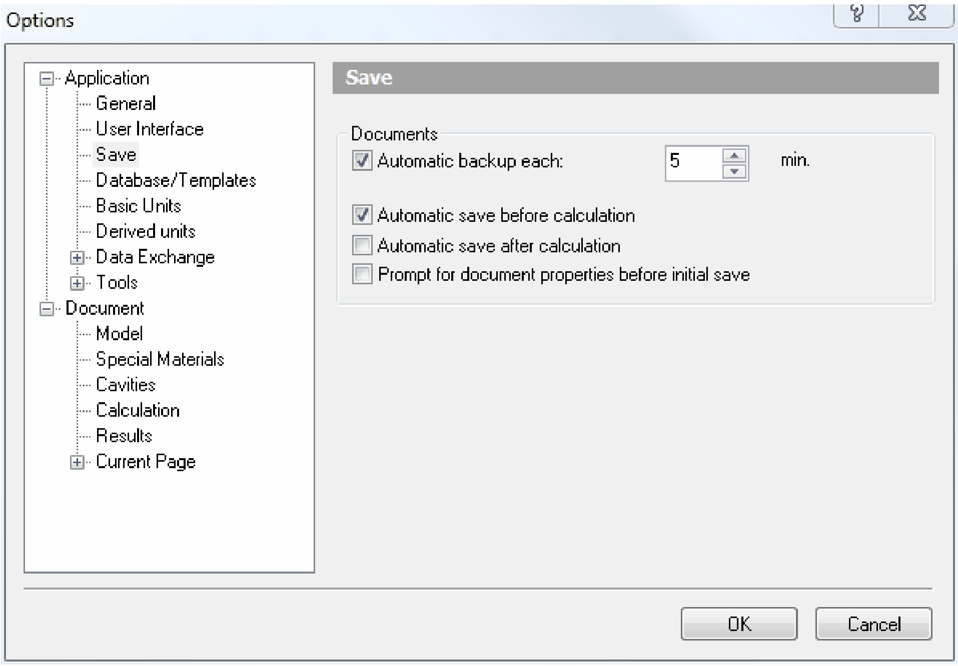
Determines whether a break after the automatic material assignment should be inserted. If this option is marked and a material was automatically assigned, then the calculation must be restarted. By activating Zoom to next Conflict the domains

	with automatic assigned materials are displayed successively.
Break after mesh generation	Determines whether a break should be inserted after the calculation of the result of the mesh generation. If this option is marked, you have the explicit option to start or not start the calculation.
Break after the calculation	Determines whether a break after the successful calculation of the result should be inserted. If this option is marked, the calculation dialog is active until you click on the **OK** button.
Switch automatically to 1st report page after calculation	Determines whether there should be an automatic switch to the first report page after a successful calculation. If this option is marked, then the switch takes place after the calculation dialog is closed. If it is not marked, then the model page remains active.
Automatic recalculation	Determines whether all secondary results after the recalculation of a construction should be refreshed. If this option is not marked, then the recalculation of all results can be started with the command Refresh from the Results menu.

4.4.2.3 Save

Description

In the Save options dialog window the save settings are determined.



Fields

Automatic backup

This option allows to make automatic backup of all open files. The files will be restored in case of a program crash.

Automatic save before calculation

Determines whether the current file will be saved before the model is calculated. If the file has not yet been saved, then a file dialog will appear, where you can enter the file name and path.

Automatic save after calculation

Determines whether the current file will be saved after the model is calculated. If the file has not yet been saved, then a file dialog will appear, where you can enter the file name and path.

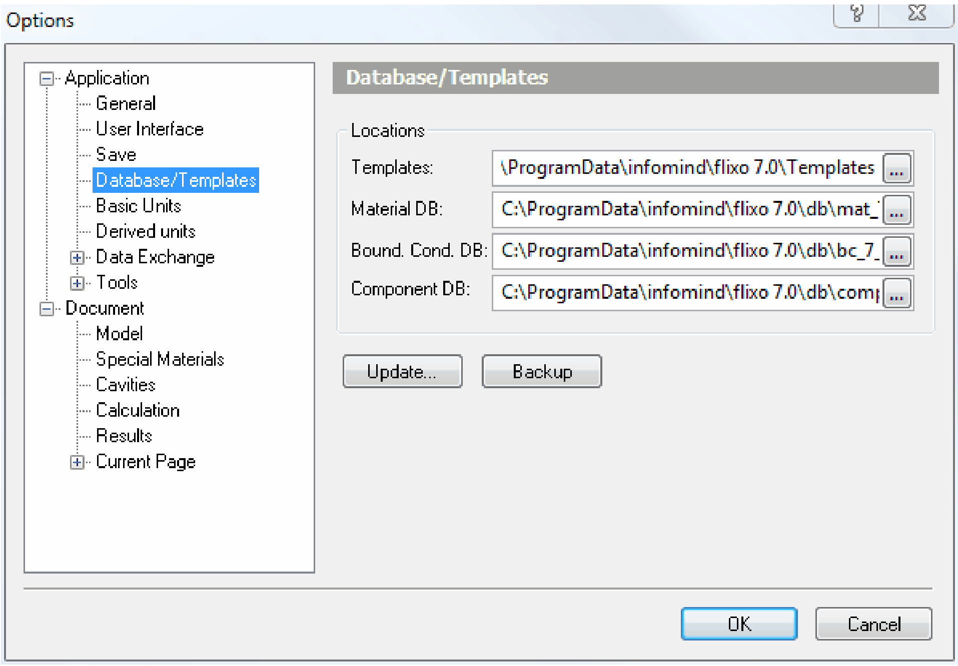
Prompt for document properties before initial save

Determines whether the Document Properties dialog window should automatically be activated before a document is saved for the first time.

4.4.2.4 Database/Templates

Description

In the Database/Templates options dialog window you can determine the settings pertinent to database and templates.



Fields

Templates

Path to the directory, where ***Document Templates*** are saved by default. More detailed information can be found in tutorial 6. By clicking on the "**...**" button a dialog window will appear where you can define the new template directory path in the text field. If the directory does not yet exist, it will be created after a respective warning.

Material DB

Name of the current ***Material Database***. More detailed information on the material database can be found in the Materials flyout chapter. After changing the name, the corresponding material database will be loaded, if the file exists. If a database does not exist, then a new database file will be generated. Upon creation of a new database file, you can choose whether to import the previous values into the new material database.

By clicking on the "**...**" button, a dialog window is opened, where a new database can be defined.

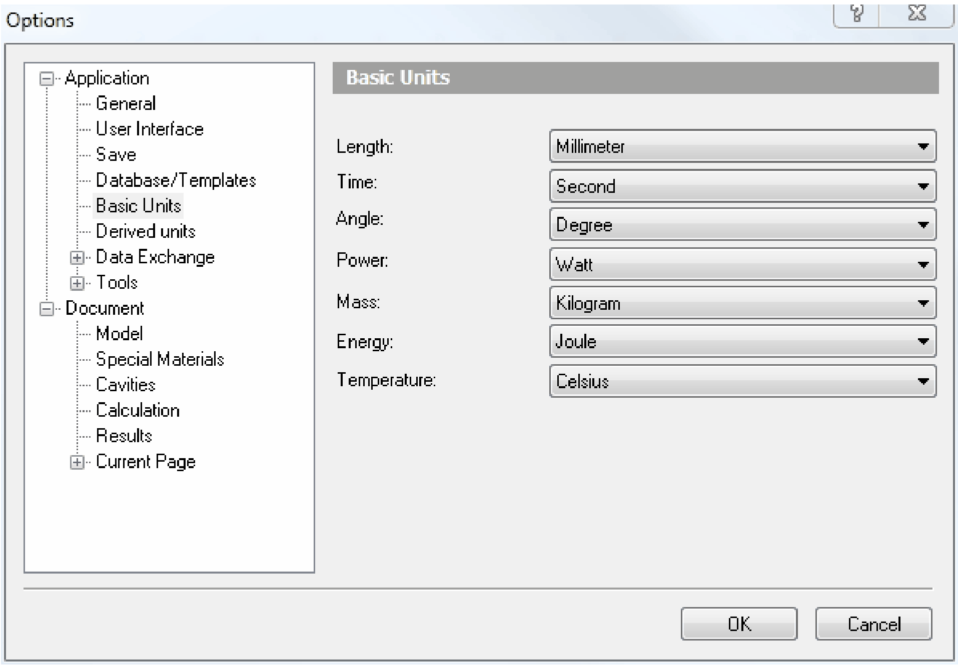
This setting is dependent on the User Interface language.

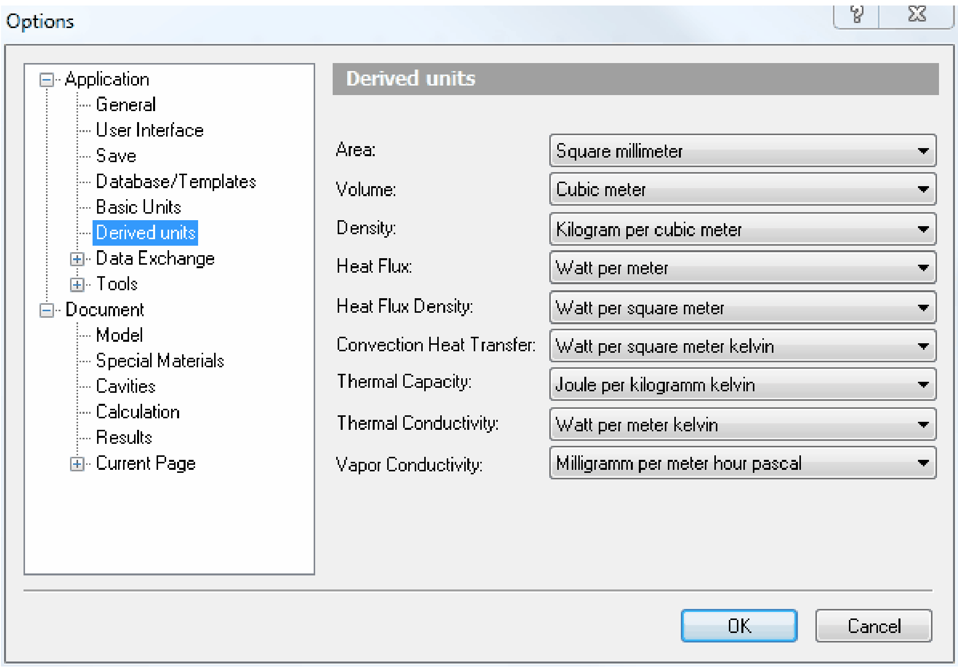
Bound. Cond. DB	Name of the current ***Boundary Condition Database***. More detailed information on the boundary conditions database can be found in the Boundary Conditions flyout chapter. After changing the name, the corresponding boundary conditions database will be loaded, if the file exists. If a database does not exist, then a new database file will be generated. Upon creation of a new database file, you can choose whether to import the previous values into the new materials database. By clicking on the "**...**" button, a dialog window is opened, where a new database can be defined. This setting is dependent on the User Interface language.
Component DB	Name of the current ***Component Database***. More detailed information on the component database can be found in the Components flyout chapter. After changing the name, the corresponding components database will be loaded, if the file exists. If a database does not exist, then a new database file will be generated. Upon creation of a new database file, you can choose whether to import the previous values into the new materials database. By clicking on the "**...**" button, a dialog window is opened, where a new database can be defined. This setting is dependent on the User Interface language.
Update...	Opens a dialog window, to check if more recent database from infomind or third-party exist.
Backup	Backs up all database and templates in a selectable folder.

4.4.2.5 Basic and Derived Units

Description

The unit settings can be found in two different options dialog windows: Basic Units and Derived Units. For each unit system, the units for inputs and outputs can be defined. If a unit is changed, this does not influence the calculation or presentation of the results.





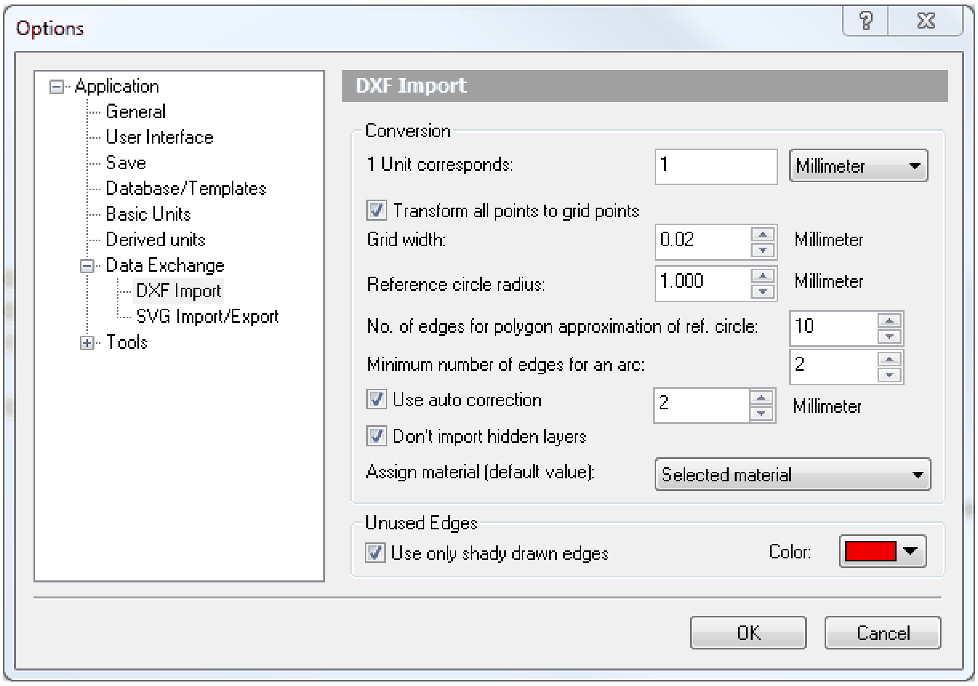
4.4.2.6 DXF Import

energy plus, frame, pro

Description

These setting are used for importing CAD files with DXF files. The importing of DXF files is described in detail in tutorial 3.

These settings must be determined before the DXF file is **opened**.



Fields

1 Unit corresponds

In DXF files geometric information is saved without units. For drawings, which are not scaled, 1 unit represents 1 millimeter in Europe. In scaled drawings, the units can be adjusted respectively (i.e. in the scale 1:1000 1 unit represents 1 meter).

Transform all points to grid points

Determines whether all DXF object points (end and midpoints) should automatically be mapped on a grid.

Grid width

Sets the distance of the squares within the grid, on which all end and midpoints of the DXF object should be mapped.

Reference circle radius

No. of points for the polygon approximation of ref. circle

Upon importing a DXF file, arcs and circles are divided into individual segments. You can determine the accuracy of the approximation by adjusting the radius and the number of points for the polygon approximation of a circle:

- All circles with radii smaller than the reference circle's radius will be subdivided, proportionally to radius, less than the reference circle.

	- All circles with radii bigger than the reference circle's radius will be subdivided, proportionally to radius, more than the reference circle;
Minimum number of edges for an arc	This setting determines the minimum number of edges for arc approximation. Determines an internal calculation for the number of edges of a specific arc (taking into account the "Reference circle radius"), which according to the polygon circular approximation, would have fewer edges than this parameter value. If this is the case, this parameter is used instead.
Use auto correction	Is this option is selected, all open polylines or not connected lines will be automatically closed, if the gap is smaller than the defined value.
Assign material	In this list you can define the standard characteristics when converting imported DXF data. **Level:** The material which is defined in the layer material mapping table is assigned automatically to all objects lying on the DXF-layer. **Layer - only closed objects:** The material which is defined in the layer material mapping table is assigned automatically to all closed objects (e.g. polylines) lying on the DXF layer. **Selected material:** The selected material from the Material list is assigned to all objects. You can also change the kind of material assignment by clicking on the "**Options**" button in the message line after importing the DXF file. The automatic material assignment is explained in tutorial 3.
Don't import turned off layers	If this option is selected, the hidden layers in the DXF file are not imported. Otherwise they are imported but not displayed. You can change the visibility of the layers in the Layers flyout

Use only shady drawn edges — This option determines, whether the entire line segment or just the end points of unclosed lines should be highlighted or not.

Special Color — Determines if and which color should be used when there is a problem spot.

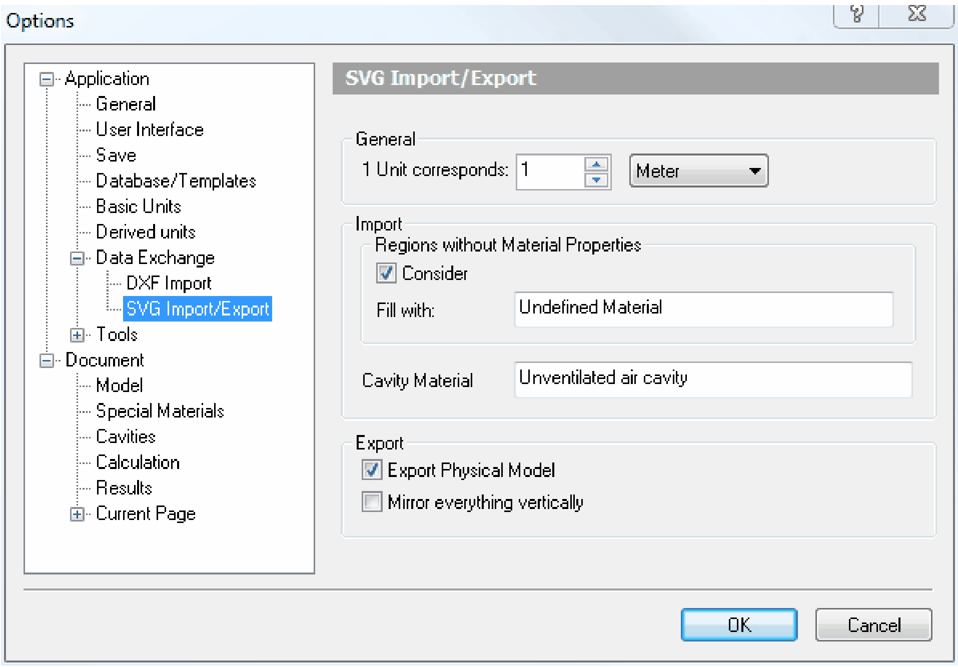
4.4.2.7 SVG Import/Export

pro

Description

Settings that are used when importing and exporting SVG files.

Note that these settings must be set before you open or save a SVG file.



Fields

1 Unit corresponds — If no other respective value has been defined in the imported file, this default value is taken into account for non-dimensional coordinate entries in Building SVG files.

Consider — If this option is marked, all domain definitions without any material specifications will also be imported and assigned with the corresponding material. If not, they will be ignored.

Fill with — Material name used for filling domains without any specific material properties. If the material is not present in the document, a new one will be created.

Cavity Material — Cavity material name used for filling cavities in a domain (like a tube). If the material is not present in the document, a new cavity material will be created according to EN ISO 10077-2.

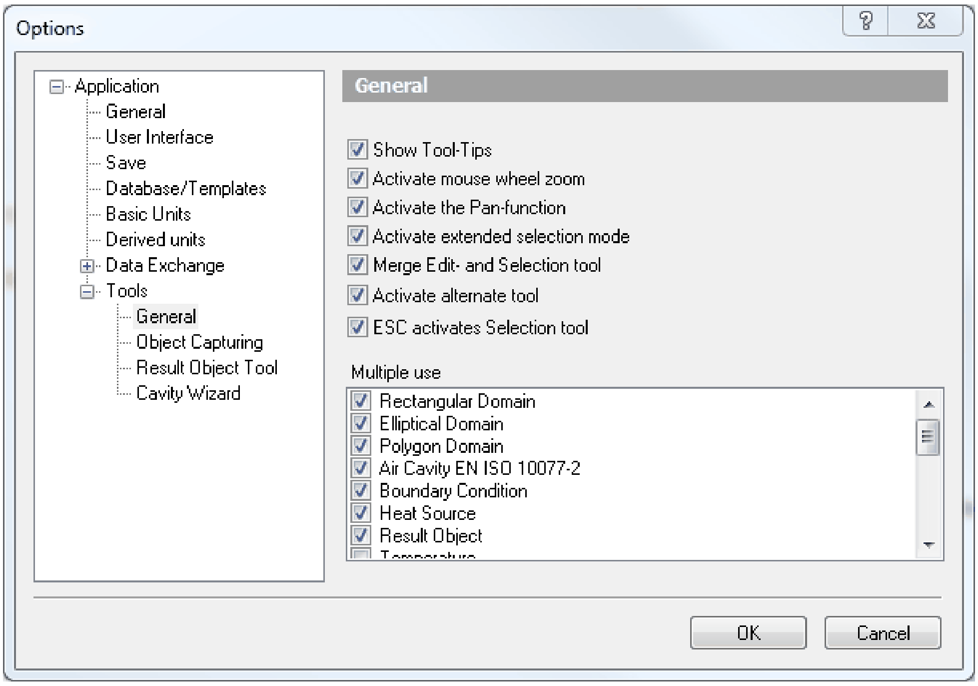
Export Physical Model — If this option is selected, the physical model will also be exported.

Mirror everything vertically — If this option is selected, all data will be mirrored vertically. This option should be set for applications - e.g. browsers - where the positive y-axis points to the bottom.

4.4.2.8 Tools General

Description

The following settings are generally used by all tools and in part (Zoom, Pan) these settings overlap with the functioning of the tools.



Fields

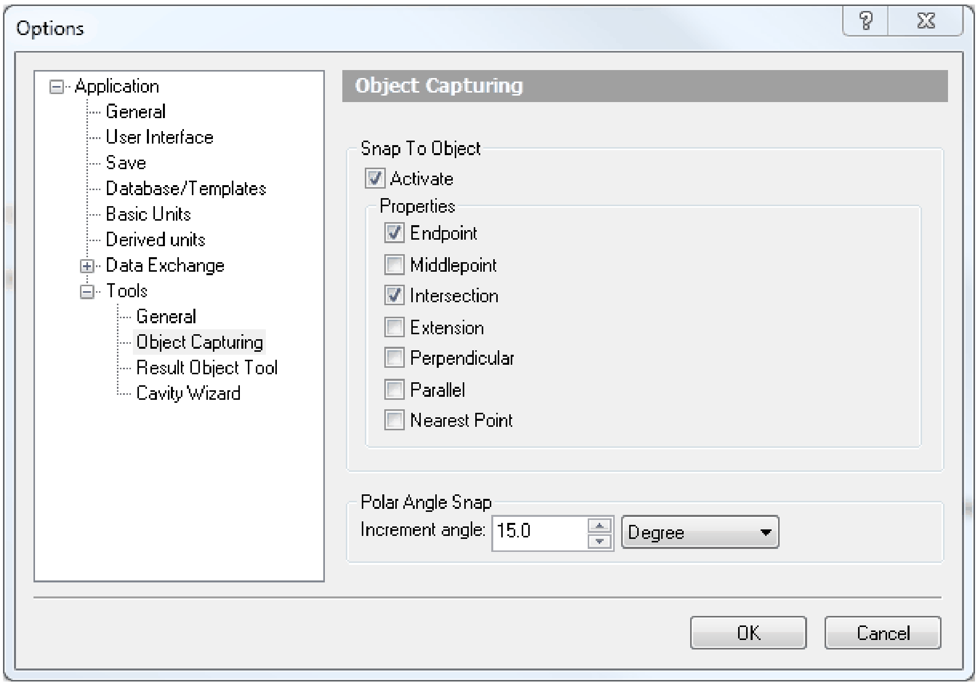
Show Tool-Tips — Determines whether Tool-Tips (short help texts next to the mouse cursor) should appear.

Activate mouse wheel zoom	Determines whether the mouse wheel should directly zoom the illustration or whether it should take on the normal scroll function. If this option is not marked, the illustration is enlarged or reduced by turning the mouse wheel while the **CTRL key** is held down. The activated mouse wheel can be used in an active window only. To activate a window, simply chick somewhere within the window.
Activate the Pan-function	Determines whether the Pan-function is activated when clicking on the middle mouse button. "Pan" is a function that moves the visible section of the application window without changing the illustration size.
Extended selection mode	The extended selection mode is dependent on the position of the start and end point. If the start point lies left of end point of the selection rectangle, then not only all objects located within this rectangle will be selected, but also objects that are crossed by the selection rectangle will be marked. In the normal selection mode, only objects lying completely within the selection rectangle are highlighted.
Merge Edit- and Selection tool	If this option is selected, then the Edit tool and the Select, Move, Scale tool is merged on the report pages in a single tool.
Activate alternate tool	If this option is selected, you can temporarily activate the Select, Move, Scale tool by pressing the **ALT key**. After releasing the **ALT key** the previous tool is active again.
ESC activates Selection tool	If this option is selected, you can activate the Select, Move, Scale tool by pressing the **ESC key**. After releasing the **ESC key** the current selection is canceled.
Multiple use	Using this tool list you can define, which tools should have a multiple use mode by default and which tools automatically activate the Select, Move, Scale tool after finishing an action. Tools with a single use mode can temporarily be set in the multiple use mode by double clicking on the symbol in the Tools flyout.

4.4.2.9 Object Capturing

Description

The following settings are generally used by all tools to capture on other objects during the input. These settings can be adjusted via the corresponding tool context menu or by clicking on the icon on the standard toolbar.



Fields

Activate Snap To Object
If this option is selected, then the mouse position will snap to selected characteristic points of a nearby object when you enter data.

Endpoint
Snaps to vertices and end points of edges of neighboring objects.

Middlepoint
Snaps to middle points of edges of neighboring objects.

Intersection
Snaps to intersections of edges of neighboring objects.

Extension
Snaps to extensions of edges of neighboring objects.

Perpendicular
Snaps to perpendiculars to edges of neighboring objects.

Parallel
Snaps to parallels of edges of neighboring objects.

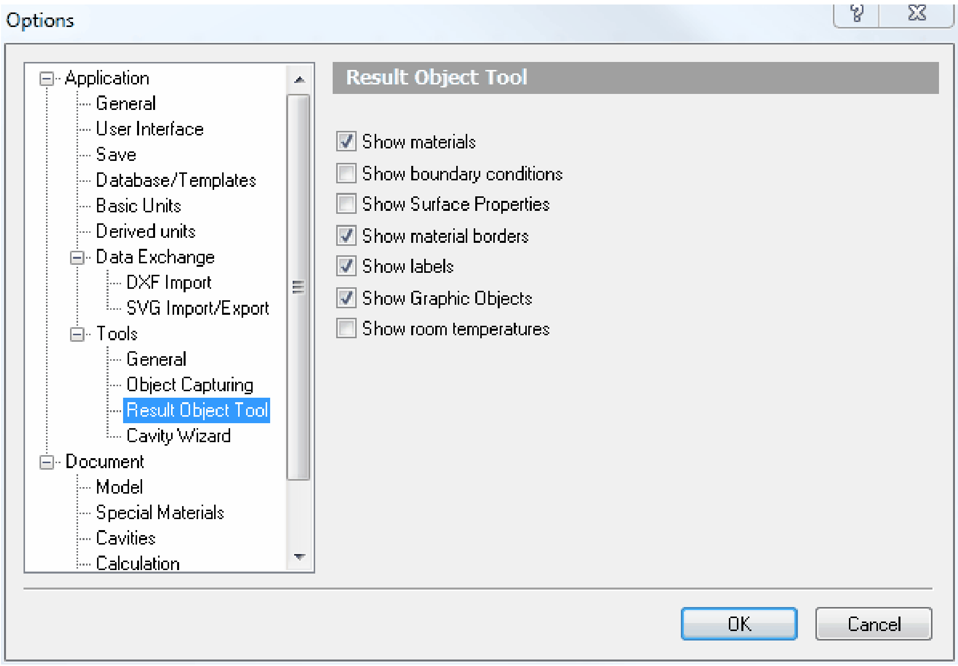
Nearest Point
Snaps to the nearest point of the edge of a neighboring object.

Increment angle
The mouse tracking is limited to whole-numbered multipliers of the selected ***increment angle***, if the **Shift** key is pressed during the definition.

4.4.2.10 Result Object Tool

Description

These settings determine the standard properties of the Result Object tool.



Fields

Show materials	Determines whether the materials of a result object should be displayed by default.
Show boundary conditions	Determines whether the boundary conditions of a result object should be displayed by default.
Show Surface Properties	Determines whether the radiation conditions of a cavity surface should be displayed by default. Radiation conditions are only displayed if they are calculated automatically by flixo (cf. options Cavities).
Show material borders	Determines whether the material boundaries of a result object should be displayed by default.
Show labels	Determines whether the point labels of individual result objects should be displayed by default.
Show Graphic Objects	Determines whether the graphic objects of a result object should be displayed by default. Graphic objects facilitate the legibility of the results, but are not considered in the calculation.

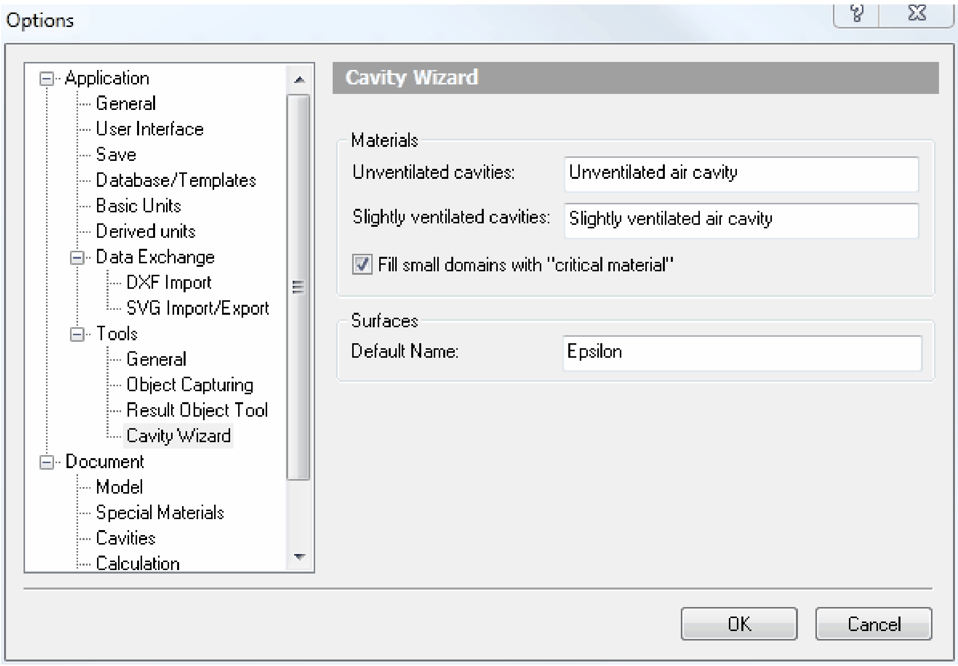
Show room temperatures Determines whether the room temperatures of a result object should be displayed by default. The room temperatures are indicated in the colors of the temperature field in the result domain.

4.4.2.11 Cavity Wizard

frame , pro

Description

These settings determine the standard properties of the Cavity Wizard.



Fields

Unventilated cavities Name of the unventilated cavity material according to **EN ISO 10077-2**, which is used by the cavity wizard to create unventilated cavities. If the material is not present in the document, you will be prompted to choose or newly define a material in a corresponding dialog window.

This setting is dependent on the User Interface language.

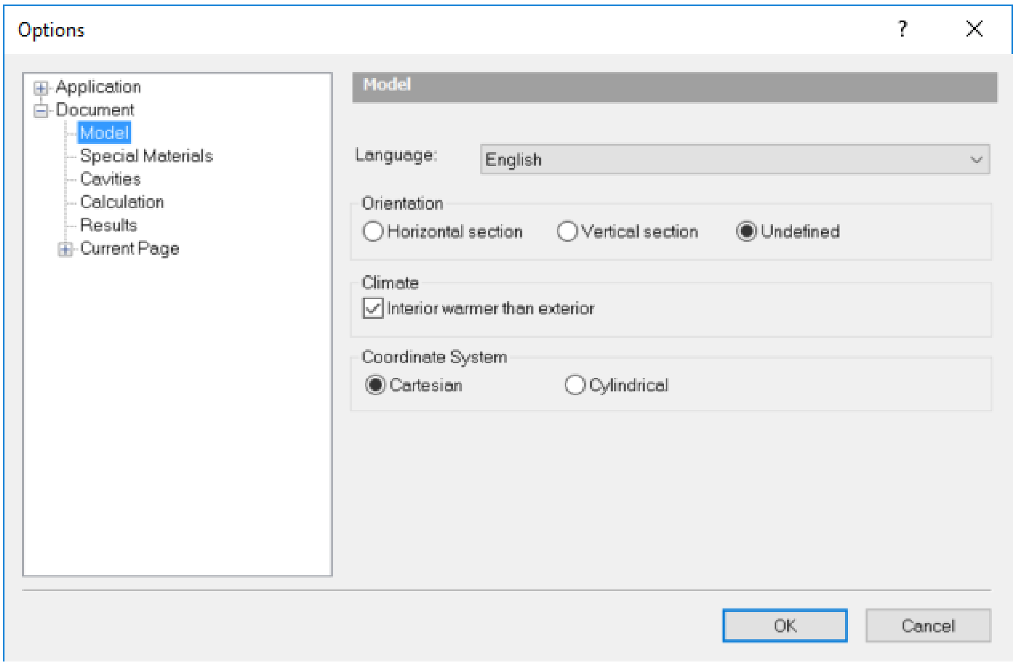
Slightly ventilated cavities Name of the slightly ventilated cavity material according to **EN ISO 10077-2**, which is used by the cavity wizard to create slightly ventilated cavities. If the material is not present in the document, you will

	be prompted to choose or newly define a material in a corresponding dialog window. This setting is dependent on the User Interface language.
Fill small domains with "critical material"	If the option is marked, small cavities (see also Special Materials) will be assigned this "critical material", rather than the unventilated cavity material. In a later step, these cavities will automatically be assigned the neighboring material.
Default Name	Prefix for the names of the automatically created radiation surface properties. This setting depends on the User Interface language.

4.4.2.12 Model

Description

In the **Model** options dialog window, the language, orientation and scale settings are determined.



Fields

Language	Choose one of the available languages from the list.

Orientation

Here, you can determine whether the construction input is a horizontal or a vertical cross section. This option is only necessary if you are using special glass unit materials .

If the orientation is defined, you can adjust it by either activating the context menu (right click) on the  icon or by double clicking.

Interior warmer that exterior

Select this option if the exterior temperature is warmer than the interior temperature in the model.

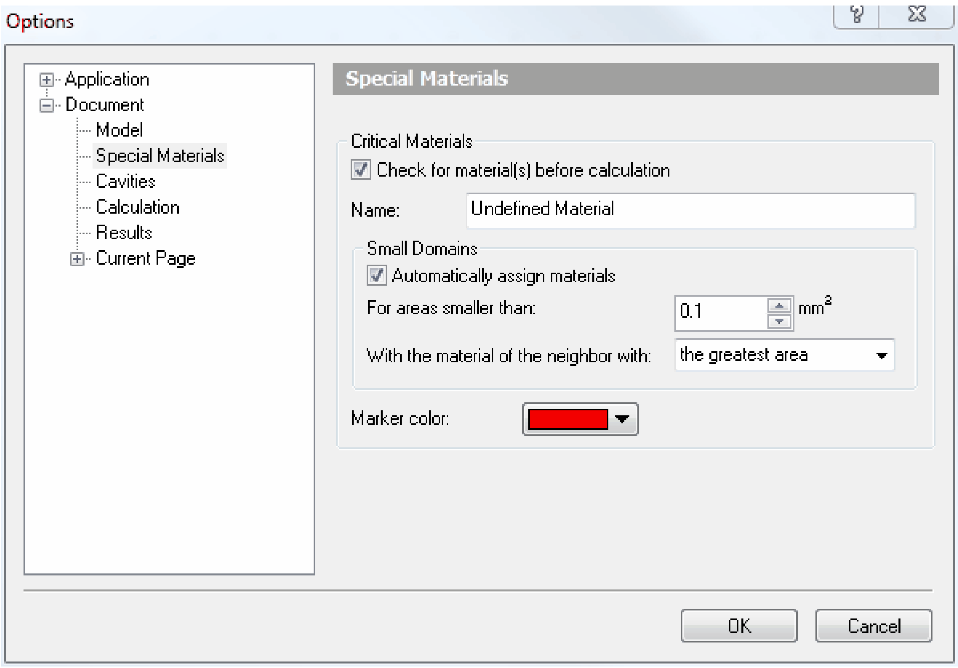
This option only has an influence on the surface temperature labeling.

4.4.2.13 Special Materials

Description

In this dialog window, you can set the necessary parameters for the material wizard. In tutorial 4, the use of this wizard is explained in further detail.

Before calculating, domains with specific materials can automatically be edited.



Fields

Check for material(s) before calculation

Checks all visible domains to see if a certain material, which should not be in use is being used. If this option is checked, then the model will not only be searched for critical material, but small domains with critical material will be assigned material according to determined standards, and larger domains will be highlighted.

Name
The name of the critical material(s), which should not be present in the model. If more than one type of material is critical, then each material name should be surrounded by quotation marks, and separated by a comma. E.g.:

```
"Critical Material 1", "Critical Material 2"
```

Automatically assign materials
Determines whether domains that do not exceed a certain size and have a critical material should automatically be assigned a material according to certain standards. The critical size and the standards can be determined separately.

For areas smaller than
Critical size for a domain assigned a critical material, which should be assigned another material. All domains with an area less than this value are automatically assigned a material according to the current assignment standards.

With the material of the neighbor with
Here, you can choose the standards by which smaller domains with critical materials should be assigned a material. The following criteria can be chosen:

- greatest density (the domain with the critical material takes on the material of the neighbor with the greatest density)

- greatest area (the domain with the critical material takes on the material of the neighbor with the greatest area)

- greatest lambda (the domain with the critical material takes on the material of the neighbor with the greatest thermal conductivity)

- smallest lambda (the domain with the critical material takes on the material of the neighbor with the smallest thermal conductivity)

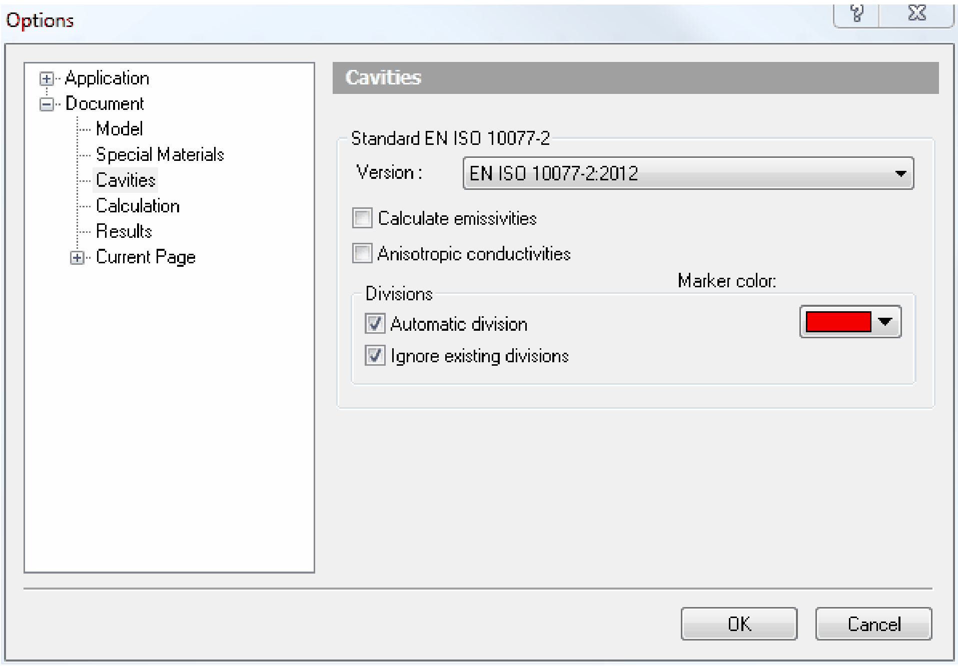
Critical material color
Color that domains with critical materials should be highlighted with.

4.4.2.14 Cavities

frame , pro

Description

In this dialog window you can define the properties of air cavities according to EN ISO 10077-2. The use of the wizard is explained in detail in tutorial 4.



Felder

Version

Selection of the standard version for the computational algorithms used for the calculation of the equivalent conductivities resp. the energy transport for cavities.

Calculate emissivities

Emissivities (long-wave radiation properties) in the main heat flux direction according to EN ISO 10077-2 should be calculated as average based on the set radiation surface properties by flixo.

If this option is not activated, the average emissivities have to be defined in the cavity materials.

Anisotropic conductivities

If this option is selekted, then per cavity in both x and y direction an equivalent conductivity according to EN ISO 10077-2 will be calculated and applied.

Automatic division

Determines whether air cavities should be divided according to 10077-2 into smaller air cavities:

Division of air cavities can have a significant influence on the results of the calculation. According to the standard, only air cavities with outlets smaller than 2mm are allowed to be subdivided into smaller air cavities.

Ignore existing colors	Determines whether manually defined, visible boundaries of air cavities to other air cavities with the same properties should be ignored or taken into account in the division calculation. Division of air cavities can have a significant influence on the results of the calculation. According to the standard, only air cavities with outlets smaller than 2mm are allowed to be subdivided into smaller air cavities.
Marker color	Color with which additional division points should be highlighted.

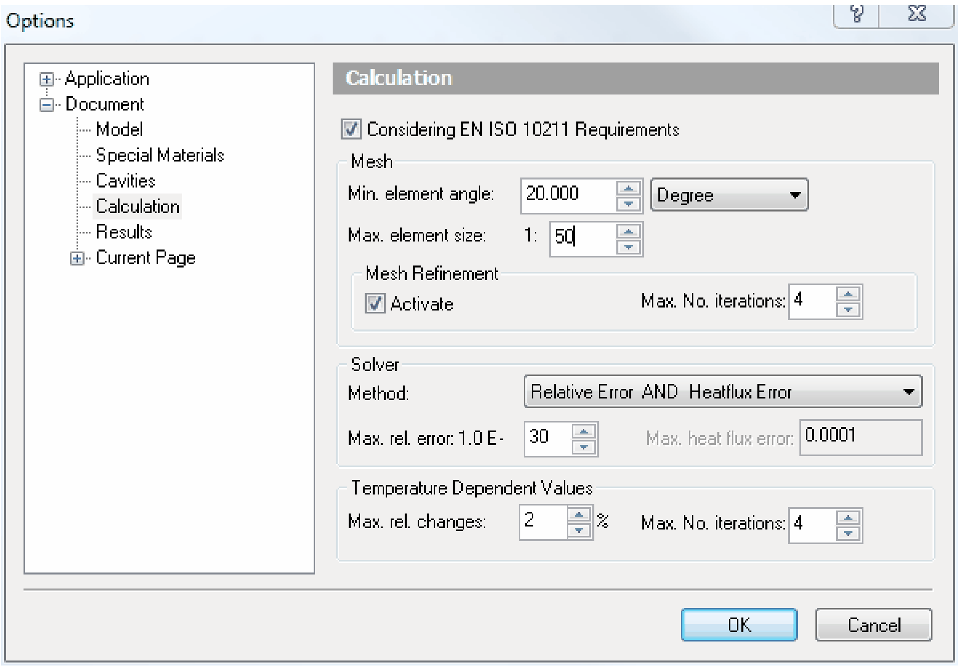
4.4.2.15 Calculation

Description

In the calculation dialog window, the parameters for the calculation process can be determined. The correct parameter settings are important to obtain accurate (and sensible) results.

The calculation process is comprised of four parts:

- Generation of the element grid
- Calculation of temperature dependent conductivity values
- System equation solutions
- Automatic Mesh Refinement



Fields

Considering EN ISO 10211 Requirements

If this option is selected, the conditions of calculation will automatically meet the standard EN ISO 10211 :

- The maximum heat flow error is less than 0.0001. The heat flow errors is the ratio of the sum of all signed heat flows and half of the sum of absolute values of the heat flows.

- The model is calculated twice, once with the number of elements characterized in **Max. element size**, once with the double number of elements. The sum of the absolute values of all heat flows will be determined for both meshes. If the relative difference of the two sums is greater than 1%, the grid will automatically refined.

Min. element angle

Minimum angle, which the individual mesh elements (finite elements) area is allowed to have. This value should not be too small (physical reasons) and should not be too big (because otherwise mesh generation is not possible).

Max. element size

Maximum permitted size for a mesh element (finite element). This number is a ratio that is relevant to the size (maximum measurement of the circumscribing rectangle).

Mesh Refinement

If this option is selected, the element mesh is refined automatically in those areas, where the relative error of the current solution is greater than a critical limit. The mesh refinement will continue as long as the relative error at any point is larger than this critical value or the maximum number of runs is reached.

Method

The termination criteria for the solver. The **Relative Error** refers to the calculated temperatures and the **Heat Flow Error** refers to the heat flow. Either, only a single criterion or a combination of both criteria can be used.

Max. rel. error:

Determines the maximum relative error of the solution vector. If this method is selected, the resulting system equation will be solved iteratively until the relative error is less than this value, or until the maximum number of iterations has been reached.

Max. heat flow error

Determines the maximum heat flow error according to standard EN ISO 10077-2 (see above). If this method is selected, the resulting system equation will be solved iteratively until the relative heat flow error is less than this value, or until the maximum number of iterations has been reached.

Max. rel. changes

The calculation of temperature dependent conductivity values will continue until either the relative change of the conductivity value of all domains with temperature dependent conductivities are smaller than the maximal tolerated relative change, or the maximum number of iterations has been reached.

Max. No. iterations

Determines the maximum number of iterations (for the calculation of temperature dependent conductivities as well as the automatic mesh refinement).

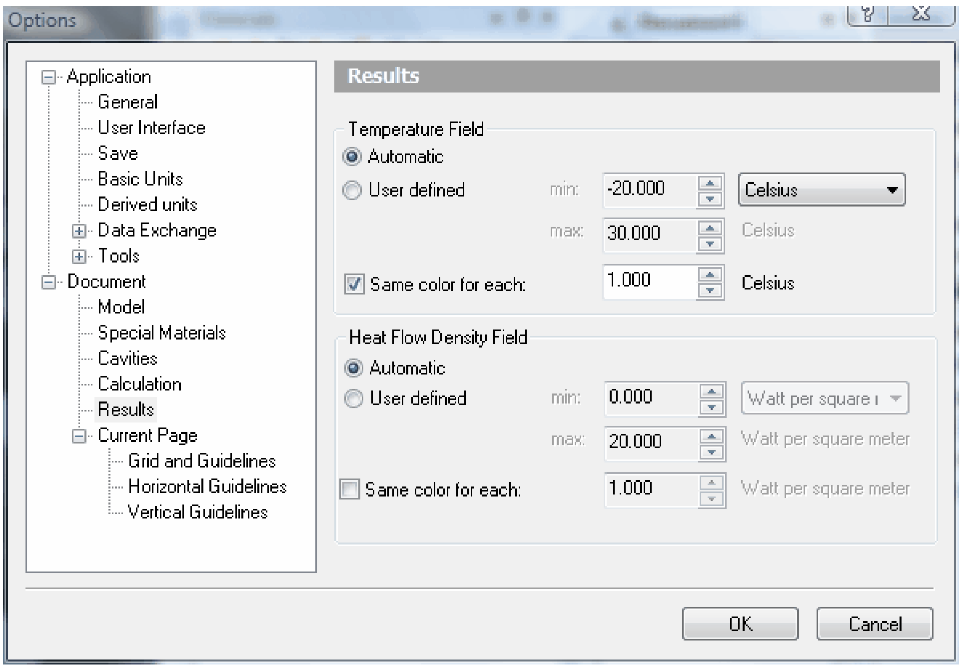
4.4.2.16 Results

Description

In the results dialog window, the coloring of the temperature and heat flux fields can be determined.

For both fields, two settings are possible:

- Automatic
- User defined



Fields

Automatic

The colors will automatically be distributed according to the minimum and maximum values which appear in the construction.

User defined — This option allows you to define the minimum and maximum values yourself (i.e. to be able to compare temperature (heat flux) field results across different variations). Values greater than the maximum defined value will take on the color of the maximum value. Values smaller than the minimum will take on the color of the minimum value.

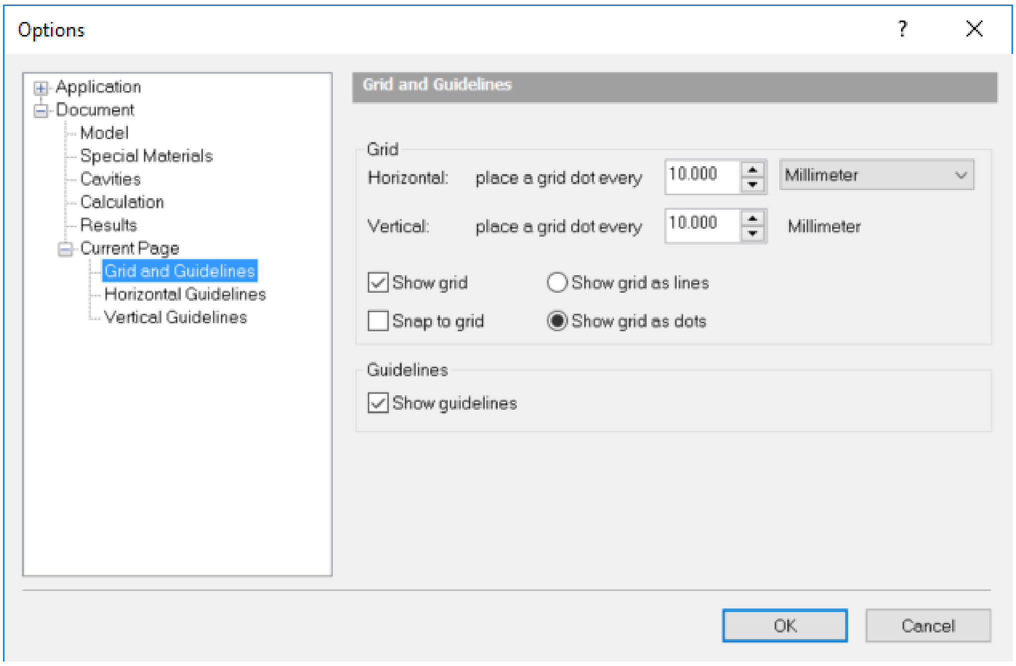
Same color for each — Determines whether values (temperature or heat flux) within a certain area should have the same color, or not, in which case a color gradient of the values is generated. If this option is selected, you can also define the size of the domain area.

4.4.2.17 Grid and Guidelines

Description

In the grid and guidelines dialog window, the parameters of the grid, guidelines, and snap functions are determined. Most of these settings can also be adjusted on the **Snap toolbar**.

These settings always affect the current page (cf. tabs on the lower edge of the application window).



Fields

Horizontal/Vertical place grid dot every — Defines the horizontal and vertical distance between grid points.

Show grid/guidelines	Hides or unhides the grid/guidelines. On the Snap toolbar these options are represented by the icons and . The same settings are also located in the Layers flyout. You can also hide guidelines by making the respective layer invisible.
Snap to grid/guidelines	Toggles the "Snap to grid"/"Snap to guidelines" functions. On the Snap toolbar these options are represented by the icons and .
Grid as Lines/Dots	Determines whether the grid is displayed as lines or dots. Note that not all snap points are shown.
Snap to object	Toggles the "Snap to Object" function. On the Snap toolbar function is represented by the icon.

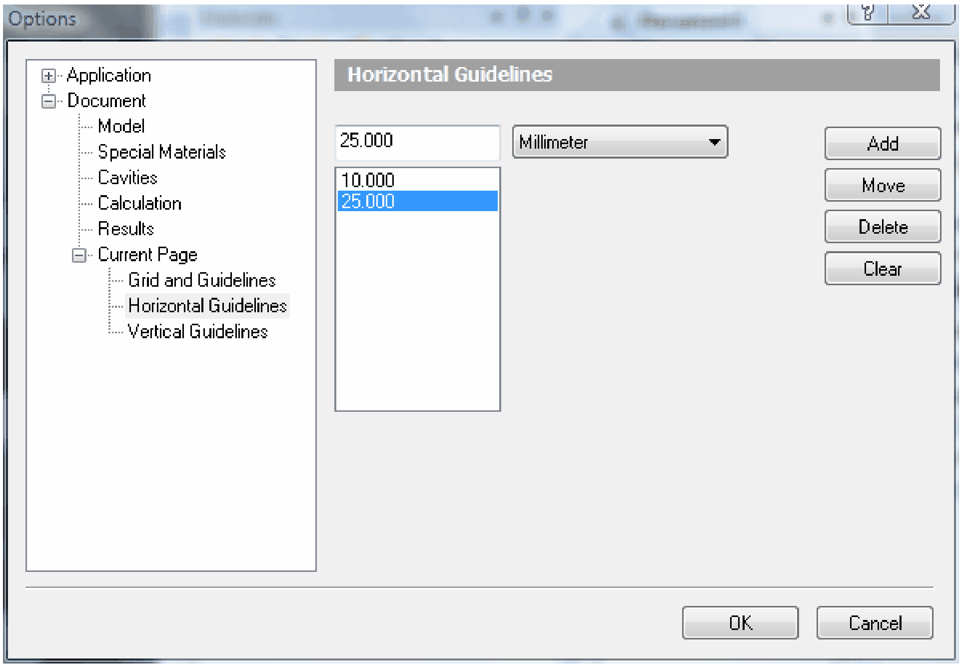
4.4.2.18 Guidelines

Description

The two dialog windows: **Horizontal Guidelines** and **Vertical Guidelines** are used to set, modify, and delete guidelines. The pages are identical, except for that they apply to two different sets of guidelines: one to the horizontal guidelines and the other to the vertical guidelines. Use of guidelines is described in more detail in tutorial 6.

The settings always correspond to the active page (see tabs on the lower edge of the application window).

A detailed description:
Enter the exact coordinate of the guideline into the text field and then click **Add**, or adjust the coordinate of an existing guideline by selecting the guideline from the list, entering the new coordinate for the guideline and then clicking on **Move**. The unit can be chosen from the drop down list. The default unit is set in the Unit Options dialog window.



Fields

List	Shows all defined guidelines.
Add	Adds a new guideline with specific coordinates.
Move	Adjusts the coordinate of the selected guideline.
Delete	Deletes the selected guidelines.
Clear	Deletes all the guidelines in the list.

4.4.3 Object Properties Dialog Window

In the following paragraphs, the various dialog windows, which you can use to show and modify the properties of selected construction elements, are described. Every object on any given page of a flixo document is code bound to a dialog window. The dialog window can be accessed with the context menu command (right click) **Properties...** of the object or with menu command Edit.Properties.... In the latter case, the dialog window will be shown for the object which is currently active object.

Alternatively most of the properties can be adjusted in the Properties Flyout.

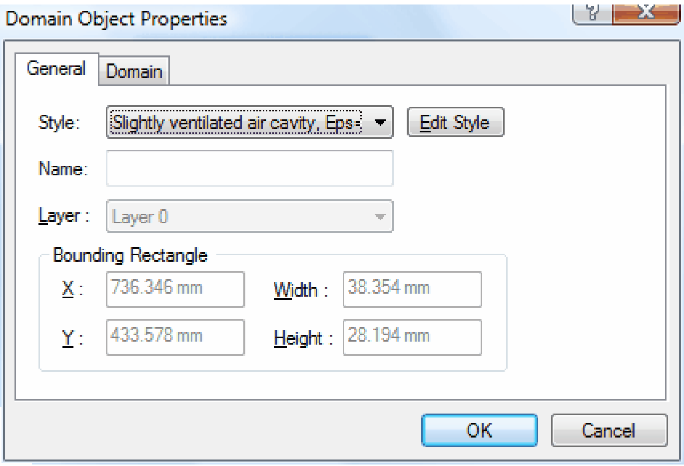
Here is a list of all the object properties windows, which are described in this section:

- General Properties dialog window
- Line Object Properties dialog window
- Domain Object Properties dialog window
- Result Object Properties dialog window
- Frame or Joint U-Value Properties dialog window
- Multiple Selection Properties dialog window

4.4.3.1 General Properties

Description

The ***General*** Properties dialog window is displayed in all object properties dialog windows. All objects in a flixo document have a **General** Properties dialog window. It contains information about the object's general properties.



Fields

Style In this drop down list, all the styles of a construction are listed. The topmost style corresponds to the currently active object; you can assign any style in the list to that object by selecting a style from the list.

Edit If you activate this button, you can edit the style that is at the top of the drop down list.

Name Here, you can name the object individually. This name should be different than the names of other objects of the same type.

Layer This field indicates the layer which contains the object.

X This is the x-coordinate of the upper left corner of the object, respectively of the bounding rectangle of the object (indicated in the selected unit).

Y This is the y-coordinate of the upper left corner of the object, respectively of the bounding rectangle of the object (indicated in the selected unit).

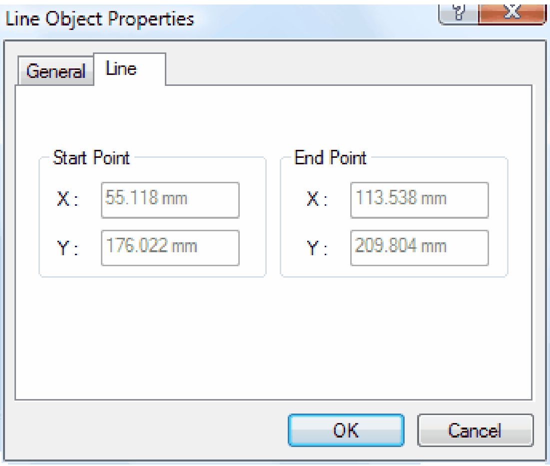
Width This is the width of the object indicated in the selected unit.

Height This is the height of the object indicated in the selected unit.

4.4.3.2 Line Object Properties

Description

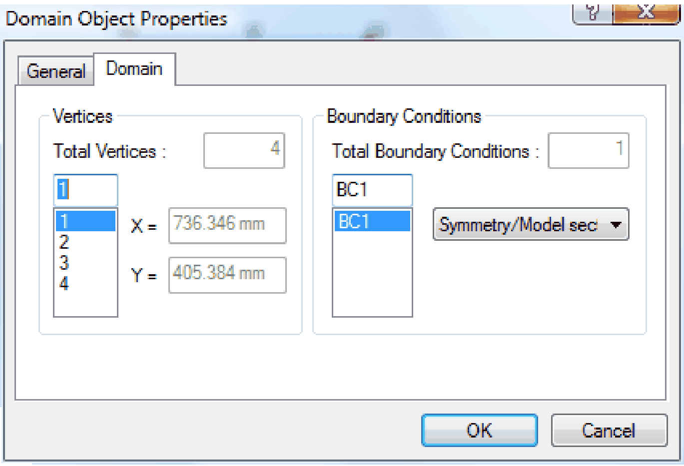
The dialog window for line objects has two tabs: A General tab as was described above, and a line tab containing information about the coordinates of the start and endpoints of a line object in the selected unit. The coordinate values cannot be modified in this dialog window.



4.4.3.3 Domain Object Properties

Description

This is the dialog window for domain objects (see Rectangular Domain tool, Elliptical Domain tool, and Polygon Domain tool), on the model page, which should not be confused with the result objects on the report page. The domain object dialog window has two tabs: a General tab as described above, and a domain tab containing specific settings for domain objects.



Fields

Total Vertices — Number of corner points for the domain

Vertices — x- and y-coordinates, which apply to the vertex that is currently selected from the list

X — x-coordinate of the currently selected vertex indicated in the selected unit

Y — y-coordinate of the currently selected vertex indicated in the selected unit

Total Boundary Conditions — Total number of boundary conditions for a specific domain

Boundary Condition — Here, you can choose the boundary condition, whose style should appear first in the drop down list or whose style can be edited.

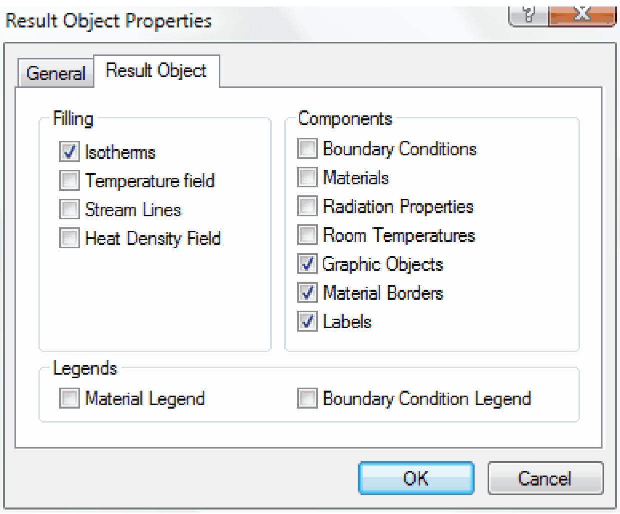
<boundary condition name> — Here, you can edit the style of the currently selected boundary condition.

Drop down list

4.4.3.4 Result Object Properties

Description

This is a dialog window for result objects (see Result Object tool) on report pages. This is not to be confused with the domain objects on the model page. The dialog window has two tabs: a General tab and a Result Object tab containing specific settings for result objects.



Fields

Isotherms

This option allows you to display the isotherms for the result object.

Temperature Field

This option allows you to display the temperature field for the result object.

Only one of the following settings can be selected: heat density, temperature, or material.

energy plus, pro
Stream Lines

This option allows you to display the stream lines for the result object.

Heat Flux Field

This option allows you to display the heat flux field for the result object.

Only one of the following settings can be selected: heat flux, temperature, or materials.

Boundary Condition

This option allows you to display the selected boundary condition for the result object.

Materials

This option allows you to display the selected material for this result object.

Only one of the following settings can be selected: heat density, temperature, or materials.

frame, pro
Radiation Properties

This option allows you to display the radiation properties of cavity surfaces. Radiation properties are only displayed if they are calculated automatically by flixo (cf. options Cavities).

Room Temperatures

This option allows you to display the room temperatures in the result object. The colors of the temperatures correspond to those of the temperature landscapes.

Graphic Objects

This option allows you to display graphic objects that facilitate the interpretation of the illustration, but that are not taken into account in the calculation.

Material Borders

This option allows you to display the material borders in the result object.

Labels

This option allows you to display point labels for the results of the result object.

Material Legend

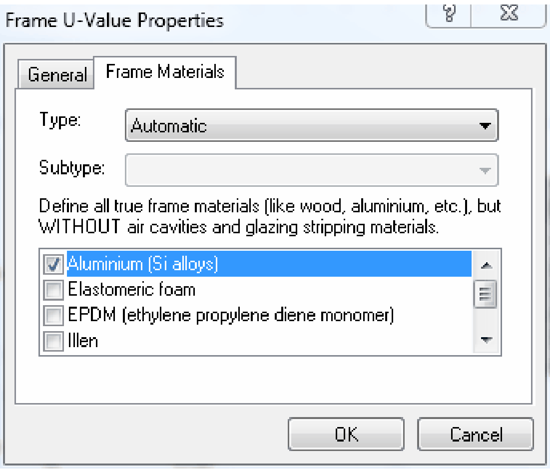
This option allows you to display a material legend for the result object.

Boundary Condition Legend This option allows you to display a boundary condition legend for the result object.

4.4.3.5 Frame or Joint U-Value Properties

Description

This is a dialog window for **Frame Uf-Value** objects and **Joint UTJ-Value** objects on report pages. The dialog window contains two tabs: the General tab and the Uf-Value respectively UTJ-Value tab containing frame materials.



Fields

Type Here, you define the type, how flixo determines the frame width:

- **Automatic**: The frame width will be determined automatically based on the selected frame materials.

- **2 Constructions**: The model consists of 2 components; Panel and frame.

- **3 Constructions**: The model consists of 3 components; Panel, frame and panel.

Subtype Here, you define the orientation for the type **2 Constructions** and **3 Constructions**.

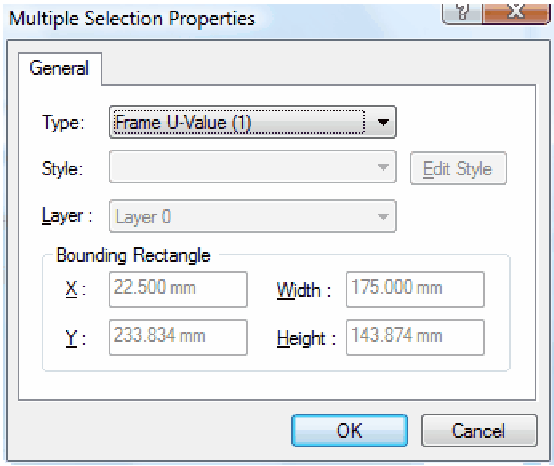
Frame Materials This option allows you to define the frame materials for the window frame cross section for the type Automatic. The settings will be needed for the automatic calculation of the window frame U-value (Uf-Value) according to EN ISO 10077-2 respectively the the joint U-value (UTJ-Value) according to EN ISO12631. In tutorial 4, the Uf-value calculation is described in more detail. Tutorial 5 explains how to calculate UTJ-Value.

Do not select any sealant material between the frame and the panel, which is used for glazing purposes.

4.4.3.6 Multiple Section Properties

Description

The dialog window for the multiple selection is displayed if the selection includes more than one object. The dialog window has only one General tab indicating general object characteristics.



4.4.4 File Properties Dialog Window

In this chapter, the properties of a flixo file are described in detail. Knowledge of these properties will help you when doing advanced Windows Explorer computer or network searches. You can activate the same dialog window by prompting the properties of a file in Windows Explorer.

You can also add the properties as a field object to the current document. More information on field objects can be found in the chapter Fields.

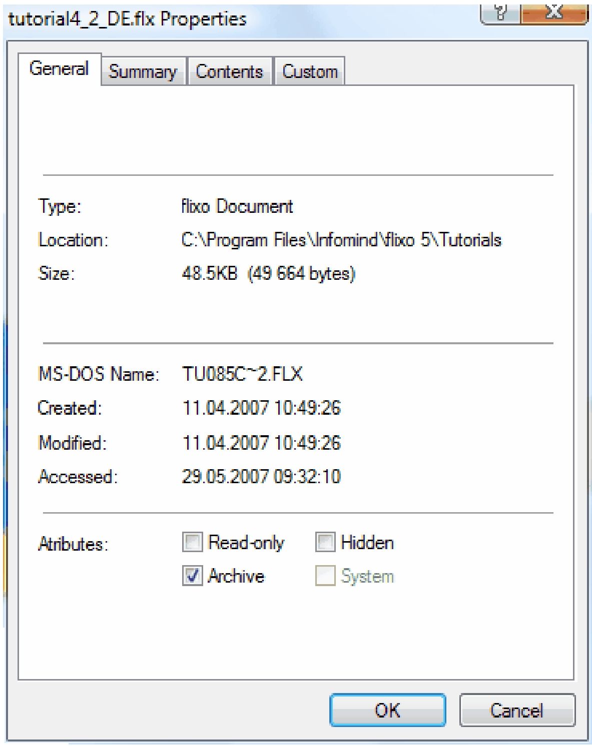
Below you will find information on the various tabs in the properties dialog window:

- General
- Summary
- Contents
- Custom

4.4.4.1 General Properties

Description

This tab shows the general information of a file such as the file path, the save date, and general file properties.



Fields

Type File Type, usually flixo documents

Location File path of the document

Size File size in MB and bytes

MS-DOS Name Name of the file in MS-DOS format. These names have a maximum of 8 characters and a 3 character data type ending.

Created Shows the date and time when the document was created.

Modified Shows the date and time when the document was last modified.

Accessed Shows the date and time when the document was last accessed.

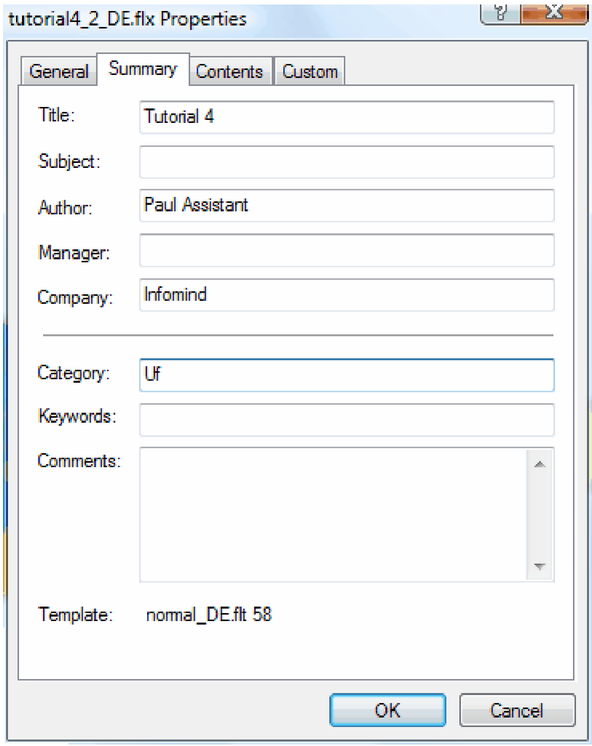
Attributes Shows the attributes of the file and allows you to modify them.

If a file is a read-only document then the document cannot be modified until the write protection is removed.

4.4.4.2 Summary Properties

Description

This tab contains information that facilitates the search for a file. These keywords also allow for a detailed description of the construction, which can also be inserted using field functions into a document (see tutorial 5).



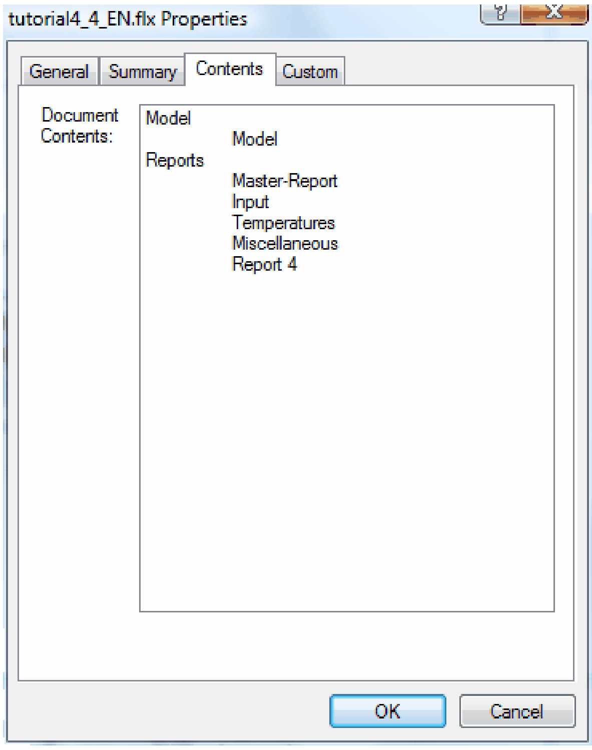
Fields

Title	Title of the document, the title can be used to search for a file.
Subject	Subject of the document, the subject can be used to search for a file.
Author	Author of the document
Manager	Manager of the document
Company	Company of the document
Category	Category of the document, the category can be used to search for a file.
Keywords	Keywords that can be used to search for a file.
Comments	Desired comments can be entered
Template	Name of the template that the document is based on

4.4.4.3 Content Properties

Description

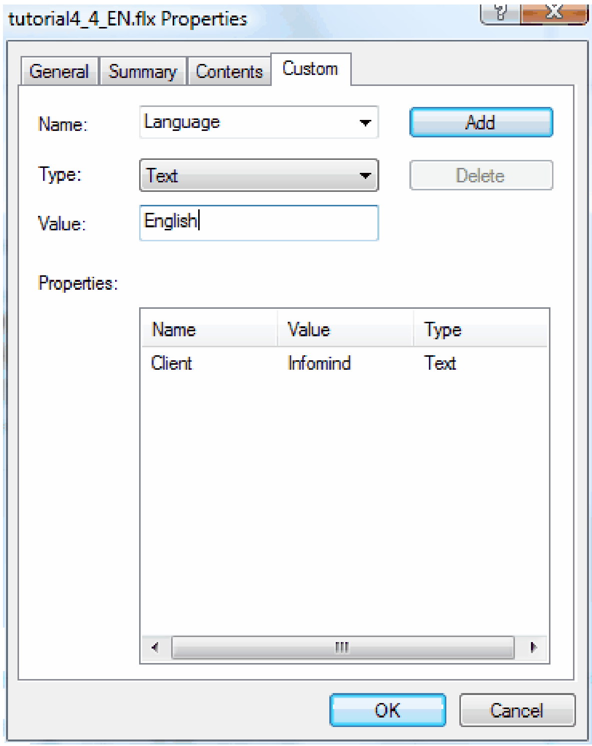
This tab shows the content of the document with the names of all the report pages.



4.4.4.4 Custom Properties

Description

In this tab, individual values of various characteristics are defined.



Fields

Name	Name of the variables, the name can either be chosen from the drop down menu, or it can be added directly into the field
Type	Type of variable selected in the "Name" drop down menu
Value	Value of variable selected in the "Name" drop down menu
Add **Modify**	Adds a new variable to the list or modifies the value of a variable, if the name corresponds to an entry in the "Properties" list.
Delete	Deletes the entry marked in the "Properties" list.
Properties	Displays a list of all the defined variables.

4.4.5 Various Dialog Windows

In this chapter, we shall discuss the various dialogs, which have not been addressed in the other chapters:

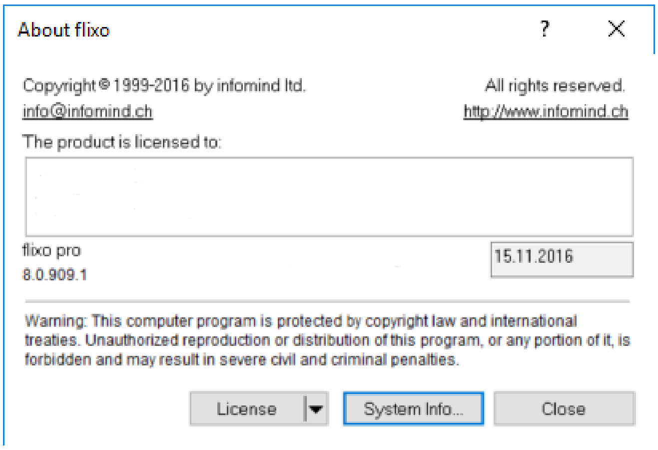
- About flixo
- Overwriting an old File
- Batchsolver
- Update Database
- Boundary Condition Temperatures
- Model Orientation

- Frame Materials
- Adjusting the Size
- Adapt Material
- Adapt Boundary Condition
- New File
- Document Templates
- Opening Pictures
- Page Setup
- Print
- Layers
- Materials
- Boundary Conditions
- Air Cavity Material
- New Components
- Calculation
- Fields
- Material Search
- Document Search
- Layer-Material Assignment
- 3D equivalent Object
- Importing Constructions
- Exporting Pictures
- Materials and Boundary Conditions
- Protect Objects
- Protect Report
- Protect Document
- Layer Object
- Glass Unit Object
- Various Warnings

4.4.5.1 About flixo

Description

In this dialog window, you can find copyright information and details about your license. The dialog window can be activated with menu command Help.



Fields

License	**Activate...** : Launches a program in which you can enter your ***ticket*** for activating your license. **Info...** : Determines your license and saves it to a file.
System Info...	Displays the Windows System Info dialog window, where you can find information about your computer, programs, and components.
http://www.infomind.ch	When you click on this hyperlink your web browser opens and goes to this address. There you can find the newest info about flixo.
info@infomind.ch	When you click on this hyperlink your mail program opens and creates a blank email addressed to infomind.

4.4.5.2 Overwriting an old file

Description

If you save a file that has been generated with a previous version of flixo, then the following warning will appear:

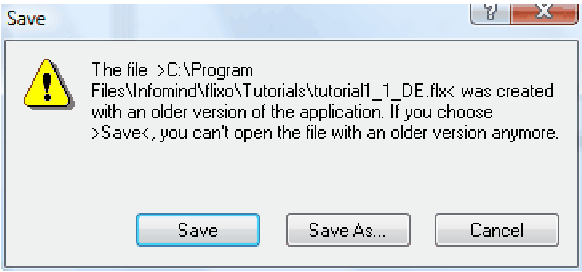


Figure 2: Save dialog window

Save Overwrites the file with the most current file. After the file has been saved in this way, it can no longer be opened with an older version of flixo.

Save As... Displays a standard file dialog window, where you can enter a new name and directory path for the file.

Cancel Cancels the action.

4.4.5.3 Translations

Description

In this dialog window you can change the translations of names, descriptions, labels etc. individually and configure the support of new languages for the active document. The translations can be typed into the corresponding cell or into the text box below.

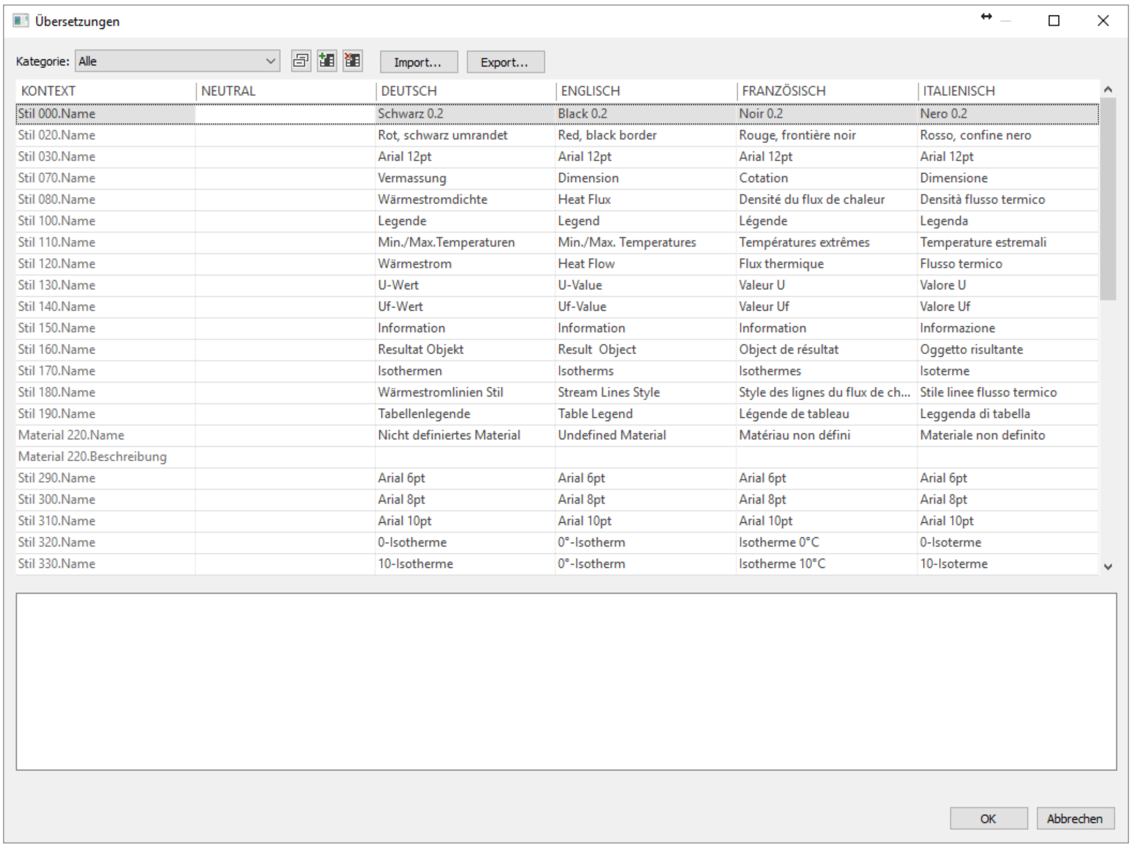


Figure 1: Translations dialog window

Category Selection of the category of descriptions (e.g. all, boundary conditions, materials) that shall be displayed in the table.

Copy	Copys the translations from one column to the next. The source (from) and the end column (to:) are determined in a dialog window.
New Language	Adds a new language, that shall be supported in the active document.
Delete	Deletes the support of the currently marked language. The support of the standard languages neutral, german, english, frenhce and italian cannot be deleted, instead all translations for these languages will be deleted.
Import ...	Imports the translations of another flixo document or form an exported translation table. The properties for the import are determined in a seperate dialog window.
Export ...	Exports the translations.
OK	Applies the customized translations.
Cancel	Cancels the action.

4.4.5.4 Translations import

Description

In this dialog window you can choose the source file, the languages you want to import and the options of the import.

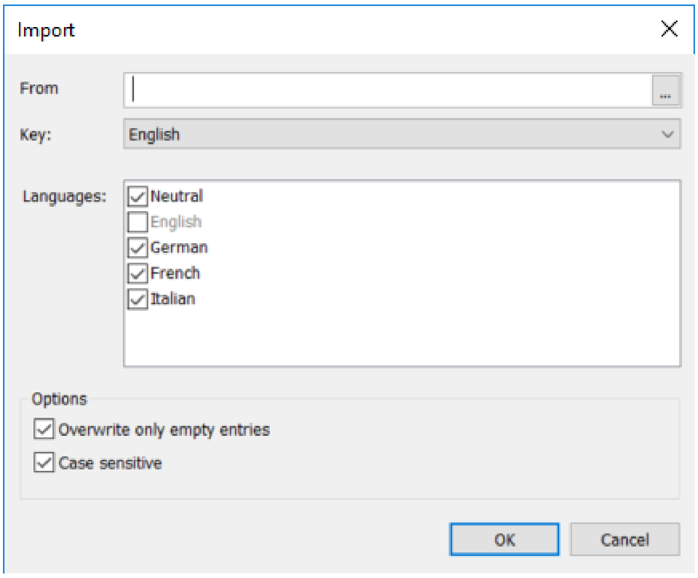


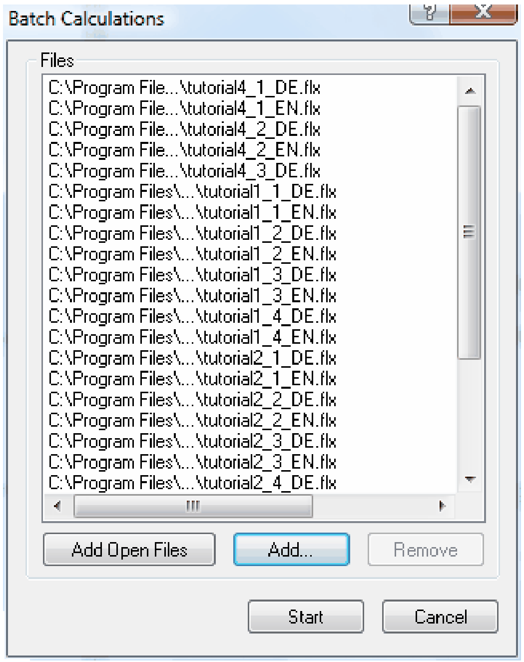
Figure 1: Translations import dialog window

From	Selection of the flixo-file or the translations file that you want to import.
Key	Language that shall be used as key for the import. The translations of the matching labels of the key language in the active document and the file that is to be imported are determined through the properties in **languages** and **options**.
Languages	Choose the languages, that are to be translated.
Overwrite only empty entries	If this option is activated. the translations are only applied for empty entries in the active document.
Case sensitive	If this option is activated, the search for matching entries is case sensitive.
OK	Imports the translations of the selected languages.
Cancel	Cancels the action.

4.4.5.5 Batchsolver

Description

You have the option to calculate many files at once, e.g. over night. The files can be defined in a special dialog window.



Fields

Files List of all files that are to be calculated.

Add Open Files Adds all the open flixo document files to the list for calculation.

Add Opens the standard Open dialog window, where you can select one or more files to calculate.

Remove Removes the selected files from the file calculation list.

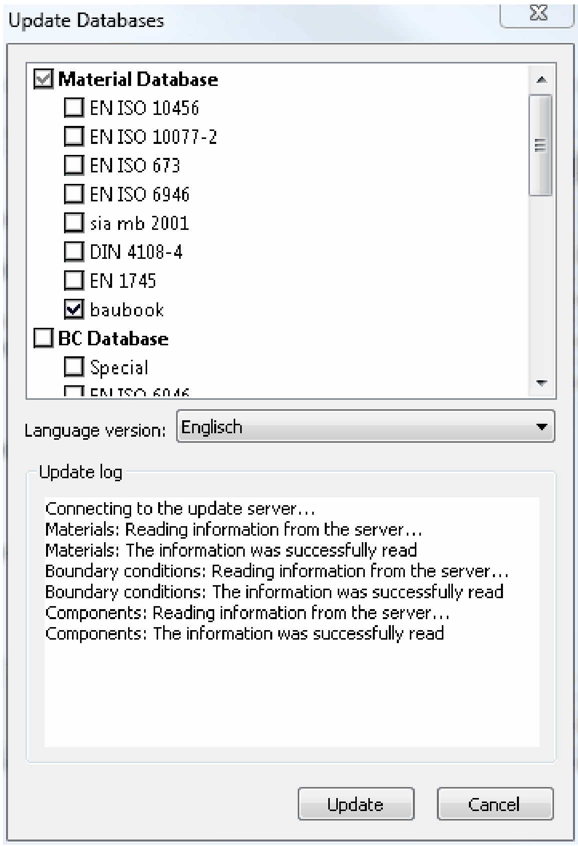
Start Begins the calculation of all files located on the file list.

Cancel Cancels the action.

4.4.5.6 Update Database

Description

You can periodically update database maintained by infomind or third-party.



Fields

Database List

Select the database, which should be updated. All databases containing not up-to-date data, are automatically selected when you open the dialogue.

Language version

Define the language of the database.

Update

The selected database are updated. The update can take several minutes.

An Internet connection is required to update the databases.

You will need **write permissions** on the database directory to update the database.

Cancel

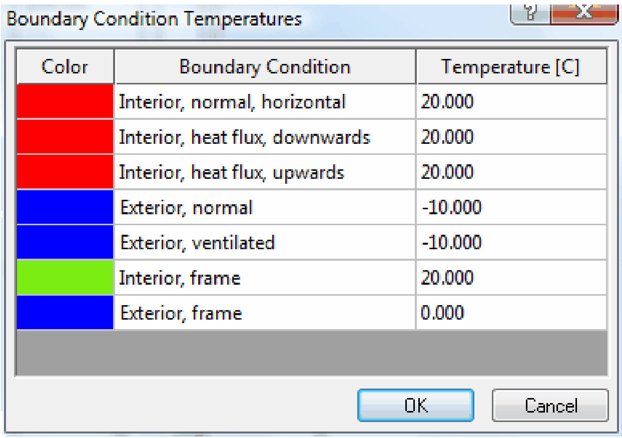
Cancels the update.

4.4.5.7 Boundary Condition Temperatures

Description

In the boundary condition temperatures dialog window, you can view all the boundary conditions and their properties.

Click on a temperature to edit its value. This dialog window can be activated from the context menu Boundary Conditions flyout.



Fields

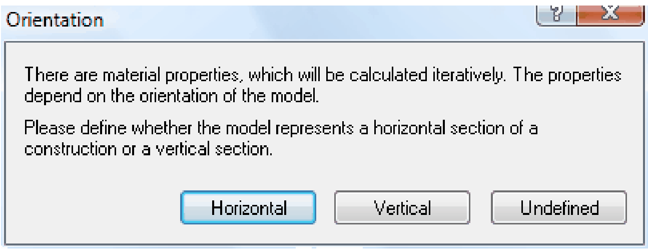
OK Assumes all new temperatures and adjusts all values of the boundary conditions.

Cancel Cancels the entire action. No temperatures will be changed.

4.4.5.8 Model Orientation

Description

For EN ISO 673 standard materials with temperature dependent thermal conductivities (cf. Materials) the orientation must be known if a calculation is to be made. You can define the model orientation before hand in the Model Options dialog window.



Fields

Horizontal section The model is a horizontal section of a construction.

Vertical section The model is a vertical section of a construction.

Cancel Cancels the calculation action.

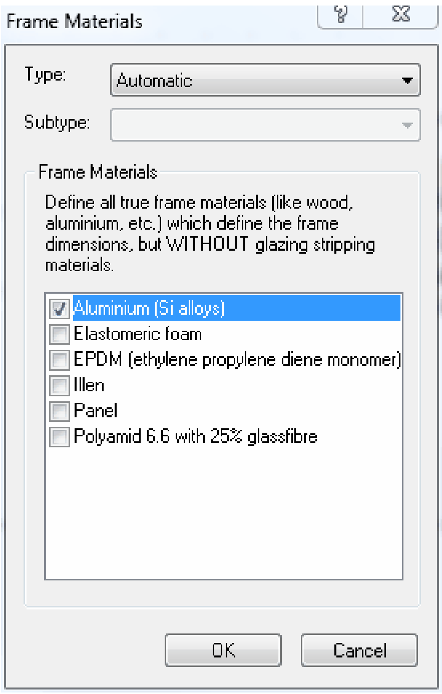
4.4.5.9 Frame Materials

frame (only Uf-Value), pro (Uf-Value and UTJ-Value)

Description

For the automatic calculation of window frame U-values (Uf-Value; cf. also tutorial 4) and joint U-values (UTJ-Value; cf. also tutorial 5) all frame materials must be known.

Alternatively, you can also set the type and the orientation of the model in the dialog window, and then adapt the dimension manually by moving the position of the start points of the dimension objects using the Select, Move, Scale tool or the Edit tool respectively.



Fields

Type Here, you set the type, how flixo determines the frame width:

- **Automatic**: The frame width will be determined automatically based on the selected frame materials.

- **2 Constructions**: The model consists of 2 components; Panel and frame.

- **3 Constructions**: The model consists of 3 components; Panel, frame and panel.

Subtype Here, you set the orientation for the type **2 Constructions** and **3 Constructions**.

Material List Here, you have to determine all the frame materials of the frame section which define the frame dimensions. The settings will be needed for the automatic calculation of frame U-values (Uf-value) according to EN ISO 10077-2 respectively for the automatic calculation of joint U-values (UTJ-value) according to EN ISO 12631 .

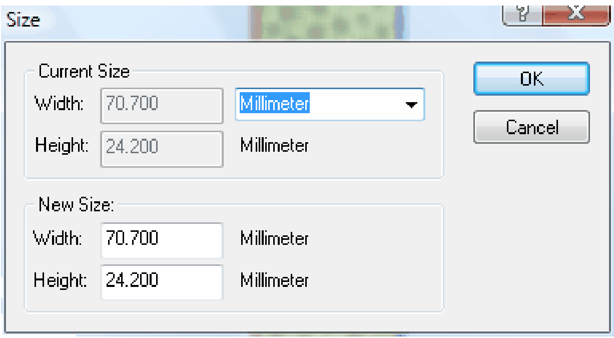
Do not select seal materials between the frame, glass units, and/or panel, which were used instead of the glass units.

Cancel Cancels the entire action; no U-value will be calculated.

4.4.5.10 Adapt Size

Description

The size of the entire construction can be adjusted by defining the size for a single domain.



Fields

Current Size In this group, the width and the height of the circumscribing rectangle are shown.

New Size Here, you can either enter the new width or the new height of the circumscribing rectangle. The second value will be calculated automatically, so that the ratio between the two values remains the same.

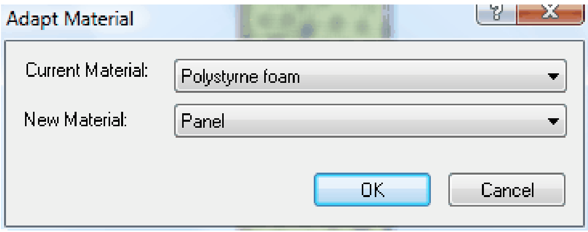
The entire construction then is enlarged or reduced according to the factor ***new width*** to ***old width***.

Cancel Cancels the entire action; the size of the construction will not be altered.

4.4.5.11 Adapt Material

Description

The materials of the entire construction can be adapted by defining the new material and the material, which is to be replaced.



Fields

Current Material In this list box, you can determine the material name, which should be replaced by the new material.

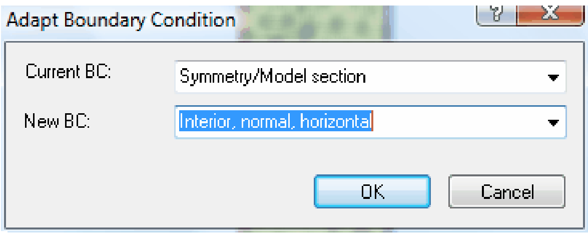
New Material In this list box, you can define the material name, which should replace the old material.

Cancel Cancels the entire action; the materials of the construction will not be altered.

4.4.5.12 Adapt Boundary Conditions

Description

The boundary conditions of the entire construction can be adapted by defining the new boundary condition and the boundary condition, which is to be replaced.



Fields

Current BC In this list box, you can define the boundary condition, which should be replaced by the new boundary condition.

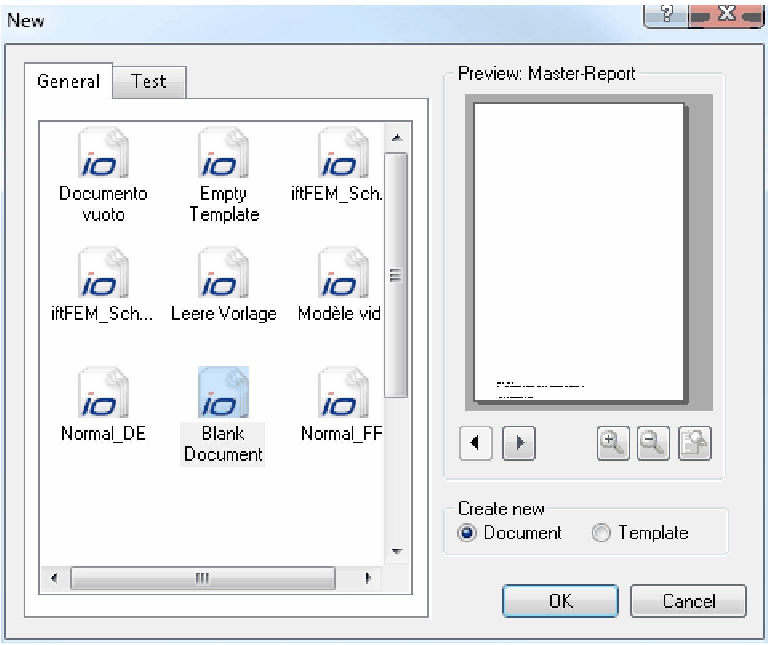
New BC	In this list box, you can define the boundary condition, which should replace the old boundary condition.
Cancel	Cancels the entire action; the boundary conditions of the construction will not be altered.

4.4.5.13 New File

Description

The New File dialog window can be activated by using the context menu command New from the File menu. You can either create a new document on the basis of a template, or you can create a new template. Template creation is described in more detail in tutorial 6.

All flixo templates in the template directory and its subdirectories are shown. You can change the template file path in the Save Options dialog window (menu command **Tools**.**Options**...).



Fields

Preview

You can get a preview of all report pages.

By clicking on the button, the previous report is shown. By clicking on the button, the next report will be shown. By clicking on the icon, the current preview will be enlarged. By clicking on the icon, the current preview will be reduced. By clicking on the icon, the entire report will be shown. You can also magnify individual regions of a report by dragging a rectangle around the area you would like to magnify.

Document When you choose this option, a new document based on the selected template in the list will be created.

Template When you choose this option, a new document template based on the selected template in the list will be created.

Cancel Cancels the action; a new document will not be created.

4.4.5.14 Document Template

Description

The Document Template dialog window can be activated by using the command Template from the File menu. All reports and settings will replaced by the selected template. Creation of a new document template is explained in further detail in tutorial 6.

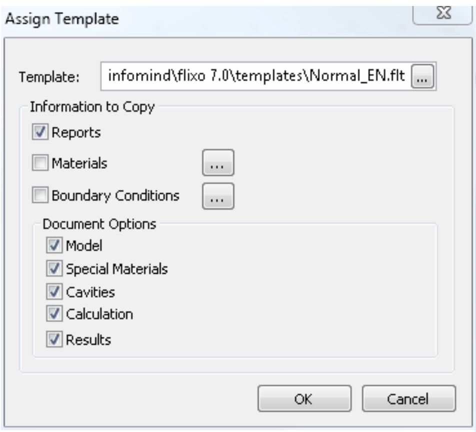


Figure 2: Selection of templates and options

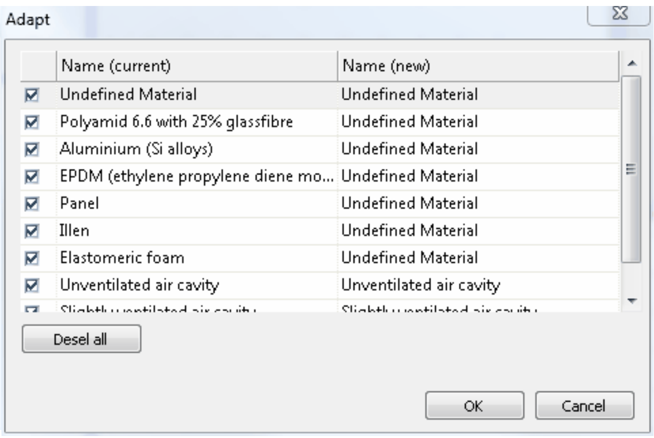


Figure 3: Replacement of materials and boundary conditions

Fields

Template Here you set the file name and path of the template, which newly serves as the basis of the document. Click on the **...** button to open a dialog window in which you can select the template.

Reports If you choose this option, all reports of the document are replaced by those of the document template.

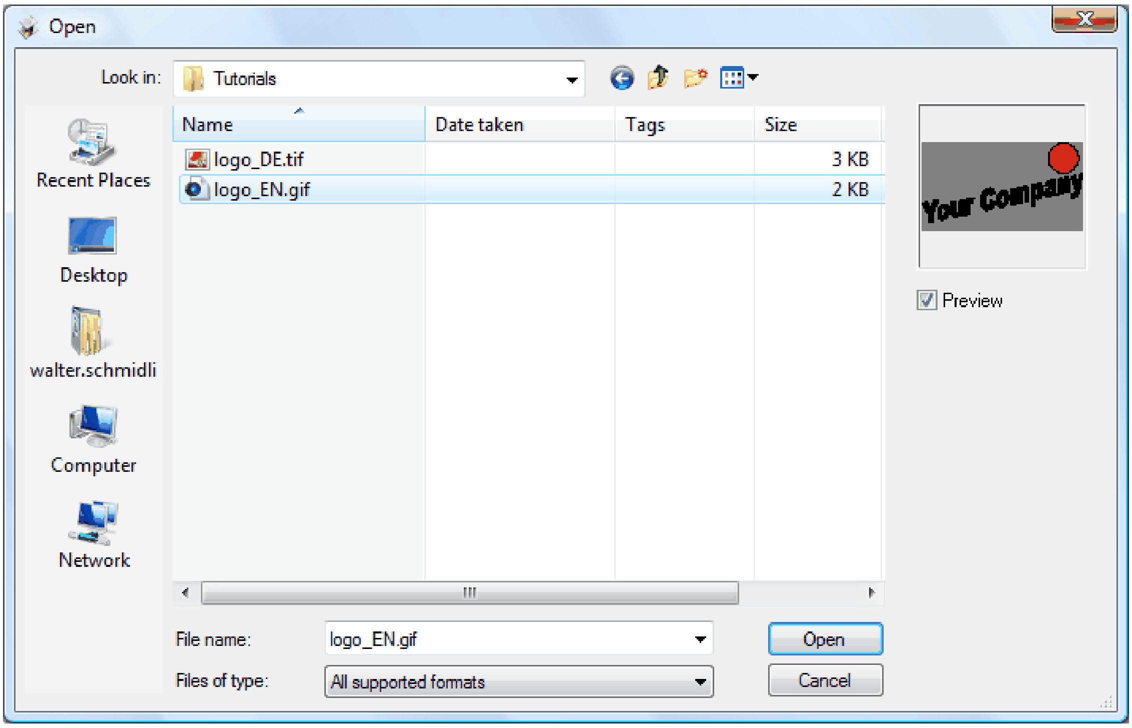
Materials If you choose this option, all materials selected in a dialog window of the document are replaced by those of the template.

	Click on the **...** button to open the dialog window (see Figure 2). In the dialog you can select the materials which will be replaced by the existing materials from the template. The new material from the template can be selected from a list by clicking into the last column.
Boundary Conditions	If you choose this option, all boundary conditions selected in a dialog window of the document are replaced by those of the template. Click on the **...** button to open a dialog window (see Figure 2). In the dialog you can select the boundary conditions which will be replaced by the existing boundary conditions from the template. The new boundary condition from the template can be selected from a list by clicking into the last column.
Model	If you choose this option, the Model Options of the template will be applied.
Special Materials	If you choose this option, the Special Materials Options of the template will be applied.
Cavities	If you choose this option, the Cavities Options of the template will be applied.
Calculation	If you choose this option, the Calculation Options of the template will be applied.
Results	If you choose this option, the Results Options of the template will be applied.
Cancel	Cancels the action.

4.4.5.15 Opening Pictures

Description

This dialog window is opened with menu command Picture.... from the Insert menu. The dialog window corresponds to a standard windows open file dialog window, except for the few points explained below.



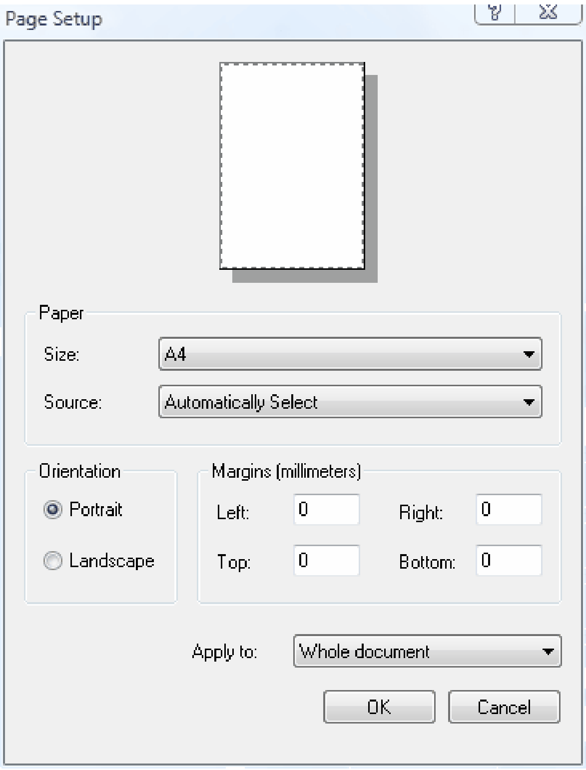
Fields

File Name	Name of the file picture you wish to add.
Files of type	Filter for the various picture files. Only those files that correspond to the file type selected here will be shown.
Preview	If this option is selected, then a preview will be displayed.

4.4.5.16 Page Setup

Description

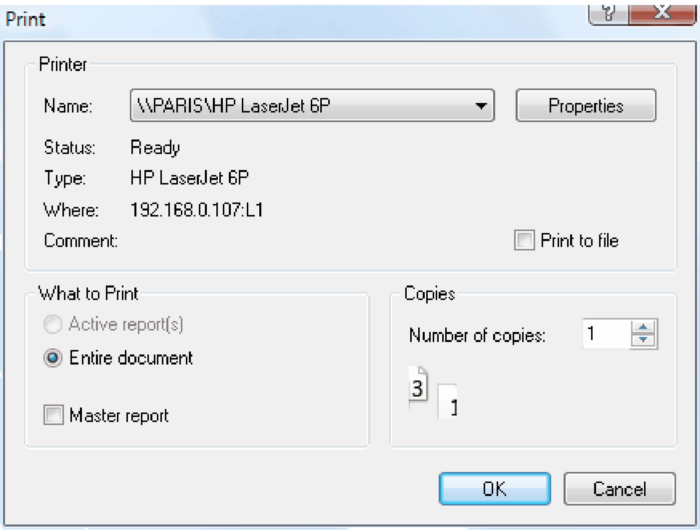
This dialog window is opened with menu command Page Setup from the File menu. The dialog window corresponds to a standard windows page setup dialog window.



4.4.5.17 Print

Description

This dialog window is opened with menu command Print From the File menu. The dialog window corresponds to a standard windows print dialog window with a few exceptions. Only the differences will be described.



Fields

Active Report(s) — If this option is selected, only the current page will be printed (cf. tabs just below the main work area). Depending on the **Master-Report** setting, the Master-Report will be printed in addition.

Entire Document — If this option is selected, the entire document will be printed (cf. tabs just below the main work area). Depending on the **Master-Report** setting, the Master-Report will be printed in addition.

Master Page — If this option is selected, the master-report will be printed.

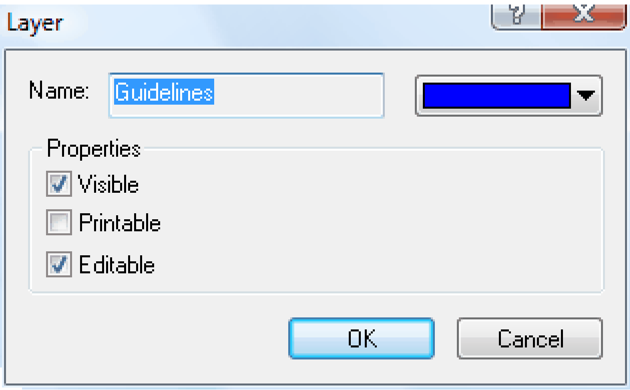
Limitations

The Model page cannot be printed.

4.4.5.18 Layers

Description

In the Layers option dialog window, the properties of the layers can be adjusted. Alternatively, the layer properties can directly be adjusted in the Layers flyout.



Fields

Name — Here the name of the layer is defined. You cannot leave the field blank. The name must be unique, i.e. it cannot be identical to the name of another layer on the same page. Aside from these restrictions, you can choose any name.

Color — Here, you can determine the color of the objects in the Draft view.

Visible — If this option is selected, all the objects on the selected layer are visible.

Printable If this option is selected, then all the objects on the selected layer will be printed.

Editable If this option is selected, then all the objects on the selected layer can be edited. This option cannot be changed for System layers.

OK Applies the settings to the selected layers.

Cancel Cancels the entire action.

4.4.5.19 Materials

Description

The dialog window ***Materials*** contains different fields to adjust the properties of a material. This dialog window can be activated from the Materials flyout or the Domain Object properties dialog window.

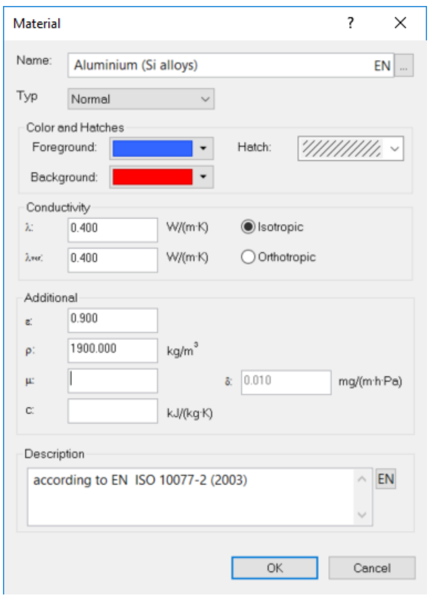


Figure 1: Material dialog window

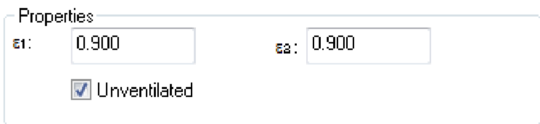


Figure 2: Air cavity properties

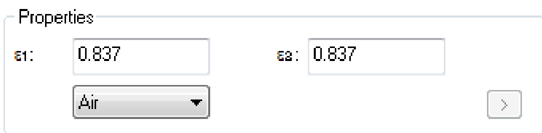


Figure 3: Gas properties

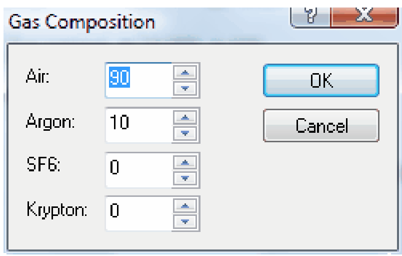


Figure 4: Gas composition properties

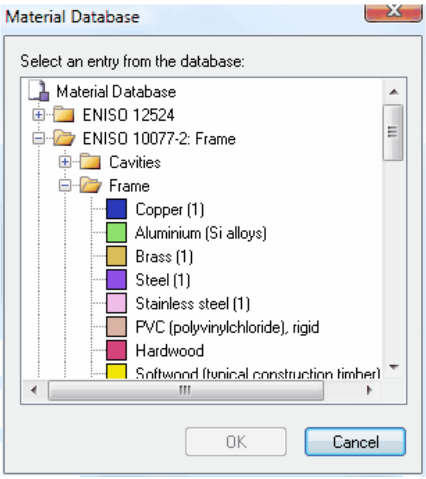


Figure 5: Materials database window

Fields

Name

Here the name of the material can be defined. You cannot leave this field empty, and the chosen name must be unique, i.e. it cannot be identical to the name of another material be it in an open document or be it in the application. Aside from these restrictions, you can choose any name.

Click on the language abbreviation in this field to set the name in all supported languages.

...

After clicking on the "..." button, a dialog window appears with the materials database. From the database, you can choose an entry. The properties of this material will be assumed.

Color

Defines the color, which along with the hatch, characterizes the material in the construction.

In the Materials flyout a small square before the name of a material defines its color. This color indicates the color of the material, which will be used when entering materials for a construction.

You can assign materials to a domain with the Assign Properties tool or by dragging the desired material onto the domain.

Hatch

Defines the hatch, which along with the color, characterizes the material in the construction.

Type

There are three different types of materials:

- Standard, for materials that do not have a temperature dependent thermal conductivity.

- Air cavity, for small air cavities according to EN 10077-2. The equivalent thermal conductivities are calculated iteratively according to EN 10077-2. The precision and the maximum number of iterations can be defined on the corresponding page of the Options dialog window.

- Gas, for gas of glass units according to CEN 673. This type should only be used for longer, rectangular glass unit cavities. The equivalent thermal conductivities are calculated iteratively according to CEN 673.

pro *with module radiation*

- vacuum for the modeling of vacuum.

Isotropic/Orthotropic

Defines whether the material is isotropic or orthotropic (thermal conductivities).

This option is only available for standard materials.

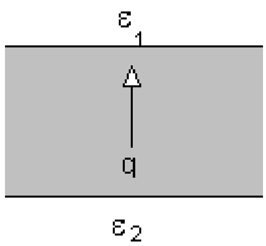
λhor, λver

Thermal conductivity. If the material is isotropic, then the second field is not available.

These values must only be entered for standard materials.

ε1,ε2

Long-wave emission properties of the two surfaces in the direction of the heat flux, if the emission properties are ***not*** directly calculated by flixo (cf. option dialog window document cavities).



These values must only be entered for "Air Cavity" and "Gas".

unventilated

Determines whether the cavity is ventilated or not ventilated.

According to EN 10077-2, air cavities are unventilated when they are completely surrounded by material with exterior exposure of ***less than 2mm***.

According to EN 10077-2, air cavities are ventilated when they have ***exposure greater or equal to 2mm but less than 10mm***.

If the air cavity is greater than this value, then a boundary condition must be set, and the air cavity cannot be assigned a "material".

This option can only be selected for air cavities.

Gas Type	A small drop down list with the supported gas types: air, argon, SF6, krypton, or gas composite. If you choose "Gas composite", then the mixture can be changed by clicking on the ">" button. A small dialog window will appear where you alter the mixture's composition. The sum of all components must equal 100%. The type of gas must only be defined for glass unit gases.
ε	Standard emmisivity of the material. This is used for calculations of the energy transport in air cavities according to EN ISO 10077-2.
ρ	Density of the material. *)
m	The ratio of the vapor conductivity of the air to the vapor conductivity of a material.
d	Vapor conductivity of the material. *)
c	Specific heat capacity of the material.*)
Description	Additional commentary Click on the language abbreviation in this field to set the name in all supported languages.

*)These values are not needed by flixo calculations.

4.4.5.20 Boundary Conditions

Description

The "Boundary Condition" dialog window contains several fields to define or adjust boundary conditions. This dialog can be activated from the Boundary Conditions flyout.

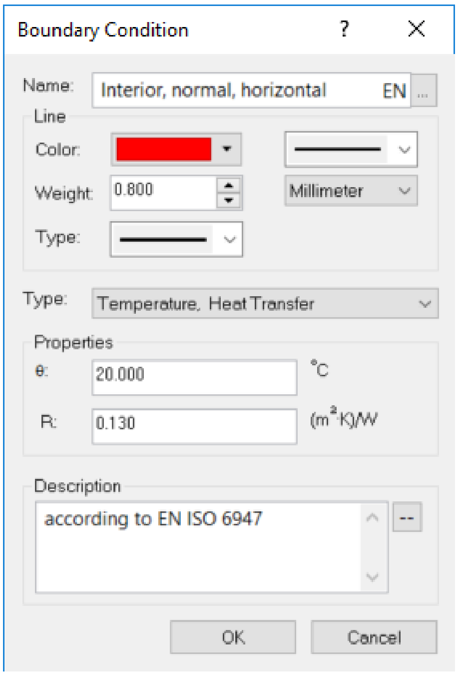


Figure 1: Boundary Conditions dialog window

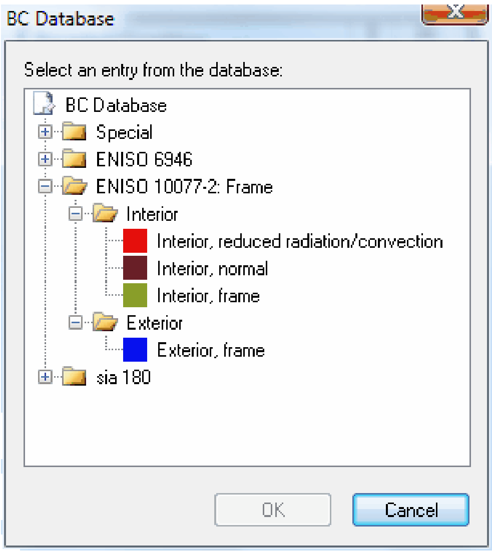


Figure 2: Boundary Conditions database dialog window

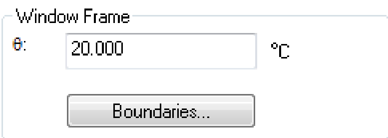


Figure 3: Window Frame Type

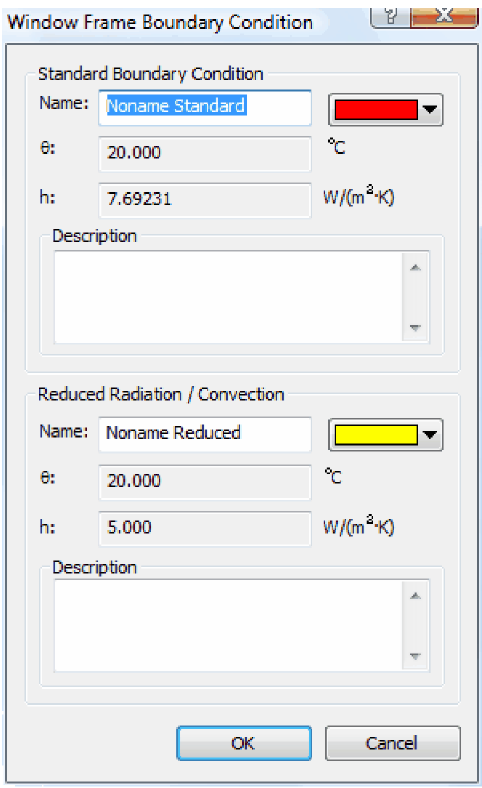


Figure 4: Window Frame boundary conditions dialog window

Fields

Name

Here the name of the boundary condition is defined. You cannot leave the field blank. And the name must be unique, i.e. it cannot be identical to the name of another boundary condition on the same page. Aside from these restrictions, you can choose any name.

Click on the language abbreviation in this field to set the name in all supported languages.

...

After clicking on the "..." button, a dialog window appears with the database. From the database, you can choose an entry. The boundary condition properties will be assumed.

Color

Determines the boundary condition color, which will appear in the model.

Type

Type of boundary condition. The following types of boundary conditions are supported:

- **Temperature, h-value**: For boundaries, where the air temperature and the heat transfer coefficient are present. This type of boundary condition is used for normal boundary conditions.

- **Temperature** (Dirichlet): For boundaries, where the surface temperature is given (e.g. the entering temperature for a floor heating system).

- **Heat Flux** (Neumann): For boundaries, where the heat flux for a surface is given.

- **Heat Flux, Temperature** (Cauchy): For boundaries, where the heat flux and the surface temperature are given.

- **Source/Sink**: For the performance of linear sources and sinks (see Heat Source Tool).

- **Radiation**: For the definition of surface radiation properties (emissivities) of air cavities, if the surface properties are calculated by flixo (cf. Option dialog window cavities).

- **Window Frame**: For interior boundaries when calculating the frame U-value according to EN ISO 10077-2 (cf. tutorial 4). The line properties, the name of the 2 dependent boundary conditions for normal domains, and the domains with reduced h-values can be adjusted by clicking on the Boundaries button and entering the values in the dialog window (see Figure 4) that appears.

q (Neumann, Cauchy)

Heat stream density on the surface.

θ (Cauchy, Dirichlet, Clima) — Surface temperature (for Dirichlet BC or Cauchy BC) or air temperature (for Climate BC and window frame BC).

h — Heat transfer coefficient

The heat transfer resistance R can optionally be entered at this point as well. The settings can be determined in the Options dialog window.

ε — Emissivity between 0 and 1.

Description — Additional commentary

Click on the language abbreviation in this field to set the name in all supported languages.

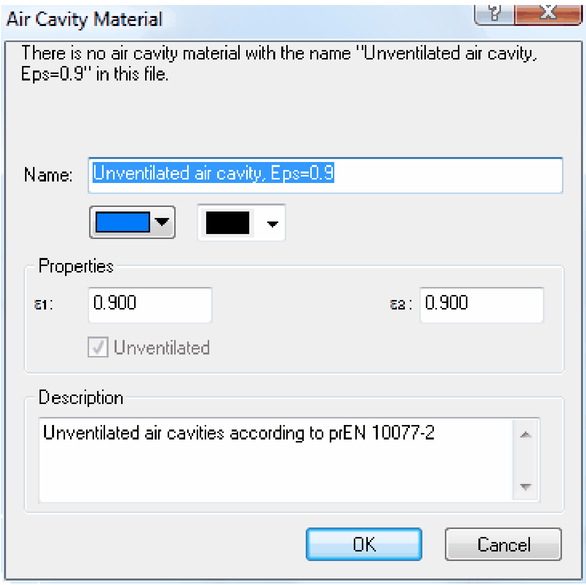
The values must be entered in the displayed units. You can change the display in the Basic Units Options dialog window.

4.4.5.21 Air Cavity Material

frame , pro

Description

The ***Air Cavity*** dialog window is activated exclusively when the Air Cavity EN ISO 10077-2 tool is prompted.



Fields

Name — The name of the material is determined here. The field cannot be left empty, and the selected name must be unique. You can either select an existing air cavity material of the correct type or you can create a new material with values entered into this dialog window.

Color

The material will be symbolized by this color (and hatch) in the construction.

You can only adjust the color if this is a new air cavity material.

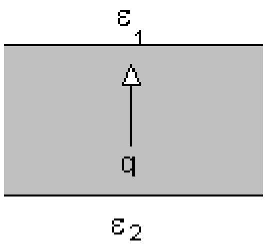
Hatch

The material will be symbolized by this hatch (and the color) in the construction. You can only adjust the hatch if this is a new air cavity material.

You can only adjust the hatch if this is a new air cavity material.

ε1,ε2

Long-wave length emission properties of the two surfaces in the direction of the heat flux.



You can only adjust these values if a new air cavity material is present and the emission properties are ***not*** calculated by flixo (cf. option dialog window document cavities).

Unventilated

Determines whether the air cavity is ventilated or not.

According to EN 10077-2, air cavities are unventilated when they are completely surrounded surrounded by material with exterior exposure of ***less than 2mm***.

According to EN 10077-2, air cavities are ventilated when they have exposure ***greater or equal to 2mm but less than 10mm***.

If the air cavity is greater than this value, then a boundary condition must be set, and the air cavity cannot be assigned a "material".

These options cannot be adjusted.

Description

Additional comments. You can only modify the description if you are creating a new air cavity material.

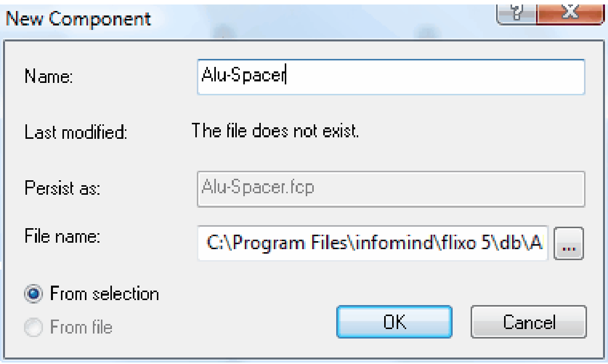
4.4.5.22 New Component

Description

In the ***New Component*** dialog window names and file paths of the component can be defined. This dialog window can be activated from the Components flyout.

This dialog window is used for both new components (components from a current selection) and for saved components.

In tutorial 2, new component creation is demonstrated.



Fields

Name — Here the name of the component is defined. You cannot leave this field empty, and the chosen name must be unique, i.e. it cannot be identical to the name of another component be it in an open document or be it in the application. Aside from these restrictions, you can choose any name.

Modified — Shows the modification date

File Name — File name under which the component was saved.

Directory — Directory, in which the components are (will be) saved. When you click on the "**...**" button, a standard fie dialog window is opened. Here you can either select the component you would like to load (radio button From file), or define the file path from a selection (From selection) to create a new component file.

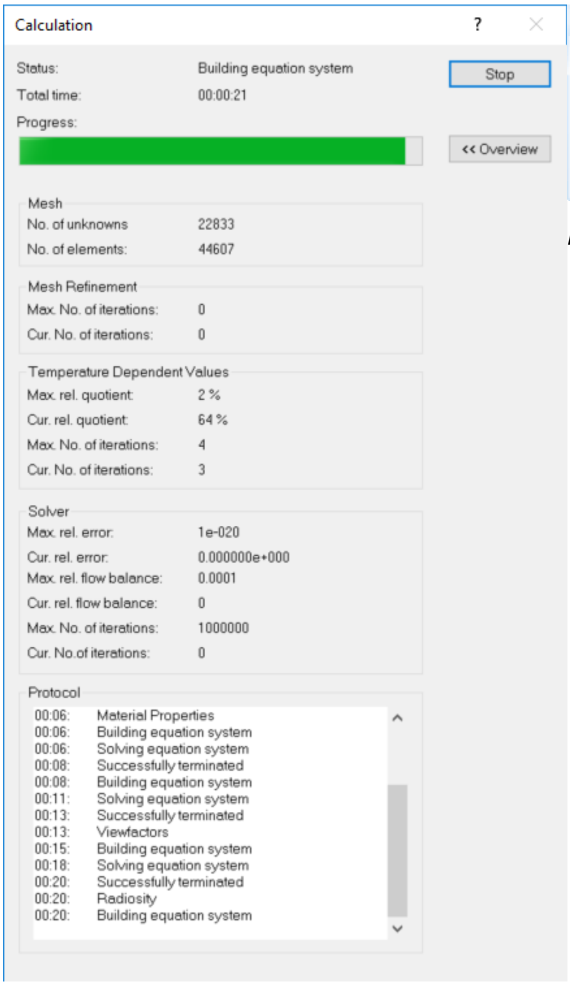
From selection — If you select this option, a new component is created from the selected object in the current document.

From file — If you select this option, a previously saved component with the entered file path will be loaded.

4.4.5.23 Calculation

Description

In this dialog window, the calculation progress and the chosen calculation options are displayed.



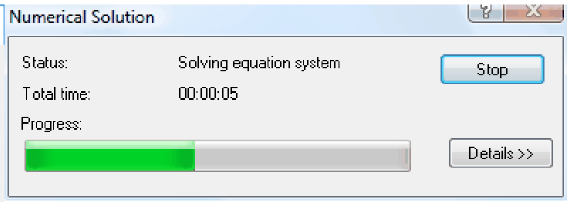


Figure 2: Overview

Figure 1: In detail

Fields

Status	Status of the calculation. After a successful calculation this reads, " **Successfully terminated**". If there is an error during the calculation, it reads, "**Error**". If the calculation has been canceled, it reads, "**Canceled**".
Total Time	Displays how long the calculation has been running for. At the end of a calculation, it displays the total time elapsed.
Progress	Progress of the calculation in percent.
Number of unknowns	Number of unknown temperatures, which must be calculated.
Number of elements	Number of finite elements for the mesh.

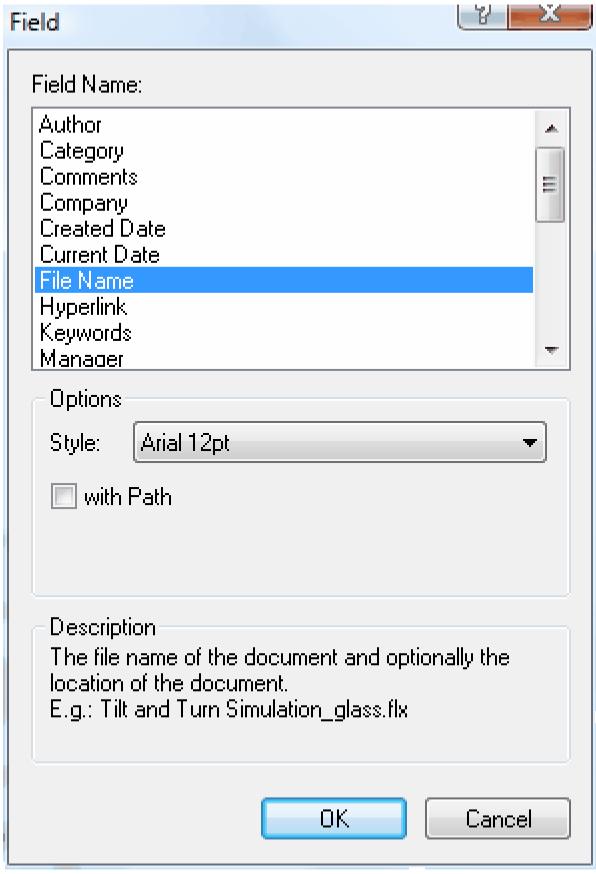
Max. Number of iterations	Maximum number of iterations for the calculation (for the calculation of the temperature dependent conductivities, equation solver, and automatic mesh refinement).
Current Number of iterations	Current number of iterations for the calculation (for the calculation of the temperature dependent conductivities, equation solver, and automatic mesh refinement).
Max. rel. quotient	The calculation of the temperature dependent conductivities will be carried out until the relative quotient of the conductivity of all domains with temperature dependent conductivities is smaller than the maximum tolerated relative quotient or until the maximum number of iterations has been reached.
Max. rel. error	Maximum relative error of a solution vector. The resulting equation system will be solved iteratively until the relative error is less than this value or the maximum number of iterations has been reached.
Current rel. error	Current relative error of the solution vector.
Max. rel. flow balance	Maximum relative error of the heat flow balance. The heat flow error is the ratio of the sum of all signed heat flows and half of the sum of absolute values of the heat flows.
Current rel. flow balance	Current relative error of the heat flow balance.
Stop	Stops the calculation. After stopping, the button caption changes to OK.
OK	Closes the dialog window. Before closing, the status of the calculation can be seen in the Status field.
<< Overview	Hides the calculation details .
>> Detailed	Displays the calculation details.

4.4.5.24 Fields

Description

In this dialog window, you can choose a number of ***field functions*** from a list, which can be inserted into the current document.

The field functions are place holders. Is the result field empty i.e. invisible then the field function descriptions can optionally be displayed instead of the results.



Fields

Current Program Version	The current version of flixo
Current Date	Current date and, optionally, the current time.
Number of FE	Number of finite elements in the current model.
Number of Pages	Total number of pages in the document
Number of Temperatures	Total number of unknown temperatures
Author	Author of the document. The value will be taken from the document properties (cf. File Properties dialog window).
User defined properties	The special, user-defined properties of the document. The value will be taken from the document properties (cf. File Properties dialog window).
File Name	File name and, optionally, the file path of the document.
Template	Template on which the document is based. The value will be taken from the document properties (cf. File Properties dialog

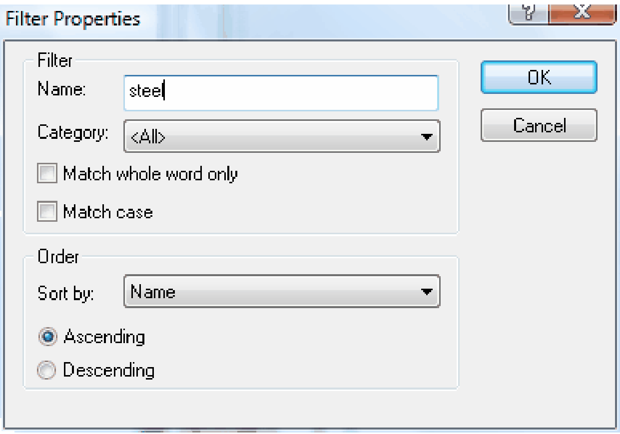
	window).
EN ISO 10077-2 Version	Version of the standard EN ISO 10077-2, which is used to calculate the equivalent conductivities of cavities.
Created Date	Date and, optionally, the time when the document was created.
Company	The company of the document. The value will be taken from the document properties (cf. File Properties dialog window).
Hyperlink	Hyperlink element. By clicking on the element, the corresponding URL will be activated and displayed in a web browser.
Category	The category(s) of the document. The value will be taken from the document properties (cf. File Properties dialog window).
Comments	The comments on the document. The value will be taken from the document properties (cf. File Properties dialog window).
Manager	The manager of the document. The value will be taken from the document properties (cf. File Properties dialog window).
Program Version Document	Program version of flixo, the last time the file was saved with.
Relative Heat Flow Error	Relative heat flow error according to EN ISO 10211.
Result as Value Object	Display the value of any result object (e.g. temperature object). The object is identified by a name. The name of the result object can be set in the Properties Flyout and must match with the one in this dialog.
Page	Current page number in the document. You can change the type of numbering (numerical, alphabetical, roman), set the first page, and define a prefix.
Page Title	The title of the current page (cf. tabs just below the work area).
Saved Date	Saved date and, optionally, the saved time of the document.
Keywords	The keywords for the document. The value will be taken from the document properties (cf. File Properties dialog window).
Sum of absolute Heat Flow	The sum of absolute values of all heat flows flowing into or out of the component.

Subject	The subject of the document. The value will be taken from the document properties (cf. File Properties dialog window).
Title	The title of the document. The value will be taken from the document properties (cf. File Properties dialog window).
Style	The selections correspond to the defined text styles (cf. Text style category in the Styles flyout). The field function will be created with the selected text style. The style can be changed afterwards with Drag&Drop.

4.4.5.25 Material Search

Description

In this dialog window, you can define the filter criteria.



Fields

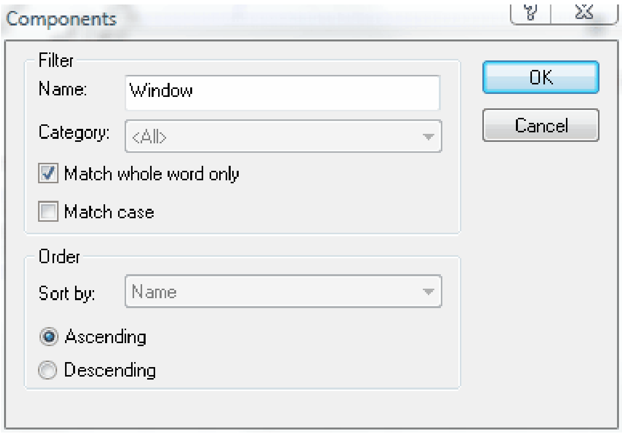
Name	Enter the name, or partial name of a material you would like to find. Place holders are not supported here.
Category	Here you define which categories you want to search in.
Match whole word only	Filter only returns materials whose names match the exact **Name:** Materials which have identical names with at least one other domain will not be returned.
Match case	Enables case-sensitive search criteria.
Sort by	Search results can be sorted by **"Name"**, **"Thermal Conductivity"**, **"Density"** or **"Specific Heat Capacity"**.
Ascending	The filter results will be arranged in ascending order (alphabetical or numerical).

Descending	The filter results will be arranged in descending order (alphabetical or numerical).

4.4.5.26 Document Search

Description

In this dialog window, you can define the filter criteria.



Fields

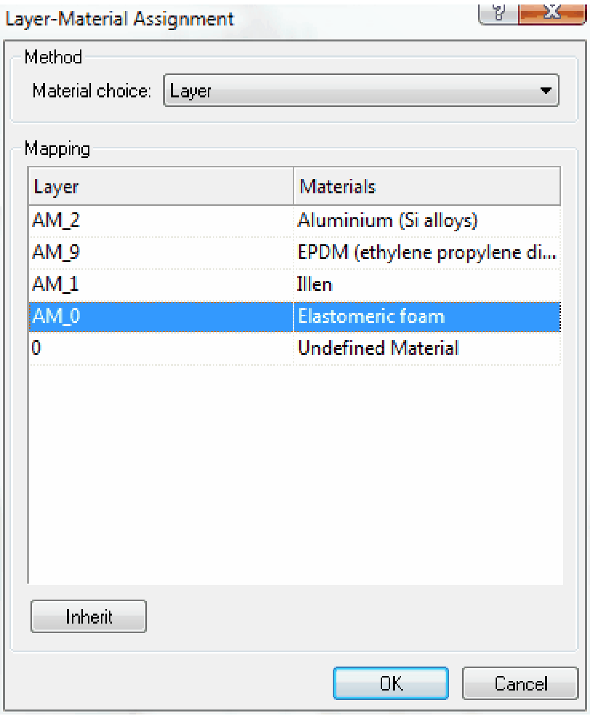
Name	Here you determine for which partial name you want to search. Place holders are not supported.
Category	You can't adjust this field.
Match whole word only	Filter only returns documents whose names match the exact **Name:** Documents which have identical names with at least one other domain will not be returned.
Match case	Enables case-sensitive search criteria.
Sort by	The search result is always sorted by the ***name.***
Ascending	The filter results will be arranged in ascending order (alphabetical or numerical).
Descending	The filter results will be arranged in descending order (alphabetical or numerical).

4.4.5.27 Layer-Material Assignment

Description

In this dialog window you can map each DXF layer to a material which will be used in the DXF conversion step. The automatic material assignment on converting DXF files is described in detail in tutorial 3.

In the table you can either select a material from the existing materials or you can create a new material in the opening Material dialog window.



Fields

Assign material

In this list you can define the characteristics of the material assignment when converting imported DXF data.

Layer: The material which is defined in the layer material mapping table is assigned automatically to all objects lying on the DXF-layer.

Layer - only closed objects: The material which is defined in the layer material mapping table is assigned automatically to all closed objects (e.g. polylines) lying on the DXF layer.

Selected material: The selected material from the Material list is assigned to all objects.

The automatic material assignment is explained in tutorial 3.

Inherit

Opens a file dialog window where you can select the file from which you want to inherit the existing DXF layer-material mapping table.

	Materials not not existing in the file materials will be inserted automatically into the material list.
OK	Inherits all data and closes the dialog window.
Cancel	Cancels the whole action. The data are not changed.

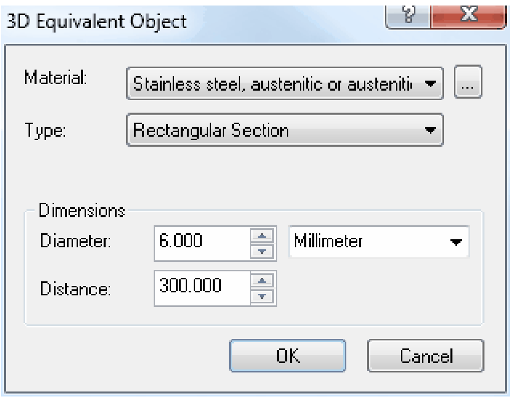
4.4.5.28 3D equivalent Object

energy, energy plus, pro

Description

In this dialog window you define the properties of periodically appearing, 3-dimensional objects according to prEN ISO 12631. The definition and use of a screw as an example of a 3-dimensional object is explained in detail in tutorial 8 .

Consider that applying an equivalent 3-dimensional object is only an approximation of a 3-dimensional calculation and doesn't replace it for critical cases.



Felder

Material	Select the material of the 3D object in this list. The equivalent material properties are calculated either by geometric mean or according to pr EN ISO 12631, depending on the diameter, the distance and the adjacent materials.
Type	Here you define the type of the periodically appearing 3D object. Following types are supported: **Screw**: periodically appearing screws. The screw head and screw pile shaft have the same material properties. **Rectangular cross section**: periodically appearing disturbances with a rectangular cross section.

pro : **Screw prEN ISO 12631**: periodically appearing screws. The equivalent lambda value is calculated according to prEN ISO 12631.

Only horizontal or vertical screws are supported. The orientation of a screw is considered as horizontal, if the horizontal dimension of the bounding rectangle of the screw is greater than the vertical one.

Distance Axis distance of the periodic 3D objects

OK Inherits all data and closes the dialog.

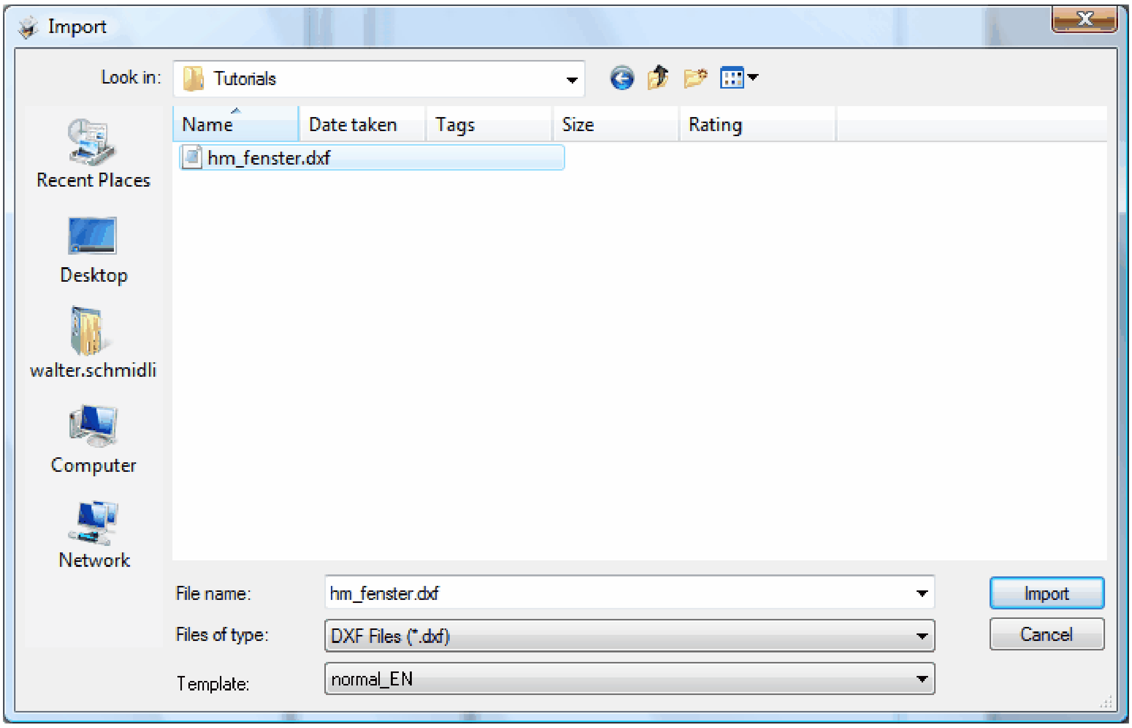
Cancel Cancels the whole action.

4.4.5.29 Importing Constructions

energy plus , frame , pro

Description

In this dialog window the file path, file name and file type, as well as the import settings can be determined.



Fields

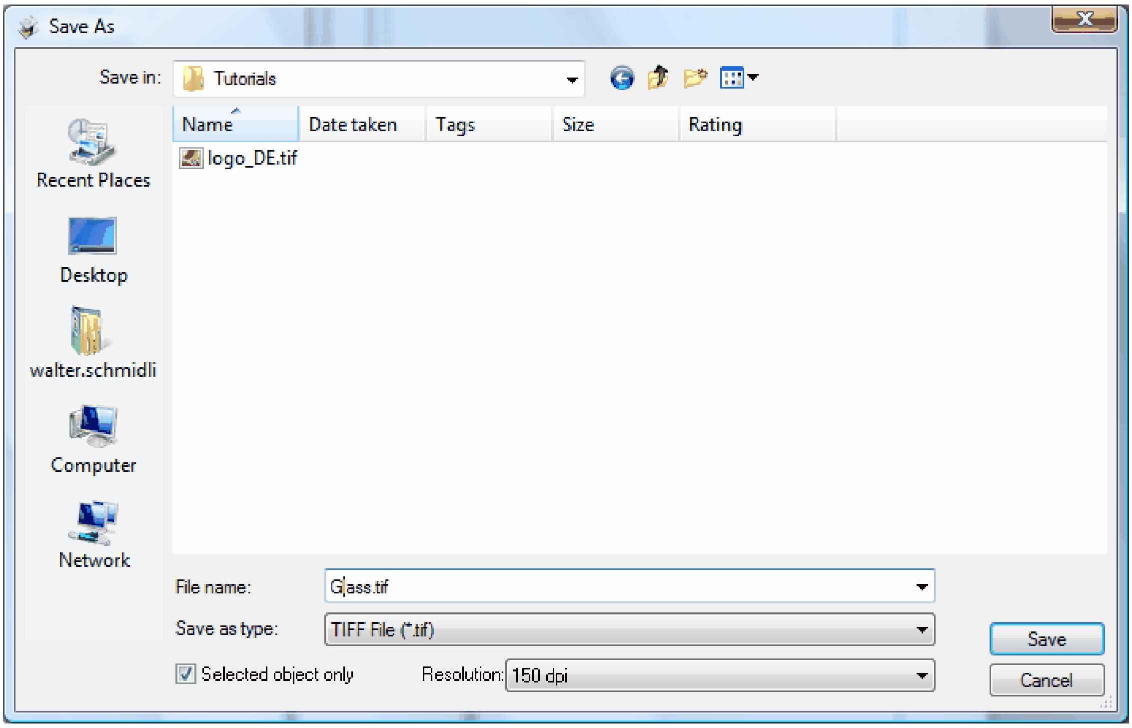
File name Name of the file you would like to import.

Files of type	Filter for the different types of construction files. The supported file types are the following: DXF format (*.dxf), SVG format (*.svg), and ISO2 files (*.con).
Template	Template file, which should be used as a basis for the imported data.
Options	Activates the Import Options dialog window, where for example the scale factor can be defined.

4.4.5.30 Exporting Pictures

Description

In this dialog window, the properties of the picture export (graphic file type, resolution) and the file path can be determined.



Fields

File Name	Name of the file under which the picture will be saved.
Files of type	File save format of the export.
Selected object only	If only the selected objects should be taken into account in the picture file, then this option can be marked. If this option is not marked then the whole report will be exported.
Resolution	Resolution at which the picture file should be exported.

4.4.5.31 Materials and Boundary Conditions

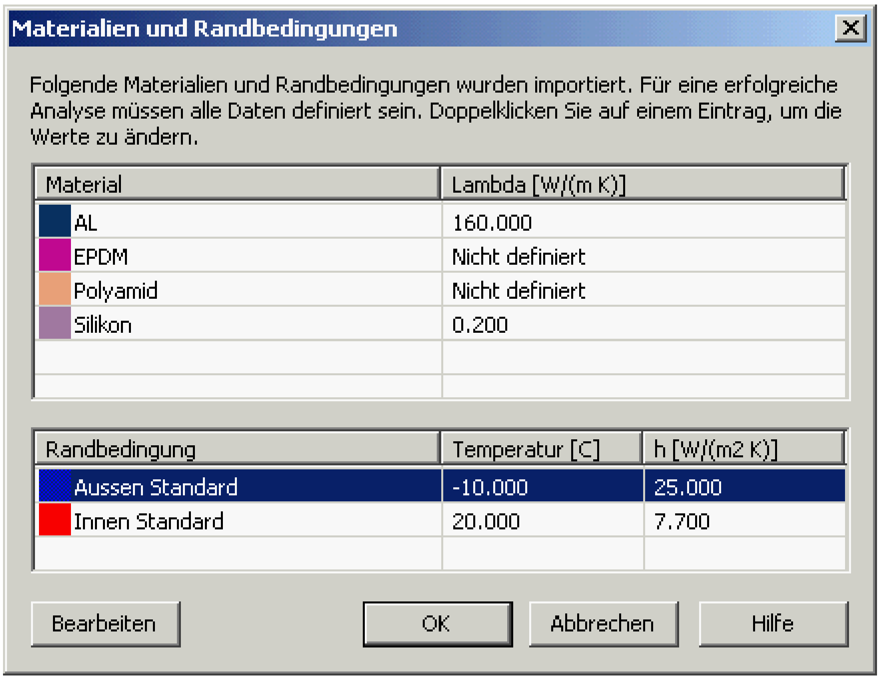
pro

Description

In the Materials and Boundary Conditions dialog window, all the materials and boundary conditions are listed when importing an ATHENA file.

For a successful import, all conductivities must be defined. By double clicking on an entry or by selecting an entry and then clicking on the Edit button, you can adjust the values.

This dialog window is only used when importing geometries and materials from the Athena CAD program.



Fields

Edit Activates, independently from the selected entries, either the Materials or the Boundary Conditions dialog window.

OK Assumes all of the new conductivity values and adjusts all of the boundary conditions.

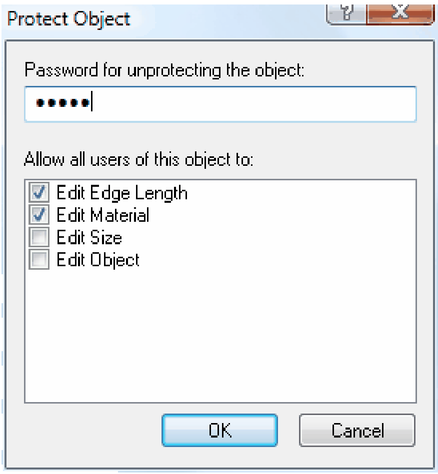
Cancel Cancels the entire action. The file will not be imported.

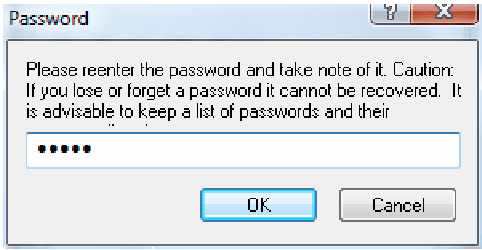
4.4.5.32 Protect Object

pro

Description

In this dialog window, you can protect certain parameters and sizes of an object against modification. It is possible to protect the object without a password, but if you did define a password a dialog prompting you for it will appear.





Fields

Password — You can enter a password or leave this field blank.

List with permitted modifications — You can protect the object selectively against changes of certain properties. The actions that are checked in the list are those which can later be modified :

- **Edit Edge Length**: Edge length can be modified with the Select, Move, Scale tool

- **Edit Material**: A domain of the object can be assigned a different material.

- **Edit Size** : The entire object can be scaled.

- **Edit Object**: The parameter and the protection properties can be changed.

OK — Selectively protects the object against changes of certain properties. The actions that are checked in the list are those which can later be modified by the user.

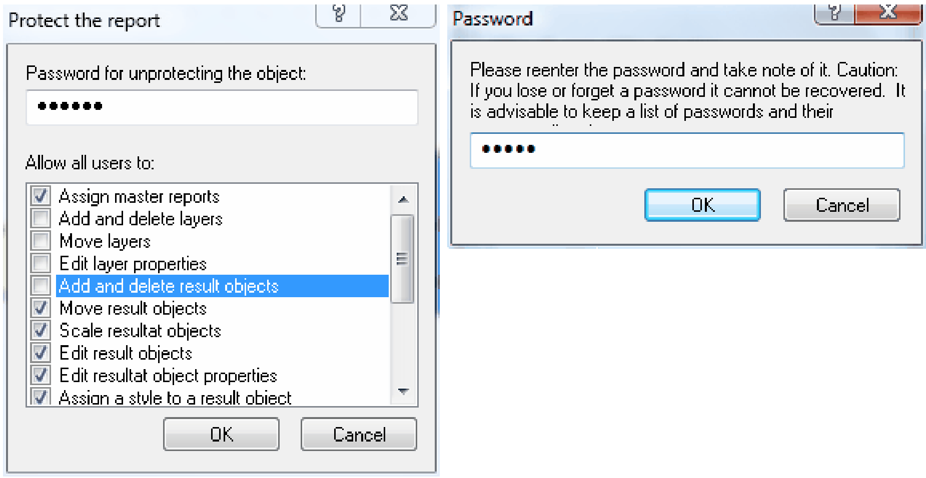
If a password has been defined, then a dialog window will appear prompting for a password.

Cancel — Cancels the entire action; the object will not be protected.

4.4.5.33 Protect Report

Description

In this dialog window, you can protect certain report properties from being revised or allow certain adjustments to reports. It is possible to protect the report without a password. If you have defined a password, a separate dialog window appears to enter the protection password.



Fields

Password

You can enter a password or leave the field blank.

Allowed adjustments list

Protects the report page selectively against changes of certain properties. Actions which are selected in the list may be adapted by users for the corresponding report page:

- **Assign master reports**: Assign another master report as a new basis to the selected report.

- **Add and delete layers**: Create new layers and delete existing layers.

- **Move layers**: Change the order - and thus the visibility - of layers.

- **Edit layer properties**: Edit the Layer properties.

- **Add and delete result objects**: Create new result objects (such as local temperatures, heat flow, etc.) and delete existing.

- **Move result objects**: Change the position of result objects.

- **Scale result objects**: Change the size of result objects.

- **Edit result objects**: Change the positions of the result object endpoints (e.g. position of the label).

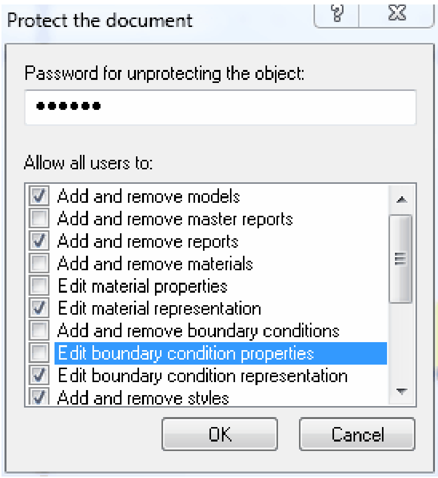
	- **Edit result object properties**: Adjust the properties (e.g. room humidity for condensation of the temperature object etc.) of a result object in the Properties Flyout or dialogue. - **Assign a style to a result object**: Choose a different appearance for a result object. - **Rename result objects**: Customize the name of a result object in the Properties Flyout. The values of the named result objects can be displayed with a field function. - **Add and delete general objects**: Create new general objects like line, text, or delete existing ones. - **Move general objects**: Adjust the position of general objects. - **Scale general objects**: Change the size of general objects. - **Edit general objects**: Adjust the position of vertices of general objects. - **Assign a style to a general object**: Select a different appearance for a general object.
OK	Protects the report selectively against changes of certain properties. Actions which are selected in the list may be adapted by users. If a password has been defined, then a dialog window will appear prompting for a password.
Cancel	Cancels the entire action. The report is not protected.

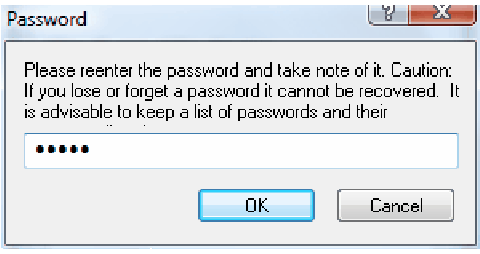
4.4.5.34 Protect Document

pro

Description

In this dialog window, you can protect certain document properties from being revised or allow certain adjustments to documents. It is possible to protect the document without a password. If you have defined a password, a separate dialog window appears to enter the protection password.





Fields

Password

You can enter a password or leave the field blank.

Allowed adjustments list

Protects the document selectively against changes of certain properties. Actions which are selected in the list may be adapted by users:

- **Add an remove models**: Add new model pages and delete existing ones.

- **Add an remove master reports**: Add new templates and delete existing ones.

- **Add an remove reports**: Add new reports and delete existing ones.

- **Add an remove materials**: Add new materials and delete existing ones.

- **Edit material properties**: Change material properties.

- **Edit material representation**: Change appearance-related properties (e.g. color) of materials.

- **Add and remove boundary conditions**: Add new boundary conditions and delete existing ones.

- **Edit boundary condition properties**: Change boundary condition properties.

- **Edit boundary condition representation**: Change appearance-related properties (e.g. color) of boundary conditions.

- **Add and remove styles**: Add new styles (appearance-related properties of result objects such as the number of decimal places) and delete existing ones.

- **Edit styles**: Change style properties.

- **Edit calculation options**: Change Calculation options.

- **Edit model options**: Change Model options.

- **Edit cavity options**: Change Cavity options.

- **Edit special material options**: Change Special material options.

- **Edit result options**: Change Result options.

OK	Protects the document selectively against changes of certain properties. Actions which are selected in the list may be adapted by users. If a password has been defined, then a dialog window will appear prompting for a password.
Cancel	Cancels the entire action. The document is not protected.

4.4.5.35 Layer Object

energy, energy plus, pro

Description

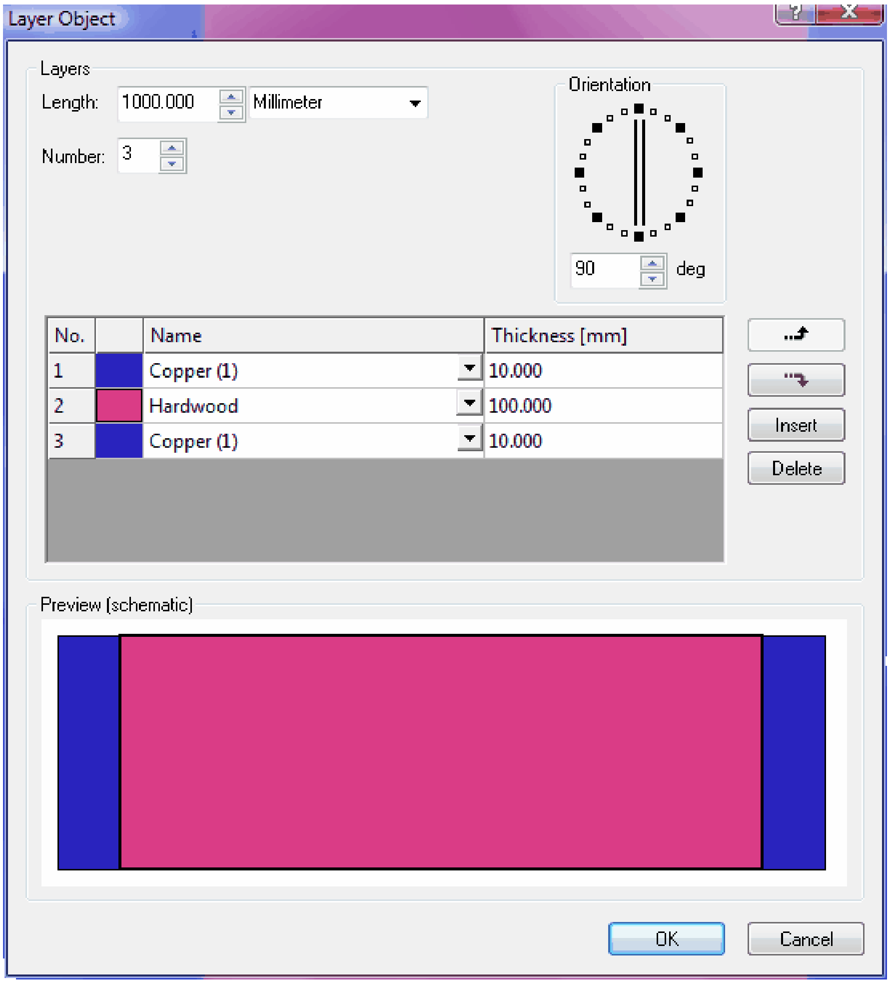
In this dialog window you can create a new parametric object constructed by material layers or you can edit an existing one.

The preview is only schematic in order to easily recognize the layers and select them directly in the preview. The layers will either be displayed horizontally or vertically and their length will be ignored.

In the orientation control group you can define the orientation of the layers by clicking on the circle representation or by numerically editing the corresponding field. By clicking inside one of the small squares in the circle representation the corresponding multiple of 15° will be set.

In the table you can choose the material property of a layer from the list of materials existing in the document or create a new material in the opening Material dialog window and you can adapt the layer thickness. With the buttons right of the table you can move, insert and delete layers.

The thickness as well the material properties can also be changed in the usual way (e.g. Select, Move, Scale tool, Drag&Drop).



Fields

Length	Length of the layers.
Number	Number of the layers.
Orientation	Orientation of the layers.
	Moves the selected layer upwards.
	Moves the selected layer downwards.
Insert	Inserts a new layer before the selected layer.
Delete	Deletes the selected layer.
OK	Adapts the layer object accordingly and closes the dialog window.
Cancel	Cancels the action. The changed data will be discarded.

4.4.5.36 Glass Unit Object

energy plus, frame, pro

Description

In this dialog window you can define a new glass unit object with a given U-value or you can edit an existing one.

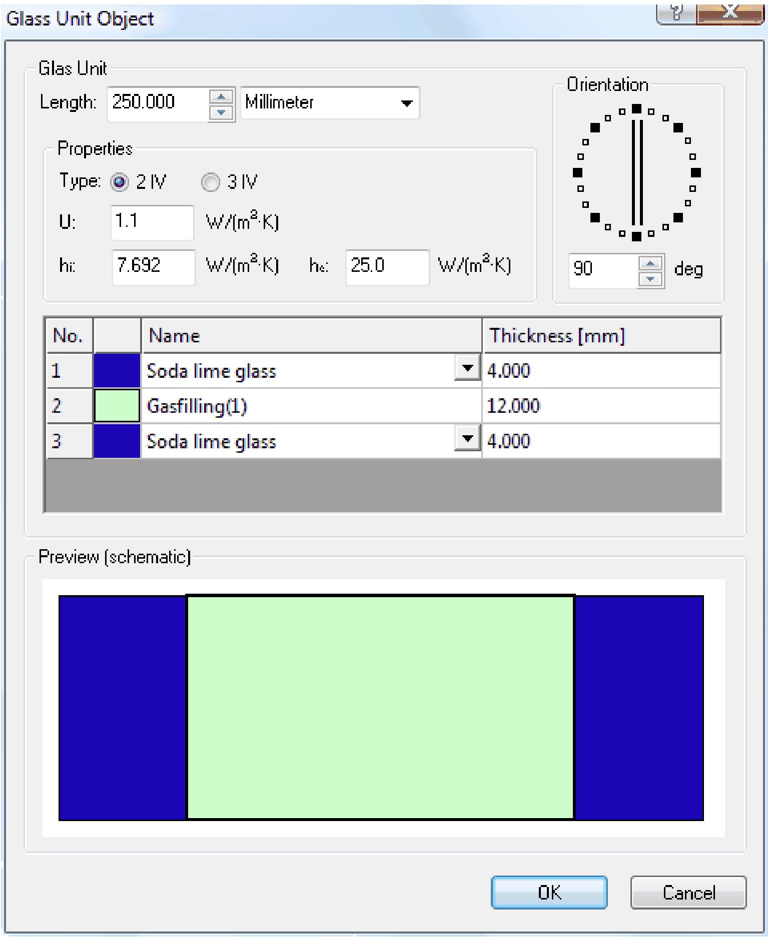
The preview is only schematic so you can easily recognize the layers and select them directly in the preview. The layers will either be displayed horizontally or vertically and their length will be ignored.

In the orientation control group you can define the orientation of the layers by clicking on the circle representation or by numerically editing the corresponding field. By clicking inside one of the small squares in the circle representation the corresponding multiple of 15° will be set.

In the characteristics group the type of glass unit can be chosen and the physical characteristics can be determined.

In the table you can choose the material property of the glass from the list of materials existing in the document or create a new glass material in the opening Material dialog window and you can adapt the layer thickness. You can also adjust the name and the color of the gas.

The thickness as well the material properties of the glass units can also be changed in the usual way (Select, Move, Scale tool, Drag&Drop).



Fields

Length Length respectively width of the glazing.

Orientation Orientation of the layers.

Type The type of the glass unit: either double (2IV) or triple glazings (3IV).

U U-value of the glass unit

hi resp. Ri Interior heat transfer coefficient, respective resistance, depending on the settings in the Option dialog window

he resp. Re Exterior heat transfer coefficient, respective resistance, depending on the settings in the Option dialog window

OK Adapts the glass unit accordingly and closes the dialog window.

Cancel Cancels the action. The changed data will be discarded.

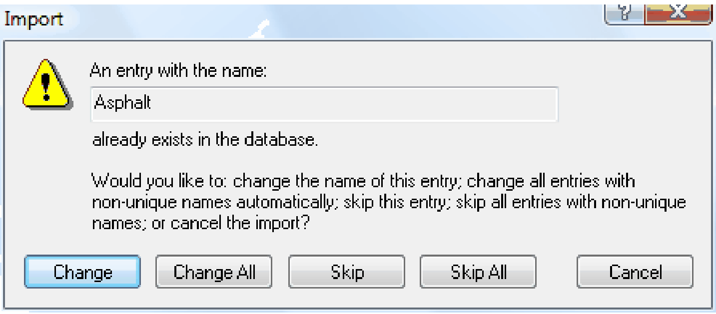
4.4.5.37 Various Warnings

In this chapter, we shall discuss various warning dialogs:

- Importing Database Entries - Existing Entries
- Importing Database Entries - Invalid Entries
- Calculation - Critical Material present
- Calculation - Radiation Properties
- Calculation - Automatic Cavity Subdivision

Importing Database Entries - Existing Entries

A material with the same name already exists in the materials database.

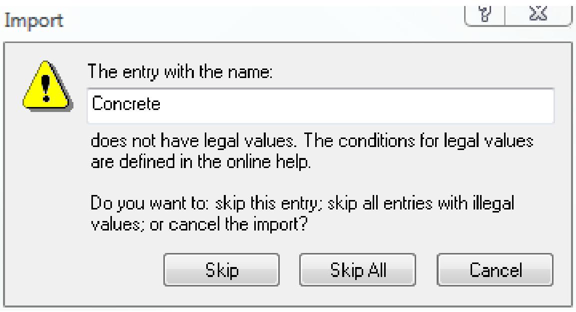


Fields

Change	Opens a dialog window in which a new name can be defined.
Change All	All names in conflict are automatically changed. All names are now unique.
Skip	Skips this entry without importing the data.
Skip All	Skips over all materials with names matching the database without further warning.
Cancel	Cancels the import process

Importing Database Entries - Invalid Entries

The definition of the listed material is either incomplete or invalid.



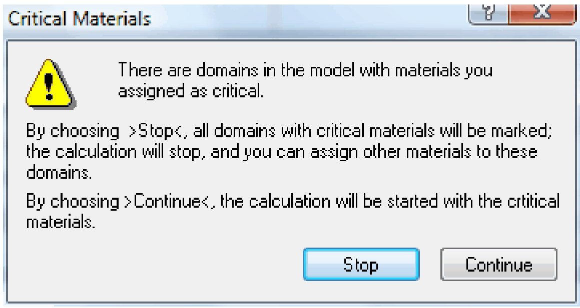
Fields

Skip Skips this entry and continues with the importing.

Cancels Cancels the import process

Calculation - Critical Material present

In the Special Materials Options dialog window you have defined materials as critical, which are visible in the construction.



Fields

Stop The calculation is stopped and all domains with critical materials are highlighted.

By repeatedly clicking on the "**Zoom In**" button in the message line, all critical material domains will be highlighted in order to assign other materials to these domains.

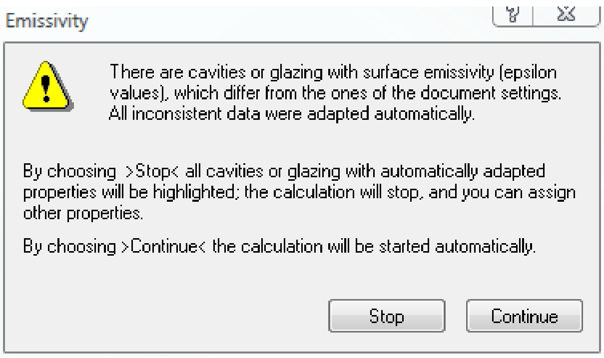
These highlights will disappear in the next calculation.

Continue The calculation continues with the present values.

frame , pro

Calculation - Radiation Properties

In the file are cavity materials according to EN ISO 10077-2 that don't match with the option settings (cf. Option dialog window).



Fields

Stop The calculation is stopped and all cavities whose properties don't match with the options will be highlighted.

By repeatedly clicking on the "**Zoom In**" button in the message line, all critical material domains will be highlighted in order to assign other materials respectively radiation properties to these domains.

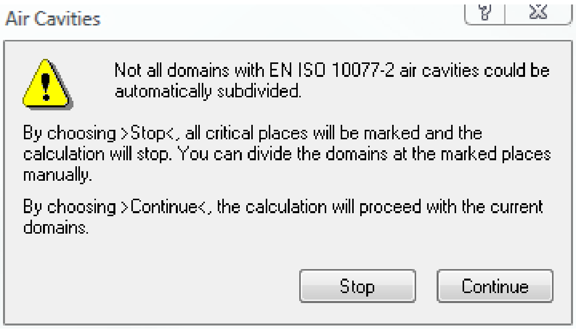
These highlights will disappear in the next calculation.

Continue The calculation continues with the current subdivisions.

frame , pro

Calculation - Automatic Cavity Subdivision

You have selected the option "**automatic subdivision of domains according to EN ISO 10077-2"** in the Option dialog window. Not all of the domains could get subdivided.



Fields

Stop The calculation has stopped and all necks will be highlighted where the cavities may be subdivided according to standard EN ISO 10077-2.

By repeatedly clicking on the "**Zoom In**" button in the message bar, all highlighted necks will be shown step-by-step in order to subdivide them manually using the Cut tool.

The markers will only disappear after the next calculation.

Continue The calculation continues with the current subdivisions.

4.5 Logical Operations

flixo offers the possibility to use logical operations on material domains. It allows you to unite, cut, or subtract domains. These types of operations are useful to create complex shapes from several separate shapes. The following chapter describes these commands.

Chapter subjects

- Union operations
- Cutting operations
- Subtraction operations

4.5.1 Unification of Objects

Description

This operation unites selected material domains. The result of the ***unification*** is a single object, which takes on the form of the sum of all participating domains. This operation allows the creation of a single object out of many different domains, or the creation of domains that contain cavities, but are composed of one single object.

The operation will unite all overlapping material domains, the border lines will be automatically removed.

If there are domains with different materials, then the new domain will assume the **material of the domain that was first selected**.

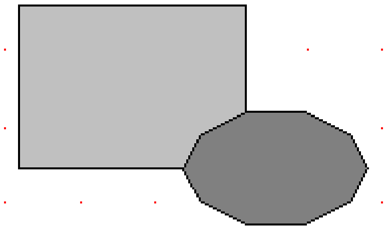


Figure 1: Before the unification

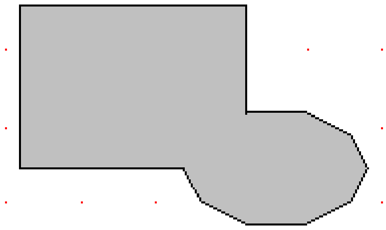


Figure 2: After the unification of the two objects

Process

- Activate the Select, Move, Scale tool, and select the domains that you would like to unify.
- Choose the command **Unite** from the **Arrange** menu or click on the icon, which is located on the **Arrange** toolbar.

Figure 3: Arrange toolbar

4.5.2 Intersection of Objects

Description

The ***Intersection operation*** lets you determine which part of a selected material domain you would like to cut. The result of the operation is the creation of a domain, which is composed out of that area where all involved domains overlap. Similar to the Unification of Objects operation, this operation can create a domain out of several independent domains belonging to the same object.

If the objects have different materials, then the result object assumes the **properties of the object that was first selected.**.

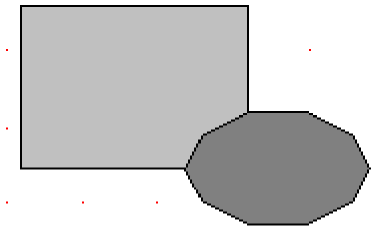


Figure 1: Before

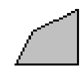


Figure 2: After: Intersection of the two objects

Process

- Activate the Select, Move, Scale tool and mark the domain you would like to cut.
- Choose the **Intersect** command from the **Arrange** menu or click on the icon, which is located on the **Arrange** toolbar.

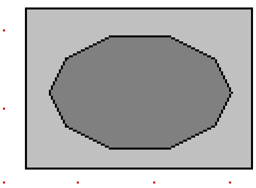
Figure 3: Arrange toolbar

4.5.3 Subtraction of Objects

Description

The ***Subtraction operation*** allows you to subtract a domain from one or more domains. The domain **which is selected last will be subtracted from the other domains**.

When only two domains are selected, then the resulting object is created from the left over area of the first selected domain. When more than two domains are involved, then the marked domain will be subtracted from all other selected domains. The resulting objects all take on the **same material properties of the first selected object**.



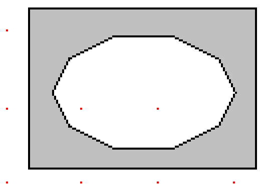


Figure 1: Before *Figure 2: After the subtraction of the two objects*

Process

- Activate the Select, Move, Scale tool and mark all the material domains, from which you want to subtract.
- Hold down the **Shift** key and select the domain you want to subtract from the others.
- Use the command **Subtract** from the **Arrange** menu or click on the icon located on the **Arrange** toolbar.

Figure 3: Arrange toolbar

4.6 Conventions

Menu Commands

The description of menu commands is done in a shortened manner: "Select the menu command X.Y" (e.g Select the menu command File.open), rather than an elaborated description such as "Select command Y from the menu X" (e.g. Select the command open from the menu file).

4.6.1 Glossar

Ctrl-Key

Control key is used with another key to give a command. When held down while using the mouse, it usually invokes secondary mouse functions. (e.g. moving a copy of an object instead of moving the object itself)

Domain (Material Domain)

A material domain is a basic element of a flixo document / cross section.. A material domain will be displayed as a conjoined area or a composition of several such areas.

Draft Mode

In draft mode the defined material domains are not filled with color. Only the edges of material domains can be seen.

Drag&Drop

Moves an object by clicking on it and holding down the mouse button, dragging the object to the desired location, and then releasing the mouse button.

ESC-Key

The escape key is used to cancel a process.

Exterior Edge

An exterior edge is a component edge, which surrounds all material domains.

Guidelines

A guideline is a vertical or horizontal line, whose position can be precisely defined. Guidelines can be used to align objects.

Grid

The grid is composed of regularly distributed points within the working area. These points can be used to align objects.

Graphic Objects

Graphical Objects are not included in the calculation. They are used to illustrate the results and facilitate legibility.

Interior Edge

An interior edge is a component edge, which is completely surrounded by a material domain, e.g. floor heating systems, and chimneys.

Isotherm

Linie in a cross-section with the same temperature.

Mesh

The mesh is a geometric subdivision of the material domain, which is necessary to create the equation system needed to do the analysis. It influences the precision of the numerical solution.

Numerical Solution

The numerical solution is the result of the mesh generation and the calculation of the physical model.

Parametric Objects

These are objects, whose edge lengths can be edited with the Select, Move, Scale tool. The coordinates of the corner points can be adjusted according to predefined corner rules. These are objects which can be adjusted rather than redrawn.

Proof Mode

In proof mode all properties of the defined material domains and objects are shown.

Shift-Key

The shift key switches from lower case letters to upper case letters as long as the key is held down.

Snap-to

Snap-to lets you move objects freely with the mouse, but the reference point will "snap-to" objects, guidelines, or grid points. Moved objects will always snap to the point which is nearest when the mouse button is released. The snap-to settings determine where the object will actually snap to.

Streamlines

These are lines where heat conducts through the construction. Between 2 adjacent streamlines flows the same amount of energy. A greater density of streamlines in a construction indicates an areas of high energy conduction.

Tab-Key

The tabulator key is often used to change foci within the program.

Tool tip

Tool tips are small text boxes that appear when the mouse hovers over a command, button, or area that has a tool tip.

Index

- 2 -

- 3 -

- A -

- B -

- C -

- D -

- E -

- F -

- G -

- R -

- S -

- T -

- U -

- V -

- W -

- Z -