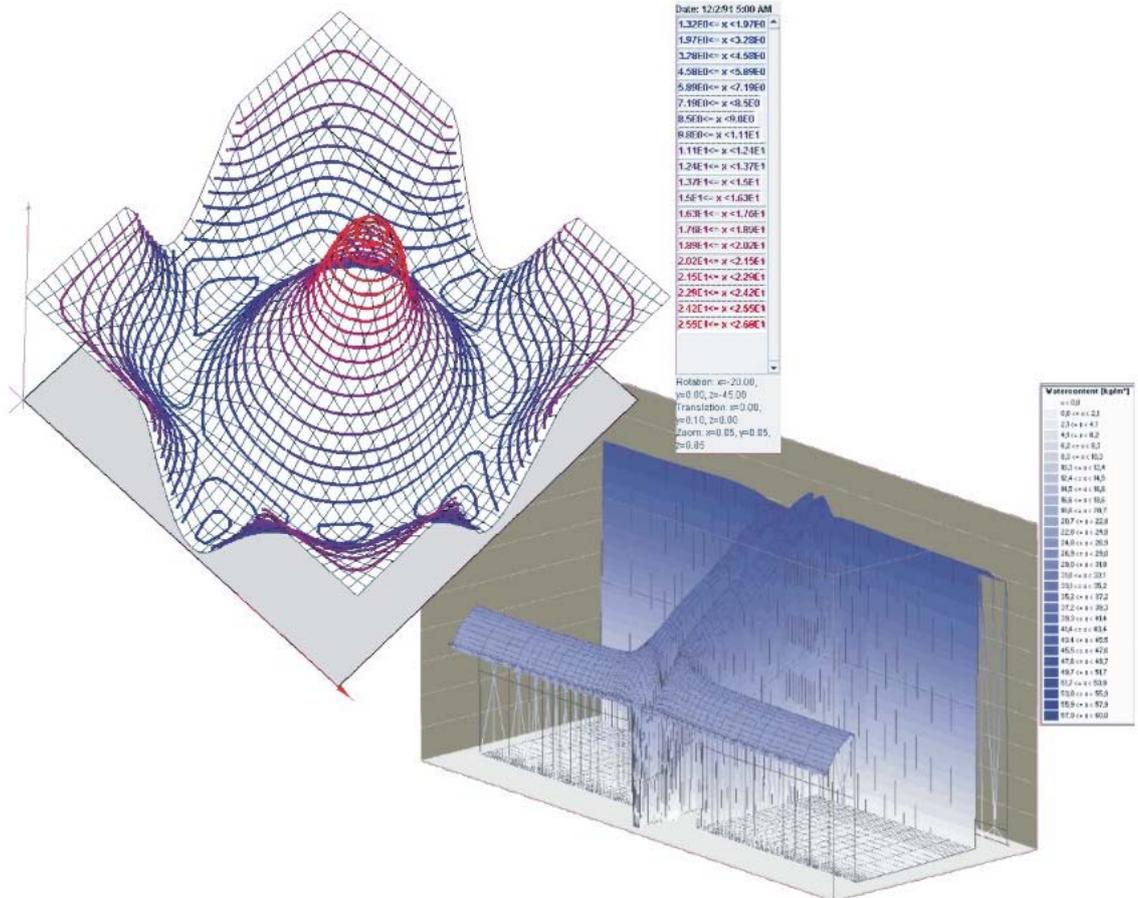




WUFI[®] 2D

Calculation Example Step by Step



WUFI[®] 2D



Fraunhofer
Institut
Bauphysik

PREFACE

This introduction will show you how to model a building component in WUFI 2D, how to specify boundary, surface transfer and initial conditions, how to perform the calculation and how to analyse the results. Many useful hints will be given. As an example illustrating each step, we will analyse the hygrothermal behavior of a concrete ceiling supported by a brick wall with exterior insulation.

For further information about the various dialogs and commands please consult WUFI 2D's online help.

Contents

1. EXAMPLE CASE - STEP BY STEP	4
Step 1: Project Notes.....	7
Step 2: Geometry of the Component	8
Step 3: Numerical Grid	12
Step 4: Material Properties	15
Step 5: Initial Conditions	20
Step 6: Boundary Conditions	21
Step 7: Computational Parameters.....	34
Intermediate Step: Save Project	36
Step 8: Computation.....	37
Step 9: Analysis of the Results	39

1. Example Case - Step by Step

This introduction will walk you step-by-step through an example case, demonstrating how to model a building component in WUFI 2D, how to perform the calculation and how to analyse the results. At the beginning of each step, **highlighted in green**, the procedure will be presented as applying to the example, then a general discussion of the program features will follow.

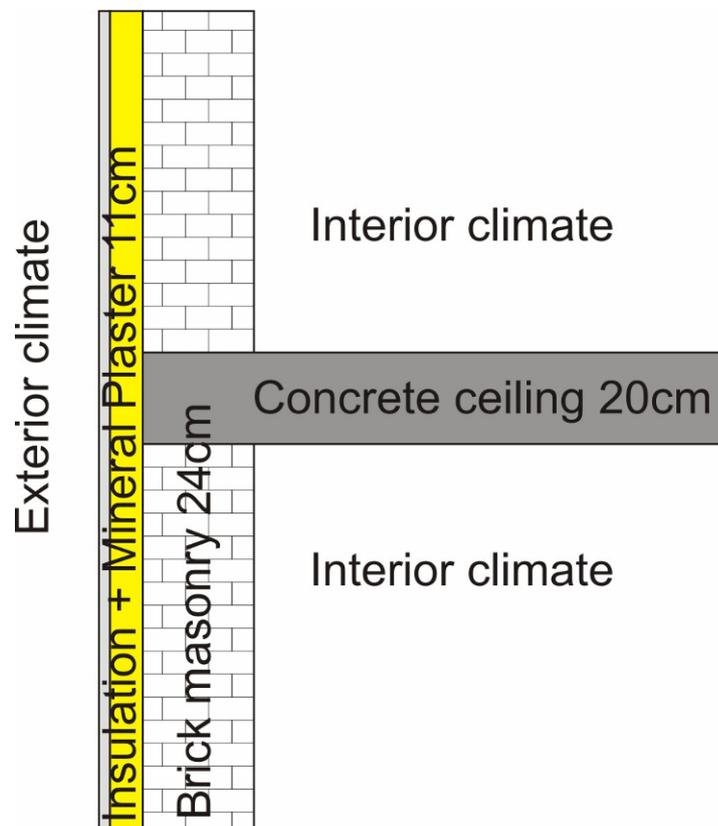
In order to model a new case, to perform the calculation and to analyse the results, please follow these nine steps:

- 1) You may give the case a name and input some notes concerning the project, the current project variant, the client etc.
- 2) You must define the geometry of the building component, that is, you build up the component from rectangular elements, using WUFI 2D's graphical component editor.
- 3) WUFI 2D automatically generates a numerical grid, subdividing the component into small grid elements for the numerical calculation. You may adjust the grid, if needed.
- 4) You must assign material data to the various elements making up the component. You may take the data from the material database or enter new data.
- 5) You must specify initial conditions for the component. For each material an initial temperature and an initial moisture content or relative humidity is needed.
- 6) You must define how the component is exposed to influences from the surroundings. This means assigning heat, vapor and liquid transfer coefficients, specifying inclination and orientation and selecting boundary conditions ("weather data") for all surface segments of the component.
- 7) You must specify a start date, the time period covered by the calculation and the time step width.
- 8) Let the calculation run.
- 9) Analyse the results.

Example Case

Step by step, we will work through the following example case:

A freshly cast concrete ceiling is supported by an outer masonry wall with exterior insulation system. The interior surfaces are assumed unfinished. Since no flooring is yet installed, there exists a symmetry axis along the center of the concrete slab. Only one half of the assembly has to be modeled therefore, reducing calculation time.



The building component consists of a 20 cm thick concrete ceiling supported by a 24 cm thick honeycomb brick wall. An exterior insulation and finish system (EIFS) composed of mineral wool insulation (10 cm thick) and mineral plaster (1 cm thick) has been applied to the outside of the wall.

The initial moisture content of the concrete, due to the mixing water, is assumed to be at free saturation (the equilibrium moisture at 100% RH).

Climate:

- Interior climate: Medium moisture load as defined by WTA-Recommendation 6-2-01/E
- Exterior climate: weather data from Holzkirchen, facade facing west, rain taken into account

Calculation period: 1 year, starting in October

The analysis shall consider in particular the temperature distribution in the component and the drying time of the concrete slab.

In the following screen shots this icon will be used to denote a mouse click (the finger points at the object to be clicked):



Step 1: Project notes

Dialog "Project Notes"

The screenshot shows the WUFI@2D 3.2 software interface. The main window is titled "WUFI@2D 3.2" and has a menu bar with "File", "Input", "Output", "Database", "Options", and "Help". Below the menu bar is a toolbar with various icons. On the left side, there is a "Project" tree view with the following items: Geometry, Grid, Materials, Initial Conditions, Surface/Climate, Computational Parameters, and Processing. The main area of the window is titled "Project: untitled" and contains a form for entering project information. The form has the following fields:

- Project name: Betondecke
- Project number: 12345-67
- Client: Mustermann AG
- Contact person: (empty)
- Street: (empty)
- Town / ZIP: (empty)
- Phone: (empty) Fax: (empty)
- e-mail: (empty)
- Person in charge: (empty)
- Remarks: (empty text area)
- Date: 16.04.2007 (dropdown menu)

At the bottom of the dialog, there is an orange button labeled "Project Notes".

This dialog serves as a memo pad for some administrative project info which can thus be kept together with the project data and the calculation results. All these inputs are optional, they only have an informative character for the user.

In the text field "Date" you may use the drop-down list to select a different date than the creation date of the project file (the first time it was saved).

Step 2: Geometry of the Component

Example Case

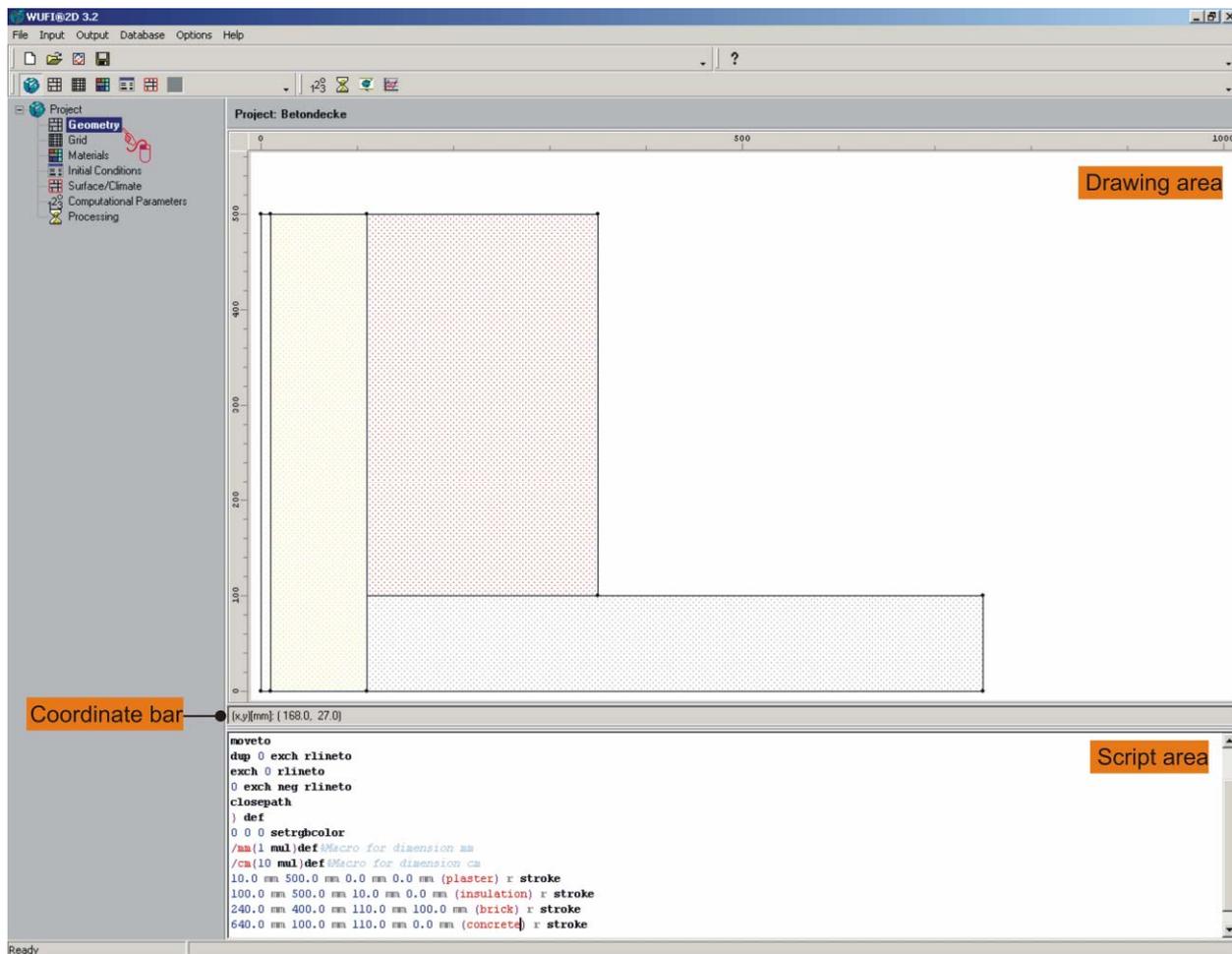
Use the graphical interface of this dialog to build up the component from rectangular elements. Each material is drawn as an individual rectangle; more complicated shapes can be composed of several rectangles.

In this case, we make use of the symmetry axis passing through the ceiling and only consider one half of the component. We need four rectangles: mineral plaster, insulation, brick masonry and concrete ceiling.

While you are drawing, WUFI 2D automatically creates a graphics script (displayed below the drawing area) describing your graphical input. You can edit the script to add elements, to give names to the elements, to refine coordinates, to rotate the component diagram etc. The script describing our example component looks like this:

```
10.0 mm 500.0 mm 0.0 mm 0.0 mm (plaster) r stroke  
100.0 mm 500.0 mm 10.0 mm 0.0 mm (insulation) r stroke  
240.0 mm 400.0 mm 110.0 mm 100.0 mm (brick masonry) r stroke  
640.0 mm 100.0 mm 110.0 mm 0.0 mm (concrete ceiling) r stroke
```

Dialog "Geometry"



Click on the item "Geometry" in the explorer tree on the left side of the main window to open the dialog panel shown above. The dialog contains the empty drawing area and the script area which already holds a few definitions.

Drawing Area

Use the drawing area to define the construction in a graphical way. Click into the area with the left mouse button, hold the button down and drag open a rectangle. For each rectangle you draw, a definition line will automatically be added to the script in the script area (see below). In order to add another rectangle, click on a corner of an existing rectangle and drag open a new one. New rectangles are always appended to existing ones.

Further functions in the drawing area:

Scroll

To scroll the diagram in the drawing area, press the **CTRL** key and the left mouse button, and drag the diagram into a convenient position.

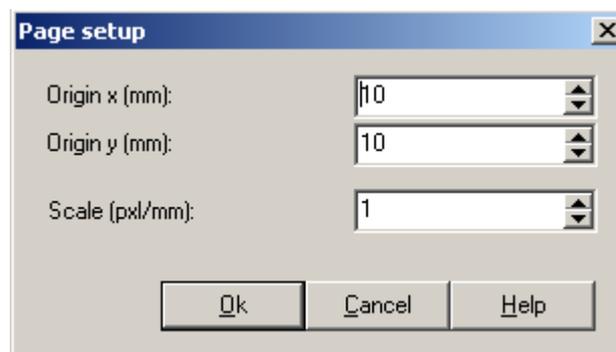
Zoom

To zoom into the diagram, press the `SHIFT` key and the left mouse button, and drag a zoom frame from the top left corner to the bottom right corner of the region to be zoomed.

In order to reset the zoomed view, press the `SHIFT` key and the left mouse button, and drag open a zoom frame of arbitrary size at an arbitrary location, moving from bottom right to top left.

Page setup

Right-click in the drawing area to open the page setup:



Here you can specify the default position of the coordinate origin in x- and y-direction, counted from the lower left corner of the drawing area.

„Scale“ defines the default magnification to which the drawing area returns when you zoom out. If the diagram becomes too large to be viewed in its entirety, you may specify a smaller value for the scale. Which value is appropriate depends on your screen resolution.

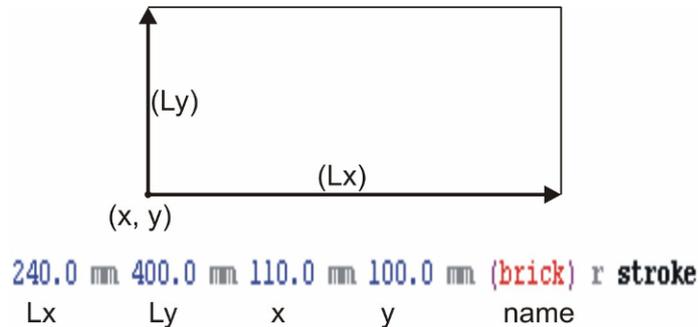
Coordinate Bar

When the mouse cursor is within the drawing area, its coordinates are displayed in the coordinate bar.

Script Area

The script area holds a script which describes the graphic you created in terms of drawing commands. Definitions for mm and cm are already in place, you may add your own definitions in PostScript style.

You can use the script to change the names and the sizes of the rectangles, which is useful for fine-tuning the construction. As soon as you have drawn a rectangle in the drawing area, the corresponding commands appear in the script area. A script line describing a rectangle looks as follows:



The first two numbers are the side lengths of the rectangle in x- and y-direction (including their units), followed by two numbers specifying the coordinates of its lower left corner, and the rectangle's name in parentheses. WUFI 2D assigns the names r1, r2 etc. to the rectangles, you may assign a more meaningful name by editing the script. A name must be unique and can only be used for one rectangle. The „r stroke“ terminates each line (the "r" macro, defined at the beginning of the script, processes the preceding numbers into a complete set of coordinates for the four rectangle corners, and the "stroke" command draws the rectangle).

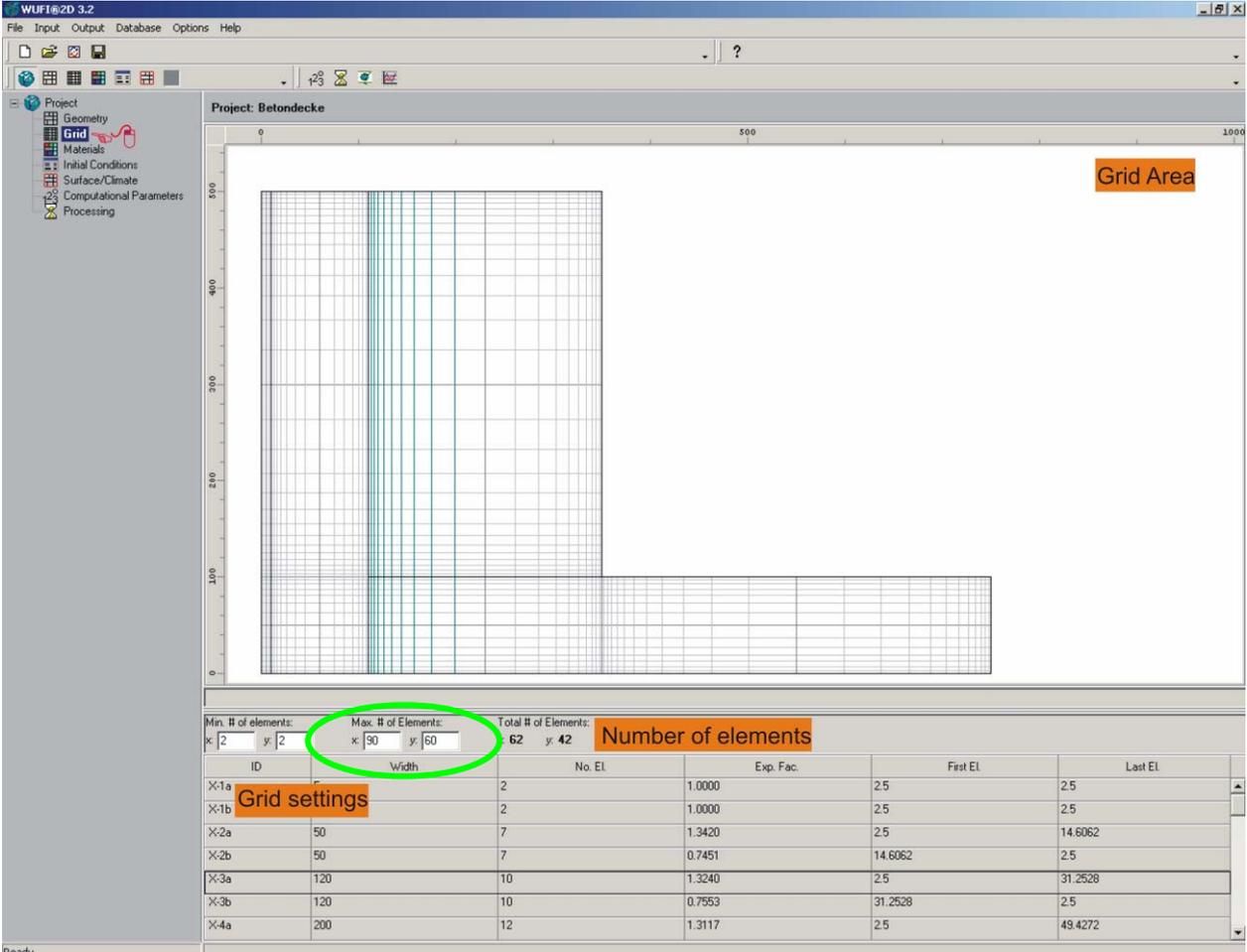
After a change to the script, press the **ENTER** key to update the diagram in the drawing area. Press **CTRL+ENTER** to insert a new line.

Step 3: Numerical Grid

Example Case

As soon as the dialog panel "Grid" is opened, WUFI 2D's automatic grid generator creates a numerical grid for the component. For the example case, an automatic grid is sufficient, but we increase the maximum number of grid elements. We set the maximum number of elements to 90 in x-direction and to 60 in y-direction (see screenshot below).

Dialog "Grid"



The screenshot shows the WUFI 2D 3.2 software interface. The main window displays a 2D grid overlaid on a construction geometry. The dialog panel at the bottom shows the 'Grid' settings. The 'Max. # of Elements' is set to x: 90 and y: 60. A table below the dialog shows grid settings for various elements.

ID	Width	No. El.	Exp. Fac.	First El.	Last El.
X-1a		2	1.0000	2.5	2.5
X-1b		2	1.0000	2.5	2.5
X-2a	50	7	1.3420	2.5	14.6062
X-2b	50	7	0.7451	14.6062	2.5
X-3a	120	10	1.3240	2.5	31.2528
X-3b	120	10	0.7553	31.2528	2.5
X-4a	200	12	1.3117	2.5	49.4272

Click on the explorer tree item "Grid" to open the dialog panel shown above. WUFI 2D displays the construction you defined in the previous step and overlays it with an automatically generated numerical grid. You may use this dialog to fine-tune the grid if needed.

Grid Area

The grid area shows a graphical display of the component, with the numerical grid superimposed for optical control. You may scroll or zoom using the `CTRL` or `SHIFT` key, respectively, as in the dialog "Geometry" (see above).

The numerical solver of WUFI 2D works most reliably if the grid is as fine as possible and if the sizes of two neighbouring grid elements are not different by more than, say, a factor of two. Towards the component boundaries and towards boundaries between different materials the grid should become finer because here steeper temperature and moisture profiles are to be expected which need to be resolved by a sufficient number of grid elements. At adiabatic boundaries, such as symmetry axes, this is not necessary, because here the temperature and moisture profiles tend to remain flat. The grid should try to strike a balance between a high number of grid elements and an acceptable calculation time.

Min. and Max. Number of Elements

The "minimum number of elements" is the minimum number of grid elements the grid generator is allowed to allot to a material. You may increase this number if you feel that a thin material in your component should have more grid elements but you do not want to make the whole grid finer.

This may be useful if you want the moisture profile across the material to be resolved more finely (e.g. in a moisture-adaptive vapor retarder), or if you want to reduce the effect that the properties of neighboring materials tend to bleed into thin materials because of the averaging WUFI 2D must perform between adjacent grid elements.

The "maximum number of elements" is the number of elements the grid generator has available for distributing across the component. For the fine grid the generator attempts to use the full number of elements, for the medium and coarse grids it uses proportionately less.

The minimum and maximum number of elements may be specified separately for the x- and the y-direction.

Grid Settings

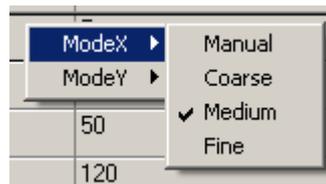
This table displays the properties of the individual grid sections. The sections are numbered serially in x-direction (from left to right) and in y-direction (from bottom

to top). The grid sections always come in pairs, „a“ and „b“; the grid is slightly expanding in section „a“ and contracting in section „b“.

ID	Width	No. El.	Exp. Fac.	First El.	Last El.
X-1a	5	2	1.0000	2.5	2.5
X-1b	5	2	1.0000	2.5	2.5
X-2a	50	7	1.3420	2.5	14.6062
X-2b	50	7	0.7451	14.6062	2.5
X-3a	120	10	1.3240	2.5	31.2528
X-3b	120	10	0.7553	31.2528	2.5

Each line of the table describes one of these sections of the grid. The column „ID“ identifies the grid section in question. „Width“ shows the width of the grid section, „No. El.“ gives the number of grid elements contained in the respective grid section. The expansion factor „Exp. Fac.“ determines by which factor two successive grid elements expand or contract. „First El.“ shows the width of the first element in the grid section, and „Last El.“ the width of the last element.

Right-click while the mouse is anywhere in the table in order to open a pop-up menu:



Here you can select the grid fineness for the x- and the y-direction:

Manual: The grid properties can be manually specified by entering the desired number of elements and the expansion factor for each grid section.

Coarse, Medium and Fine: A coarse, medium or fine grid will automatically be generated, depending on the minimum and maximum number of grid elements (see above).

Step 4: Material Properties

Example Case

For the present example, the following material properties are read from the material database and assigned to the different elements of the building component:

- Mineral plaster
- Mineral wool (heat conductivity: 0,04 W/mK)
- Aerated clay brick (density: 600 kg/m³)
- Concrete C35/45.

Dialog "Materials"

The screenshot shows the WUFI@2D 3.2 software interface. The main window displays a 2D cross-section of a wall component. The wall is composed of several layers: a thin yellow layer (plaster) on the left, a thick grey layer (mineral wool) in the middle, and a thick grey layer (concrete) at the bottom. The top of the wall is a thin grey layer (aerated clay brick). The vertical axis is labeled in mm, ranging from 0 to 500. The horizontal axis is labeled in mm, ranging from 0 to 1000. The project name is 'Betondecke'. Below the diagram is a table of material assignments.

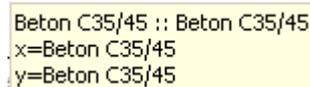
Name	X-Material	Edit	Y-Material	Edit
plaster	Mineral Plaster (stucco)	Edit...	Mineral Plaster (stucco)	Edit...
insulation	Mineral Wool (heat cond: 0,04 W/mK)	Edit...	Mineral Wool (heat cond: 0,04 W/mK)	Edit...
brick	Aerated Clay Brick (density: 600 kg/m ³)	Edit...	Aerated Clay Brick (density: 600 kg/m ³)	Edit...
concrete	Concrete, C35/45	Edit...	Concrete, C35/45	Edit...

Click on the explorer tree item "Materials" to open the dialog panel shown above. It shows the building component as defined previously. Click on one of the rectangles to assign the material properties.

The material properties in x-direction may be the same as in y-direction (isotropic material), or they may be different (anisotropic material):

1. If no material has yet been assigned to the rectangle, a click with the *left mouse* button will assign the material specified in the next dialog *both to the x- and the y-direction*.
2. Otherwise, a click with the *left mouse* button will assign the material to the *x-direction*...
3. ...and a click with the *right mouse* button will assign the material to the *y-direction*.

The list displayed below the construction diagram shows the materials assigned to the construction. Alternatively, you may hover with the mouse above a rectangle to pop up a little window showing the material(s) assigned to this rectangle:

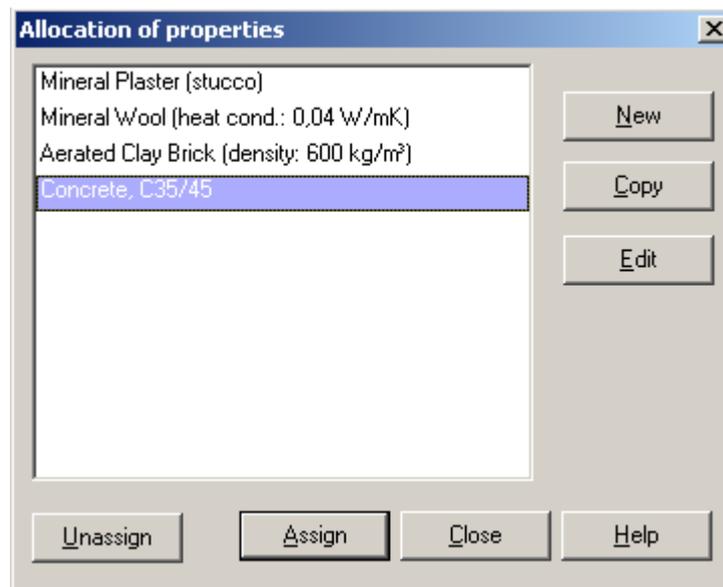


```
Beton C35/45 :: Beton C35/45  
x=Beton C35/45  
y=Beton C35/45
```

The left- or right-click on the rectangle opens the dialog „Allocation of properties“.

Dialog "Allocation of Properties"

This dialog lists the materials that have already been allocated to one of the rectangles. You may re-use materials from the list or add new materials. Initially the list is empty.

**Buttons:**

- New:** Add a new material to the list. WUFI 2D opens the dialog „Material Database“ where you can select the material (see below).
- Copy:** Create a copy of the highlighted material in order to edit it or to use an independent copy of the material. Use independent copies if you want to assign different initial conditions (see below) to elements containing the same material.
- Edit:** WUFI 2D opens the dialog "Material Properties" for editing the highlighted material (see below).
- Unassign:** Removes the assigned material from an element (separately for x- and y-direction, see above).
- Assign:** Assigns the material to the rectangular construction element (for x- and/or y-direction, see above).
- Close:** Closes the dialog.
- Help:** Opens the online help for this dialog.

Dialog "Material Database"

The dialog "Material Database" displays the materials contained in the material database and offers them for selection. (If you want to add a new material to the database, use the database editor which you can access via the menu „Database → Materials...“.)

WUFI: Material database

Source: Fraunhofer Institut für Bauphysik (IBP) - Holzkirchen, Deutschland

Catalog: All Catalogues

Name	Den. [kg/m³]	Ther. Cond. [W/mK]	Diff. Res. Fac. [-]	Free Sat. [kg/m³]	Poro. [m³/m³]	H.Cap. [J/kgK]
Number of Materials: 104						
Fired Clay Brick (used for cellars)	800			33	0.6	850
Gypsum Board	850			30	0.65	850
Hardwood	650	0.13	200	370	0.47	1500
HOIZ® S 45 Wood Shavings	65	0.045	2.5	426	0.95	2100
INTELLO	115	2.4	2.6E4	84	0.086	2500
Interior Plaster (Gypsum Plaster)	850	0.2	8.3	400	0.65	850
KlimatexFlock cellulose fiber	50	0.038	1.8	426	0.95	2000
Krensheimer Shelly Limestone	2440	2.25	140	75	0.13	850
Laminated veneer lumber	462	0.13	156	525	0.63	2500
Light Expanded Clay Aggregate	700	0.13	7	97	0.67	850
Lime Cement Plaster (stucco)	1900	0.8	19	210	0.24	850
Lime Plaster (for salt extraction)	1900	0.7	12	186	0.33	850
Lime Plaster (stucco)	1600	0.7	7	250	0.3	850
Lime Silica Brick (density: 1900 kg/m³)	1900	1	28	250	0.29	850
medium density fibreboard #1	528	0.1	12	667	0.8	2000
medium density fibreboard #2	508	0.12	15	667	0.67	1700
Mineral Insulation Board	115	0.043	3.4	667	0.95	850
Mineral Plaster (stucco)	1900	0.8	25	210	0.24	850
Mineral Wool (heat cond.: 0.04 W/mK)	60	0.04	1.3	210	0.95	850
Oak, longitudinal	685	0.3	8	500	0.72	1500
Oak, radial	685	0.13	140	500	0.72	1500
Oberkirchner Sandstone	2150	2.3	32	110	0.14	850
Oriented Strand Board (density 553 kg/m³)	553	0.12	134	610	0.61	1700
Oriented Strand Board (density 595 kg/m³)	595	0.13	165	814	0.9	1500
Oriented Strand Board (density 615 kg/m³)	615	0.13	175	636	0.9	1500
Oriented Strand Board (density: 630 kg/m³)	630	0.13	650	636	0.6	1500
PA-Membrane	65	2.9	4.38E3	11.2	0.001	2300

Info-Text: A-Value 0.1 kg/m²h0.5 (SI-Units) - from H.M. Künzel: Austrocknung von Wandkonstruktionen mit WDVS

Thickness [cm]: 0.01, 0.02

Optional Data: Moisture Storage Function, Liquid Transport Coefficient, Suction, Thermal Conductivity, moisture-dep, Water Vapour Diffusion Resistance

Show optional data: Assign, Help, Close

In order to read a material from the database,

1. Select the respective source and catalog.
2. Select the desired material.
3. Press the „Assign“ button to add the material to the materials list in the dialog „Allocation of properties“.

For each material, the basic material properties are displayed in the columns after the material name. The remaining properties are shown as "Hygic Extensions" if the option „Show optional data“ is checked.

Dialog "Material Properties"

This dialog displays the properties of a material. You can view and edit the properties. If you want to edit the data, you first have to unlock the material by clicking on the padlock icon. Once a material has been unlocked, its name begins with an asterisk (*) to distinguish it from the original material in the database.

Material Properties X

Material Name: Aerated Clay Brick (density: 600 kg/m³)

Material Data | Info | Details

Basic Values

Bulk Density [kg/m ³]	600
Porosity [m ³ /m ³]	0.77
Spec. Heat Capacity [J/kgK]	850
Thermal Conductivity dry [W/mK]	0.12
Water Vapour Diffusion Resistance Factor dry [-]	16

Generation of the Moisture Storage Function

	RH [-]	W.Cont. [kg/m ³]
Reference Water Content	0.8	11
Free Water Saturation	1.0	188

Generation Parameter

Thermal Conductivity Supplement [%/M.-%]	10
--	----

Additional

Material Color	
Typical Built-In Moisture [kg/m ³]	100

Moisture Storage Function

Liquid Transport Coefficient
Thermal Conductivity, moisture-dependent
Water Vapour Diffusion Resistance Factor, moisture-depend

Graph | Table

Generate Simple
 Enhanced

Water Content [kg/m³]

Relative Humidity [-]

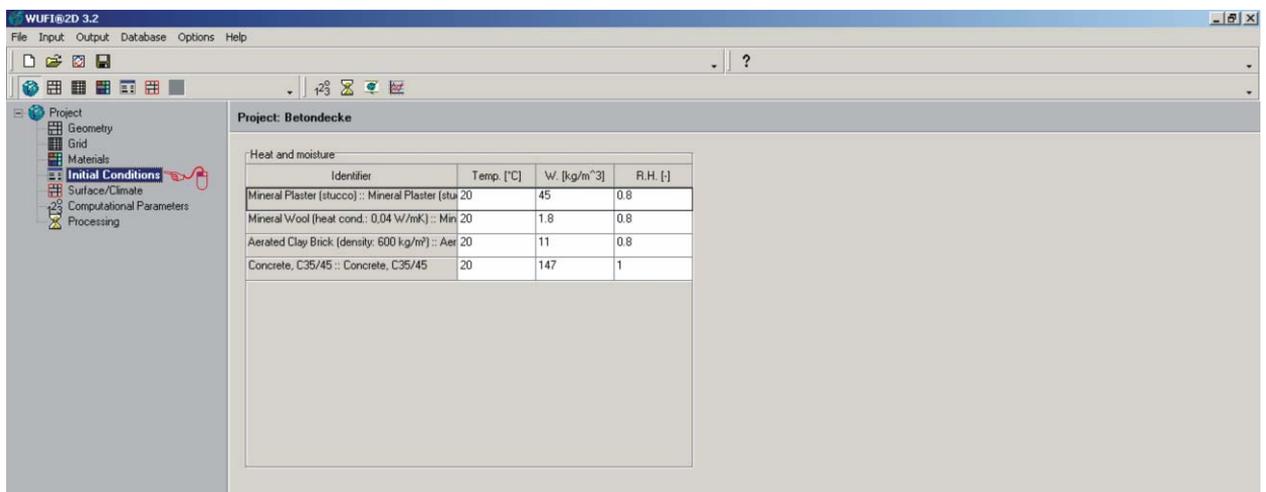
? Help OK Cancel

Step 5: Initial Conditions

Example Case

In our example case, we set the initial water content of the concrete to free saturation (147 kg/m^3), and leave the other materials unchanged.

Dialog "Initial Conditions"



Click on the explorer tree item "Initial Conditions" to open the dialog panel shown above. For each material assigned to the component, you can specify the initial temperature and the initial water content or relative humidity. (All elements of the component with the same material will start with the same initial conditions. If you want to specify *different* initial conditions for elements with the same material, assign *copies* of the material to them; see the dialog "Allocation of Properties" above).

Since the thermal conditions in a building component usually adapt very quickly to the boundary conditions, it is not necessary in most cases to specify elaborate initial temperatures. Within a few hours, the thermal conditions will usually have become independent of the initial temperatures anyway. The predefined initial temperature of 20 °C within the whole component will usually be good enough. If you simulate short-term processes (e.g. laboratory tests), it may be necessary to give more thought to the initial temperatures.

Step 6: Boundary Conditions

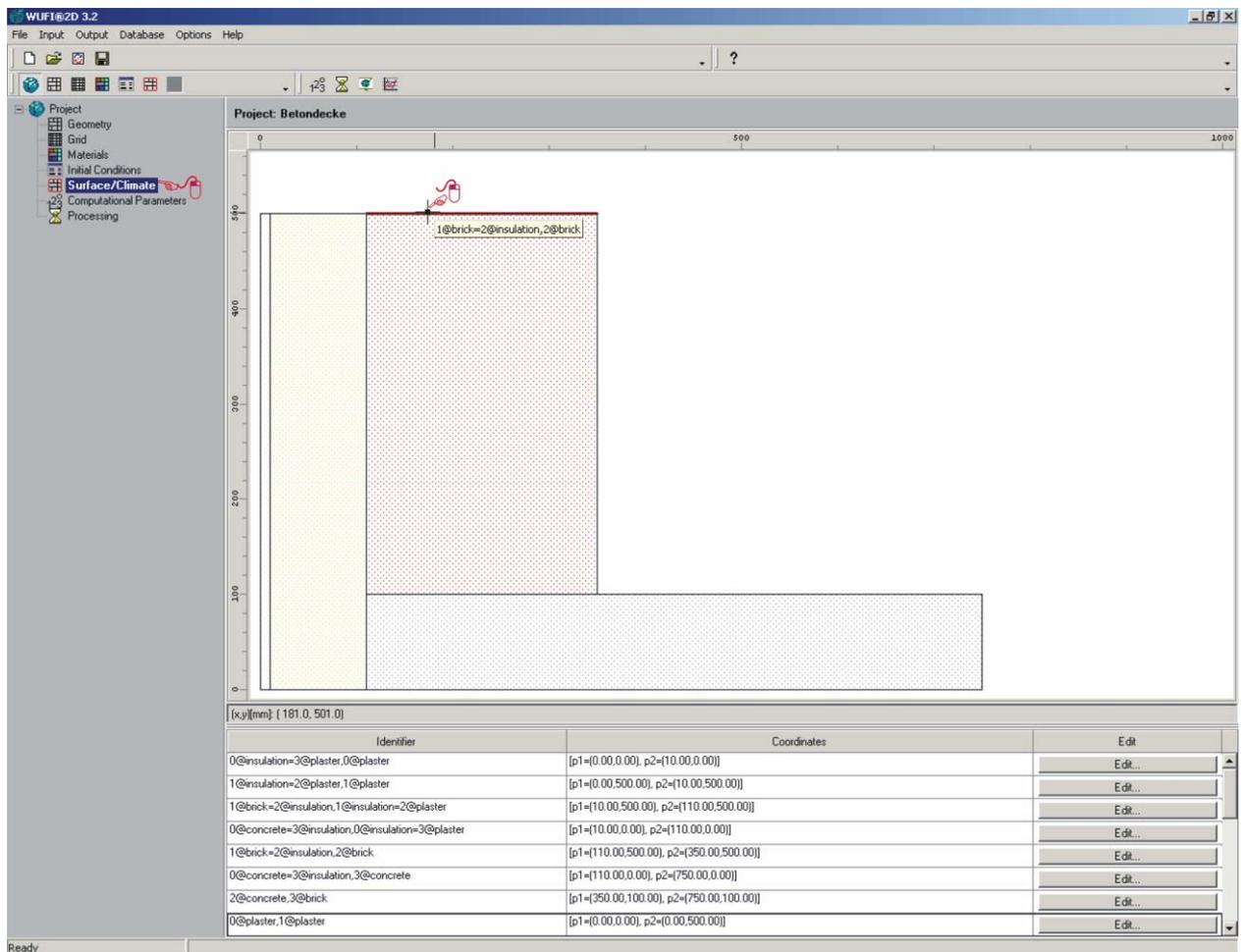
Example Case

The data necessary for defining the boundary conditions in our example are listed in the following table:

	Ext. climate	Int. climate	Adiabatic
Surface Coefficients			
Adiabatic/ System Border			X
Treat as Inner Surface		X	
Heat Transfer Coefficient [W/m ² K]	17	8	
Short-Wave Radiation Absorptivity [-]	0.4 (Bright plaster)		
Long-Wave Radiation Emissivity [-]	0.9		
Rain Water Absorption Factor [-]	0.7		
Color in Model	●	●	●
Climate File	Holzkirchen	WTA	
Azimuth [deg]	90	-	
Inclination [deg]	90	-	

For where and how to enter these boundary conditions see the following sections.

Dialog "Surface/Climate"



Click on the explorer tree item "Surface/Climate" to open the dialog panel shown above. Here boundary conditions must be specified for each surface segment of the building component.

In the example shown here, the exterior facade is exposed to the Holzkirchen weather, including rain. The symmetry axis and the borders where the continuing component has been cut off are treated as adiabatic "surfaces" (i.e. no heat or moisture can flow across them).

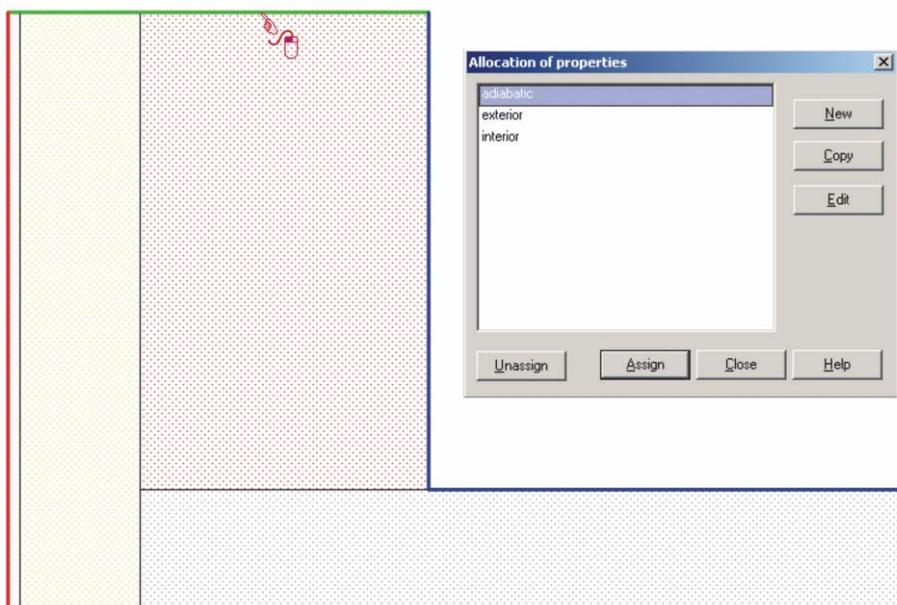
Left-click on one of the surface segments to open the dialog „Allocation of properties“.

Dialog "Allocation of Properties"

This dialog lists the boundary conditions that have already been allocated to one of the surface segments. You may re-use boundary conditions from the list or add new ones. Initially the list is empty.

In order to add a new set of boundary conditions click on „New“. A new list item is added, with a name automatically generated by the program (e.g. 1@brick=2@insulation,2@brick). If you want to replace the name by something more intelligible (like "exterior facade"), double-click on it to change it.

In order to edit the boundary conditions themselves, click on the button "Edit".



The screenshot above shows the building component and the dialog „Allocation of properties“ with the three sets of boundary conditions used in the example (the "adiabatic" set of boundary conditions has been assigned to the three surface segments marked green in the diagram, the "exterior" conditions to the red segment and the "interior" conditions to the two blue segments).

Buttons:

New: Adds a new set of boundary conditions to the list. The program assigns a name describing the surface segment for which you created the list entry.

Copy: Creates a copy of the highlighted list item. This may be useful if you want to create a set of boundary conditions which differs only little from an existing set.

Edit: Opens the dialog „Surface/ Climate“ for editing the highlighted set of boundary conditions (see below).

Unassign: Unassigns the highlighted set of boundary conditions from the respective surface segment.

Assign: Assigns the highlighted set of boundary conditions to the previously selected surface segment.

Close: Closes the dialog.

Help: Opens the online help for this dialog.

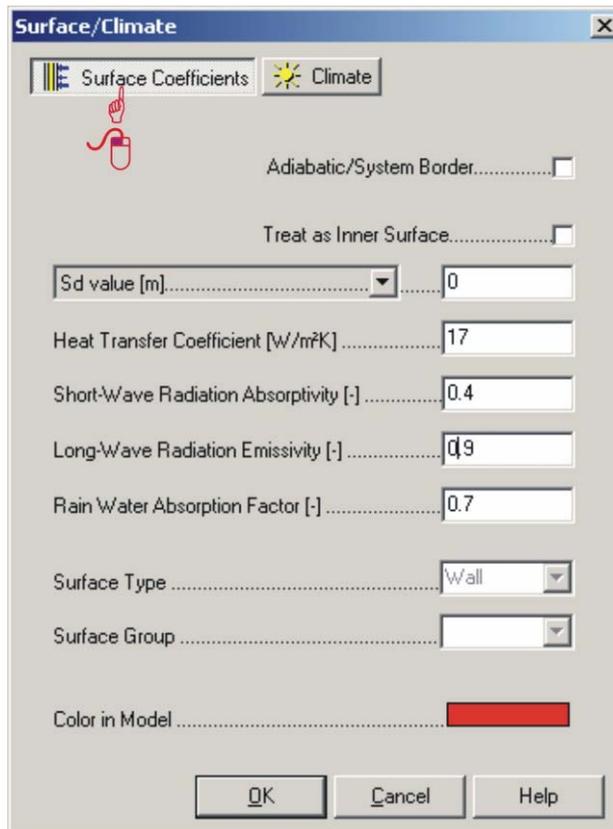
TIP: Double-click on a list item to edit its name.

TIP: The two surface segments meeting at the corner of a house often are assigned the same boundary conditions, except that they are facing different directions. In such a case, instead of creating a new and empty set of boundary conditions with „New“, „Copy“ the existing one and simply „Edit“ it to change the direction the facade is facing.

Dialog "Surface/ Climate"

Click on the button „Edit“ in the dialog „Allocation of properties“ to open the dialog shown below. This dialog consists of one page for the "Surface Coefficients" and one page for the "Climate" acting on the surface.

Surface Coefficients



The screenshot shows the settings for the "exterior" surface in our example. In general, the following settings can be edited:

- Adiabatic/System Border:** This option makes the surface an adiabatic system border. The surface is assigned infinite resistances for heat and moisture flows and is thus made impermeable to heat and moisture exchange. The main application for this option are symmetry axes, since no heat or moisture can be exchanged across these. If the option is checked, no further boundary conditions need to be specified.
- Treat as Inner Surface:** This option specifies the surface as an interior (room-side) surface. WUFI 2D automatically com-

puts the surface transfer coefficient for vapor flow from the surface transfer coefficient for heat flow (unless overridden, see below), and the details of this computation depend on whether the surface is an interior or an exterior surface.

Sd-Value [m] or Vapor Transfer Coefficient [$\text{kg/m}^2\text{sPa}$]:

WUFI 2D allows two different ways to specify the diffusion resistance at the surface:

s_d -value:

Allows to model a diffusion-impeding surface layer (if any, e.g. wallpaper, tiles, vapor barrier...) by specifying its s_d -value, thus avoiding the need to include the layer explicitly in the component model.

The natural diffusion resistance caused by the boundary air layer is automatically computed and added by WUFI 2D.

Vapor transfer coefficient:

Allows to specify the *total* diffusion resistance of the surface (the natural resistance plus the influence of a surface layer, if any) if you wish to use a different value than what is automatically determined by WUFI 2D.

Add all resistances you wish to account for and take the reciprocal to obtain the coefficient.

In most cases (using standard values for the natural diffusion resistance and no surface layer), simply enter s_d -value=0. For further details, please refer to the online help.

Heat Transfer Coefficient [$\text{W/m}^2\text{K}$]:

The heat transfer coefficient of the surface.

Short-Wave Radiation Absorptivity [-]:

The fraction of incident solar radiation absorbed by the surface.

Long-Wave Radiation Emissivity [-]:

The emissivity of the surface for thermal radiation. We recommend to set this number to zero; for further information please refer to the online help.

Rain Water Absorption Factor [-]:

This reduction factor takes into account that some of the rain water hitting the surface splashes off on impact and is not available for capillary absorption. For vertical walls, a value of 0.7 is usually adequate.

Surface Type:

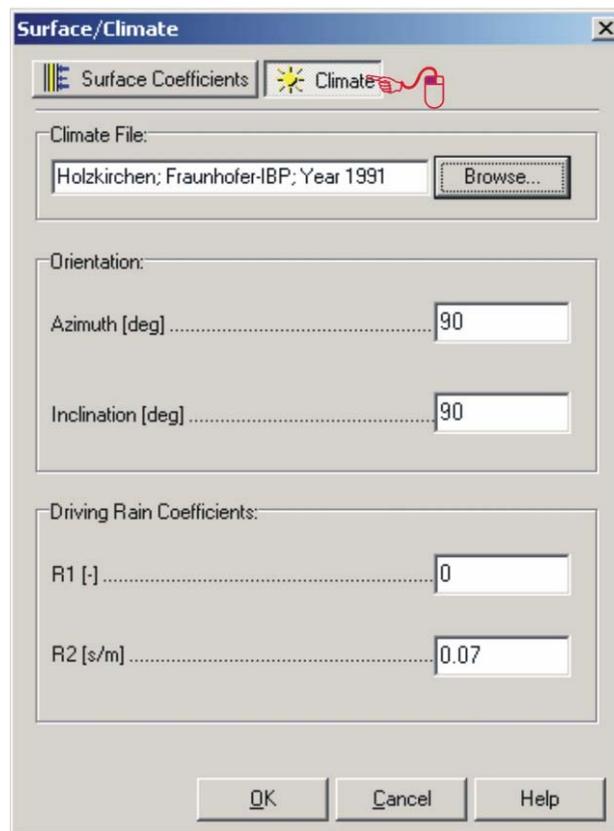
Surface Group:

These options are not used in the current version of WUFI 2D

Color in Model:

Select the color with which the surface segment being edited shall be displayed in the component diagram.

Climate



The screenshot shows the settings for the "exterior" surface in our example. In general, the following settings can be edited:

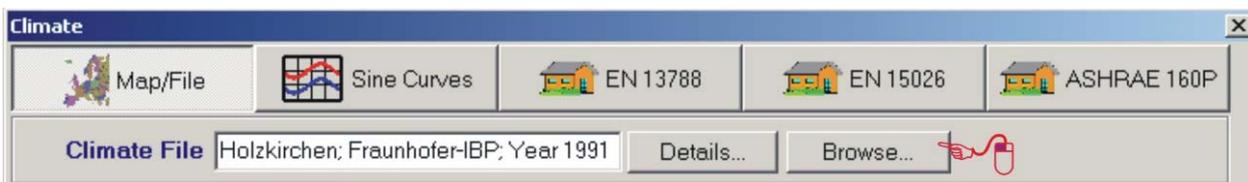
- | | |
|---------------------------|--|
| Climate File | Specifies the climate acting on the surface segment. Click on the „Browse...“ button to select a climate file or climate specification in the dialog „Climate“ (see below). |
| Azimuth [deg] | The compass direction towards which the surface is facing. South is 0°, 90° is west, -90° is east and ±180° is north. Intermediate values are allowed. |
| Inclination [deg] | The tilt of the surface. A vertical surface has a tilt of 90°, a flat roof has a tilt of 0°. |
| Driving Rain Coefficients | These coefficients are used to estimate the driving rain load on the surface. The driving rain load is estimated as
rain load = normal rain*(R ₁ +R ₂ * wind speed).
For details on R1 and R2 please refer to the online help. |

Dialog "Climate"

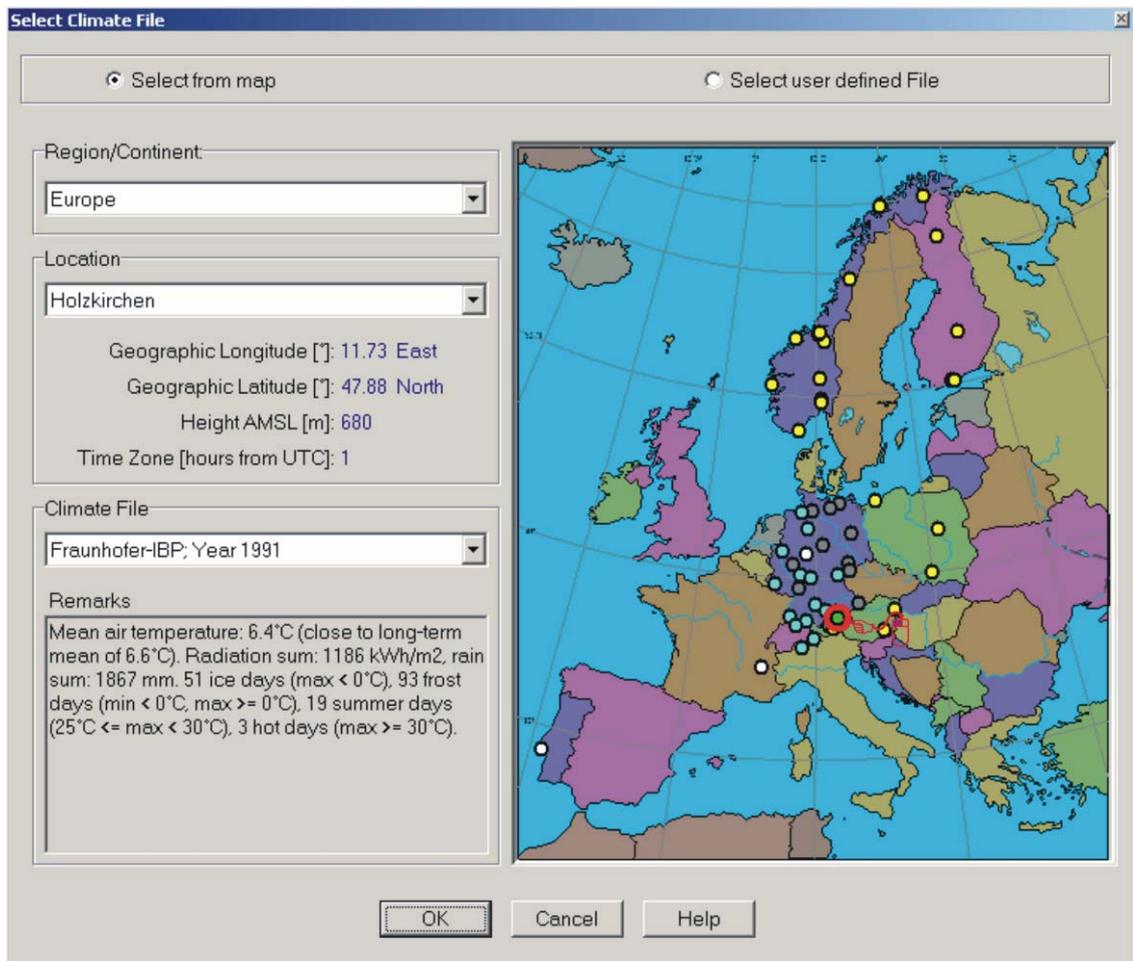
The „Browse...“ button on the "Climate" page of the dialog „Surface/Climate“ opens the following dialog where you can select the appropriate exterior or interior climate that shall act on the surface currently being edited. You may select cities from a map or specify a climate file provided by yourself. Several simplified climate models (mostly for indoor climates) as defined by several standards can be used as well.

Exterior climate (accessible via the tab „Map/File“)

A number of files with measured exterior climate data for various cities is provided with WUFI 2D; you may also use climate files supplied by yourself. WUFI 2D's files are simply selected from a map showing the respective cities; your own files are selected with from a file browser. First click on the tab „Map/File“, then on the button „Browse...“.



The „Browse...“ button opens the dialog „Select Climate File“.

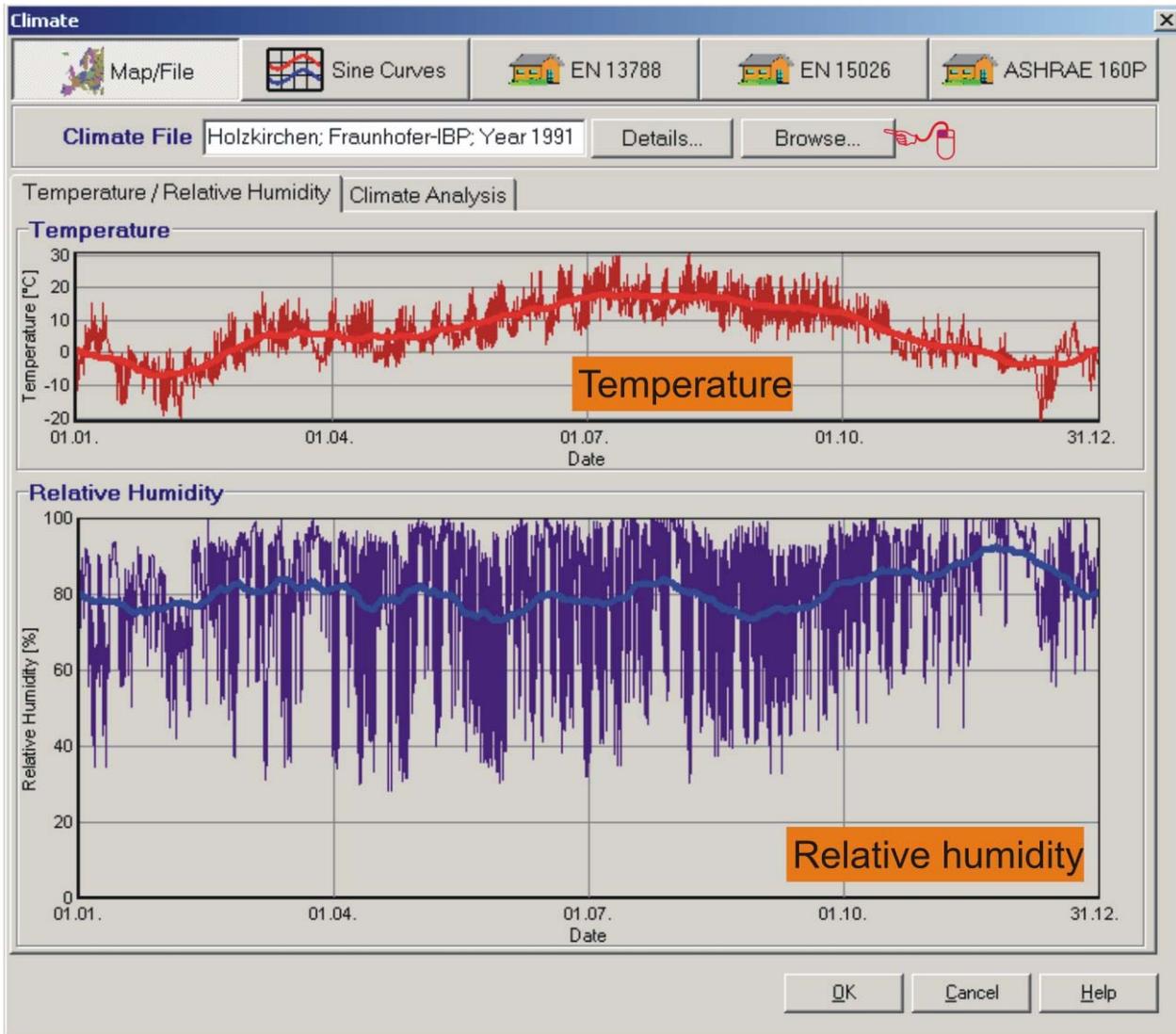


Use this dialog to either select from the map a city for which WUFI 2D provides climate data, or to open a file browser for selecting a climate file from a different source.

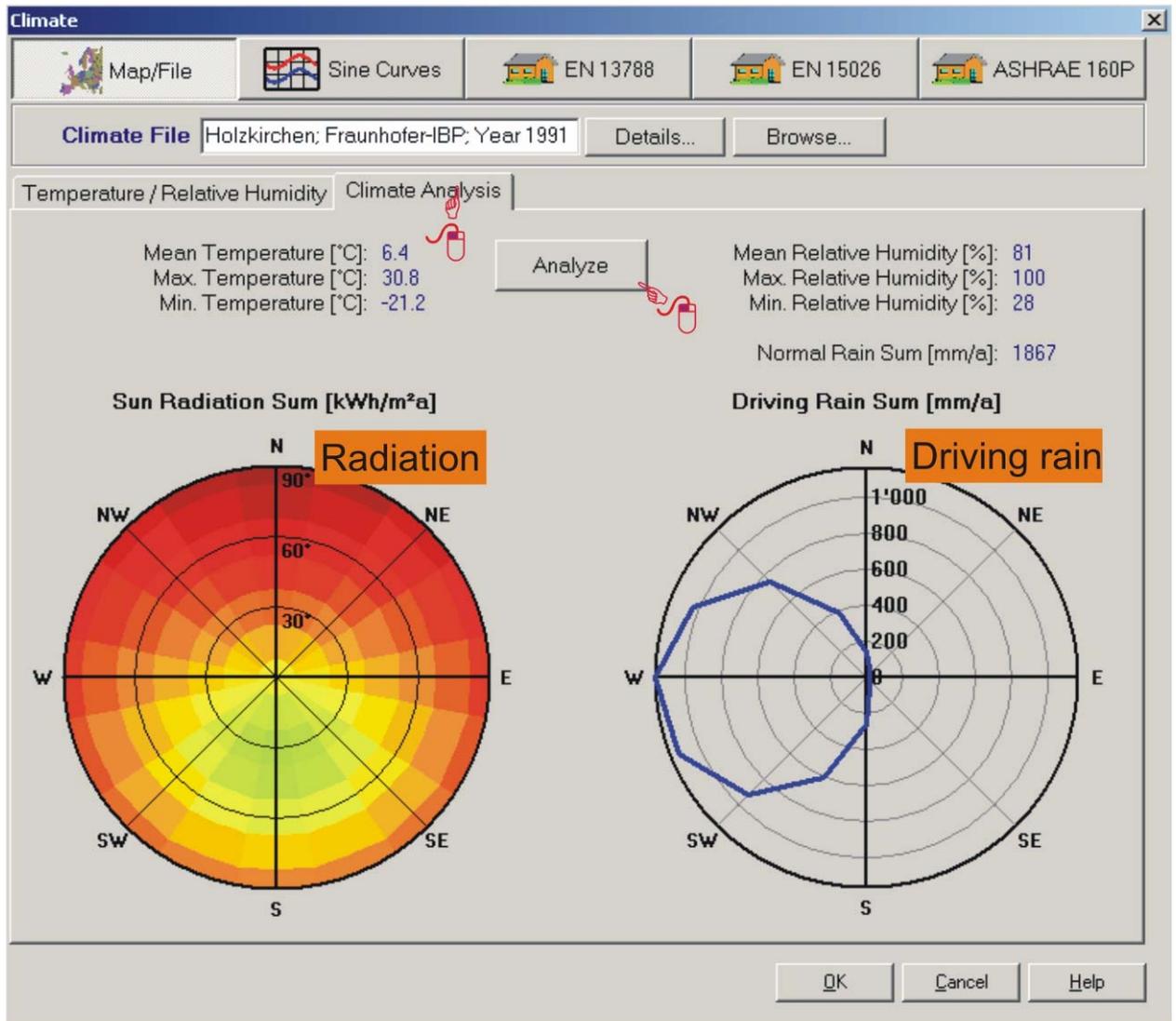
Switch between different maps using the drop-down list "Region/Continent". Select the desired city either from the drop-down list "Location" or by directly clicking on it in the map (our example uses the location Holzkirchen). The colored dots mark cities for which WUFI 2D provides climate data. The grey dots mark cities whose geographic data have been registered in WUFI 2D's database but whose climate data are not included with WUFI 2D. These data can be purchased separately, for example the Test Reference Years of the German National Meteorological Service. If you have acquired such a file, just copy it into WUFI 2D's climate directory.

Once you have selected the climate file, close the dialog by clicking on "OK".

When a climate file has been selected, WUFI 2D automatically displays its temperatures and relative humidities on the tab „Temperature/Relative Humidity“. The diagrams show the hourly values (thin curves) and moving monthly means (bold curves) of temperature (top, red) and relative humidity (bottom, blue) for the time period covered by the climate file.



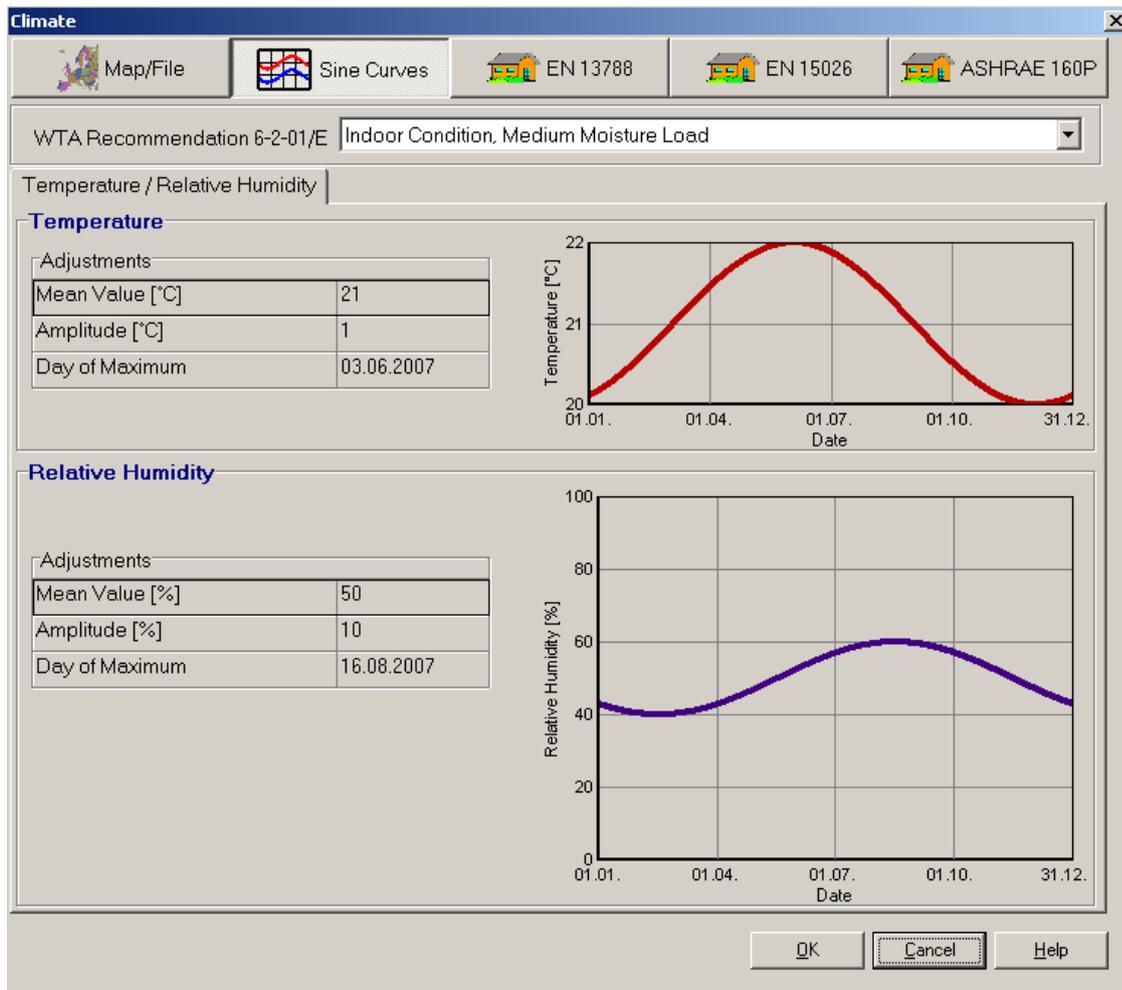
WUFI 2D also offers analyses of average and extreme values for temperature and relative humidity and the distributions of solar radiation and driving rain. Access these by clicking on the tab „Climate Analysis“.



Since this analysis takes a while, it only starts when you click on „Analyze“. After a few seconds WUFI 2D displays the mean, maximum and minimum temperatures in the top left corner of the page and, in the radiation rose below, the yearly sum of the solar radiation as incident on surfaces with various orientations and inclinations. Low energy input is represented by dark red colors, high values by bright blue-green colors. The color scale is always the same, independent of the location, so that you can discern immediately whether this is a location with low or high solar radiation load.

On the right side, WUFI 2D displays some statistics for the relative humidity and the yearly sum of the normal rain. Below that, the driving rain rose shows the yearly sum of driving rain as incident on vertical surfaces with various orientations. For each location, the driving rain scale is individually adapted to the maximum amount encountered there. As for the normal rain, the amount of driving rain is given in mm/year.

Interior climate (sine curves, interior climate models)



For the interior climate, WUFI 2D offers simple sine curves and several interior climate models.

Click on the tab „Sine Curve“ to specify simple sine-shaped curves with periods of one year for temperature and relative humidity, with maximum values in summer and minimum values in winter. You may use the drop-down list to select predefined curves for the exterior climate and for interior climates for low, medium or high moisture load, as recommended in WTA Guideline 6-2-01/E „Simulation of Heat and Moisture Transfer“.

Select „User-defined sine curve parameters“ from the drop-down list if you wish to define your own mean values and amplitudes for the sine curves. Check the option "constant" to specify constant conditions (e.g. as a simple model for air-conditioned spaces). For most calculations the standard settings for average living space (medium moisture load, temperature 21 °C with amplitude 1 °C) are reasonable and sufficient. These settings are also used for our example case.

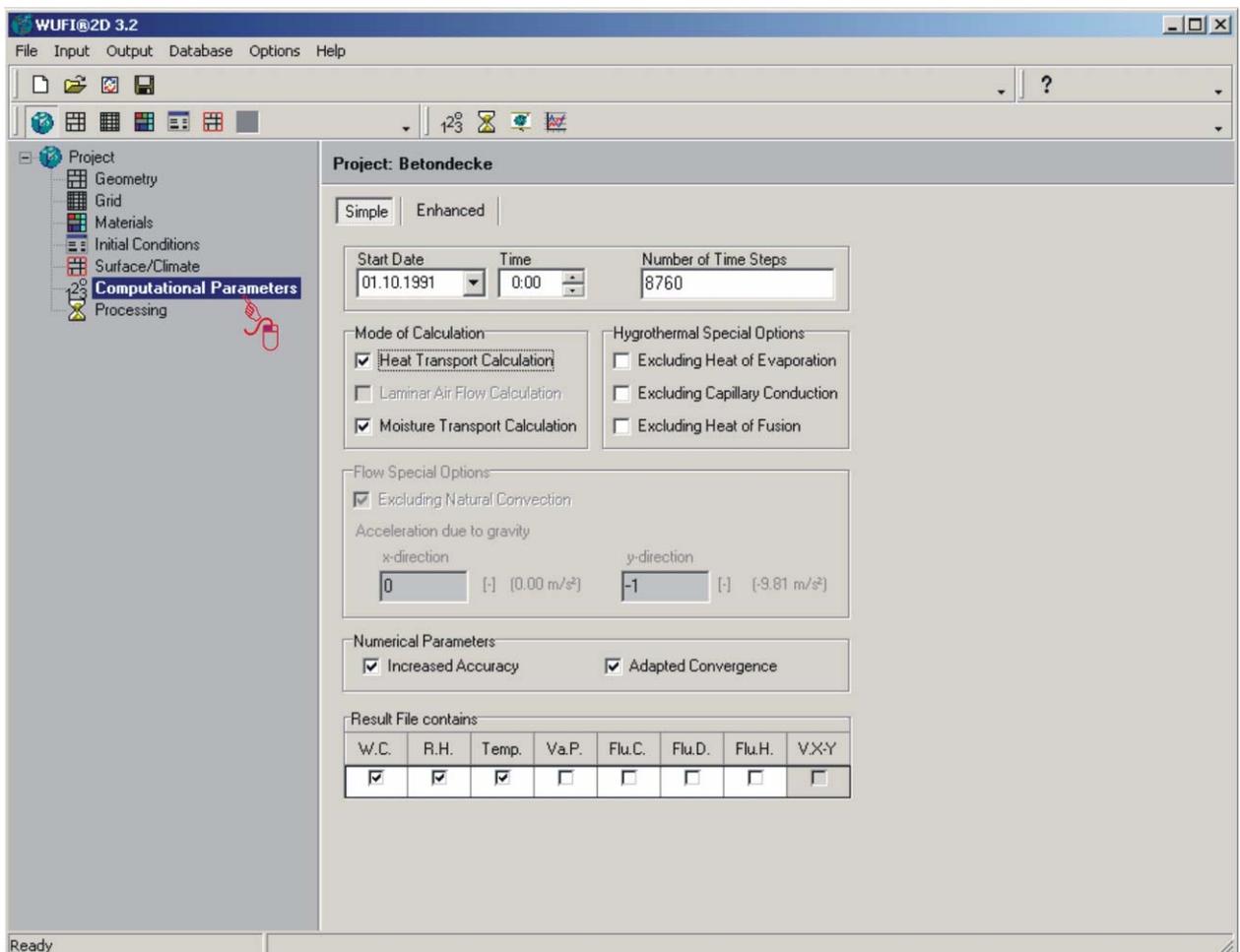
Step 7: Computational Parameters

Example Case

The example case shall be computed for a period of 8760 hours - that is, one year. The start date is 01.10.1991. 1991 is the year of the Holzkirchen weather file, but the year number is irrelevant and ignored by WUFI 2D. We want WUFI 2D to write water content, relative humidity and temperature to the result file.

Dialog "Computational Parameters"

Click on the explorer tree item "Computational Parameters" to open the dialog panel shown below. Here you specify the details of how the computation is to be performed.



Adapted Convergence

Use stronger convergence damping (increases the computation time).

Result File Contains

Specify here which computed quantities shall be written to the result file. Make sure all those quantities are checked which you wish to analyse later with WUFI2DMotion or WUFIGraph.

W.C.	Water Content [kg/m^3]
R.H.	Relative Humidity [0..1]
Temp.	Temperature [$^{\circ}\text{C}$]
Va.P.	Vapor Pressure [hPa]
Flu.C.	Capillary Flux [$\text{kg}/\text{m}^2\text{s}$]
Flu.D.	Diffusion Flux [$\text{kg}/\text{m}^2\text{s}$]
Flu.H.	Heat Flux [$\text{J}/\text{m}^2\text{s}$]
V.X-Y	not used in this WUFI 2D version.

Intermediate Step: Save project

In order to save the input you have entered so far, choose File → Save Project As... or click on the disk symbol in the toolbar. This opens a dialog showing a directory tree. The root of the tree is the „Home Directory“ specified in Options → Settings (see the relevant section in the introduction manual).

WUFI 2D saves the project data into several files in a project directory. It suffices therefore to select the desired location in the directory tree and to create there a folder with an appropriate name.

In order to reload the project later, you just need to specify the desired project directory; WUFI 2D will know which files to load.

Step 8: Computation

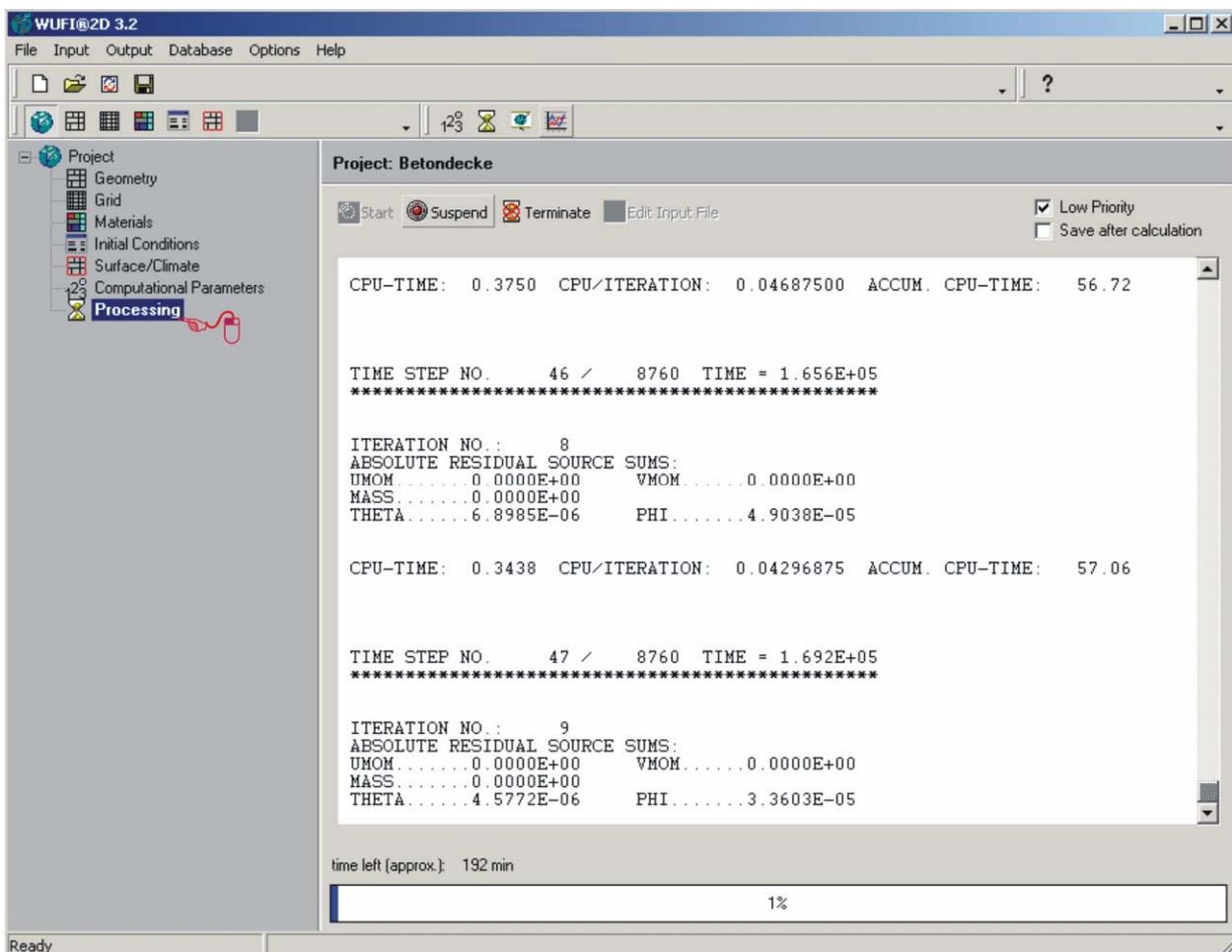
In this step the computation of the transient thermal and hygric transport processes in the component is performed. This will take a while. Depending on the size of the construction, the fineness of the numerical grid, intensity of the transport processes and number of time steps, the computation may require hours or even days.

Example Case

Click on "Processing" in the explorer tree, then click on the "Start" icon.

Dialog "Processing"

Click on the explorer tree item "Processing" to open the dialog panel shown below. WUFI 2D automatically creates an input file (input.dat) for the numerical core, which may take a few seconds.



The „Processing“ dialog offers these functions:

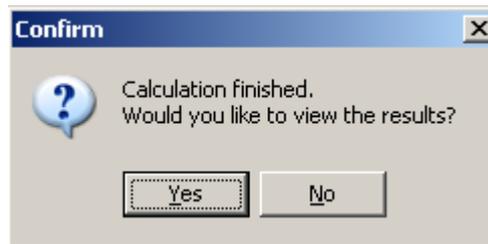
 Start	Start the calculation
 Resume	Resume a suspended calculation
 Suspend	Suspend the current calculation
 Terminate	Terminate the current calculation
 Edit Input File	Edit the input file for the numerical core (recommended for experts only)

Low Priority Assigns low priority to the computation process. Other programs executed on the computer are less sluggish.

Save after calculation

Saves the project (and the results) when the calculation has finished.

After the calculation has finished, a "confirm" dialog asks you whether you wish to view the results.



Click "Yes" to launch the analysis tool WUFI2DMotion which displays the computed quantities.

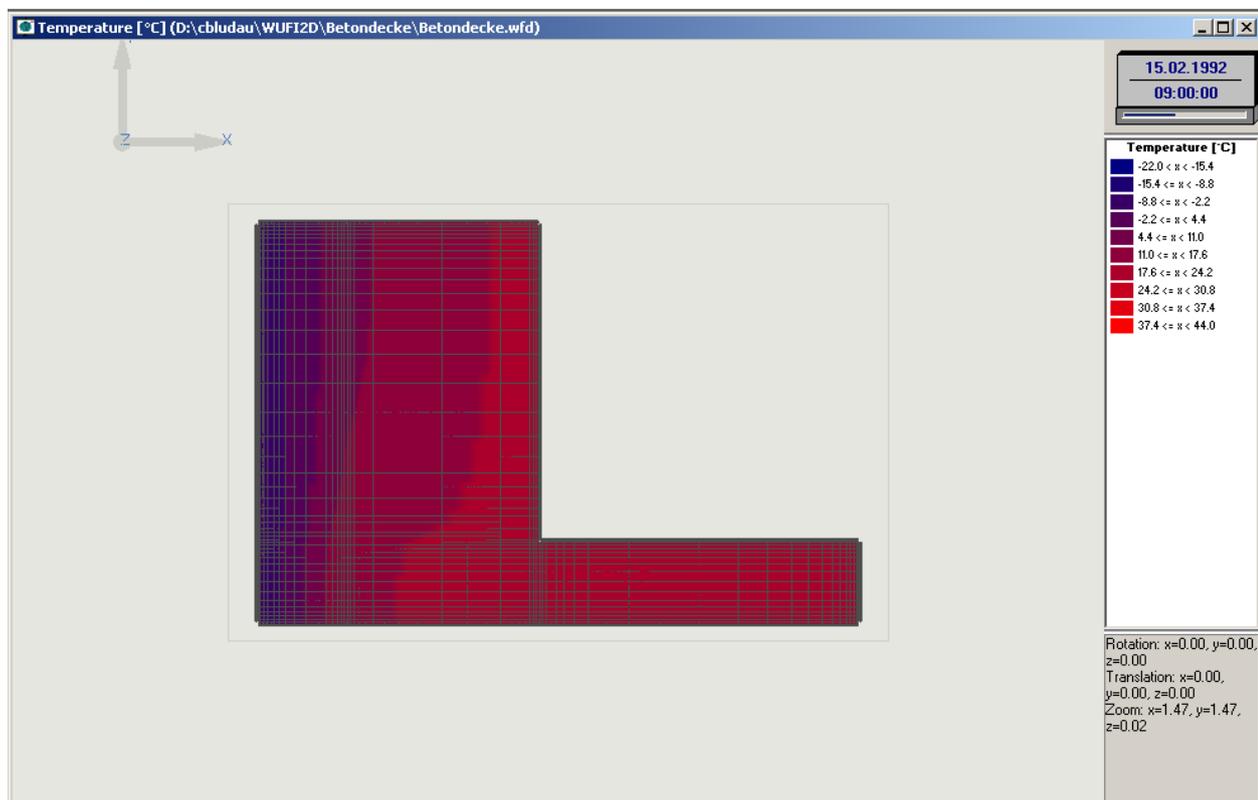
Step 9: Analysis of the Results

In this step we analyze and assess the results. The results may be viewed with WUFI2DMotion as a 2D-animation or with WUFI2DGraph as curves showing the temporal evolution of the various quantities. In the present example case we are mainly interested in the behavior of the temperatures; water content and relative humidity are only checked to make sure there are no conspicuous problems.

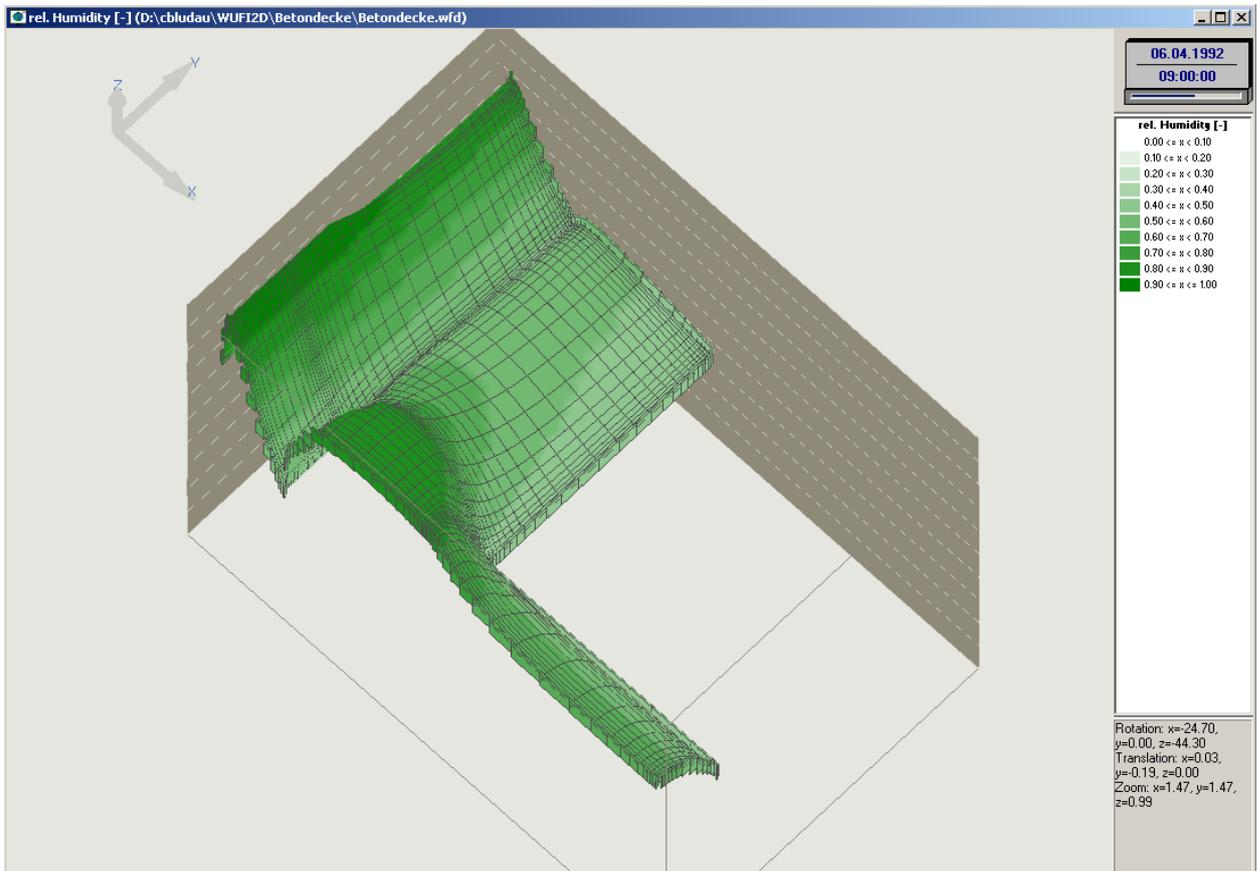
Viewing the Results with WUFI2DMotion

In order to view the results in WUFI2Dmotion, answer "Yes" to WUFI 2D's question "Would you like to view the results?" when it has finished the calculation, or click on the WUFI2Dmotion icon (🎬) in the toolbar, or choose Output → View the Results.

WUFI2DMotion displays one window for each of the calculated quantities you selected for output. In our case, these are temperature, relative humidity and water content, which are shown as two-dimensional color-coded fields. In the example, click on the temperature window to bring it to the foreground, then start the animation (Movie → Start). The temperature distribution on February 15 clearly shows the effect of the thermal bridge created by the concrete slab.



Now switch to the window „rel. Humidity“. On the April day shown below the concrete slab has already dried noticeably in the part exposed to the indoor air, while the part enclosed in the wall has retained more humidity.



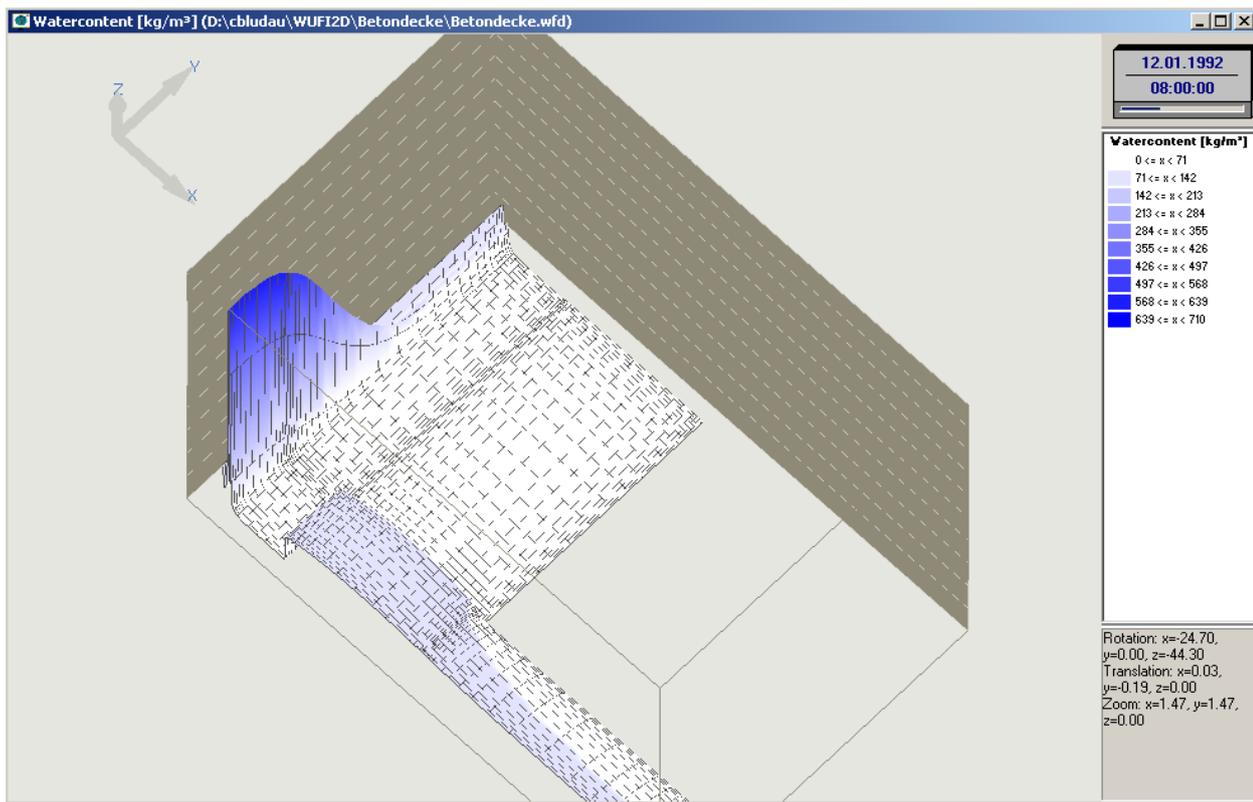
TIP: You may grab the diagram with the cursor and - with the mouse button held down - rotate it into any desired view. In addition to the color coding, the different values in the fields are then visualized by their different heights along the z-axis.

TIP: In order to return to simple top view, click on the button "Reset" () in the toolbar.

TIP: In order to zoom in and out, use the `SHIFT` key as described in step 2 (dialog "Geometry").

TIPP: In order to scroll the diagram in the window, use the `CTRL` key as described in step 2 (dialog "Geometry").

Now activate the window "Water content", reset the film () and start it again. Here we see, starting in November, noticeable amounts of water accumulating below the plaster near the concrete slab. However, the accumulated water will have dried out by June.

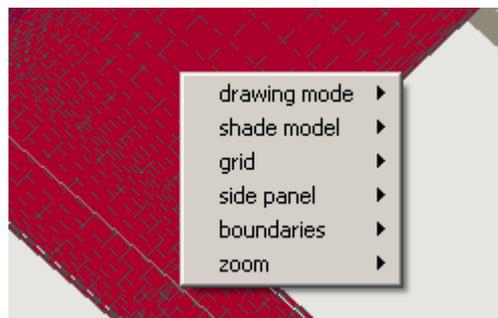


TIPP: The movie runs faster if you minimize the windows you do not need. Maximize the minimized windows to make them reappear.



Further Functions of WUFI2DMotion

A right-click on one of the diagram windows opens this pop-up menu:



Use this menu to fine-tune the appearance of the diagrams.

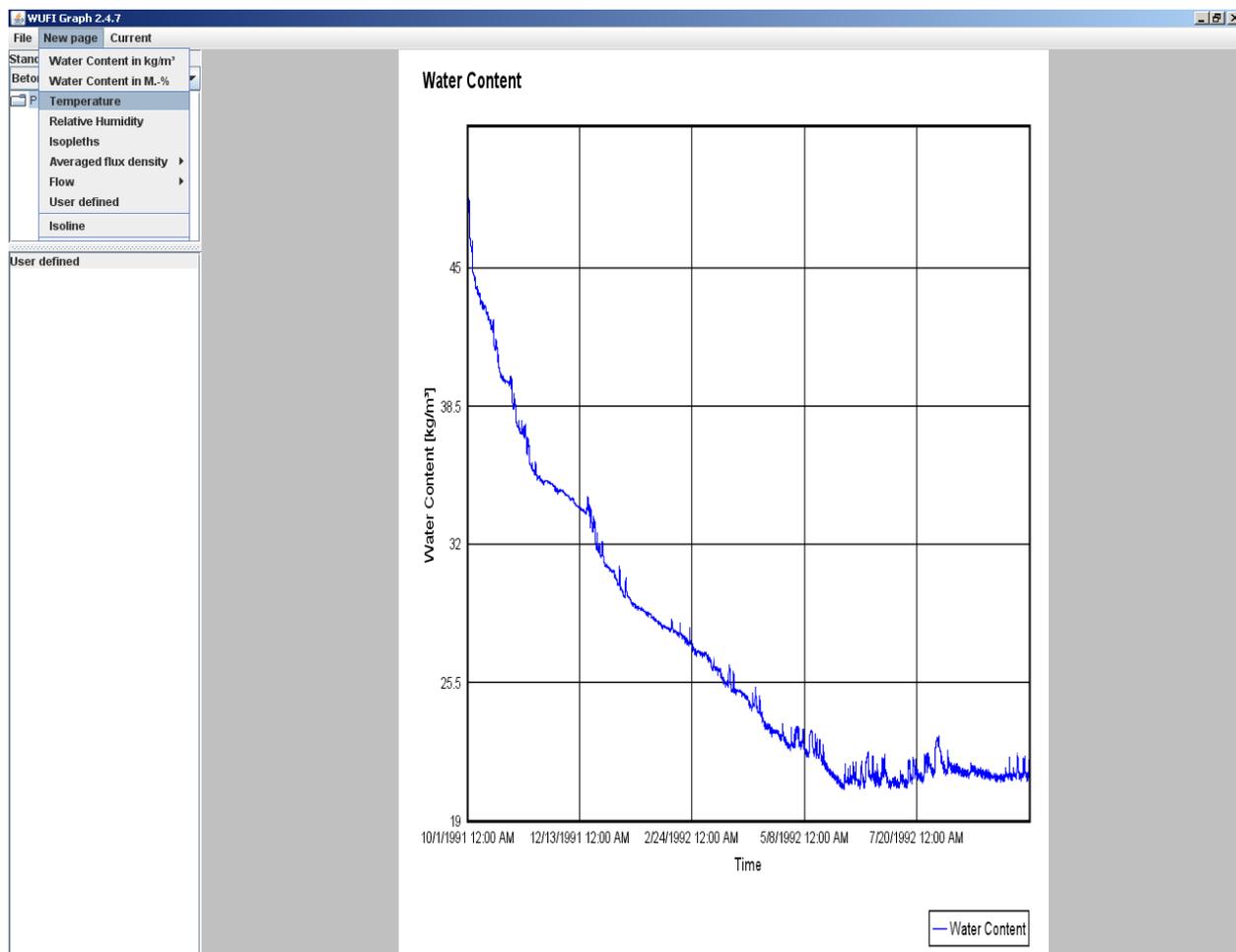
drawing mode → colorized	If this option is unchecked, the values are only indicated by the field height in z-direction. If the option is checked, the 2D-field is also color-coded to indicate the computed values. The (editable) color scale is displayed to the right of the diagram.
shade model → smooth / flat	switches smoothing of color gradients on or off.
grid → visible	switches the grid lines on or off.
grid → depth test	switches display of hidden parts of the grid on or off.
side panel → visible	shows or hides the side panels of the diagram.
side panel → lighted	switches the side panels between a light and a dark color.
side panel → numbering	switches numerical labels for the z-axis on or off.
boundaries	shows or hides the boundary conditions (separately for top, left, bottom, right)
zoom → factor → x, y, z	stretches or shrinks the diagram by the specified factor in x-, y-, or z-direction.
zoom → in	zooms into the diagram.
zoom → out	zooms out of the diagram.

Viewing the Results with WUFI Graph

In order to view and analyze the results with WUFI Graph, click on the corresponding button in the toolbar () or run the program from the Windows "Start" menu. If you choose the latter, the WUFI 2D result file (*.wfd) has to be opened manually.

WUFI Graph displays the results as curves showing the temporal evolution of the various computed quantities. The display is organized in pages, diagrams and curves. Each page contains one or several diagrams, and each diagram contains one or several curves.

Upon launch, WUFI Graph opens with a predefined "page 1" showing the total water content of the construction.



The behavior of the total water content allows a first assessment of the results. It shows whether the overall water content of the component increases or decreases during the period under investigation.

Increasing total water content

If a calculation over several years shows a long-term accumulation of moisture, the reason for this must be assessed. A possible reason might be that the initial water content was estimated too low and that the construction simply tries to reach the normal equilibrium water content which corresponds to the ambient climatic conditions. A more serious reason might be that a flawed design causes moisture accumulation in the construction.

You should try to determine the normal equilibrium water content of the construction, either by extending the calculation over a longer period of time, or by repeating the calculation with a higher initial moisture content. If the water content keeps rising without reaching an equilibrium state, or if the long-term water content is so high that damage by rot, mould, corrosion or frost wedging can occur, it must be concluded that the investigated construction would fail under the applied conditions.

Decreasing total water content

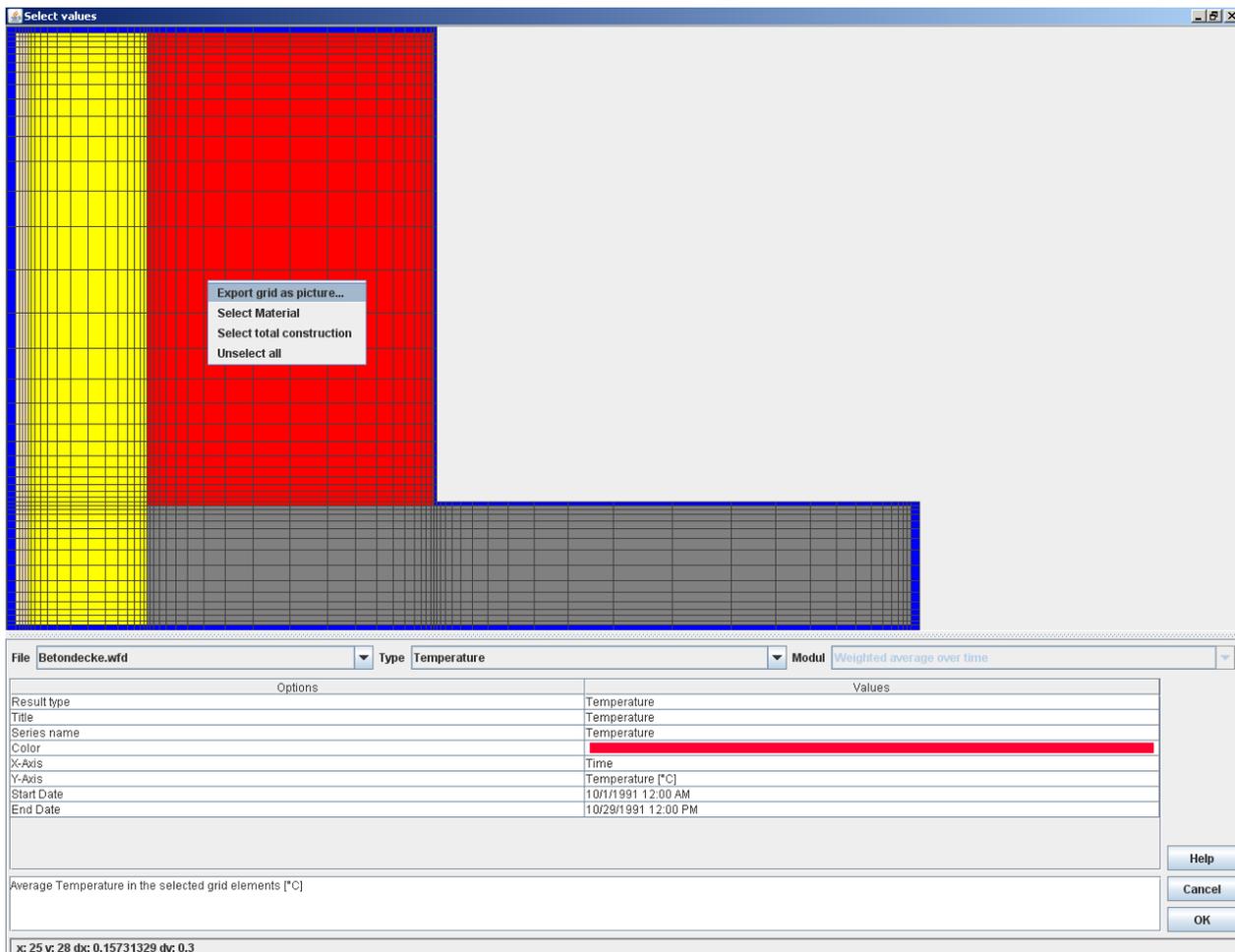
If the water content falls below the initial values, the construction dries out. If the initial values already were quite low (e.g. 60 % r.H., averaged over the construction) you may proceed to analyse the behavior of the individual construction elements. Even here, however, it is more advisable to definitely reach the equilibrium state of the construction first, by extending the calculation over a longer period of time.

Total water content remaining constant

If the water content remains constant from one year to the next (except for the usual variations in different seasons), the construction is in periodic equilibrium (that is, the transient moisture profiles repeat from one year to the next). This state is independent of the initial conditions and represents the typical behavior of the construction under the applied climate conditions. In this case you can proceed to analyse the moisture contents of the individual construction elements.

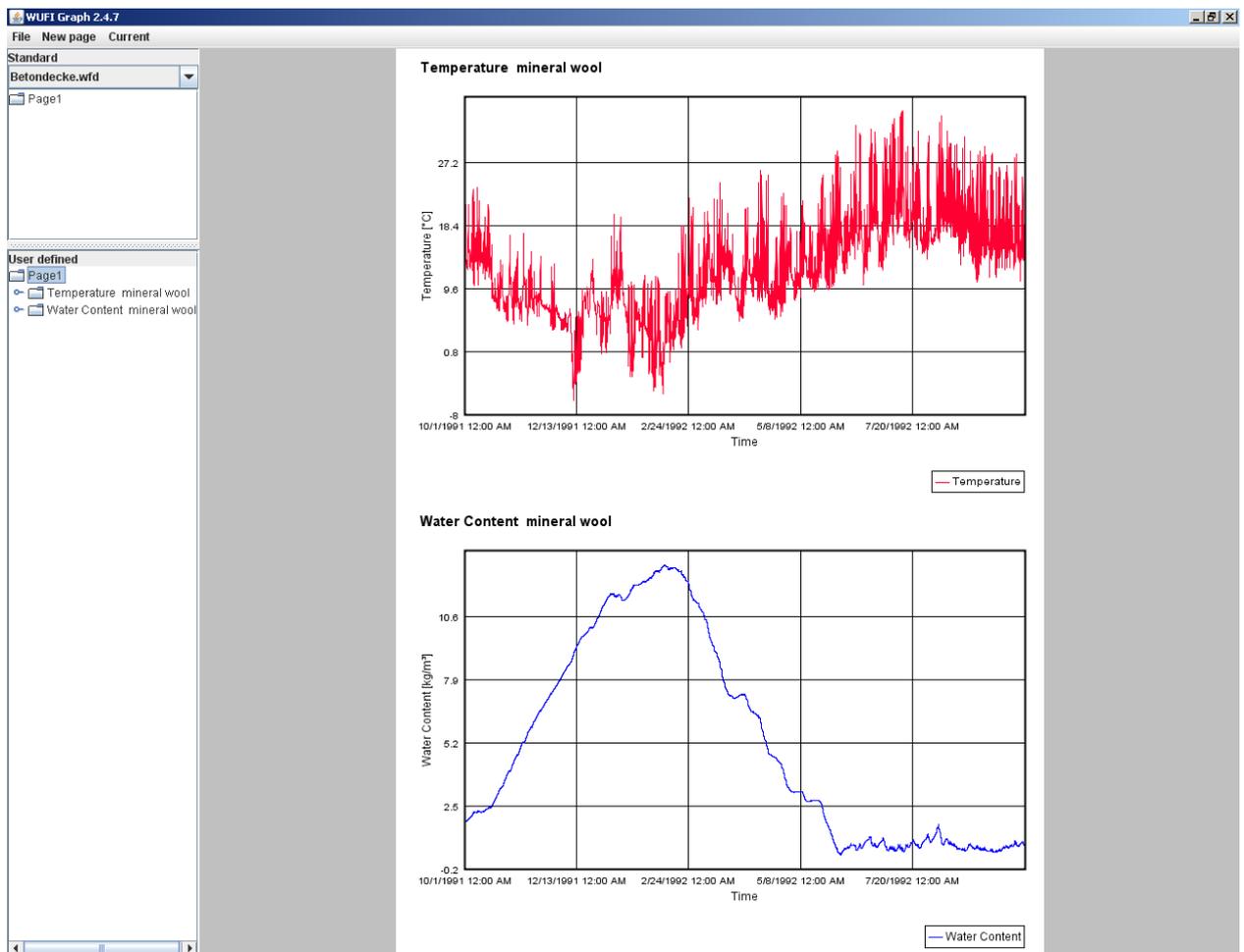
An assessment of the water content in the different materials allows a more detailed analysis of individual locations in the building component.

In our example, add a new page with temperature curves: open the menu "New Page" and choose the menu item "Temperature". WUFIgraph opens the dialog shown below. Right-click on the mineral wool insulation (yellow) and, in the pop-up-menu, choose "Select Material". This will select that part of the grid which is filled with mineral wool (the selected region is marked in blue) and tell WUFI-Graph to display the temperature averaged over the mineral wool. Enter the text "Temperature mineral wool" in the text box "Title"; this text will be displayed at the top of the new page. Click „OK“ to create the page.



Now click on "Page 1" in the list of "User-defined" pages in order to highlight it, open the menu "Current" and choose "Add water content in kg/m³". The "Select" dialog opens again, with the mineral wool still highlighted. Change the title to „Water content mineral wool“ and click „OK“.

You now have created a page which shows the evolution in time of the mean temperature (top) and the mean water content (bottom) in the mineral wool.



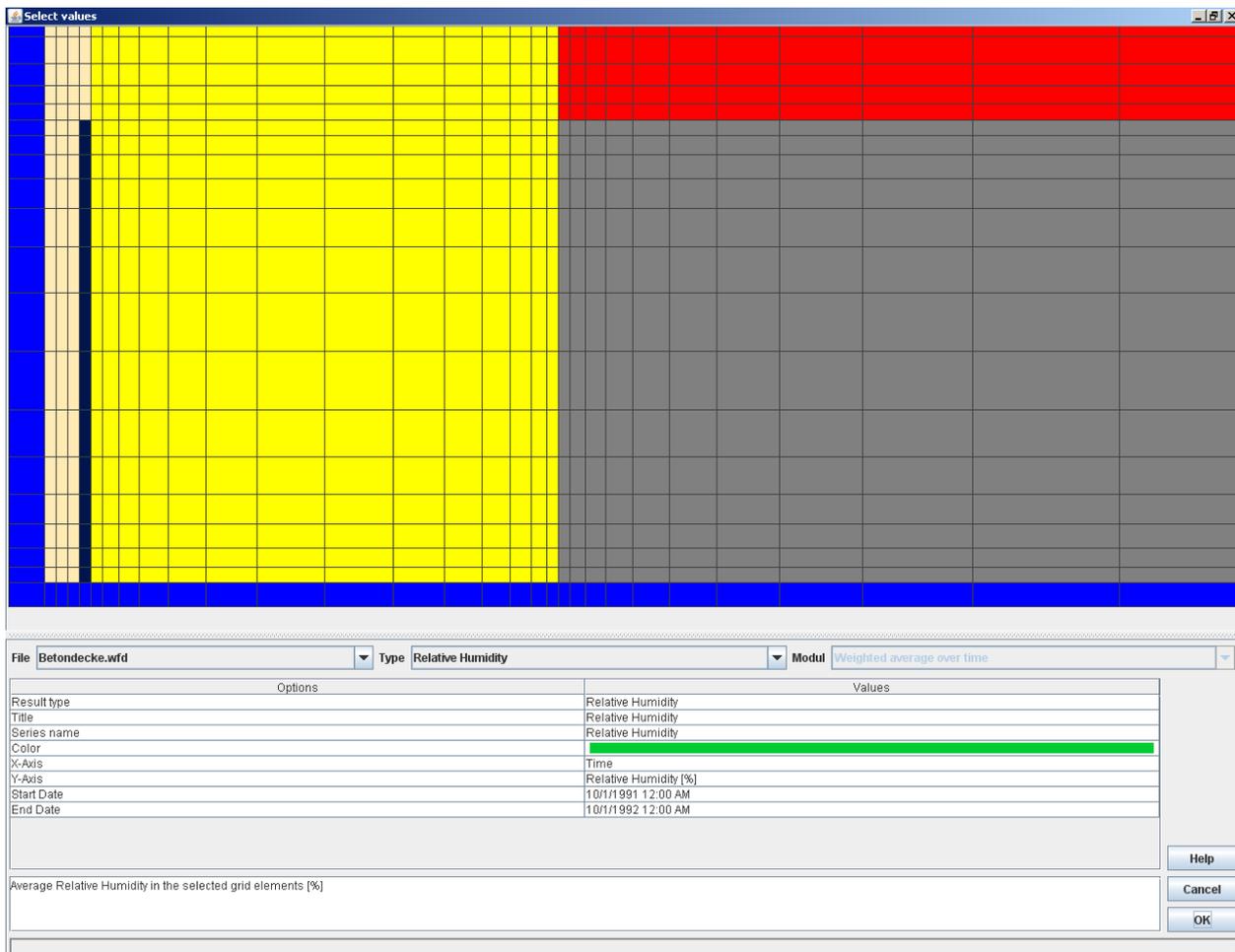
TIP: The contents of the menu "Current" change in dependence of what you have highlighted in the "User-defined" pages list. If nothing is highlighted, the menu lets you create a new page. If a page is highlighted, it lets you add a new diagram to the page. If a diagram is highlighted, it lets you add a new curve to the diagram. If a curve is highlighted, it lets you edit the properties of this curve.

TIP: Instead of using the menu "Current" you can also open a pop-up menu with the same choices by right-clicking on an item in the "User-defined" pages list.

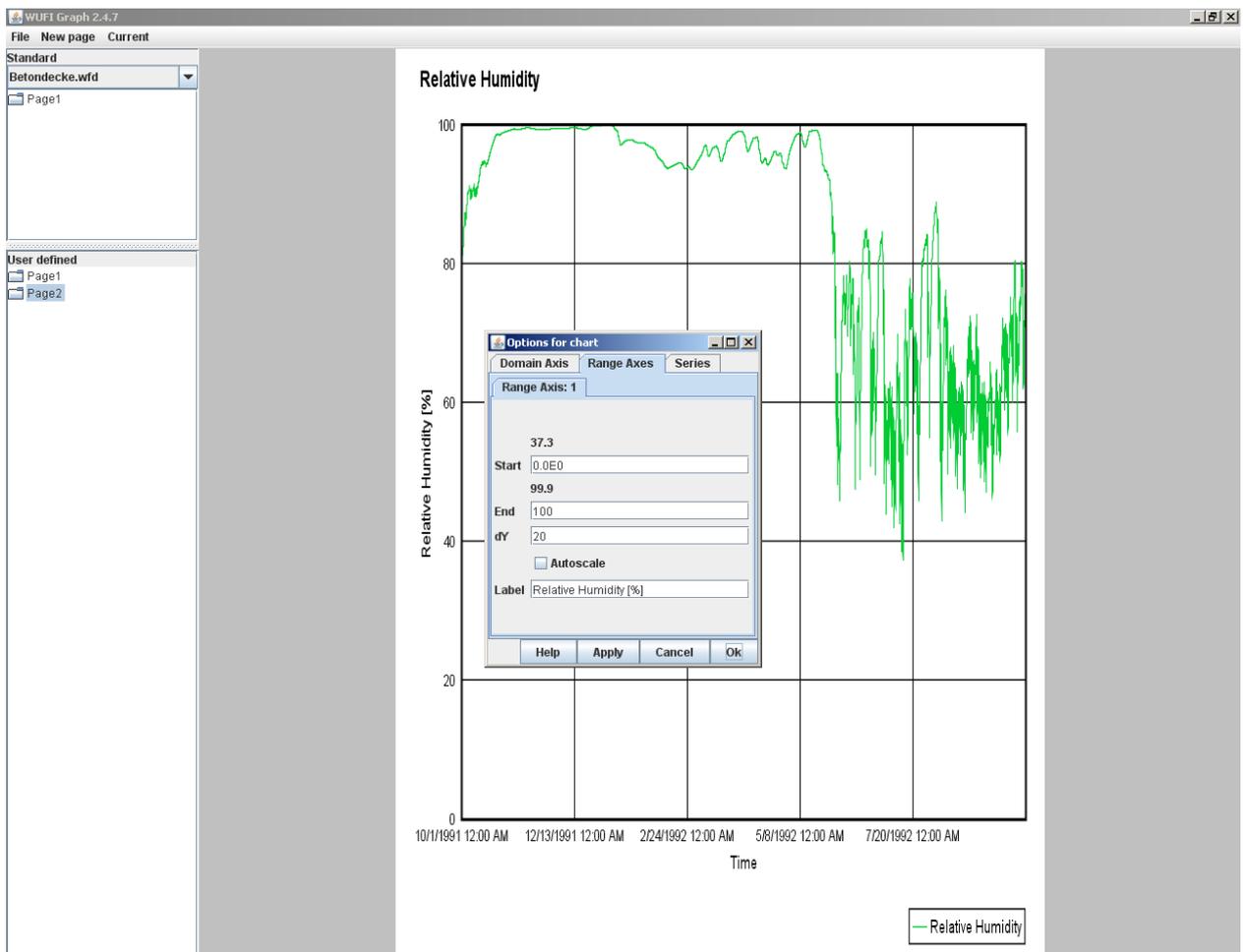
TIP: Please note that the last selection in the "Select" dialog is still active when you re-open the dialog for a new page, diagram or curve. If you still want to analyze the same selected region, you need not select anything new. If you want to analyze a different region, you must remove the old selection (right-click on the grid). If you define a new selection without removing the old selection, the average will be taken over the new and the old selection.

In the next step, we wish to create a curve showing the evolution in time of the relative humidity in the plaster, close to the boundary between plaster and mineral wool. To do this, create a "New page" for the quantity type „Relative Humidity“. Zoom into the grid by dragging open a zoom frame over the lower left corner

of the grid, with the `SHIFT` key held down. If necessary, scroll the zoomed grid into a convenient position by dragging it with the mouse while you hold the `CTRL` key down. Right-click on the grid and choose "Unselect all" because we do not want the still-selected mineral wool contributing to our new curve. Then select the desired grid elements in the plaster by clicking on each of them or by drawing a selection frame over them. The selected region is shown in a darker color (see screenshot).



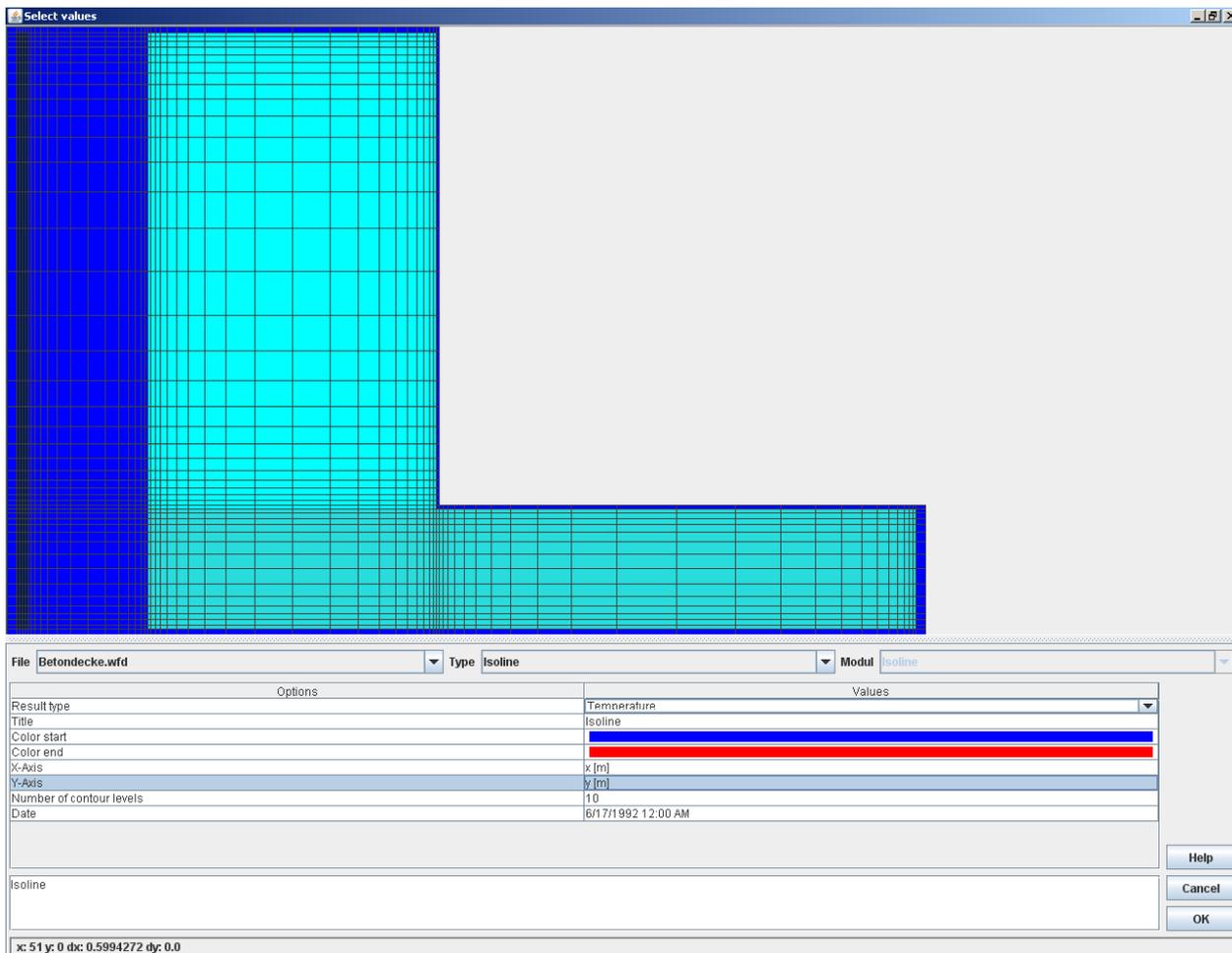
Confirm the selection by clicking „OK“. The user-defined pages list now contains the "Page 2" shown below:



Right-click on the diagram and choose "Options" in the pop-up menu to open the dialog "Options for chart". Here you may edit the axis labels and scales, the order of the curves if the diagram contains more than one curve, etc. **In our example**, deactivate the "Autoscale" feature for the y axis ("Range Axis") and manually establish a scale from 0 (Start) to 100 (End) with an increment of 20 (dy). Click on "Apply" to test your changes. Click on „OK“ to finally apply your changes and to close the options dialog.

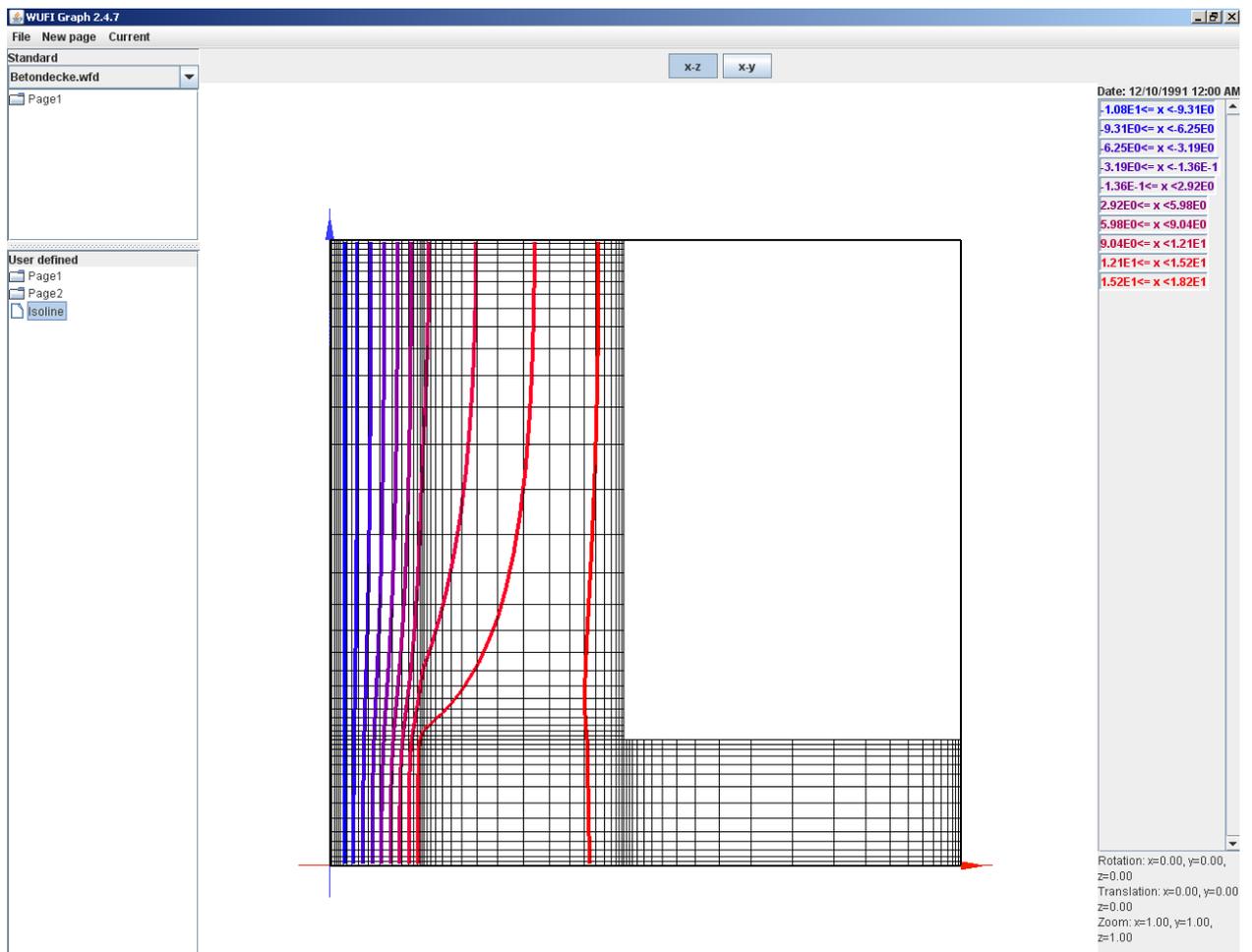
TIP: You can zoom into the diagram by dragging open a zoom frame from upper left to lower right over the area to be zoomed. Reset the zoom by dragging open a frame (anywhere in the zoomed diagram) from lower right to upper left.

Finally, we wish to create a page with isolines showing the temperature distribution on 10. December 1991 at 6:00 o'clock. Open the menu "New page" once more and choose the menu item "Isoline". The following dialog opens:



Right-click on the grid and from the pop-up menu choose "Select total construction". From the drop-down list "Result type" choose "Temperature". Then select the desired point in time as the "Date" and click „OK“. WUFIgraph displays the temperature distribution in the component at this point in time as isolines.

TIP: Sometimes the appearance of isoline diagrams is improved by including the boundary conditions acting on the component. To do this, drag a selection frame around the whole grid, including the blue border representing the boundary conditions.



In order to print the diagram pages, call up the desired view by selecting the appropriate page, diagram or curve in the "User-defined" pages list, and then in the menu "File" choose "Print selected...".

If you want to save the pages you created, open the menu "File" and choose "Save graphic project...".

NOTE:

When you save a graphic project, please note the following:

- If the graphic project is saved into the same directory where the WUFI 2D result file (*.wfd) is located, a *relative* path to the result file will be saved.
- If the graphic project is saved into a different directory than where the WUFI 2D result file (*.wfd) is located, an *absolute* path to the result file will be saved.

This is because WUFI Graph will often be used while the result file still resides in WUFI 2D's cache directory. The result file will almost certainly be moved from there, but if the graphic project file is moved with it, the relative path will still find the result file.

If an absolute path was saved and the result file is moved for some reason, WUFI Graph cannot find it any more. Its name in the drop-down list of open result fi-

les will then be displayed in red. As a workaround, open the graphic project file (somename.w2g.xml) with a text editor and correct the path pointing to the result file. You find it in the third line of the file, following the text „resultfile filename=“. The file must then be saved in UTF-8 encoding, or else it cannot be read by WUFIGraph. When you re-open the graphic project, WUFIGraph will be able to find the result file.

TIP: If you save the graphic project to the same directory where the result file resides, you avoid the risk that WUFIGraph may not be able find the result file when you re-open the graphic project later.

Further functions of the isoline diagram

The isoline diagram offers a few further functions which can be accessed via the keyboard:

Arrow keys (numpad)	scroll the diagram in the respective direction.
5 (numpad)	reset diagram
PageUp	zoom into diagram
PageDown	zoom out of diagram.

Mode selection:

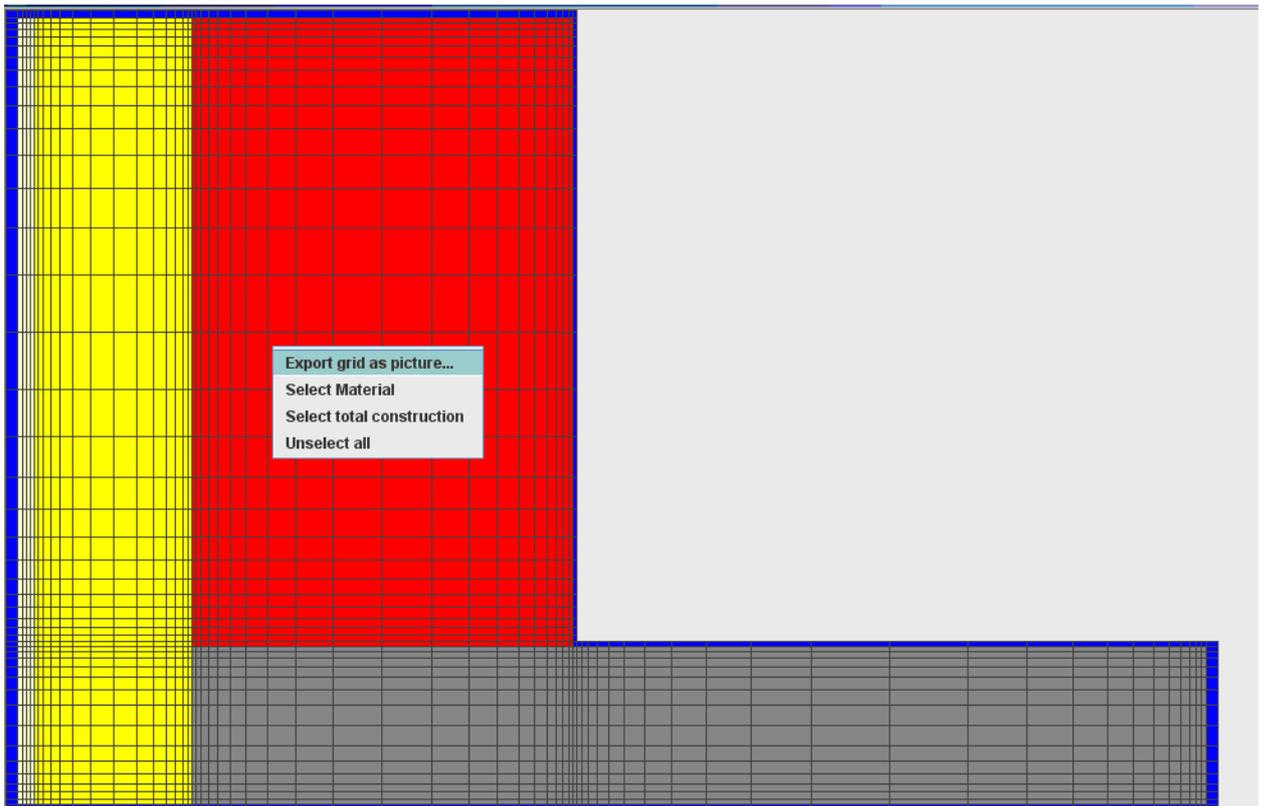
Ctrl + z	switches to mode ZOOM
Ctrl + v or Ctrl + t	switches to mode SCROLL
Ctrl + r	switches to mode ROTATE
Ctrl + s	shows the currently active mode.

x,y,z	applies the currently active mode with the respective increment to the x-,y- or z-axis.
+,-	increases or decreases the increment

The diagram can also be rotated by grabbing it with the cursor and dragging it into the desired view.

Further functions in the grid selection dialog

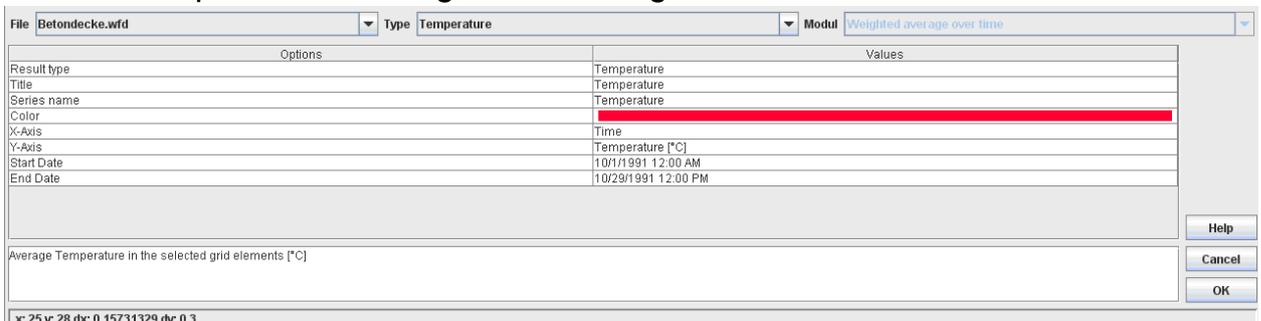
The grid selection dialog offers a few further functions:



Right-click on the diagram to open a pop-up menu with four functions:

- Export grid as picture... Saves the current grid diagram as an image file (*.png).
- Select Material Selects all contiguous grid elements which contain the same material as the grid element under the cursor.
- Select total construction Selects the whole construction, excluding the boundary conditions (blue border).
- Unselect all Removes all selections from the grid.

In the lower part of the dialog, the following functions are available:



File Select one of the open result files.

Result type	Select one of the possible types for the result quantity to be displayed: water content, temperature, relative humidity, isopleths and user-defined.
Module	This option is only available when the result type user-defined has been selected. It allows to select between „evolution in time of averaged values“ (a quantity is plotted against time on the x-axis) and „averaged x-y-values“ (for each point in time, one quantity is plotted against the other, e.g. relative humidity is plotted against temperature).
Result type	Has usually been fixed by your earlier choice. Can only be edited for type "user-defined" where relative humidity, temperature or water content can be selected.
Title	Allows to specify a title for the diagram. If the current curve is not the first one in the diagram, the title cannot be edited.
Series name	Specify a name for the current curve which will be displayed in the legend.
Color	Select a color for the current curve.
X-Achse	The title for the x-axis.
Y-Achse	The title for the y-axis.
Start date	Start date for the curve.
End date	End date for the curve.

TIP: Most objects or dialogs in WUFIGraph offer options which can be accessed with a right-click.