A-

EFFICIENT ENERGY FOR EU CULTURAL HERITAGE

D7.2 Toolkit and self-training material

EUROPEAN COMMISSION DG

Research and Innovation

Seventh Framework Programme

Theme [EeB.ENV.2010.3.2.4-1]

[Compatible solutions for improving the energy efficiency of historic buildings in urban areas]

Collaborative Project – GRANT AGREEMENT No. 260162

SEVENTH FRAMEWORK PROGRAMME

Technical References

Project Acronym	3ENCULT
Project Title	Efficient ENergy for EU Cultural Heritage
Project Coordinator	Alexandra Troi EURAC research, Viale Druso 1, 39100 Bolzano/Italy Alexandra.troi@eurac.edu
Project Duration	1 October 2010 – 31 March 2014 (42 Months)

Deliverable No.	D7.2
Dissemination Level	PU
Work Package	WP 7
Lead beneficiary	P09 TUD
Contributing beneficiary(ies)	P011
Author(s)	Ayman Bishara
Co-author(s)	Rudolf Plagge ,Heiko Fechner, Ulrich Ruisinger
Date	31 March 2014
File Name	WP7_D7.2_Toolkit and self-training material

Table of Content

Abstract

A particular advantage of numerical simulation is that the investigation of constructions consisting of different materials under various climatic loads requires relatively little work in comparison to field experiments. Thus a hygrothermal assessment of constructions can be evaluated in a short time. Construction building details and building materials can be optimized using the numerical simulation as well as renovation measures can be planned.

The aim of present deliverable is to facilitate the application of simulation tools, which the project partners used within the project, for planners.

• Elaboration of tutorials, based on simulations performed within the project

• Develop training material

• Elaboration of a guideline on how to use simulation results in design tools as PHPP

Purpose of this report is to deliver a documentation of the Delphin-training material and the way of using Delphin-tools and its material data files (simulation, program data base, user data base). The reader should understand a hygrothermal tools and material file and learn how to create and manipulate own material data files and how to import and export material data into his hygrothermal simulation.

0 INTRODUCTION

There has been a change in trends in the European building sector, from new constructions to insulation, retrofitting and restoration of existing buildings. Thus, building reconstruction has an outstanding position in the building market in Europe. This is based on the fact that a comparatively large number of older buildings exists in Europe which are nevertheless worth preserving. Taking a closer look at the East-European countries that have recently joined the EU, this aspect of building will become even more important. Many buildings in the stock have to be improved considerably by means of an additional thermal insulation of the building envelope. Safe wall construction in connection with an appropriate thermal insulation is generally done by computational prognosis.

The rapid development of the modern hard- and software tools, e.g. CHAMPS, DELPHIN and Energy Plus and others, enable the simulation of the hygric and thermal behaviour of building components under transient climatic boundary conditions. Meanwhile building simulation programs are generally used in research projects and engineering applications as well. The simulation of building components, e.g. constructive details, is a part of the whole building simulation or integrated building simulation. Here the hygrothermal behavior of the building envelope and the microclimate close to the construction surfaces are the main focus of interest. Despite an adequate description of the material configuration (layering, dimensions, etc.), the quality of simulation results generally depends on the material properties and climatic boundary conditions.

Climatic data, such as temperature, relative humidity, short- and long-wave radiation, precipitation, wind velocity and wind direction are generally available from weather services, but still the problem of local climate is not solved. The determination of microclimatic data locally at a given building façade position from general climatic data from weather station, especially regarding driving rain and surface heat or mass coefficients, is a difficult task. The other major problem are the respective hygrothermal properties of building materials. Published data usually lacks an adequate general material description and often the data sets are incomplete. Very often the measured data are not documented and the sets consists of results coming from different laboratories and often do not represent the same material batch. Another problem arises from the requirements of the simulation codes themselves. The numerical solution of coupled differential equations uses moisture and temperature dependent transport and storage functions [4]. Laboratory experiments deliver material parameters that have to be interpreted in terms of material functions to be useful for simulation models.

The Institute for building climatology in Dresden University of Technology is the developer of such computational technologies. On simulation tool is called Delphin and a license has been given for each 3ENCULT project partner to do hygrothermal simulations for their case studies.

To support these simulations, the TUD provided guidelines on:

(i) the proper analysis of construction and identification of material used in the regarded detail,

(ii) makes available the complete information from the Delphin material property database,

(iii) offered the opportunity to measure in its lab characteristic material data (as e.g. bulk density, thermal conductivity, moisture content at e.g. 80% relative humidity) of material samples provided by case studies

(iv) offered support in the preparation of the needed climatic data, by preparing the climatic data file - case study responsible partner have just to submit the climatic data of the location, at least the climate data close or nearest by the test house.

In this part of the 3ENCULT deliverable teaching and training courses and workshops on the hygrothermal simulation for project partners have benn already organized and carried out by TUD at the industrial partner location Remmers in Löningen (Germany) between 3rd to 5th September 2012. Since evaluation of measured data by simulation is an important task, the workshop was held together with the monitoring group in Löningen. The teaching and training materials developed for the course as well as the experience of the course itself were the basis for training modules and tutorials as described in Annex I. the Documentations of the training workshop (Workshop program, teaching materials, Delphin tutorials, presentations, pictures) are included in this file.

1 Hygrothermal simulation tools & Interior Insulation Workshop

3Encult Workshop 3rd to 5th September 2012
In the conference room „Forum"of Remmers Baustofftechnik GmbH, 49624 Löningen

Programme

Monday, 3rd September 2012

09:00 hrs	Workshop 1:30h
10:30 hrs	Coffee/Tea
10:45 hrs	Workshop 1:45h
12:30 hrs	Lunch
13:00 hrs	Guided tour through the company Remmers
15:00 hrs	Workshop 1:30 h
16:30 hrs	Departure to Cloppenburg (by bus)
17:00 hrs	Guided tour (English) through the Open Air Museum in Cloppenburg
18:30 hrs	Dinner in the "Dorfkrug" in the Open Air Museum
21:00 hrs	Return to Löningen (by bus)

Tuesday, 4th September 2012

09:00 hrs	Workshop 1:30h
10:30 hrs	Coffee break
10:45 hrs	Workshop 1:45h
12:30 hrs	Lunch break
13:15 hrs	Workshop 1:30h
14:45 hrs	Coffee break
15:00 hrs	Workshop 2:00
17:00 hrs	End of workshop
18:30 hrs	Dinner in a local restaurant "Gambrinus" (own account)

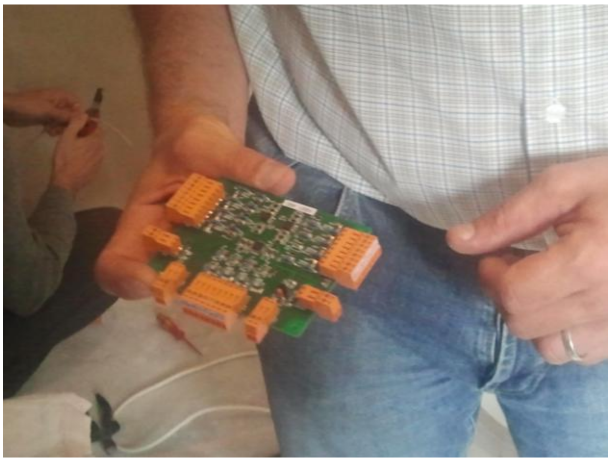
Wednesday, 5th September 2012

09:00 hrs	Workshop 1:30h
10:30 hrs	Coffee break
10:45 hrs	Workshop 1:45h
12:30 hrs	Lunch break
13:00 hrs	End of workshops

Photo documentation of the workshop

Hygrothermal (Delphin) workshop

Preparation of monitoring installation



Installation of monitoring system

Installation of Sensors

Insulation of Sensors

Monitoring system

Complete the installation of Monitoring system

IQ-Therm Interior insulation

IQ-Therm Interior insulation

Appling of interior insulation

Applying of interior insulation

Applying of interior insulation

Applying of interior insulation

Complete Applying the wall interior insulation with installation different sensors

Complete the wall interior insulation with installation of different sensors

2 Teaching materials

2.1 Delphin tutorials

Tutorial (1) outside wall with interior insulation

In this tutorial a small example of a typical old masonry wall will be used that is modeled and simulated before and after fitting of an inside insulation.

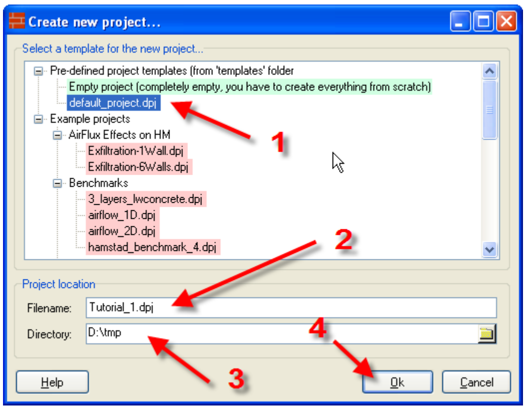
First, the previous construction (without inside insulation) shall be evaluated with respect to hygric and thermal performance. Afterwards a variation study is done to find a suitable inside insulation system.

Part 1: Simulation of Previous Construction.

This part of the tutorial covers the principle steps in creating Delphin simulation projects.

Project Setup and Modeling of the Construction

At first, after starting the Delphin program, only the small main menu is visible at the top of the screen, where the different buttons for the project setup and control can be found. The first step is the creation of the new project. The New-button (or File → New...) opens the dialog for creating a new project:



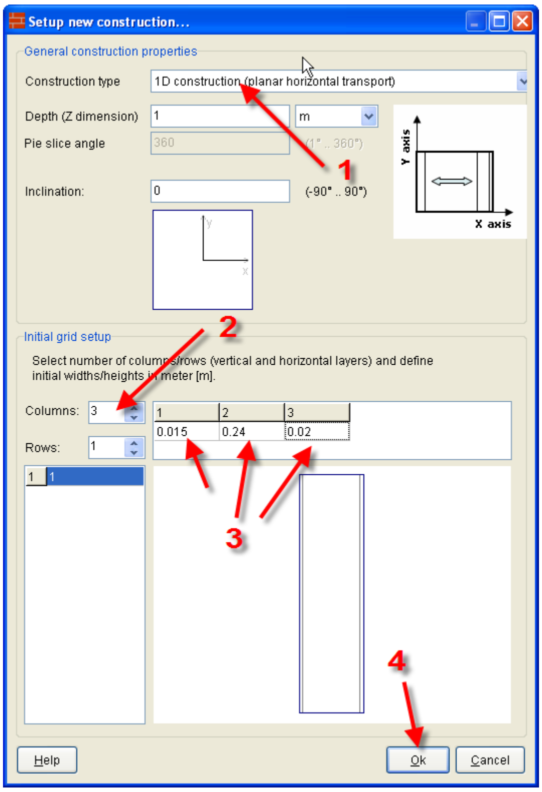
Important in this dialog is the selection of the project template. This can be either an empty default project, or one of the example projects. Select here **default_project.dpj**, enter file name and path and confirm the dialog.

Please make sure to select the project template **default_project.dpj**, because some of the steps below depend on this.

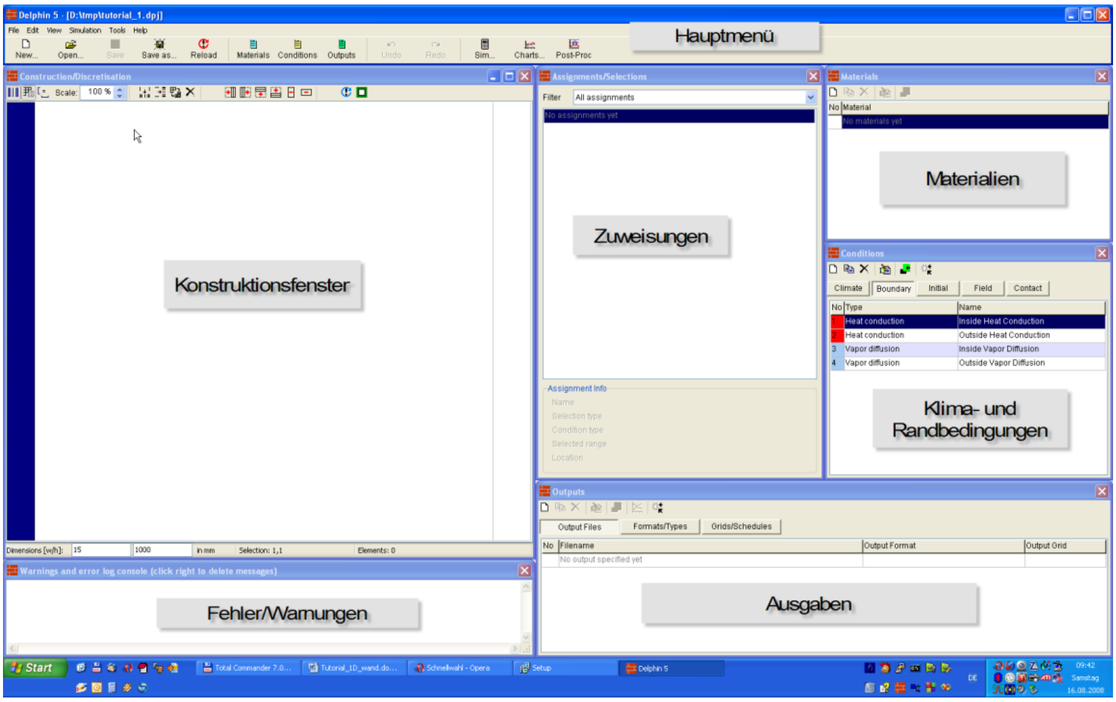
If later similar projects need to be created, you can copy your own project into the **templates** directory within the Delphin installation directory.

After confirmation of the dialog the construction dialog opens (see screenshot on next page). Here, the principle construction type is selected and the initial dimensions are entered.

Please select **1D construction (planar horizontal transport)**, and use 3 as number of initial columns of the construction. Then you can enter the column widths 0.015, 0.24 and 0.02 (in meter). For one-dimensional constructions it is important to ensure that the height (or width, in case of vertical constructions like roofs) is set to 1 m. This simplifies the analysis and interpretation of the results later.

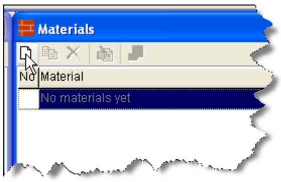


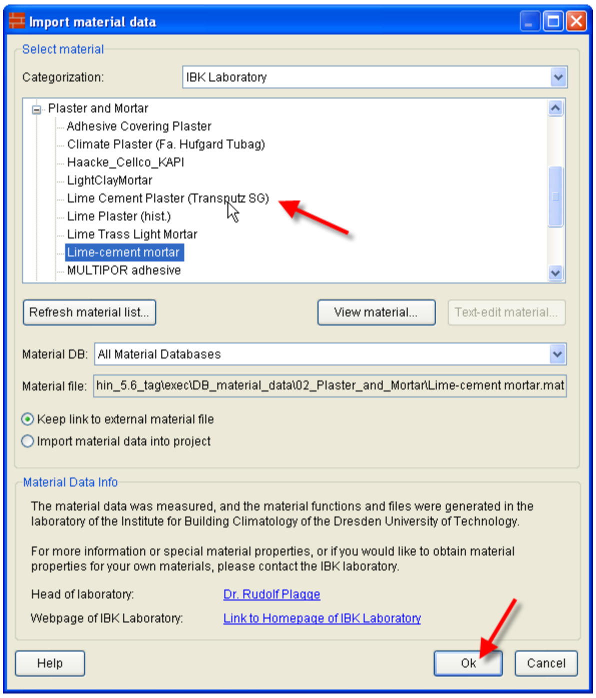
After confirming the dialog the program views appear showing a 3-layered construction, yet without materials.



The layout of the windows can be adjusted at will. The View-menu contains the commands for the layout of the views. The layout shown above is suitable for higher screen resolutions.

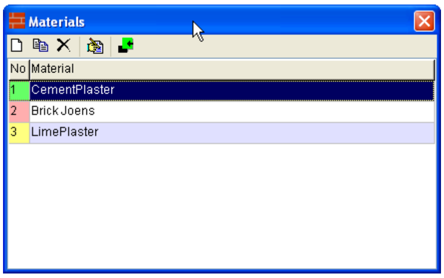
The shown construction is still empty and in the next step the materials need to be imported. First click on the New-button in the material list view (see left screenshot below).





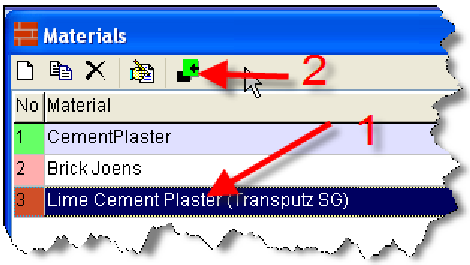
You can now select materials in the material import dialog (screenshot to the right). To import specific materials, switch the categorization (screenshot to the right top) to “Alphabetically Sorted List”.

Now select one after another the materials **CementPlaster**, **Brick Joens** and **Lime Cement Plaster (Transputz SG)**. You can select multiple materials with Ctrl+click and Shift+click. After the import the materials appear in the material list view.



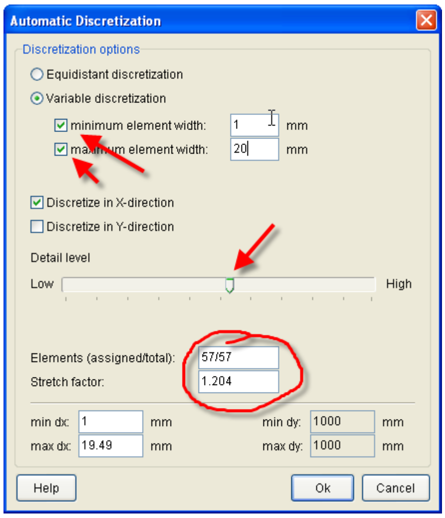
Now the materials are assigned to the appropriate layers of the construction.

To do this, you need to select a layer (or several layers) with the mouse. Then select the desired material in the material list and finally press the green assignment button.



After assigning all materials the construction view should show the layers of the construction with the corresponding material colors. If you now select a material in the material list, the corresponding layer or layers will be highlighted. This *highlighting* is independent of the actual *selection* in the construction view.

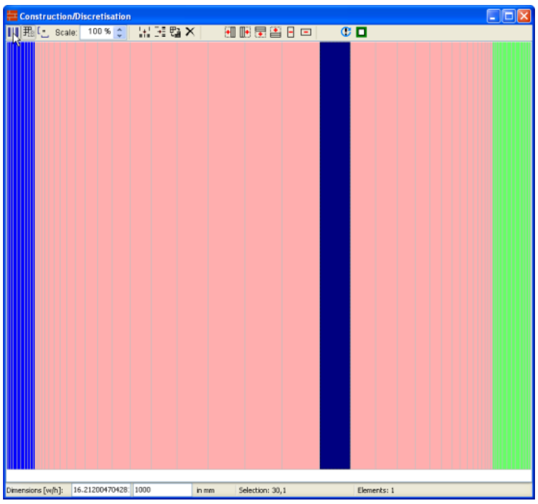
Next you can discretize the construction, i.e. divide in many small elements. For this you can use the dialog for Automatic Discretization, accessible from the menu buttons in the construction view (see left screenshot below).

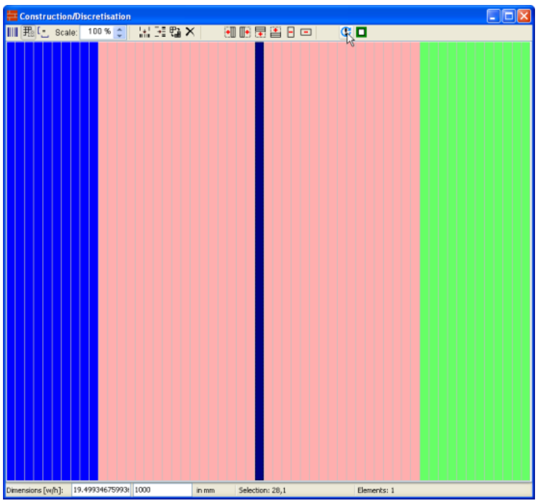


In the Automatic Discretization dialog we will only discretize the construction in X direction (since we have wall with only horizontal transport direction). We use variable discretization, that generates smaller elements near the boundary of the construction and at material interfaces. Inside of the construction the element widths are gradually enlarged.

Important parameters in this dialog are: Minimal and maximal element widths (1mm and 20mm are good default values), and the detail level. The discretization detail can be adjusted with the slider which adjusts the stretch factor and consequently the number of elements. A stretch factor of 1.2 is suitable for 1D-constructions. For 2D-constructions a higher stretch factor of approx. 1.5 is advisable due to long simulation duration.

As soon as this dialog was confirmed, the construction is shown as collection of many elements. Because of the small size of the boundary elements, it is useful to switch to equidistant display of the construction (button to the very left of the button bar in the construction view). In this view mode all elements are shown with the same dimensions, regardless of their actual size. This simplifies the selection of boundary elements.





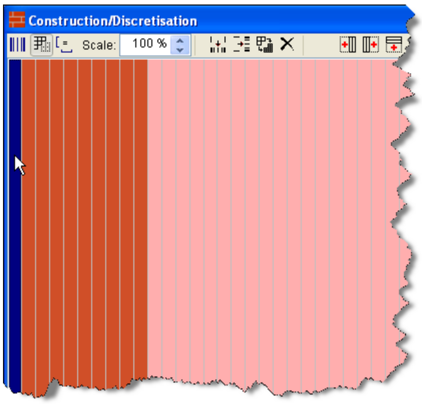
The left screenshot shows the normal proportional view whereas the right screenshot shows the equidistant view.

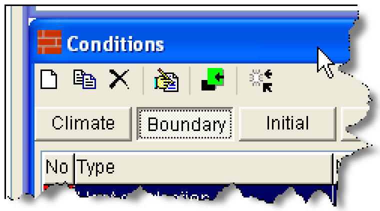
Boundary, Initial, and Simulation Conditions

As the next step all boundary conditions need to be specified and assigned. The project template already contains boundary conditions for the inside and the outside. The specification of own boundary conditions from scratch is covered in the next part of the tutorial.

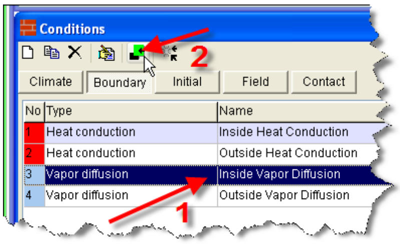
The assignment of boundary conditions is the next step, so that the construction 'knows' to which surface the boundary condition is applied. The usual order is: select a range of elements in the construction view, select a condition, and execute the “assignment” action (the green assignment button).

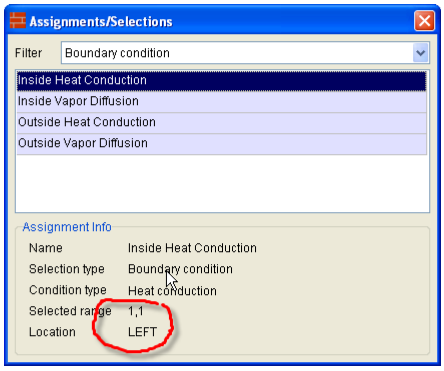
In the case of the boundary conditions, select at first one of the outer layers of the construction. In this tutorial the left side is supposed to be the inside, so the left-most layer should be selected first (see left screenshot below).





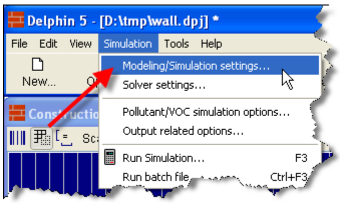
Then, switch in the conditions view to the tab with the boundary conditions “Boundary” (right screenshot above). Finally, select one or more boundary conditions (see left screenshot below) and use the green assignment button (see left screenshot below) to assign the conditions to the selected layers.



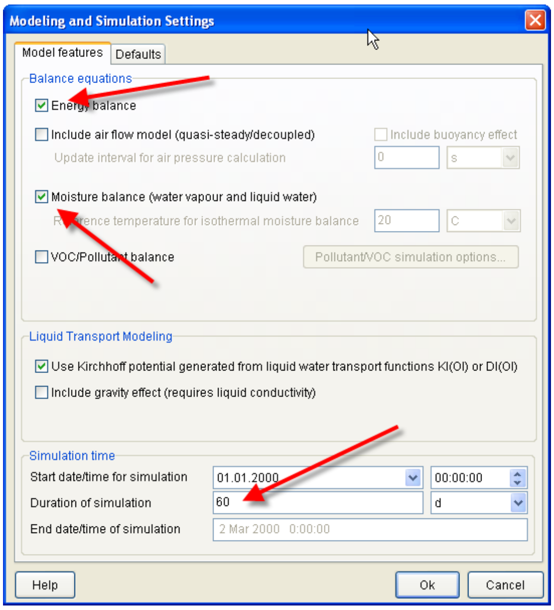


The assignment list shows all assigned boundary conditions and also provides information about the location and the side (surface) of the assignment (right screenshot). All boundary conditions in the template project need to be assigned to the appropriate construction side (2 left and 2 right, thermal conductivity and vapor diffusion respectively), as shown in the screenshot below.

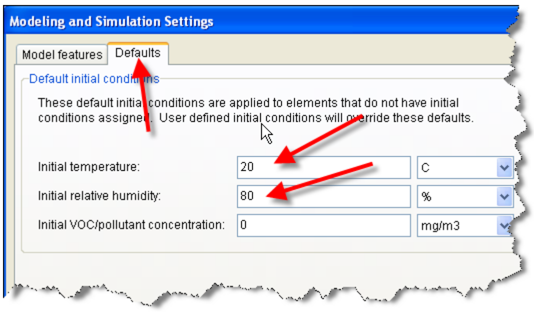
After specifying the boundary conditions it is time to select the simulation and modeling options. Open the modeling dialog:



In this dialog you can select the basic properties of the physical model. For this example you need to enable the balance equations for heat and moisture transport:



Other important settings in this dialog are the simulation duration and the start time point of the simulation. 60 days should be used as duration of the simulation. The start date is not important for a design simulation. However, the initial conditions need to be specified. These can be globally set in the “Defaults” tab:

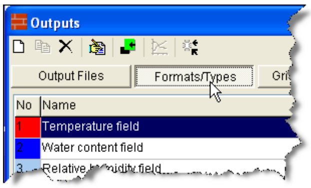


Typical settings are 20°C and 80% relative humidity, which should also be used in this example.

After confirmation of the dialog all settings required for the simulation are given in the Delphin project. However, so far no outputs have been requested. The next steps are selection of desired outputs.

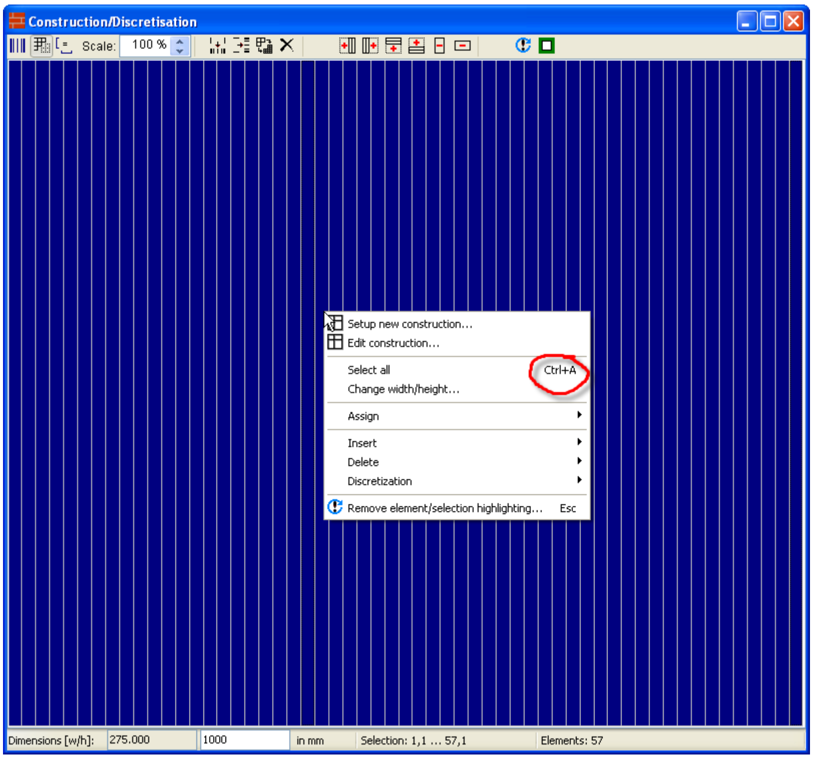
Outputs

Some outputs are already pre-defined in the default project template. These are shown in the output format list, visible in the “Formats/Types” tab of the outputs list view:



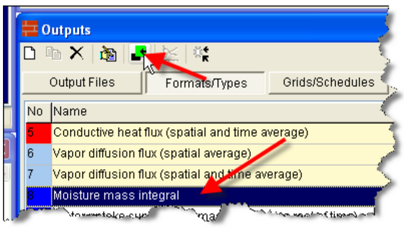
The formats define which quantities (temperature, relative humidity, mass of condensate, etc.) will be monitored. Also, the formats specify whether spatial or temporal averages or integrals should be calculated.

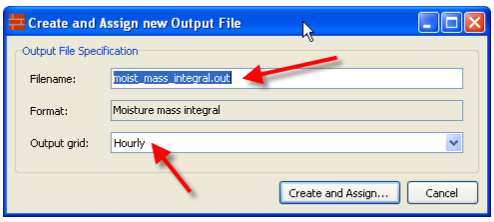
The formats can now be assigned to ranges of the construction, which are of interest. As usual, at first a range of elements needs to be selected in the construction view. In this tutorial, all outputs should be made for the entire construction, so you can simply select all layers.



As shown in the screenshot, the context menu of the construction view contains the option "Select all". Alternatively, you can use the usual shortcut Ctrl+A to select all layers.

After selecting the range of elements of the construction, you can select an output format and assign it with the green assignment button (see left screenshot below).

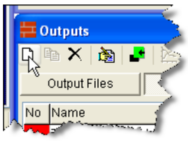


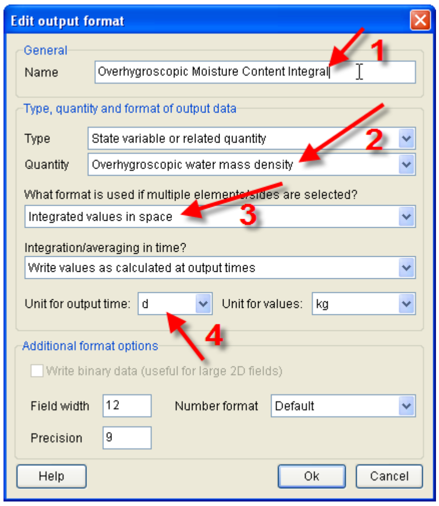


In this tutorial the Moisture Mass Integral shall be assigned first.

After pressing the assignment button the dialog for creating/defining an output file is shown (see right screenshot). Here you need to type a unique file name (without path !). Furthermore, you need to select an output grid. Output grids define when and how often outputs should be made. For integral values, corresponding to a single value per output, you can use hourly values. For fields and profiles you should use larger intervals (e.g. daily output intervals).

Sometimes the desired output formats are not included in the list. In this case you can create your own format, using the “New” command in the output format tab (see screenshots on next page).

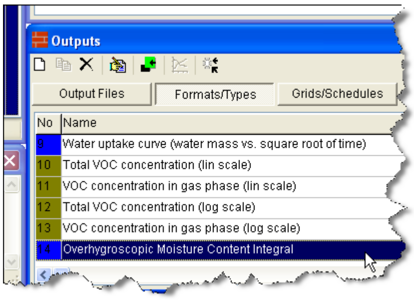


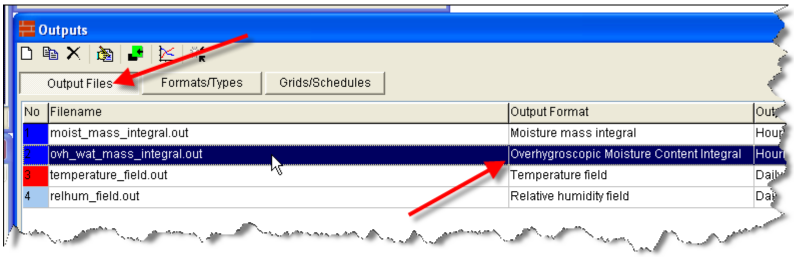


For this example we need to monitor the over-hygroscopic moisture content (= condensate). Im Output Format edit dialog (right screenshot), the following inputs are required: Unique identification name, “Overhygroscopic water mass density“ as Quantity, “Integrated values in space“ (spatial integration, since we are interested in the total mass of condesate) and as unit for output time points “d“ (we calculate 60 days, so we better plot the outputs also in days).

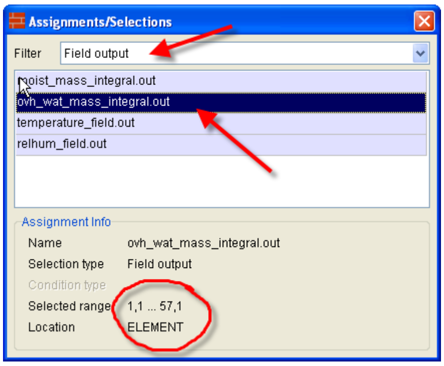
Once the dialog has been confirmed, the new format appears in the list of output formats/types (see left screenshot below). We can now assign the format to the whole construction and create a new file for this output.

For all defined outputs a separate output file is created. All output files are shown in the output file list in the “Output Files” tab. The right screenshot below shows the newly created and assigned output format for the overhygroscopic moisture content.



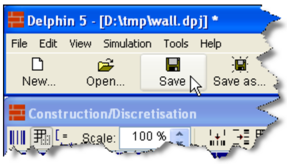


As already described, the connection between output files and the construction is created via assignments. These are also shown in the assignment list, where you can switch the assignment filter to “Field output”:



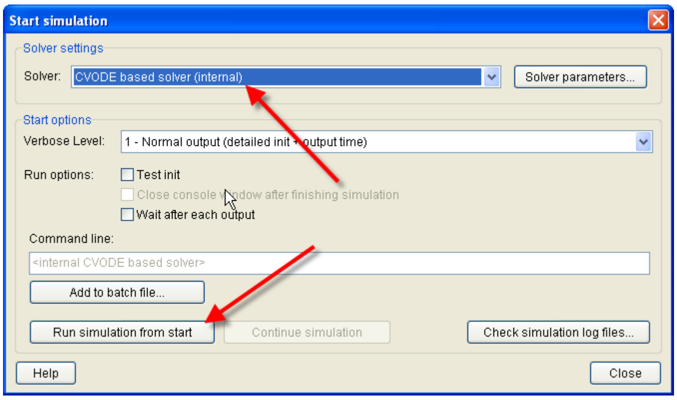
For each assignment you can see the range of elements, the type, and the location of the assignment. ELEMENT indicates here an element specific assignment. For boundary conditions the selected side of an element is shown instead. Note: depending on the discretization used in your project, the range may differ from the one shown in the screenshot.

Now all project settings are complete and it is advisable to save the project (Ctrl+S).



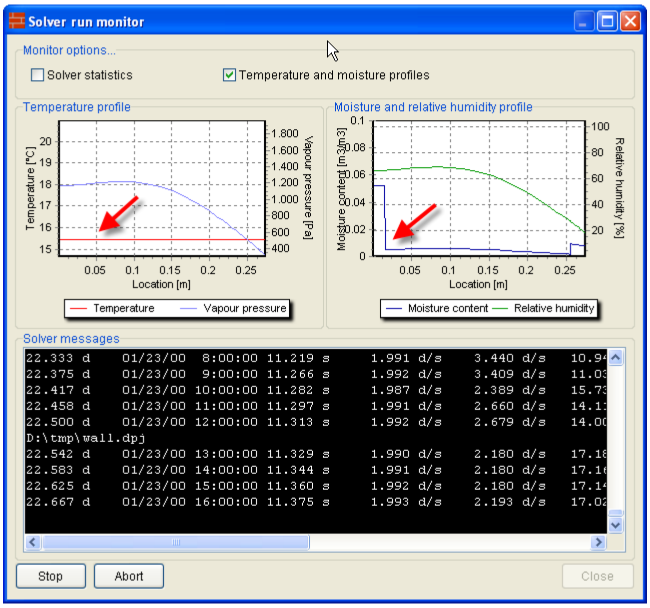
Run Simulation

The simulation is run in the simulation window, which is found in the main menu: Simulations >> Run simulation... or via the command button Sim...:



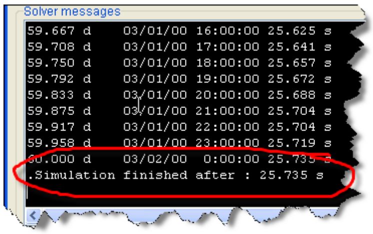
In this dialog you can choose between the internal solver (with graphical output of the calculation process) and the external solver. The latter is much faster and particularly for 2D simulations the recommended choice. In this example, and generally for 1D simulations, the internal solver is advisable.

The simulation is started with “Run simulation from start“and the simulation window opens:

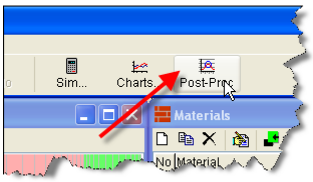


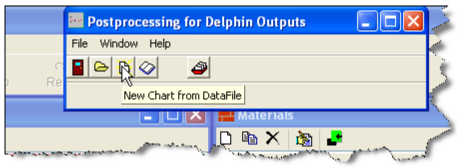
The simulation window shows the current temperature and moisture profiles that can be used to quickly check the results. Also, the current solver performance statistics can be shown.

Once the simulation is complete



the calculation results can be visualized and analyzed with the post-processing. The command button “Post-Proc” starts the post-processing tool:





In the post-processing window you can create new charts using the “New” button (see right screenshot above).

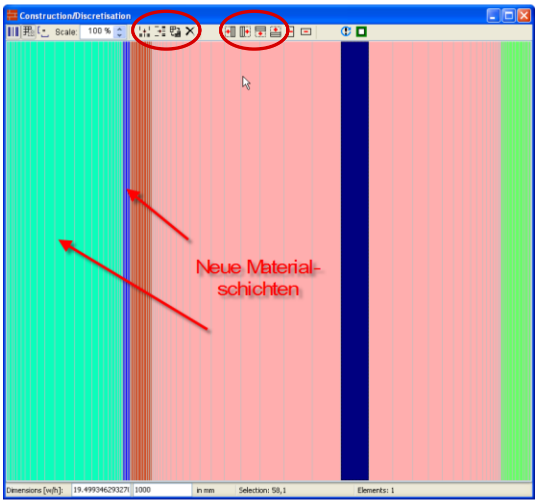
Part 2: Adding a Capillary Active internal Insulation

Since the thermal insulation does not meet modern requirements of buildings, an additional insulation is added. In this tutorial we assume that adding insulation at the outside is not possible (perhaps a historical facade, etc.) Therefore, a capillary active insulation system (using in this example the calcium silicate insulation) shall be used. The CaSi insulation is fitted using a glue mortar.

At first, the materials „Adhesive Covering Plaster“ and „Calciumsilicate“ need to be imported into the project from the material database. Then you can add two new layers in the construction view to the left of the innermost layer. (see command buttons in menu bar of construction view, right ellipse).

After specifying the thicknesses of the new layers (5mm coving plaster/glue mortar, 80mm insulation) using the input fields at the bottom of the construction view, you can assign the materials to the respective layers (see construction sketch at the very begin of this document).

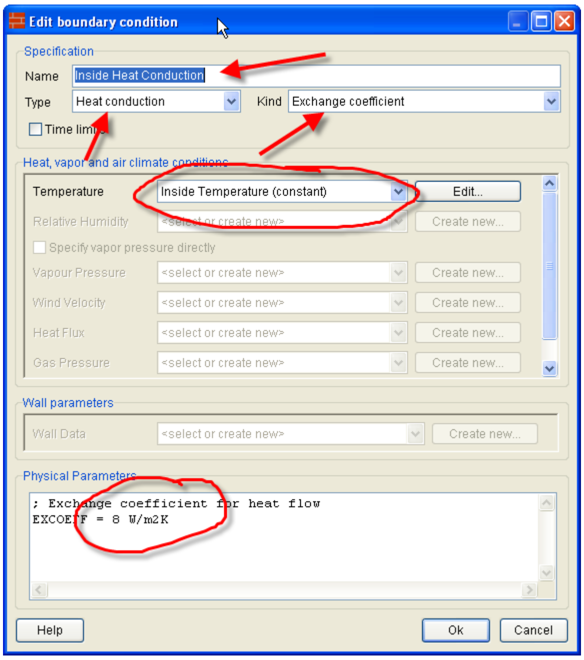
Finally, you can use the Automatic Discretization dialog again to create a grid for the new material layers, or manually discretize each layer with the Discretization dialog (see respective buttons in menu bar, left ellipse).



Whenever the construction has changed it is advisable to check that all boundary conditions and outputs are still assigned to the correct layers.

Now the simulation can be repeated (save the project with different file name first, so that the previous results remain for comparison).

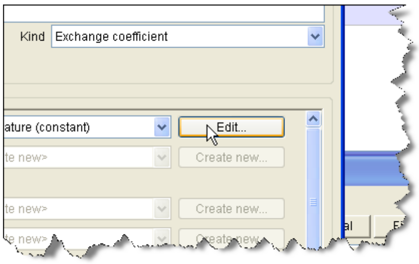
However, the selected boundary conditions in the project template do not quite match the EN/DIN requirements. Therefore, as the next step in this tutorial we will adjust the boundary conditions. The project template contains boundary conditions for heat conduction and vapor diffusion. Open, for instance, the heat transfer boundary condition with name “Inside Heat Conductction” (select the RB in the conditions view, “Boundary” tab, and click on the “Edit” tool button). Now you see the boundary condition dialog:

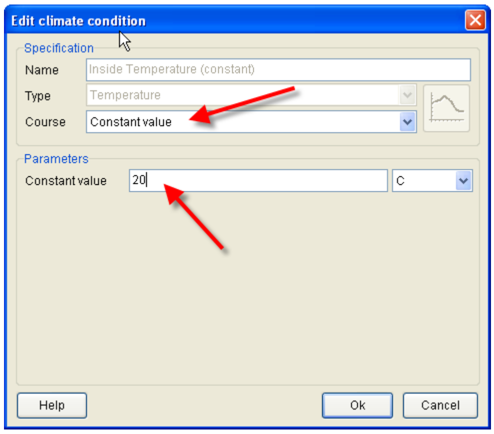


The highlighted inputs are necessary, to sufficiently define the boundary condition. Important properties for boundary conditions are always

- the unique identification name,
- the type (for instance, *Heat Conduction* or *Vapor Diffusion*),
- the kind of boundary condition (corresponding to a certain physical model),
- the required climatic conditions, and finally
- the parameters for the boundary condition model.

In this tutorial the default parameters for the boundary condition are correct, except for the climatic conditions. Depending on the boundary condition type and kind one ore more climatic conditions may be required. These can be selected and modified using the drop-down lists and the edit buttons besides the lists.





The dialog for the climatic condition allows changing and viewing of the climate data. Again, a unique identification name is required, as well as the type of the climate component (for instance temperature or relative humidity), the course (can be constant, following a sinusoidal wave, or double-harmonic function, or can be read from a climate data file). The course can be defined as constant, sinus curve, double sinus curve (daily and yearly course) or as arbitrary e.g. measured data course from a file.

For the definition of the climate on the inside and outside wall surfaces, the corresponding temperatures and relative humidities need to be adjusted. The outside climate should be adjusted to -10°C and 80% RH, whereas the inside climate should be set to 20°C and 50% RH.

If the simulation is repeated with the changed conditions, the amount of interstitial condensate should increase. If the inside insulation system works as expected, the total overhygroscopic mass obtained at the end of the condensation period should still be within the acceptable limits.

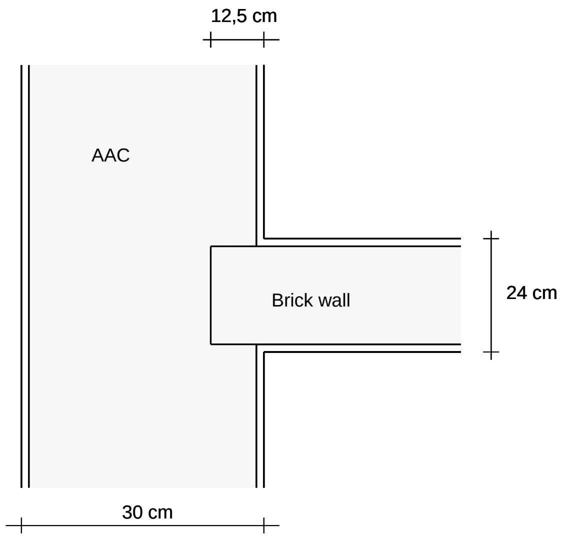
… End of 1st Tutorial …

Tutorial (2) Simulation of 2D construction details

This tutorial demonstrates how 2D construction details can be simulated with DELPHIN. The general approach on setting up the model and defining all boundary conditions and outputs is shown. A simple thermal bridge problem is modeled and calculated.

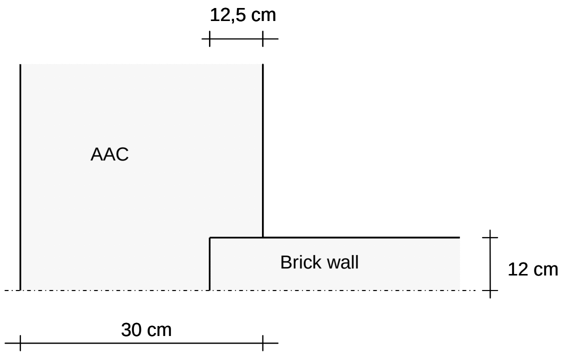
2D Thermal Bridge Problem

The detail shown below illustrates the problem of an intersecting wall in a monolitic exterior wall. This model should be simulated with DELPHIN. The exterior wall is made of autoclaved aerated concrete and the intersecting wall is a brick wall.



Simplification of the Constructional Detail

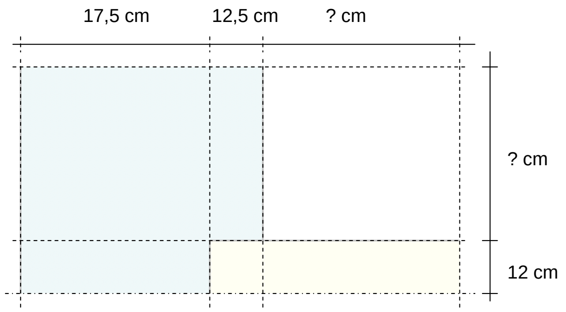
Before a construction is entered into DELPHIN it should be attempted to simplify the model. In this just thermal example, we can omit the plaster layers. Also we can make use of the symmetric geometry.



Define Construction Grid

Within DELPHN constructions are defined inside a rectangular construction grid. For this you need to define construction line first. The resulting rectangles are then filled with the corresponding materials, or in the case of whole left empty.

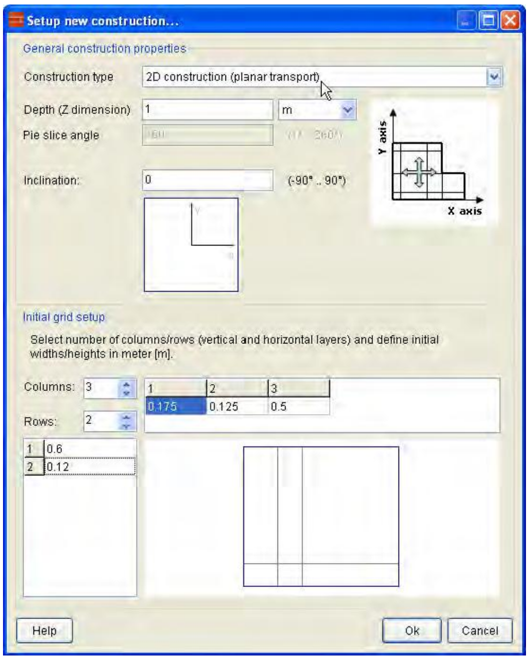
For our construction we can now define the following grid.



For the sections labeled ? we still need to define a length/width. We have to consider that the length of the intersecting wall and exterior wall are picked large enough, so that the influence of the thermal bridge is no longer important at the clip points. Usually, you can take a length of twice the wall thickness to get an estimate. In our case, we select 60 cm for the length of the exterior wall and 50 cm (rounded from 24 cm * 2) for the intersecting wall. If we should later see in the results that the influence of the thermal bridge extends to these clip points, we can just enlarge the sections a bit.

Input of the model into DELPHIN

We use the same procedure as shown in tutorial 1. When setting up a new project using the default template, we select 2D construction in the construction definition window. We also input the initial grid dimensions. Later, we can always change the row heights and column widths or add new layers.



When entering the basic grid geometry you should note that the **grid widths and heights are in m**.

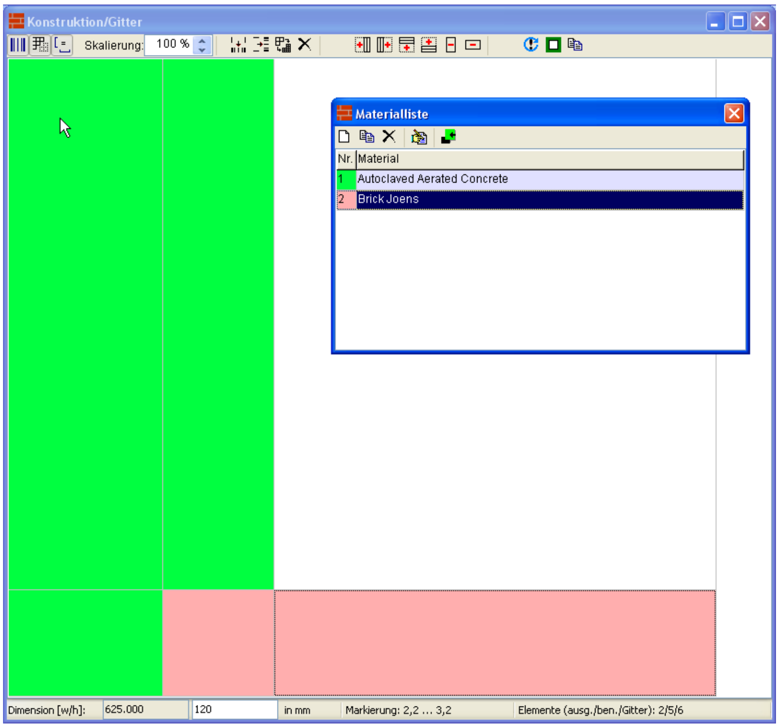
Now we import the materials and assign them to the respective sections in the construction. We use the **Autoclaved Aerated Concrete** and **Brick Joens** as materials.

In the following, we talk about the assignments. These are rectangular ranges of the construction which can be selected with the mouse or keyboard in the construction grid. In the footer of the construction window you can see the current selection, for example showing 1,1....3,2:

For 2D constructions you can choose among different possibilities to assign materials to the respective sections.

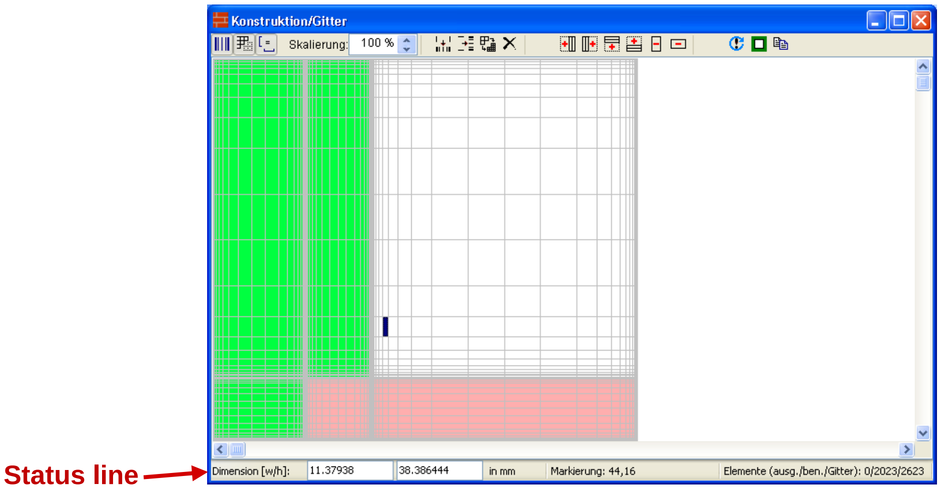
1. Subsequent selection of ranges with elements of a specific material and assignment until all elements have been assigned (can be a bit cumbersome with complex 2D details)
2. Making use of selections that overlay each other and overwrite previously made selections. In our example we could first select the range 1,1...2,2 and assign the AAC material. The we can select 2,2...3,2 and assign the brick material. With this operation we overwrite the AAC with brick in the element 2,2.

The second option allows often a quicker setup of the construction model. There is also a special case of a material assignments, particularly suitable for hollow sections or corners. In the tool bar you can use the symbol to assign an empty range. With this you can override previously made assignments (if you want to remove a material assignment, it is better to just remove the respective assignment in the assignments list window).



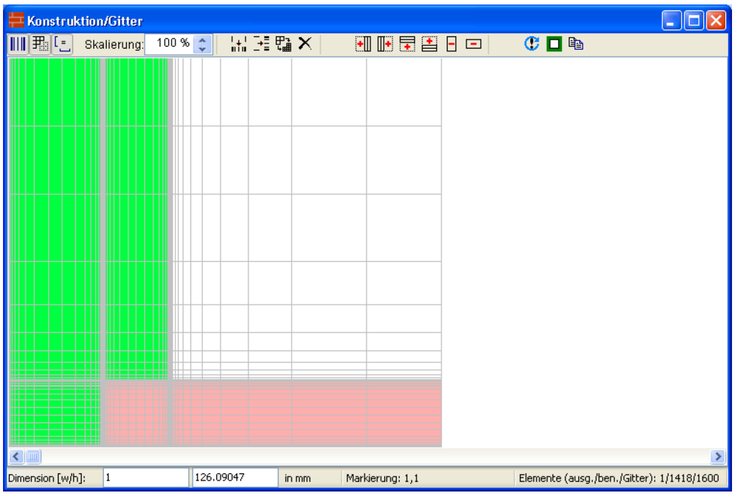
Grid generation

After we have created the construction grid and assigned the materials, the simulation grid needs to be created. This is done most easily with the dialog for automatic discretization. In this dialog you should make sure that both directions are checked (this is the default for 2D constructions), alternatively you can also specify the minimum element widths. In our example we use the standard settings and create the following computational grid.



At this point we may take a moment to explain some functions of the construction grid. In the status bar (footer) the current selection is shown, and also two input fields for the width and height of the currently selected column and row. If you have selected multiple rows or columns the respective input field just shows the sum of the widths/heights, but the input field itself is disabled. You can only change the row height or column width of a single row or column at a time. The status bar also shows the grid size in three numbers. First number is the number of currently selected elements, second number is the number of used grid cells (elements with materials assigned) and the last number is the total grid size. A value of < 6000 is generally to be recommended for quick calculations. For purely thermal calculations you may use values up to 30000 and still get reasonable performance.

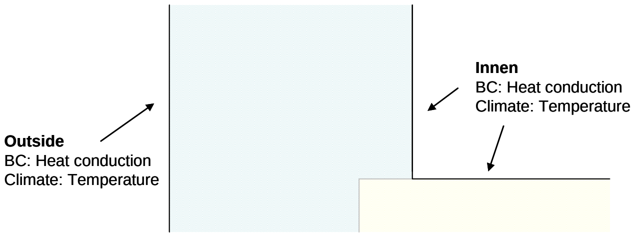
The automatic discretization algorithm has generated a grid that is refined at all boundaries. However, we have a symmetry plane at the bottom and top side of the construction. Here we don't need the refinement because we don't have temperature gradients across a symmetry line. Therefore we could save grid elements by manually enlarging the elements using the manual discretization dialog, accessible via the toolbar buttons . Saving grid cells can significantly increase the calculation speed in hygrothermal calculations of complex details. In our simple example, however, this is not necessary.



The first top tool buttons in the construction window serve to alter the display of the simulation model in the construction grid: proportional versus equidistant drawing, show/hide grid line, draw with the same aspect ratio or stretch both axes to fill the window.

Assigning boundary conditions

We need to assign boundary conditions (BC) to each boundary of the construction. The following sketch illustrates the necessary boundary conditions and the related climatic conditions.

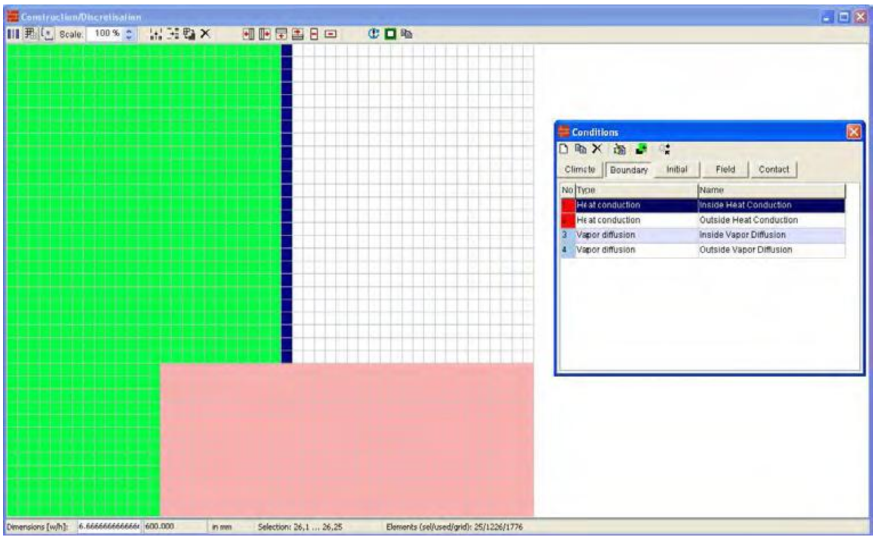


At symmetry lines and at borders of the calculation domain (where 1D conditions are expected) you do not need to assign boundary conditions. A boundary without assigned conditions is perfectly tight and adiabatic. In our example we just need heat conduction boundary conditions with the corresponding temperatures.

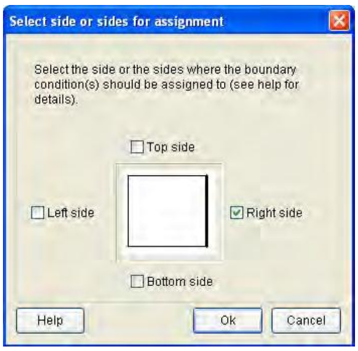
The default project template already contains climatic and boundary conditions, which we will just use here. The assignment of boundary conditions is done as described in detail in Tutorial 1:

1. Select range of elements where boundary conditions need to be assigned
2. Select boundary conditions
3. Assign boundary condition (by clicking on the green arrow button in the condition view)

The following screenshot shows how the boundary condition is assigned to the inside wall surface. It is recommended to use the equidistant drawing option of the construction grid to simplify selection of the element range.

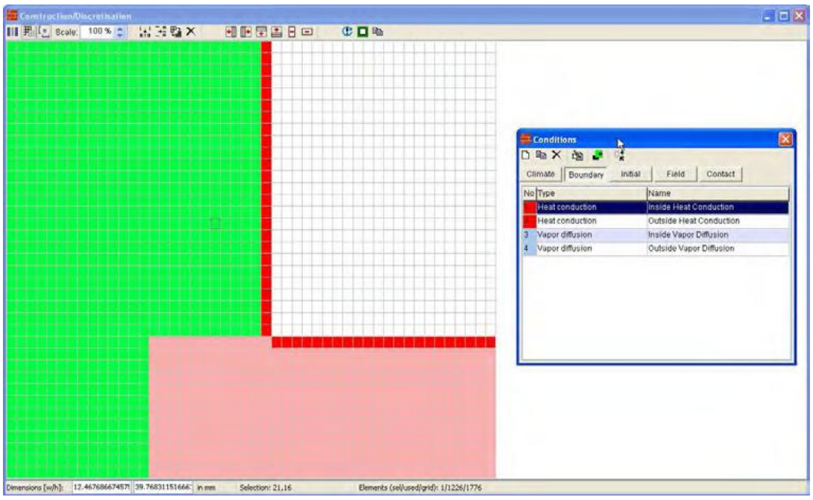


After clicking the assignment button a dialog pops up with the side selection options.



Here, you need to select the respective side of an element for the assignment of the BC. You have to remember that **boundary conditions inside the construction grid (i.e. between materials) are invalid**.

When all boundary conditions have been assigned, it is possible to verify the placement of the assignments by clicking on individual boundary conditions. For example, if you select the heat conduction boundary condition for the inside, the construction view will highlight all elements that have this assignment.



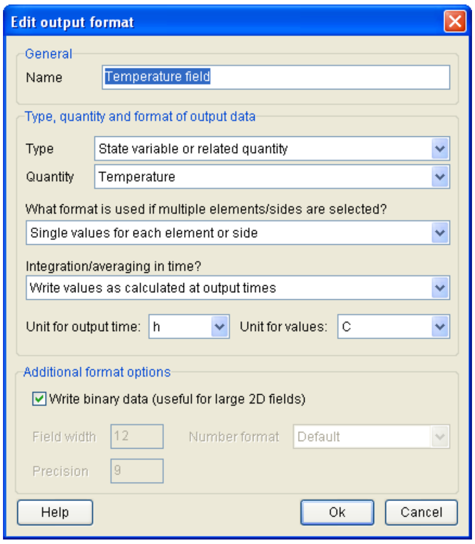
In the assignment list window you can also check all assignments individually. There should be 3 boundary condition assignments in total: one for the heat conduction at the outside, and two for the heat conduction at the inside, just as it is shown in the screenshot above.

Tasks

In a simple thermal bridge calculation the temperature field is interesting purely for visual analysis, e.g. to find the point with lowest temperatures in complex geometries. Critical for design decisions and code compliance are, however, the temperatures in critical sections, most importantly the inner wall surfaces. Also of interest are the actual heat fluxes through the inside wall surface to quantify the heat loss through the thermal bridge.

The post-processing allows extracting temperatures at specific points as well as horizontal and vertical cuts. Therefore we will first assign a temperature output to the whole construction.

The default project template already contains some pre-defined output formats. In the output window you may, however, also define your own output formats. The following screenshot shows a possible definition of a format for a temperature field output.

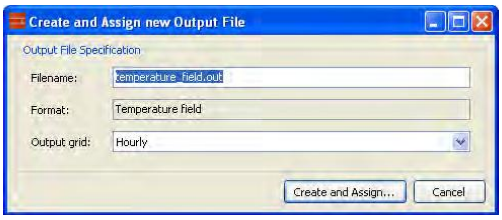


The assignment of a temperature field output is done as usual in the three steps:

1. Select range of elements for which the output is to be obtained
2. Select output format (or output file, in case you want to combine several selections into a single output file)
3. Assign the output format or output file definition

If an output format is assigned, a dialog is shown where you can specify additional information for the output file to be created (output format, output time grid, and file name of the output file).

In our example, we want to create an output for the whole construction. The keyboard shortcut Ctrl+A can be used to select all elements, alternatively you can also select the whole construction with the mouse. When specifying the output time grid, it is here meaningful to use hourly values (even though we are just interested in the final steady-state result).

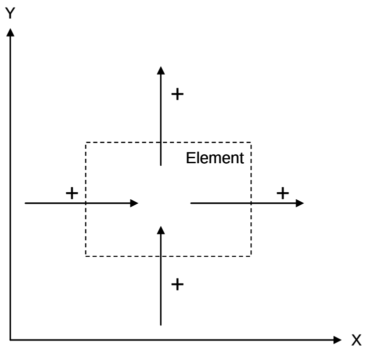


Sometimes you may just want to obtain an output of a part of the construction, for example, the total moisture mass in a certain material. In our case, we could just want to obtain the temperature field of the aerated concrete (even though this doesn't really make much sense). However, we have to assign an output now to an element range that is not rectangular. So we have to make two assignments. For such cases, the following procedure is recommended:

1. In the output window select the register card "Output Files"
2. Create a new output file definition (press "New" tool button) and select here the temperature field as output format, a time grid and a file name, for example "temperature_field_aac.out" and confirm the dialog
3. Now select the element ranges covers by the AAC material in two steps and press the assignment button in the output window. When done correctly, a click on the new output file will highlight just the AAC material.

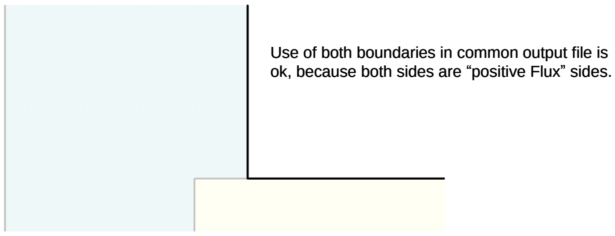
When obtaining outputs of heat flux of other flux densities, one has to keep in mind the sign definition for flux quantities in DELPHIN. If you combine flux outputs not correctly, sometimes flux densities with the same direction (for example heat flux from the inside to the outside) may cancel each other out.

The following diagram illustrates the sign definition used in DELPHIN.

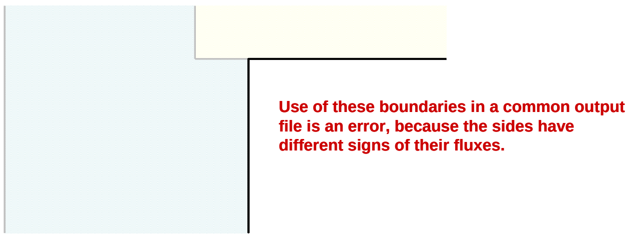


This sign definition is used independently whether a side is between two elements or a boundary side. Flux densities in direction of the coordinate axes are always positive. That has the advantage that one can define cut planes through the construction and obtain the heat flux through this cut plane, as long as the sides used in the cut plane are of the same sign.

An example may help to illustrate this. If we want to obtain the heat flux from the inside to the outside in our example thermal bridge, we need to output the integral (spatial integral) of the heat flux densities across the inner surface. If we define the construction as we did before, we could simply create an output file for integral heat flux densities and assign this output file to the two interfaces of the construction to the inside, as shown in the picture below.

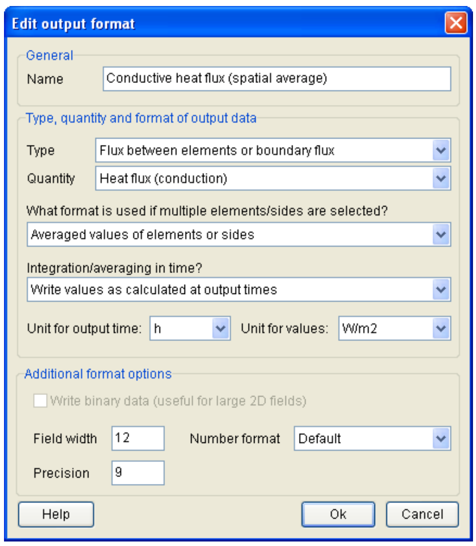


Both boundaries have the same sign definition which allows us to combine the two outputs into a single file for the integral heat flux density.



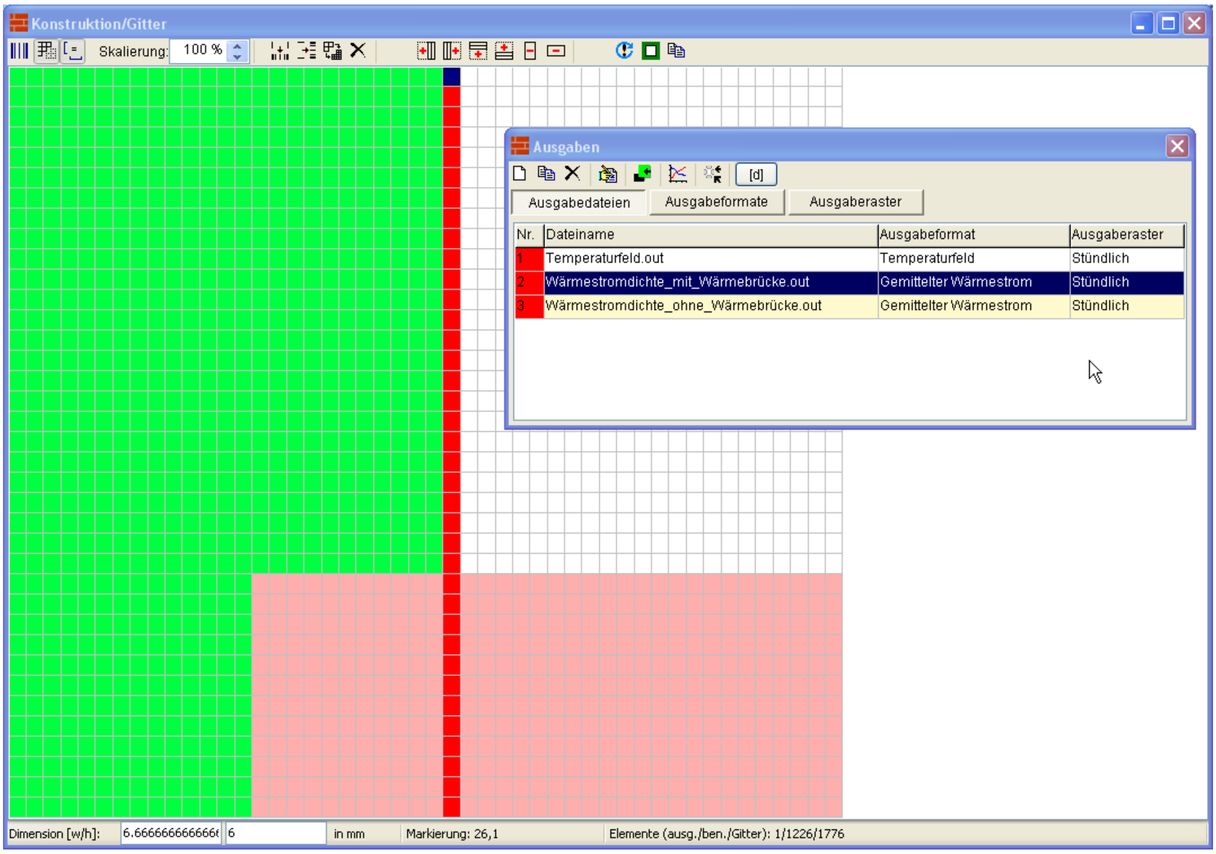
If the construction were defined as above, the assignment would be wrong. A heat flux from the inside to the outside would yield a positive flux for the top interface (flux in positive y-direction), yet the flux through the vertical interface would yield a negative flux (against the x-direction). When you add these up in an integral heat flux output the flux would cancel each other, even though both flux indicate a positive energy flux from the inside to the outside.

This strict sign definition has, however, the advantage, that the heat loss through the construction can be obtained much more easily. First we use the output format for the averaged heat flux density.



When defining the output it is important to set the output type to a "Flux type" and select the weighted average as calculation method.

Once the output format has been defined, the output can be made by placing a vertical cut through the construction, as shown in the following screenshot.



For comparison you can create another output just for the inner boundary element of the first row. Here we are far away from the thermal bridge and should have (approximately) one dimensional conditions. A heat flux obtained here corresponds to the heat flux of the wall without the thermal bridge. We can use this as reference value for the rest of the wall.

Simulation Settings and Starting the Simulation

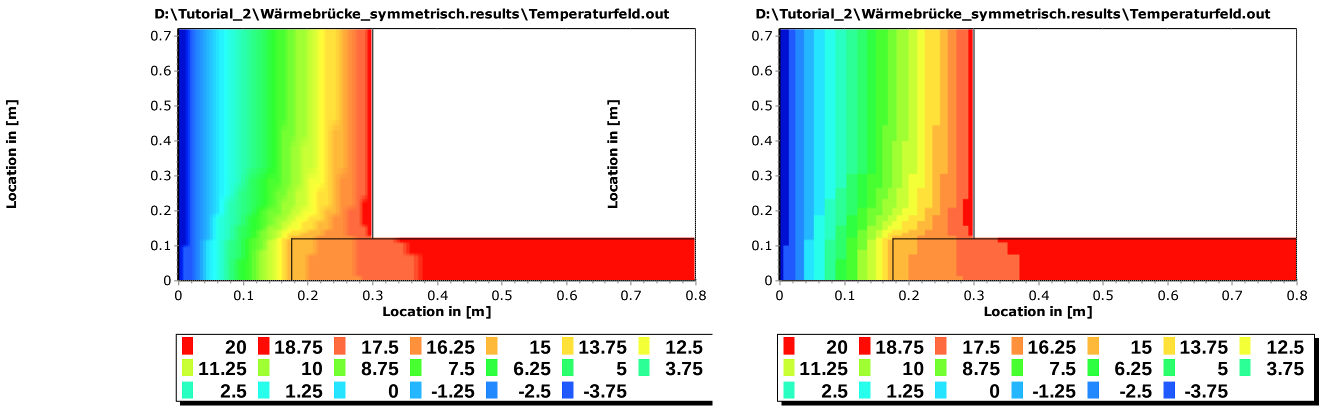
Once the construction is modeled and boundary conditions and outputs have been assigned, we only need to set the simulation options. In the simulation settings dialog we deactivate the moisture mass balance, and reduce the simulation time to 2 days. Within this time we should reach a steady state.

Now the simulation can be run and should be finished within a few seconds.

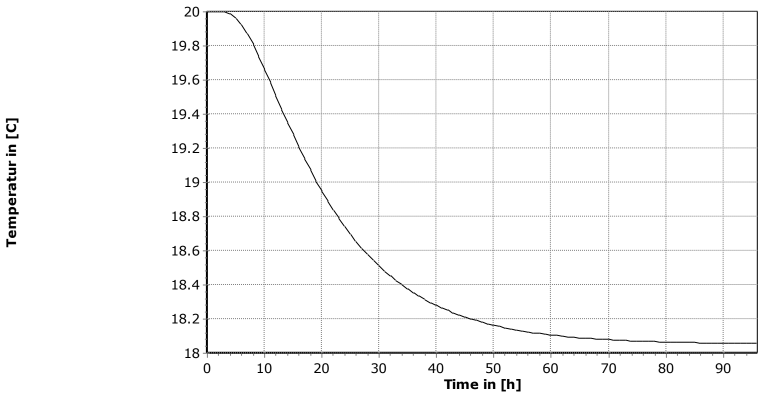
Post-Processing

The analysis of the outputs is done with the DELPHIN post-processing tool, or alternatively with other external tools such as TecPlot. The details of how to use the DELPHIN post-processing is done in a separate tutorial, here we will just analyze the results.

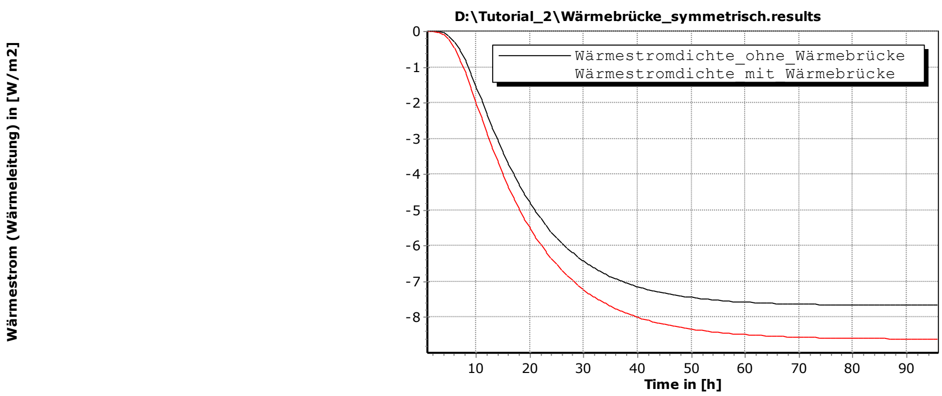
The calculated steady-state temperature field is shown below in the interpolated and discrete mode. In the latter mode (right) we can clearly see the underlying computational grid. In areas with large gradients the grid should be refined for more accuracy. Furthermore we see that at the top of the construction we just reach 1D conditions whereas the intersection wall is much too long (nothing happens further to the right). So we could shorten the wall at the right side by about 30 cm and save computation time (in our example this isn't relevant, but it may be in transient hygrothermal simulations).



When we obtain the course of the temperature at the critical pointer (inside corner), we see that even after 2 days we haven't reached steady-state yet. For the screenshot below we have continued the simulation for another 2 days (first change the total simulation duration in the simulation settings dialog then use the "Continue" button in the simulation start dialog).



If you now consider the heat flux outputs below, we notice the negative sign of the flux, which means against x-axis and therefore from inside to outside, as expected.



Also, we see that the thermal bridge does increase the heat flux through this part of the wall by about 12 % compared to a plain wall (red curve is the output with thermal bridge, black is the reference).

Summary

In this tutorial we saw the principle steps of setting up a simulation model of a 2D construction detail. Similarly, one can setup a simulation for hygro-thermal problems or much more complex geometries.

Problems

1. Model the construction above without considering the symmetry and compare the results!

2. Refine the grid by adjusting the discretization options and compare the steady-state temperature in the wall corner (mind that after modifying the discretization you may need to update the assignments for flux outputs)!

3. Add an inside insulation and compare the influence of the thermal bridge on critical temperature and heat fluxes!

4. Run the simulation with and without inside insulation as hygro-thermal simulation. In addition to the heat conduction boundary condition, you need to specify also the vapor diffusion boundary conditions.

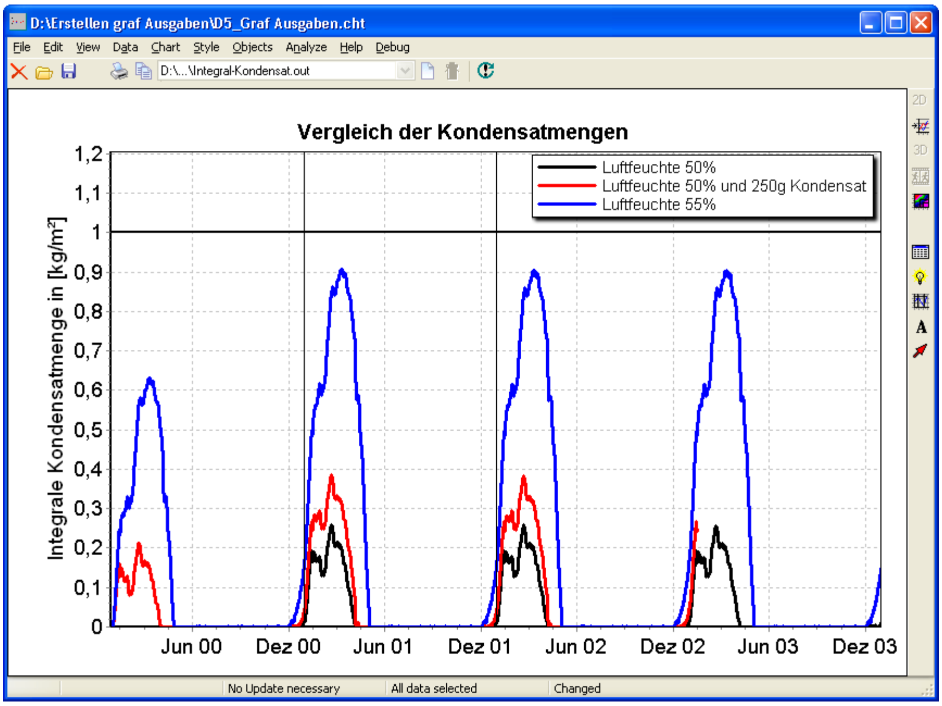
… End of 2nd tutorial

Tutorial (3) Postprocessor

Graphs of state variables

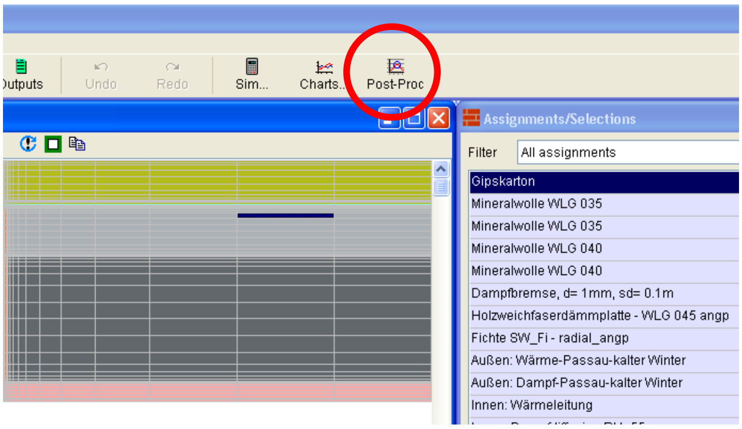
The Delphin-Postprocessor provides manifold possibilities to customize graphical outputs to own conceptions. This tutorial explains how

- the postprocessor is started and result data are selected,
- chart headlines are customized,
- multiple lines are displayed in one chart,
- a legend is added and the appearance and labelling of single lines is customized,
- axes and labelling of axes are customized and
- supplemental lines are added.

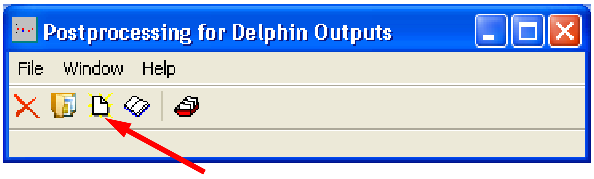


Starting the postprocessor and selection of results

Start the external postprocessor by clicking on the corresponding symbol in Delphin5.



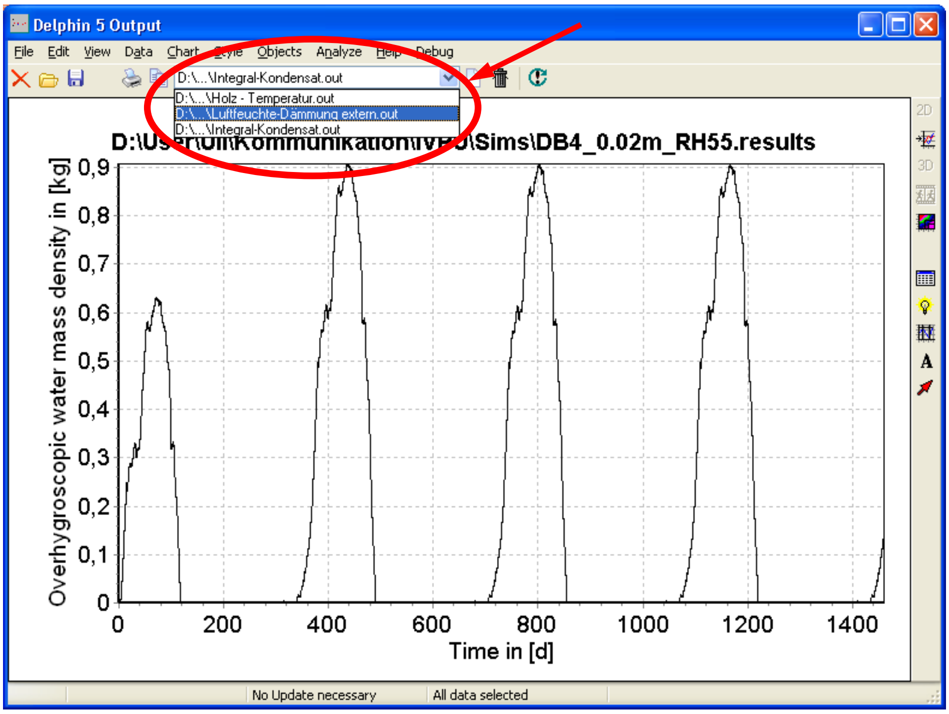
Thereupon the postprocessor will open. To open result data click on the button „New Chart from Data File“.



A conventional Windows dialog will open. Choose the desired data. If you want to work on more than one file from the result folder, keep pushing <Strg> resp. <Shift> while clicking on the files.

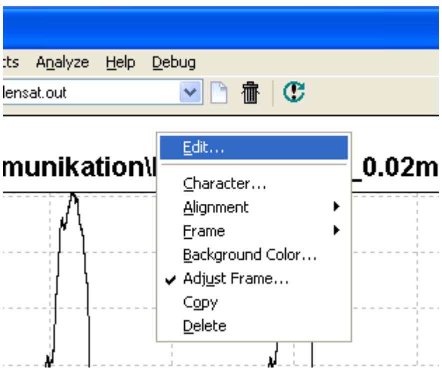
If more than one file has been chosen Delphin assigns result data with identical units, e.g. °C or kg, to one chart.

In the so called „chart“-view several charts can be regarded and edited by choosing the corresponding chart in the drop-down menu.

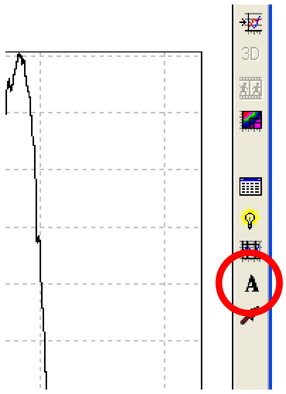


Customizing chart headlines

With a right-click ON the chart headline you have access to the headline dialog, alternatively with >> Style >> Title, too. If you intend not to display a headline choose after right-clicking on the headline >> Delete.



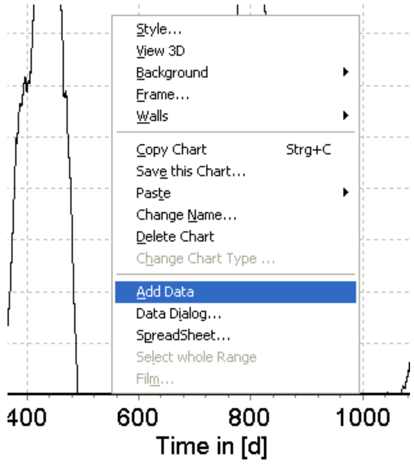
By choosing >> Edit after a right-click on the headline a dialog appears where the labelling and some other settings regarding the headline can be changed. Confirm your changes with a click on >> OK.



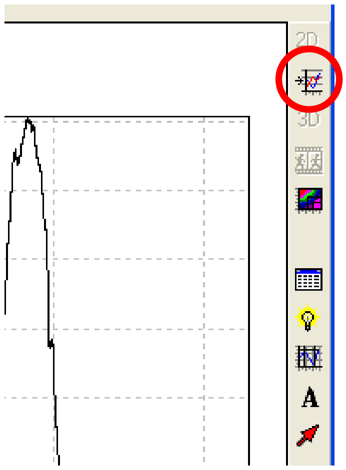
To display any text at any place in the chart you may open the Delphin text editor (right picture). But to guarantee a consistent typeface, headlines may be added better within the word processing program.

Displaying multiple lines

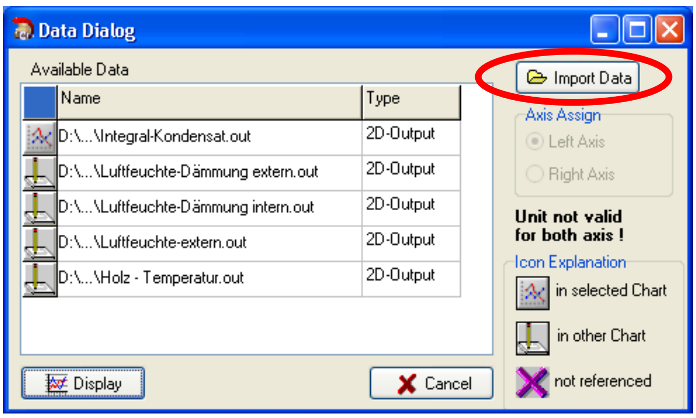
To display more than one line in one diagram or to add a new line choose neither >> Data >> Add Data or right-click on the diagram but NOT on axes, axes labelling or lines, and choose >> Add Data. Another possibility is a click on the button in the toolbar (right picture).



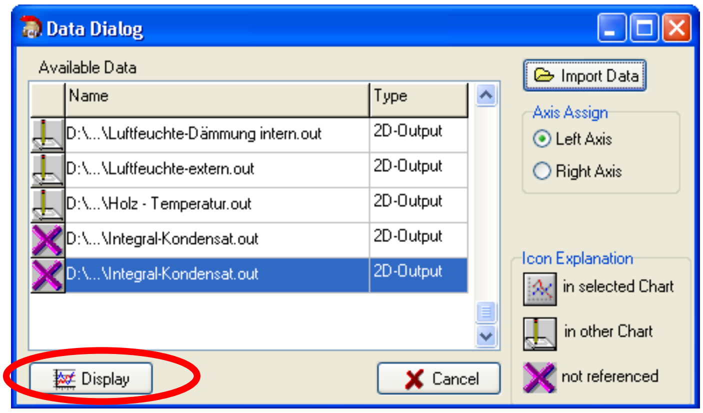
OR



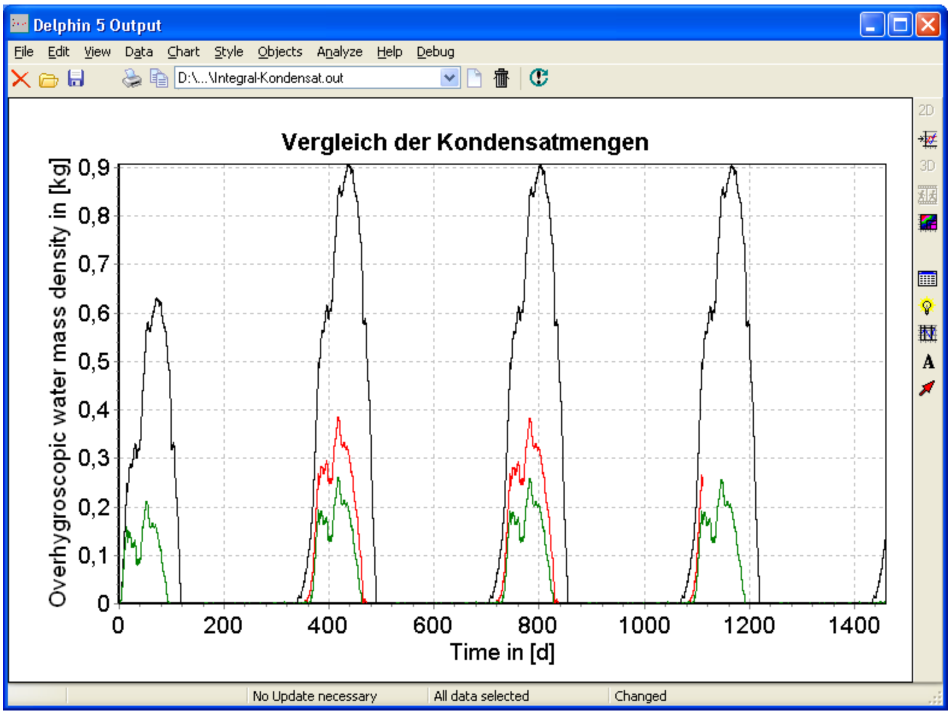
The Data Dialog pops up showing which data are stored in ALL charts of the outputfile. With a click on >> Import Data more result files can be loaded.



If you import more data, these are not displayed automatically in the chart currently opened. Therefore, a violet cross on the left of the new imported list entries is visible. To assign them to the opened chart view, choose the chart in the list and click on >> Display. To choose resp. display more than one file use the <Shift> button.

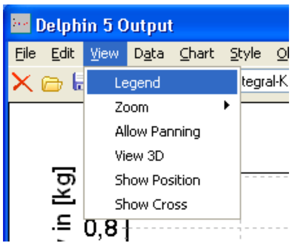


After importing and displaying two more graphs the following postprocessor output can be seen within this example, the headline has been changed in the previous chapter:



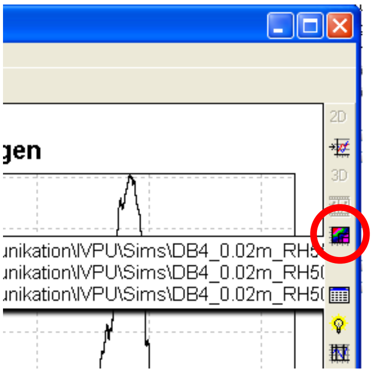
Adding a legend, customizing appearance and labeling of single lines

A legend can be mapped with >> View >> Legend. If you left-click on the legend and keep the button pressed, the legend can be moved to any place in the chart.

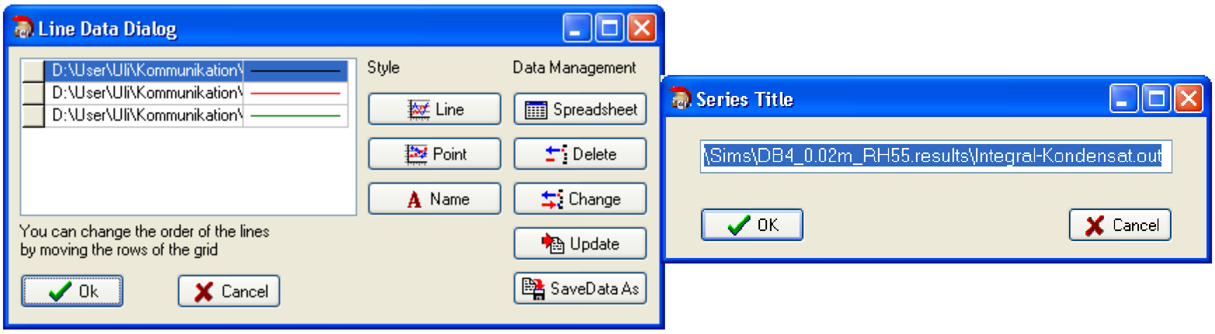


To customize the general defaults of the legend choose after a click on the right mouse button and choose
>> Legend Dialog.

If you want to change the labelling of single lines, line width or colour click on the Line Data Dialog in the right vertical bar of the chart view or in the menu bar >> Data >> Line Data Dialog. Depending on the type of the chart there are more possibilities to access this dialog.

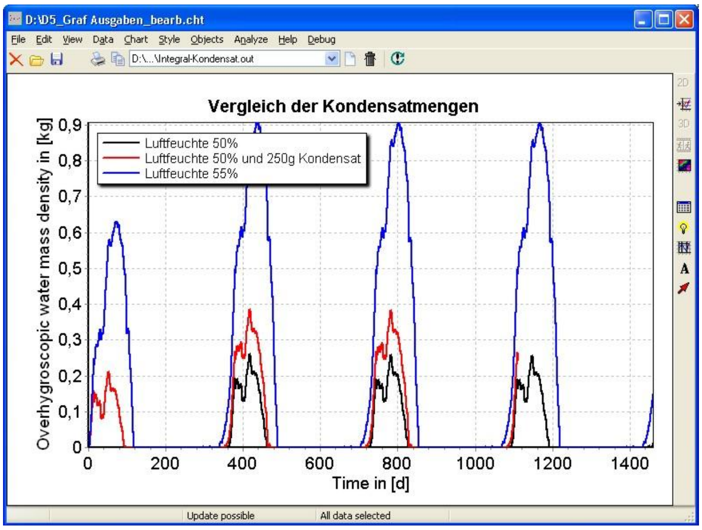


Here you have manifold possibilities to customize graph related settings according to your conceptions. To CHANGE THE LINE LABELLING double-click on the desired graph in the list, change the name and confirm it by clicking on >> OK.



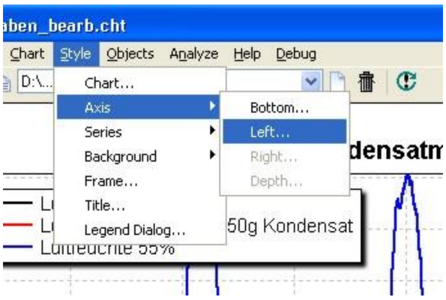
The LINE WIDTH and COLOUR can be changed by clicking on >> Line in the Line Data Dialog. In the dialog >> Point the display of lines can be specified more exactly e.g. whether in place of a continuous line rhombi or circles of different size or colour shall be used.

Via >> Spreadsheet every single stored value at every single output point in time can be revised and, if necessary, changed.

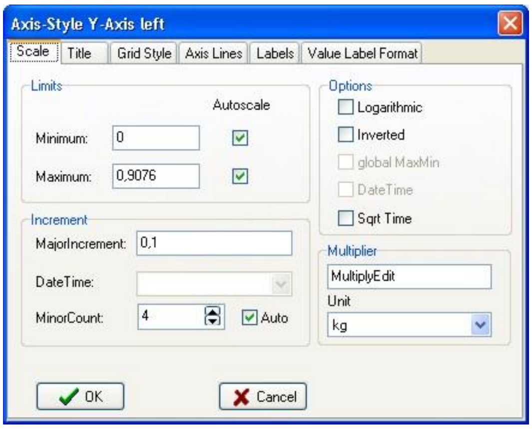


Customizing axes and labelling of axes

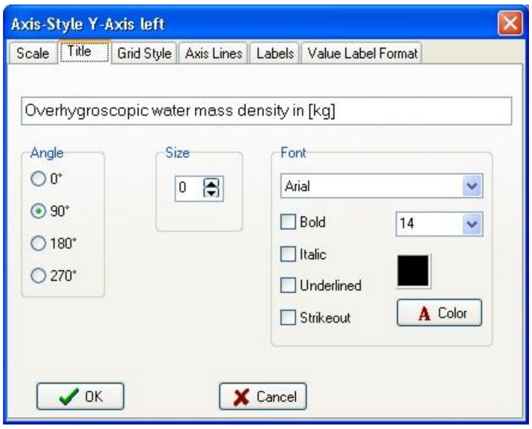
To customize axes labelling, axes section or character etc. you have to access to the favoured axis first. This can be done neither by double or right mouse click on the axes or with >> Style >> Axis >> Top resp. bottom resp.…



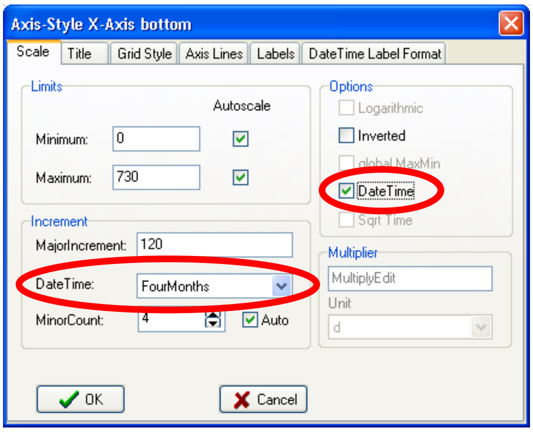
In the first tab „Scale“ MINIMUM AND MAXIMUM VALUES OF AXIS can be adjusted. In the present example the maximum for the y-axis is set on 1.2. Maximum and minimum values are set automatically if >> Autoscale is switched on. In the box „MajorIncrement“ the difference between visible values of the axis division can be defined. If the margin between single numbers becomes too small for displaying them, Delphin disregards the user value at „MajorIncrement” and uses a higher value automatically. >> Logarithmic causes the logarithmic division of the axis, >> Inverted inverts the division.



The AXIS LABELING can be changed in the second tab „Title“. Here, in the box „Size“ the MARGIN of the axis labelling to the left border can be customized. A „0“ at „Size“ causes the automatic calculation by the software Delphin. This setting sometimes causes at numbers with many decimal places overlapping.

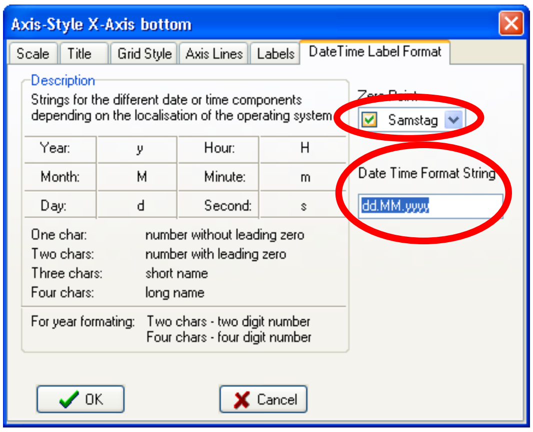


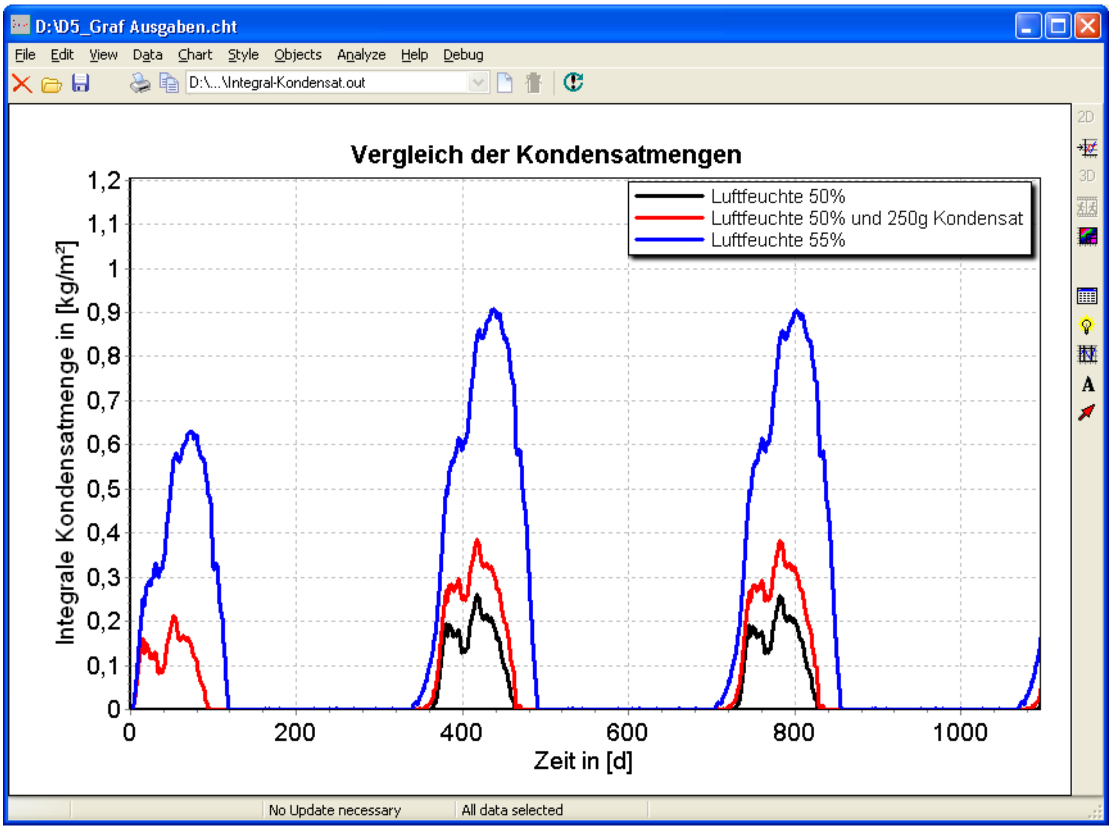
In the same manner modifications of the horizontal x-axis can be carried out. If concrete date display shall be indicated on the x-axis, activate the flag „Data Time“ on the right hand side. The time interval can be chosen on the lower left hand side.



The FORMAT of the date display as well as the SPECIFICATION OF THE FIRST POINT IN TIME can be set in the tab „Date Time Label Format“.

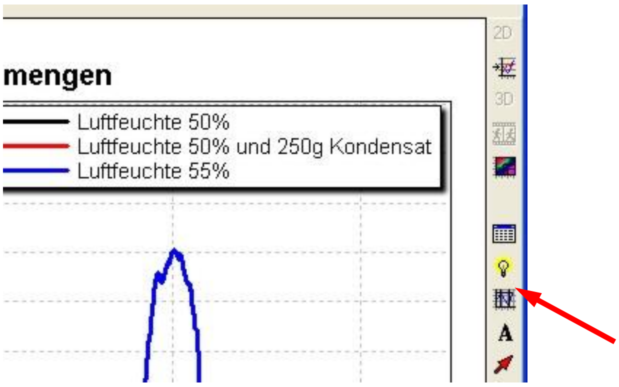
Specification of the first point in time Format



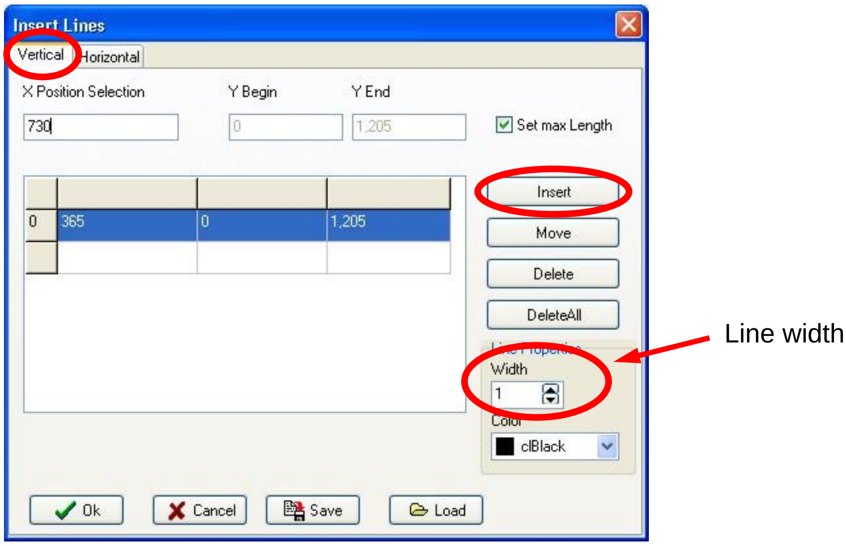


Adding horizontal or vertical lines

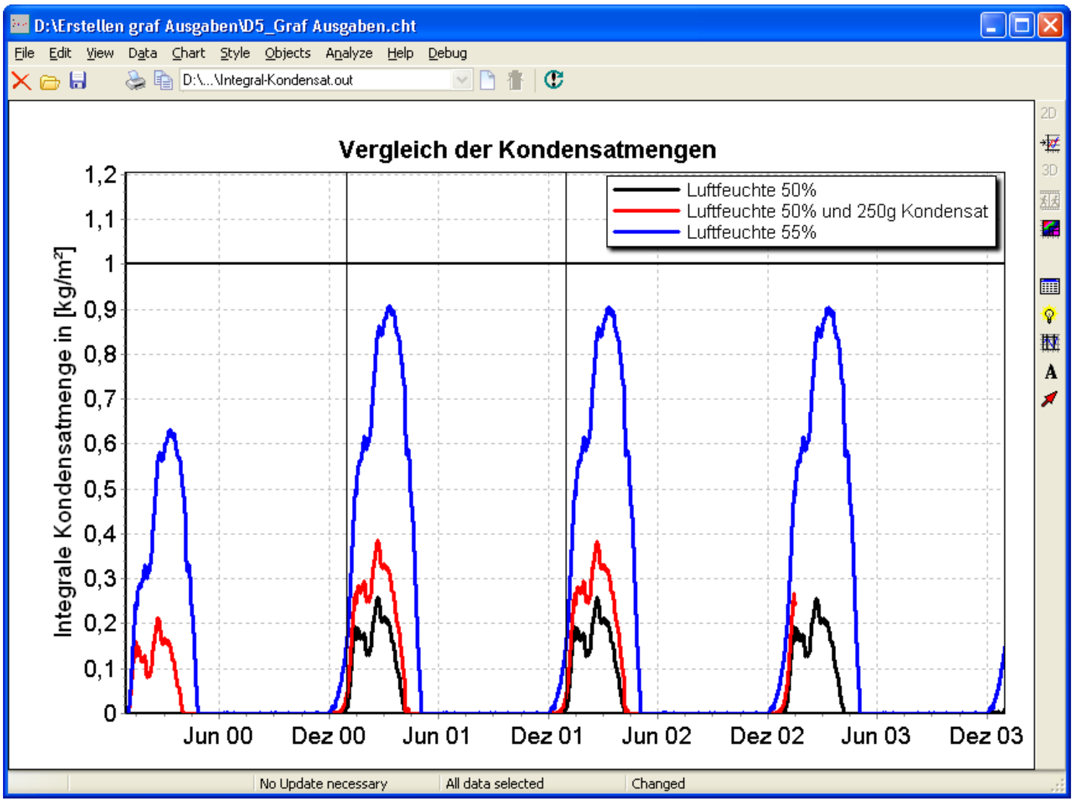
Additional horizontal or vertical lines can be added after clicking on the corresponding button in the right vertical bar or with >> Objects >> Included Lines.



For instance, to separate single years more clearly from each other choose the tab „Vertical". Input at „X Position Selection" 365 and, to safe additional time and effort in input, choose „Set max Length", if the line shall run on the entire height of the chart. Do not forget to confirm every input with „Insert", otherwise inputs get lost! It is possible to save and load the whole *collection* of inserted lines. In so doing, lines have not to be drawn in every chart of similar constructions again.



Terminatory, the limiting value of one kilogram per square meter is drawn as horizontal line and slightly thicker:

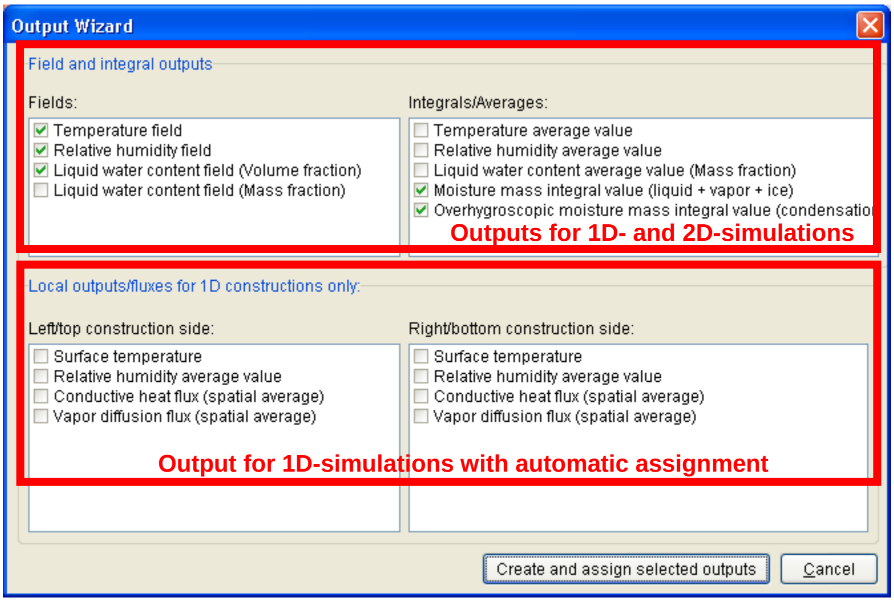


Tutorial (4) Defining of (result) outputs

Delphin offers a number of canned outputs that can be automatically generated. Nevertheless it could furthermore be useful to define further outputs. Both will be explained in this tutorial.

Usage of the output assistant

This button calls up the assistant for the result output. Thereby the following dialogue window will open:

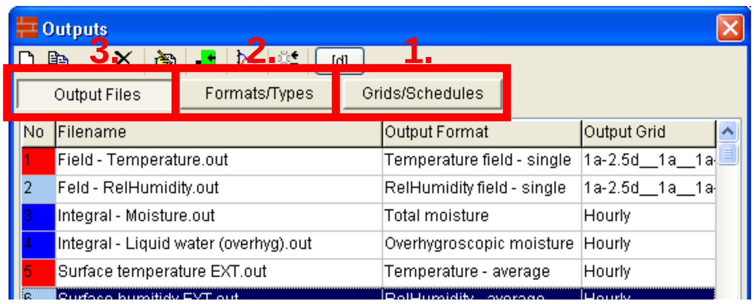


The outputs of the area above are automatically assigned to the entire construction. This assignment can be altered afterwards. The old assignment in the window >> Assignments-/Selections should be deleted in order to change an effected assignment. Afterwards the requested output can be clicked on in dialogue window >> Outputs >> Output files and can be assigned with .

Outputs of both areas can be clicked on whereat the outputs of the area below are only useful for a one-dimensional simulation. They are automatically assigned to each respective surface or more exact to the outer discretized elements.

Defining of own outputs

Three steps must be executed for the definition of own outputs:

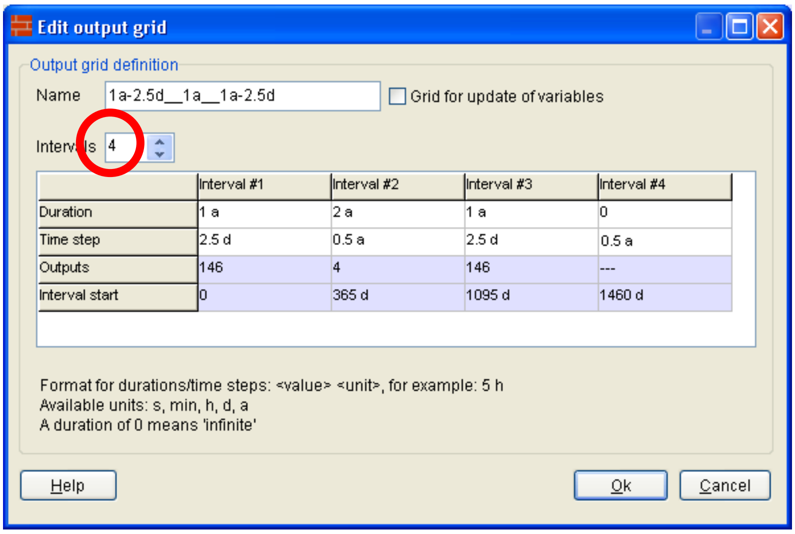


1. Grids/Schedules: Determination of the time grid
2. Formats/Types: Determination of the output type
3. Output Files: Assignment to elements with determination of the output file name

Determination of the time grid

In general output formats with hourly and daily output rhythm are already predefined under Grids/Schedules. The definition of further output grids can be initiated with a click on 🗋 or a right click on the table below and >> New output … . An output grid with different sections should be generated in the following.

Two-dimensional field outputs, being explained later on explained in a more detailed way, can result into large output files influenced by a to fine temporal output grid. Their processing in the postprocessor occupies much time. In principle the outputs are thereby often practical redundant, since the individual annual outputs differ visually little at a cyclically recurring climate. Therefore it can be useful to only display the first and the last year of simulation.

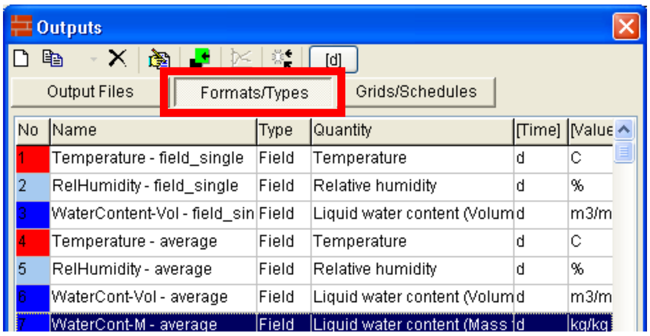


Therefore the definition of four different intervals is useful. Initially four years are estimated for one simulation displayed in the figure above. The first interval for the first year (duration = 1a) is planned with shorter time steps since at the beginning of the simulation the construction is not yet settled. Then, in dependence of the construction, a period of time may follow during which hardly any condition images are saved (here 2 years: duration = 2a). Thereupon the construction should be displayed at the settled state, which is why a third interval of the length of one year is determined. Subsequently, if the simulation is to be continued, only one output grid is defined here for ALL of the following years, which is why a ‚0' will be entered for the duration. ‚0' stands for unlimited, infinite period of time.

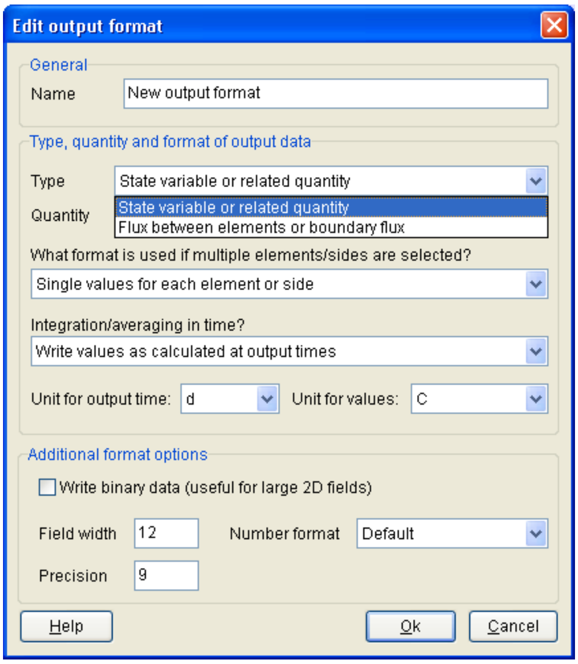
The rough output grid was selected with 0.5 a, meaning there will only be one output every half year. The finer output grids add up to 2.5 days = 2.5 d. Thereby alternating between noon and midnight an output is saved, so that no false impressions accrues especially concerning temperature images. Would a daily output rhythm be agreed upon, only the cooler temperatures at midnight would be on display.

Determination of the output type

The output types can be determined under >> Formats/Types in case it should be output more detailed.



The output types can not only be defined in this window, but also with associated with elements. The definition of additional output grid can be started with a click on or a right click on the table below and >> New output ... .



It is advisable to give the individual output types at "Name" meaningful names, so that the name already clarifies the content of the output. This avoids any possible confusion.

“Type“ and „Quantity“:

First, at >> Type it must be decided between the output of a condition variable and a flux. The >> Type: Flux between ... include, for example, the heat flux or the convectively transported water vapour flux. The >> Type: State variable ... include temperature, air humidity, degree of saturation or mass-related moisture content.

The self-explaining selection at >> Quantity automatically adjusts to select the combination of >> Type.

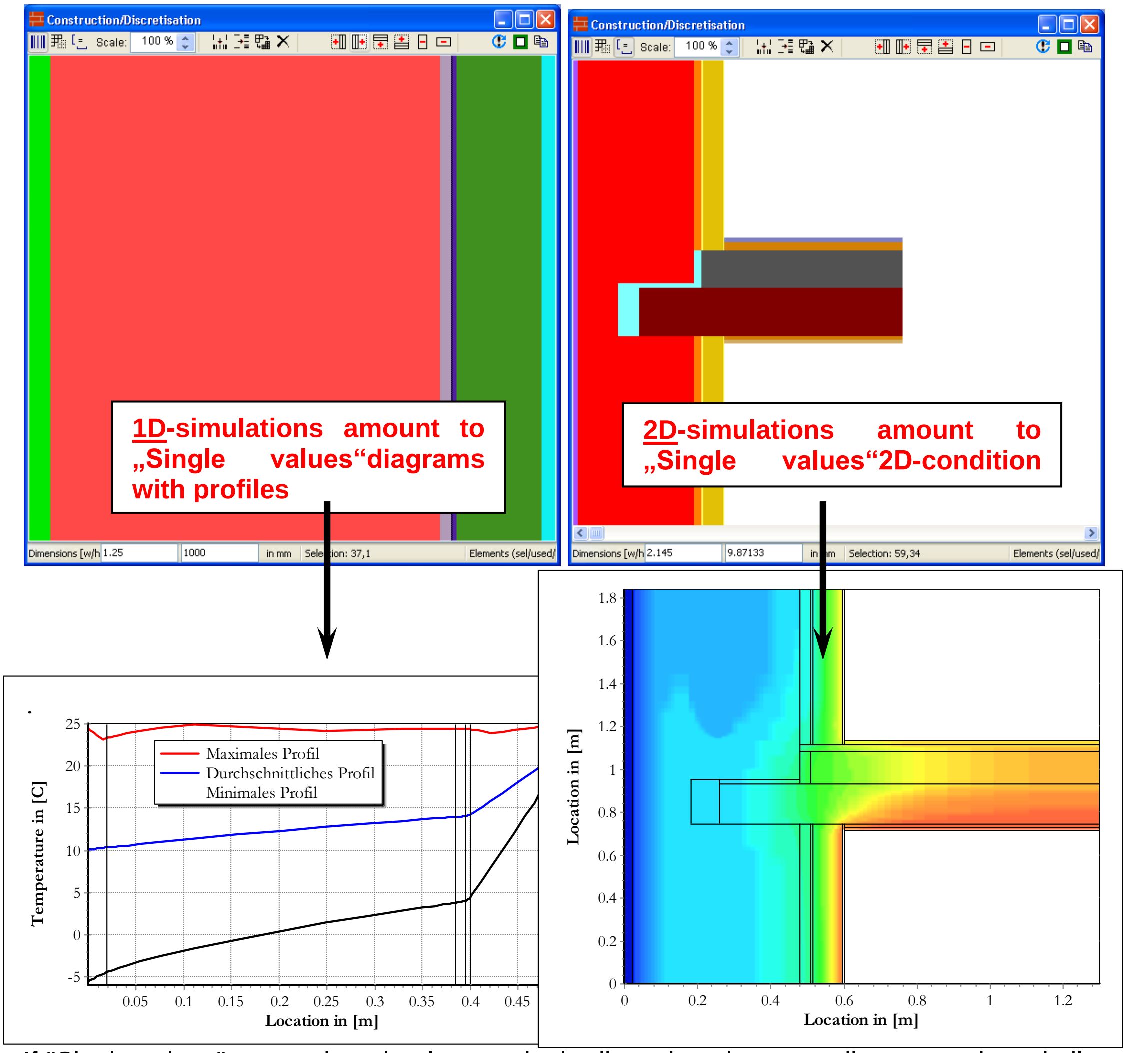
“What format is used if multiple elements/sides are selected?“:

Three different formats are distinguished here:

- “Single values for each element or side”
  If multiple elements are assigned to this format, a value of **each** element is saved. With this format one-dimensional wall profiles or profiles in two-dimensional details can generate

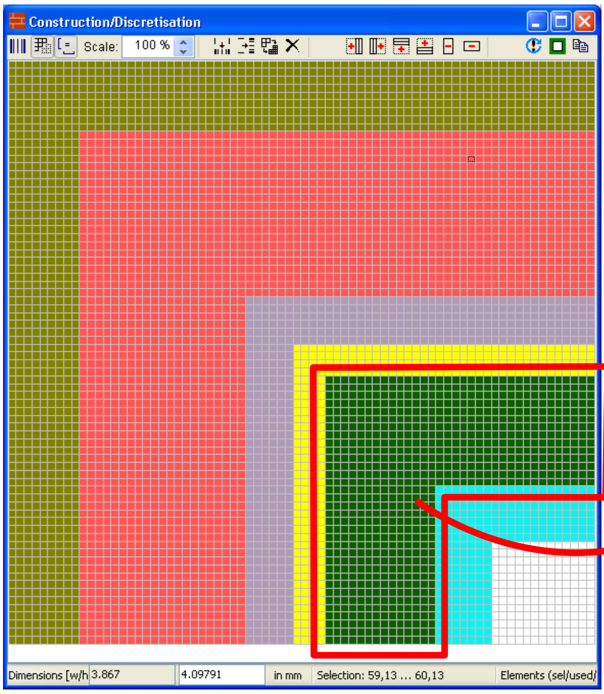
"colourful images". The outputs can get very large for two-dimensional simulations. Larger output time steps should therefore be chosen here as described in "Determination of the time grid".

One-dimensional wall construction: Two-dimensional detail:

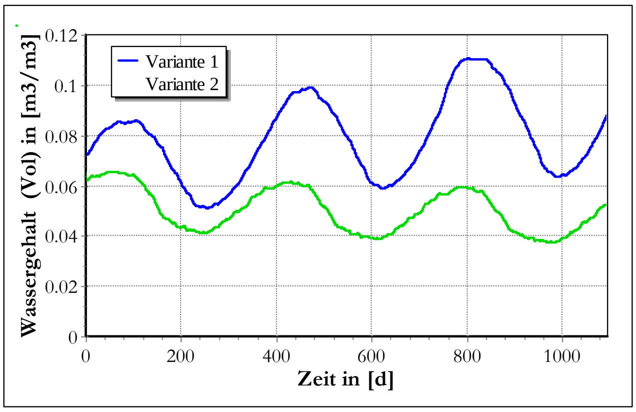


If "Single values" are assigned only to a single discretize elements, diagrams arise, similar to the assignment of "Averaged values ..." or "Integrated values ..." with the time axis as the x-axis.

- “Integrated values in space”
  If multiple elements are assigned to this format, only **one value** is saved of **all selected elements** at any time of output. However here the result values of all fields are integrated. Delphin only accepts summing of reasonable output variables, for example temperatures of several fields cannot be integrated.
  The output diagrams look similar to >> Averaged values ... .
- “Averaged values of elements or sides”
  If this format is assigned to several elements, all selected elements are saved at each time of output. This is the average value.



Averaged values amount to time-depending lines in x-y-diagrams (below two possibilities are compared)



"Integration/Averaging in time":

At present only one possibility is given for the exact definition of the time of output or area of output. There the output value is output at the end of each interval, e.g. the temperature is output at the end of one hour. Averaging over a period of time is currently not possible.

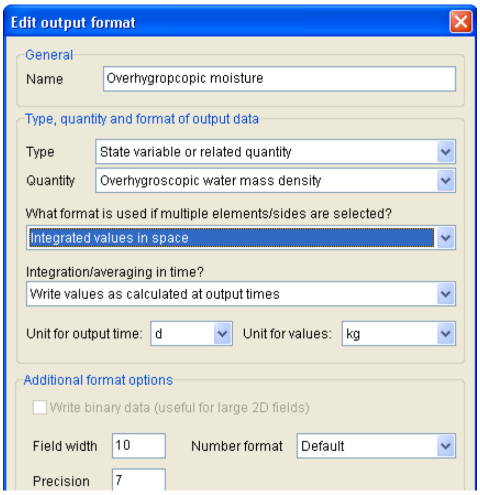
Notes:

Pay attention to what is displayed in the "Unit for values"! In "Unit for output times" normally matching units should be selected. In the simulation of multi-year periods days (d) or years (a are recommended).

Example: Definition of an output format for overhygroscopic humidity

The output of the overhygroscopic humidity is often taken to help if you want to calculate how much liquid water is generated. In Delphin5 the default is 95%. This preference may be set under >> Simulation >> Output related options >> Maximum relative humidity hygroscopic. Specifically, this means that the amount of moisture above 95% humidity is defined and output as overhygroscopic humidity. The amount of moisture below 95% relative humidity is not considered.

Insert in tab >> Format/Type a new output format, showing the following:



If at >> Quantity: Overhygroscopic water mass density is selected, only with the output format Integrated >> values in space the desired output unit kg is generated.

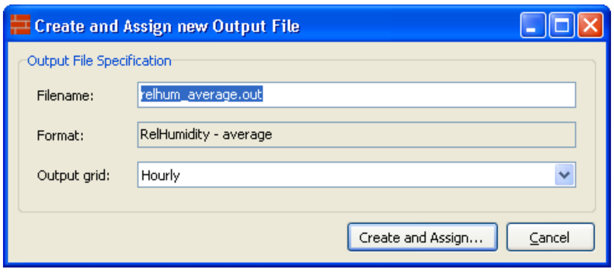
Often in calculating one-dimensional wall structures a comparison with respect to a size of m^2 is desired. This is obtained automatically when the level of the wall structure is 1 m, since by default, the length in the z-direction, meaning out of the screen plane, is also 1 m. The length in the z-direction can be changed by right-clicking in the window >> Construction/Discretisation >> Edit construction ... .

Assignment of the output format to elements

Outputs can be assigned to individual elements both in tab >> Output >> Format/Types or in tab >> Output files

Assignment in tab "Format/Types":

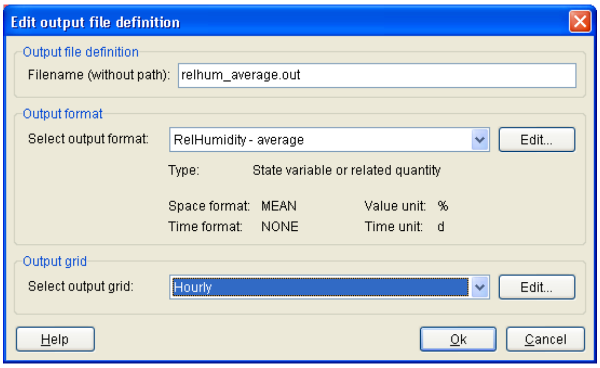
Mark the desired discretize elements and the output format. Click on then and it appears:



Assign under "Filename" a clear, understandable name. . "Out" is the Delphin5 output format. You can select the appropriate temporal output grid under section "Output grid". After clicking on "Create and assign ..." the output is generated automatically and is entered in the tab "Output files" and on the other hand, the output assignment is registered, like all assignments, in the "Assignments/Selections" window.

Assignment in tab "Output files":

Mark the desired discretize elements. Then click on or select by right-clicking on the white-backed list "New output …" and it appears:



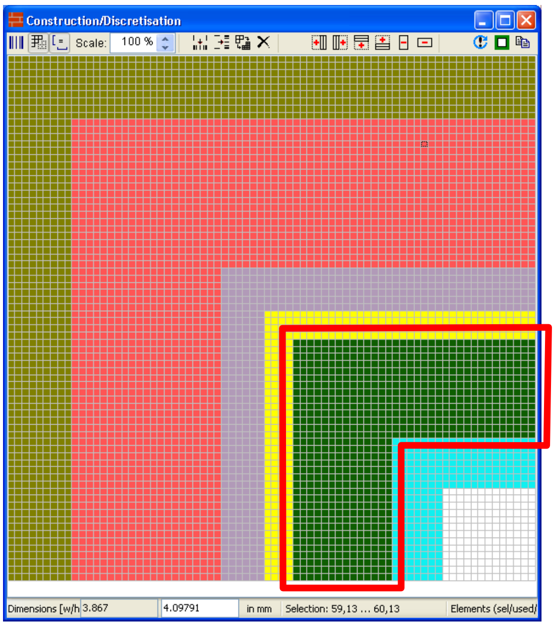
Here you also have to assign a possibly distinct name for the output file ("FileName"). In the "Select output format" you can choose between the output files that are defined under "Format/Types". In the rows below the properties of the selected output type are displayed, in particular the spatial format. Now you only have to choose a temporal output grid at "Select output grid". The output file will then be created in the list with OK.

You must mark the desired elements, select the output and assign with [icon] for the final assignment of the output. The assignment is then automatically entered in the list of the window "Assignments / Selections".

Note 1:
Sometimes it will be less time-consuming if you copy the tab "Output files" in existing outputs. The name can be changed by double-clicking on the output file.

Note 2:
If an output from the dialog >> Output files is assigned to several different areas of design, all associated elements are released in an output file. If, for example, the output file was an integral the values of all assigned elements are counted together. In the exemplary illustration below of a corner of the building, both the amount of moisture and the horizontally and vertically displayed layer is then added together. The average value of all assigned elements is made according to an average output format.

In Delphin only rectangular areas can be processed. Therefore, the vertical and horizontal layer were assigned sequentially in the corner of the building, meaning the output was first assigned to the horizontal layer and then to the vertical layer. However, since it is the same output format the order is irrelevant.



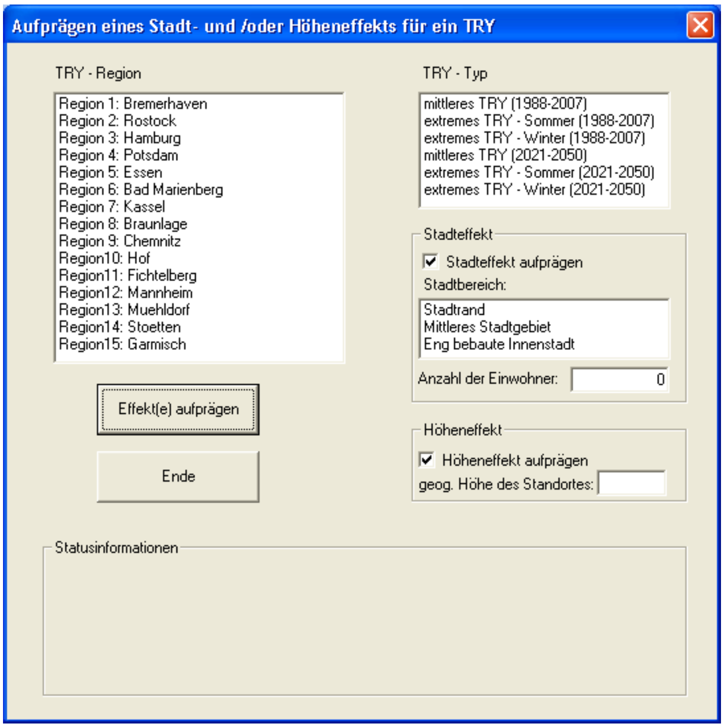
An issue has been assigned to the displayed vertical and horizontal insulation layer.

Tutorial (5) Creating and importing of TRY-climate data

On the sides of the BBSR (Federal Institute for Building, Urban Affairs and Spatial Development) free software for creating TRY climates can be downloaded (http://www.bbsr.bund.de/BBSR/DE/FP/ZB/Auftragsforschung/5EnergieKlimaBauen/2008/Testreferenzjahre/03_ergebnisse.html? nn = 436 654 or at an Internet search for "TRY download BBSR 2011" enter). With this software, climate data for Germany can be generated, while the height and city's influence can be taken into account by special algorithms.

Creation of climate data

The TRY Software "impressing a city and/or height of effect for a TRY" is kept quite clear. With a few clicks a TRY (test reference year)-climate is created:



1. 1 First, select in the window >> TRY region a climate region. The 15 climate regions are displayed in the supplied Germany map *. pdf files.

2. Then the >> TRY-type must be selected. Here you can decide whether a future climate should be created, as it is expected to prevail in the years 2021-2050, or whether a climate based on measured values of the years 1988 to 2007 is to be created. The climate data record of an average year, a year with very warm summer and a year of very cold winter are at disposal within the two climate periods.

3. If a city effect should be taken into account >> City effects impose must be clicked on. This way the location of the building within the city and the size or number of inhabitants is included.

4. A compared to the reference site other high elevation of a building is considered when is confirmed impress >> height effect. A compared to the reference site other high elevation of a building is considered if >> height effect imprinting is confirmed.

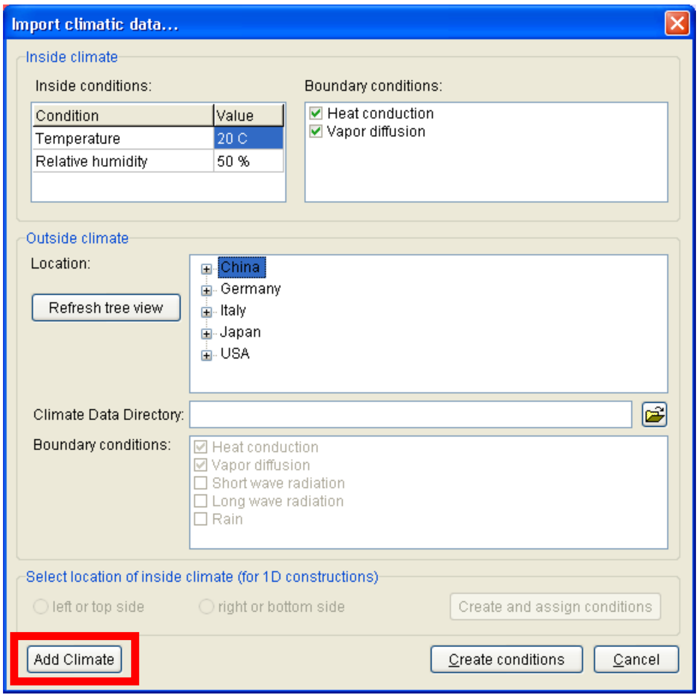
5. After clicking on >> Effects imprinting and the exclamation “Hossa! Hossa!” the climate data will be created.

The produced climate data is in a file with the extension * .dat in the folder "_xxx_ / data / results" created by the software. The file name displays all settings made. The climate data does not include rain data.

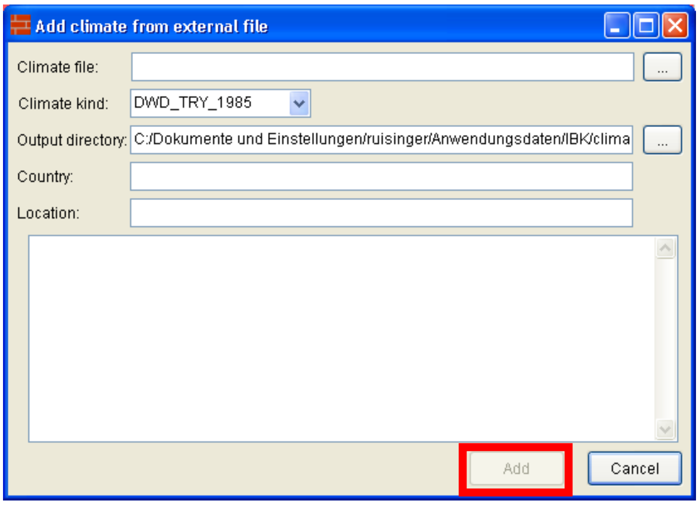
Further explanations can be found in the manual or project report.

Import of TRY-climates in Delphin

At first you must call up the dialogue of the climate data import in window >> Conditions with the button for the import of a produced TRY-climate in Delphine5.



Click on >> Add Climate and it appears:



First, the type of climate to be read should be entered in >> Climate kind, in this case DWD_TRY_2011. Then *.dat file can be selected in >> Climate file and the output folder in >> Output directory. A click on >> Add triggers the import into the Delphin project. Thereby an error message can occur for incomplete data sets, especially for cloud coverage.

If a newly created climate of Delphin is found generally to choose from, it can be saved in the folder IBK/Delphin 5.x/DB_climate_data/[x_country name_x]. The climate will always be available via the dialog >>Climate >> as a result of this.

2.2 Workshop´s Presentations

- Numerical Simulation

- Application of Delphin

- Moisture damages

TECHNISCHE UNIVERSITÄT DRESDEN

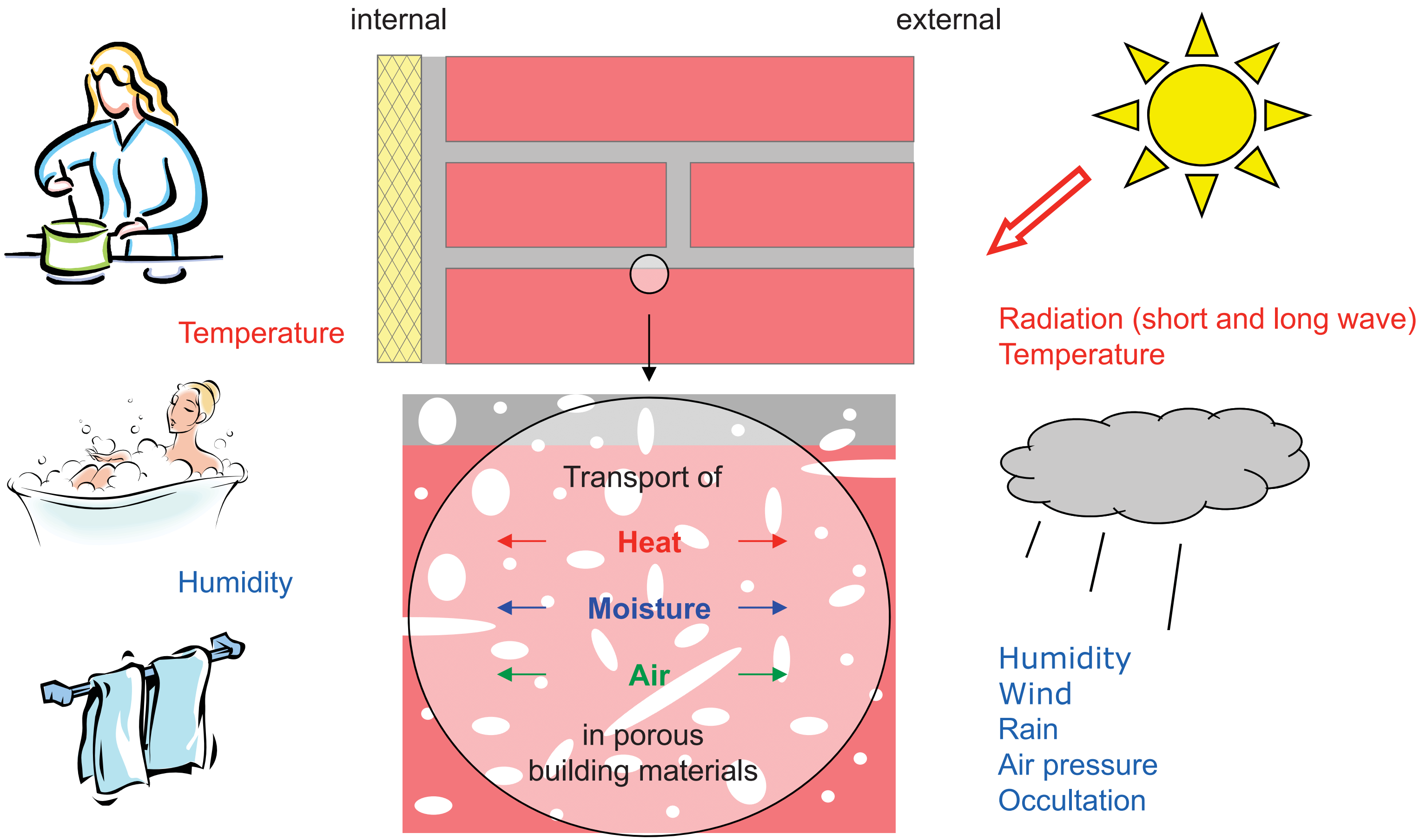
Faculty of Architecture Institute for Building Climatology Professorship of Building Physics

Numerical simulation of heat- and moisture transport in capillary-porous building materials

Basics of simulation software DELPHIN

- Transient
  - Usage of dynamical boundary conditions (external und internal climate)
  - Thermal and hygric inertia of construction is considered
- Hygro-thermal
  - Heat conductivity and storage
  - Moisture transport (vapour and capillary conductivity) and moisture storage
- Building elements
  - Materials and systems/constructions
  - Constructional details
- Simulation
  - For analysis (expertises) and prediction (feasibility study/optimisation)

Transient transport processes in capillary-porous building materials



Basic knowledge for the use of simulation software

- Material properties
- Conserved quantity
- Transport processes
- Initial conditions
- Boundary conditions

} Physical state equations

- Mathematical method
- Space discretisation
- Time steps
- Precision

} Numerical solving method

Part 1

Physical basic equations and models

Mathematical basics and nomenclature

Einsteins' summation rule:

$$j_k = \sum_k j_k$$

Usage of direction index implicates sigma sign

For cartesian coordinate systems: $k = x, y, z$

Usage for partial derivative: $\frac{\partial}{\partial x_k} = \frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z}$

Example:

$$m = m(t, x_k)$$

Conserved quantity is defined in dependency of time and space

$$\frac{\partial m}{\partial x_k} = \frac{\partial m}{\partial x} + \frac{\partial m}{\partial y} + \frac{\partial m}{\partial z}$$

Partial derivatives are summed

Conservation equations (Balance equations)

Energy conservation (transient heat conductivity equation):

$$\frac{\partial Q}{\partial t} = \frac{\partial}{\partial x_k}\left(\lambda \frac{\partial \theta}{\partial x_k}\right)$$

with $Q = Q(t, x_k)$ and $\theta = \theta(t, x_k)$

Q	J/m^3	Internal energy density
θ	$°C$	Temperature

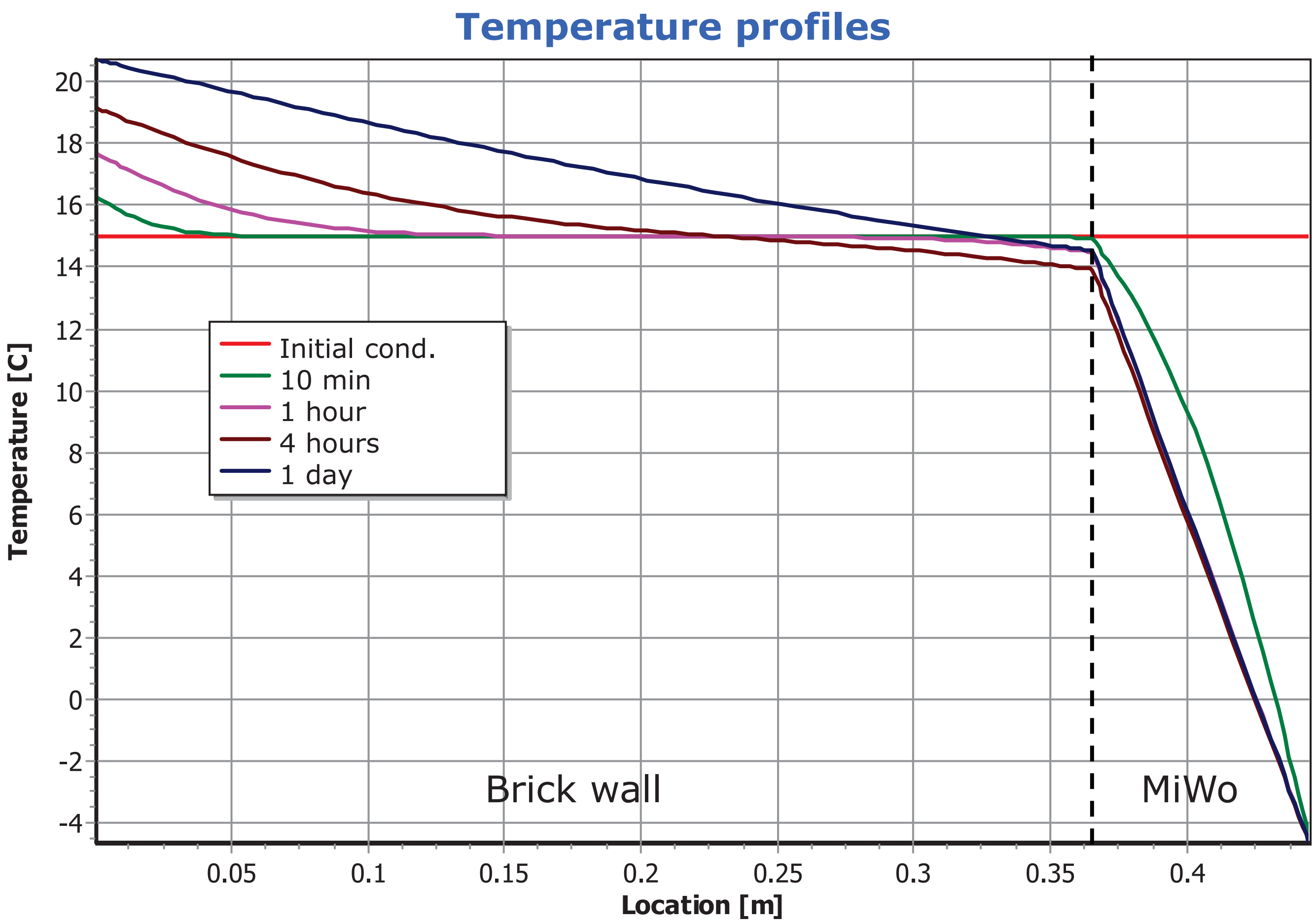
Change of internal energy in time (only heat storage):

$$\frac{\partial Q}{\partial t} = \rho_{dry} c_T \frac{\partial \theta}{\partial t}$$

Important material parameter:

ρ_{dry}	kg/m^3	Density of dry materials
c_T	J/kgK	Specific heat capacity
λ	$J/smK = W/mK$	Heat conductivity

Transient heat conductivity – till achievement of steady-state conditions



Transient heat conductivity – till achievement of steady-state conditions

Steady-state conditions = no change of conservation quantities in time anymore

$$\frac{\partial Q}{\partial t} = 0 = \frac{\partial}{\partial x_k}\left(\lambda \frac{\partial \theta}{\partial x_k}\right)$$

⇨ Heat fluxes constant: $q = -\lambda \frac{\partial \theta}{\partial x} = const$

⇨ Storage term and heat capacity are irrelevant for stationary state

At transient processes the storage term controls how quick
the system responses to boundary conditions:

- high heat capacity – slow achievement of stationary conditions
- low heat capacity – quick response

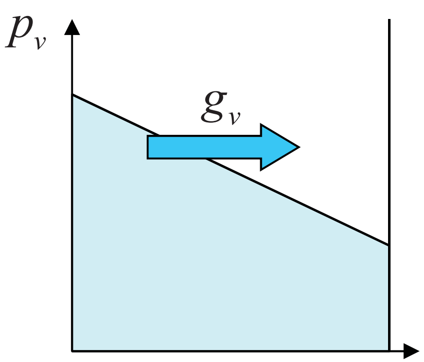
Conservation equations (Balance equations)

Water mass balance (transient conservation equation for moisture in building components):

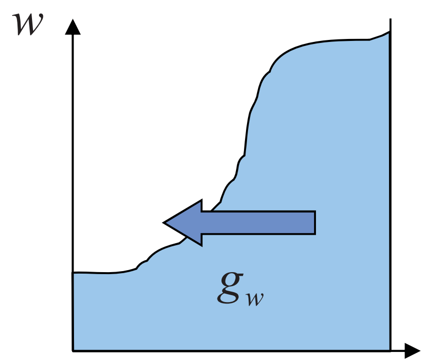
$$\frac{\partial w}{\partial t} = -\frac{\partial}{\partial x_k}\left(g_{v,k} + g_{w,k}\right)$$

with $w = w\left(t, x_k\right)$

w	kg/m^3	Water mass density
g_v	kg/m^2s	Vapour flux density
g_w	kg/m^2s	Capillary water flux density



Vapour pressure gradient ¯ Vapour diffusion



Water content difference ¯ capillary conductivity

Conservation equations (Balance equations)

Gas mass balance (transient conservation equation for gaseous phase in building components):

$$\frac{\partial \rho_g}{\partial t} = -\frac{\partial}{\partial x_k}\left(g_{gc,k} + g_{gg,k}\right)$$

with $\rho_g = \rho_g\left(t, x_k\right)$ and $\rho_g = \rho_g\left(T, w, p_g\right)$

ρ_g	kg/m^3	Gas density
g_{gc}	$kg/m^2 s$	Convective gas flux density
g_{gg}	$kg/m^2 s$	Gas flux density due to gravity

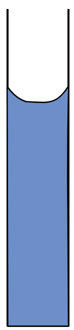
Transport processes and models

Heat flux density: $$q = -\lambda \frac{\partial \theta}{\partial x}$$

Vapour diffusion: $$g_v = -\frac{\delta_{v,air}}{\mu} \frac{\partial p_v}{\partial x} = -\frac{D_v}{R_v T} \frac{\partial p_v}{\partial x}$$

Capillary water transport: $$g_w = -K_\ell \frac{\partial p_\ell}{\partial x}$$

Gas transport: $$g_{gc} = -\frac{K_g}{\rho_g} \frac{\partial p_g}{\partial x} \qquad g_{gg} = -K_g \cdot g_z \rho_g$$



r

Capillary pressure: $$p_c = \frac{2\gamma \cos\varphi}{r}$$

⇨ The smaller the pore radius, the bigger the traction force and therefore the water height in the capillary tube

Water pressure: $$p_\ell = p_c + p_g$$

Pressure gradient in liquid phase induces water transport

Transport processes and models

Evaporative cooling and heat of condensation

Specific enthalpy of water vapour: $h_v = c_{T,v}\left(T - T_{Ref}\right) + H_{evap}$

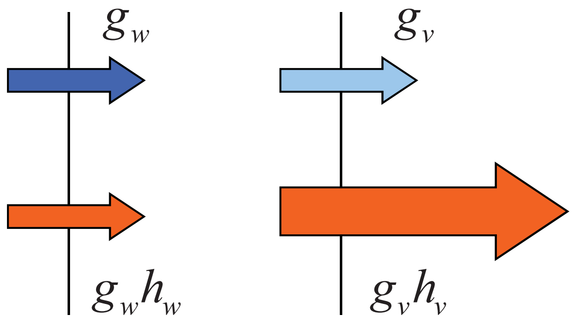
Specific enthalpy of water: $h_w = c_{T,w}\left(T - T_{Ref}\right)$

Evaporation enthalpy: $H_{evap} = 3.08 \cdot 10^6 \text{ J/kg}$

→ $h_v \gg h_w$ although $c_{T,w} \simeq 2c_{T,v}$

Enthalpy transport of water vapour is much bigger than of liquid water!

Example:



Summary

Conservation equations

Moisture mass balance: $$\frac{\partial w}{\partial t} = -\frac{\partial}{\partial x_k}\left(g_{v,k} + g_{w,k}\right)$$

Energy balance: $$\frac{\partial Q}{\partial t} = -\frac{\partial}{\partial x_k}\left(q_k + h_v g_{v,k} + h_w g_{w,k}\right)$$

Solution of equations

Initial conditions (one for each conservation equation), e.g.: T, φ or T, w

Boundary conditions (types):

Neumann (2nd)	Describes **fluxes** from surroundings into construction, z.B. radiation heat flux, Vapour diffusion flux
Dirichlet (1st)	Describes **boundary values**, e.g. surface temperature
Cauchy (3rd)	Describes **fluxes** and **boundary values**

Summary

Material parameters and material functions

General parameter:

ρ_{dry}	kg/m^3	Density of dry material
c_T	J/kgK	Specific heat conductivity

Transport parameter:

λ	W/mK	Heat conductivity
μ	$-$	Water vapour diffusion resistance value
K_ℓ	s	Liquid water conductivity

Moisture storage parameter:

$w(p_c)$	kg/m^3	Moisture retention curve (MRC)
$w(\varphi)$	kg/m^3	Sorption isotherm

Part 2

The numerical solving method

Control volume method

Used to transform partial differential equations into systems of ordinary differential equations

Analytic derivation using the example of heat conduction equation:

1. Transformed original equation:

$$\frac{\partial U}{\partial t} + \frac{\partial}{\partial x_k}\left(q_k\right) = 0$$

2. Multiplied with function:

$$\omega \cdot \left[\frac{\partial U}{\partial t} + \frac{\partial}{\partial x_k}\left(q_k\right)\right] = 0$$

3. Integrated over a volume:

$$\int_V \omega \cdot \left[\frac{\partial U}{\partial t} + \frac{\partial}{\partial x_k}\left(q_k\right)\right] dV = 0$$

Presumptions/preconditions:

$w = const$ (= 0-order FEM) and $\int_V \frac{\partial U}{\partial t} dV \simeq \frac{\partial U}{\partial t} V$

Control volume method

Analytic derivation using the example of heat conduction equation:

4. Equation transformed/simplified: $$\frac{\partial U}{\partial t} = -\frac{1}{V}\int_V \frac{\partial}{\partial x_k}\left(q_k\right)\, dV$$

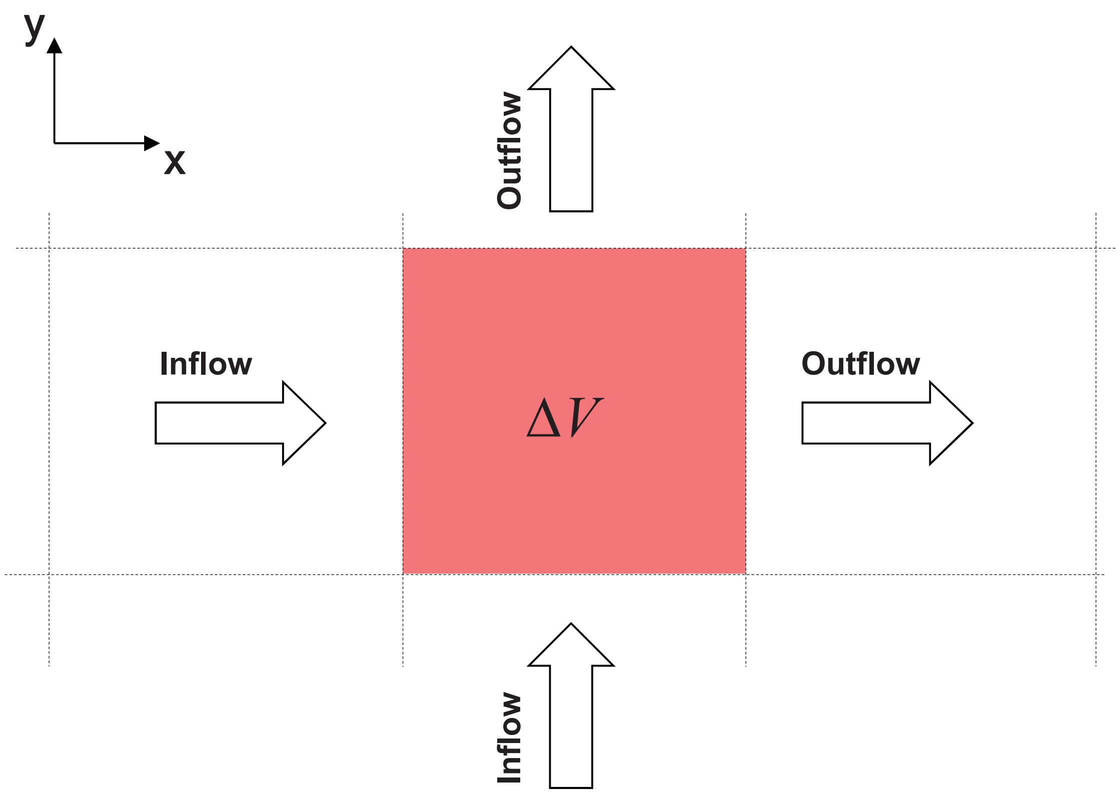
5. Gauss-Green-Theorem: $$\frac{\partial U}{\partial t} = -\frac{1}{V}\oint_A \vec{n}_k A_k \overrightarrow{q_k}\, dA$$

6. Application for discrete areas: $$\frac{\partial U}{\partial t} = -\frac{1}{V}\sum_i \vec{n}_i\ A_i\ \vec{q}_{k,i}$$

Example: 1D $$\frac{\partial U}{\partial t} = \frac{1}{V}\left[A_l q_l - A_r q_r\right]$$

$$\frac{\partial U}{\partial t} = \frac{1}{\Delta x}\left[q_l - q_r\right]$$ with the borders of the volume l = left, r = right

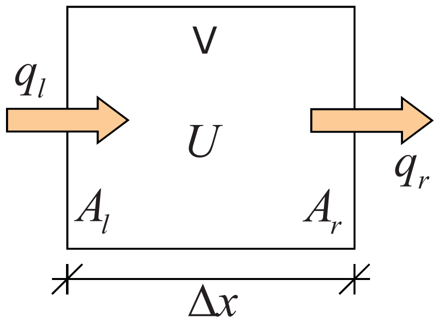
Balancing of conserving quantities (mass + energy)



Change of density in the discrete volume = Difference between inflow and outflow

Control volume method

Derivation at concrete example:



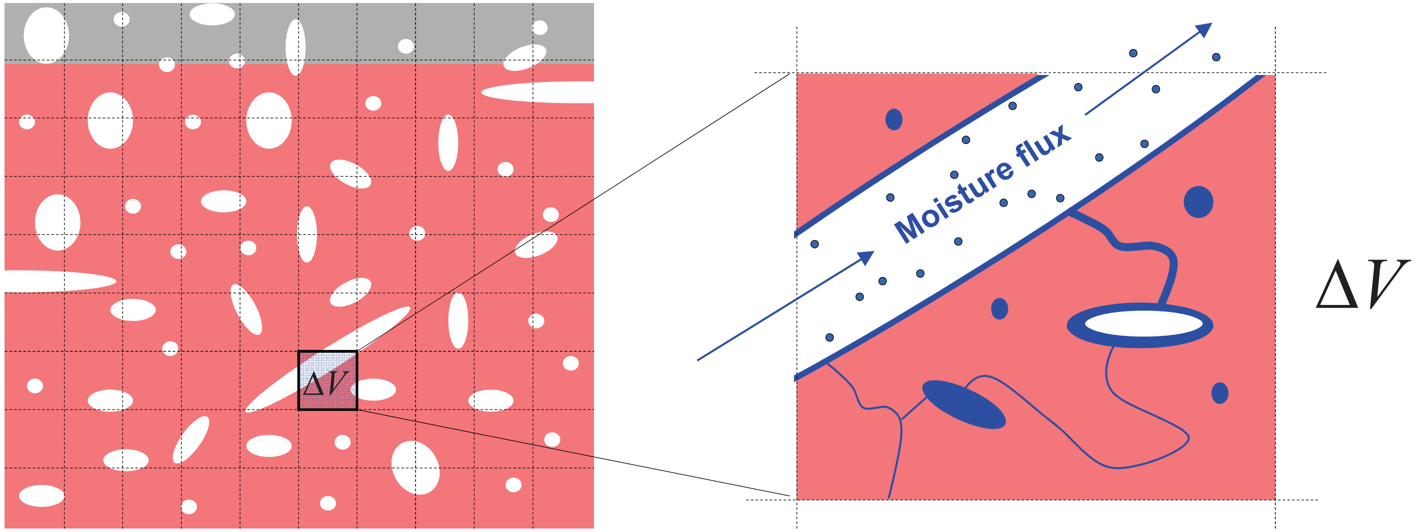
Change of absolute conservation quantities per time = difference of fluxes

$$\Delta U \cdot V = \Delta t \left(A_l q_l - A_r q_r \right)$$ at which $A_l = A_r = A$ and $V = \Delta x \, A$

hence $$\frac{\Delta U}{\Delta t} = \frac{A}{V}(q_l - q_r)$$ and $$\frac{\Delta U}{\Delta t} = \frac{1}{\Delta x}(q_l - q_r)$$

and for infinitesimal time steps: $$\frac{\partial U}{\partial t} = \frac{A}{V}(q_l - q_r)$$

Discretisation



Discretisation for numerical solution

- Material macroscopically homogenous
- Isotropic transport properties
- Properties of volume elements, representative for the material

Definition of local state variables

θ_l	*Water content*
T	*Temperature*
φ	*Relative humidity*
p_v	*Vapour pressure*
p_c	*Capillary pressure*

Discretisation of partial derivation

Example: Heat fluxes between control volumes

$$q_k = -\lambda \frac{\partial \theta}{\partial x_k}$$

Taylor series expansions:

$$f(x+\Delta x) = f(x) + \Delta x \frac{\partial f}{\partial x} + \frac{\Delta x^2}{2!}\frac{\partial^2 f}{\partial x^2} + \frac{\Delta x^3}{3!}\frac{\partial^3 f}{\partial x^3} + \ldots$$

Estimation of 1. derivation of function (with fault 2nd order):

$$\frac{\partial f}{\partial x} = \frac{f(x+\Delta x) - f(x)}{\Delta x} + O(2)$$

⇨ Discrete formulation of heat flux
Density between control volumes:

$$q_k = -\lambda \frac{\Delta \theta}{\Delta x_k}$$

Numerical solving methods at a glance

$$\frac{\partial U}{\partial t} = -\frac{\partial}{\partial x_k}\left(\lambda \frac{\partial \theta}{\partial x_k}\right)$$

+

Control volume method

+

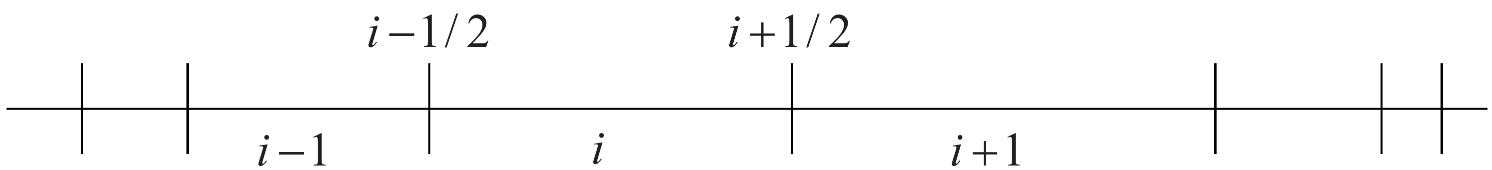
Discretisation of partial derivations

=

System of ordinary differential equation
(one equation per control volume and conserving quantity)

e.g. 1D heat conductivity equation

$$\frac{\partial U_i}{\partial t} = \frac{1}{\Delta x_i}\left(\lambda_{i-1/2}\frac{\theta_{i-1}-\theta_i}{\Delta x_{i-1/2}} - \lambda_{i+1/2}\frac{\theta_i-\theta_{i+1}}{\Delta x_{i+1/2}}\right)$$

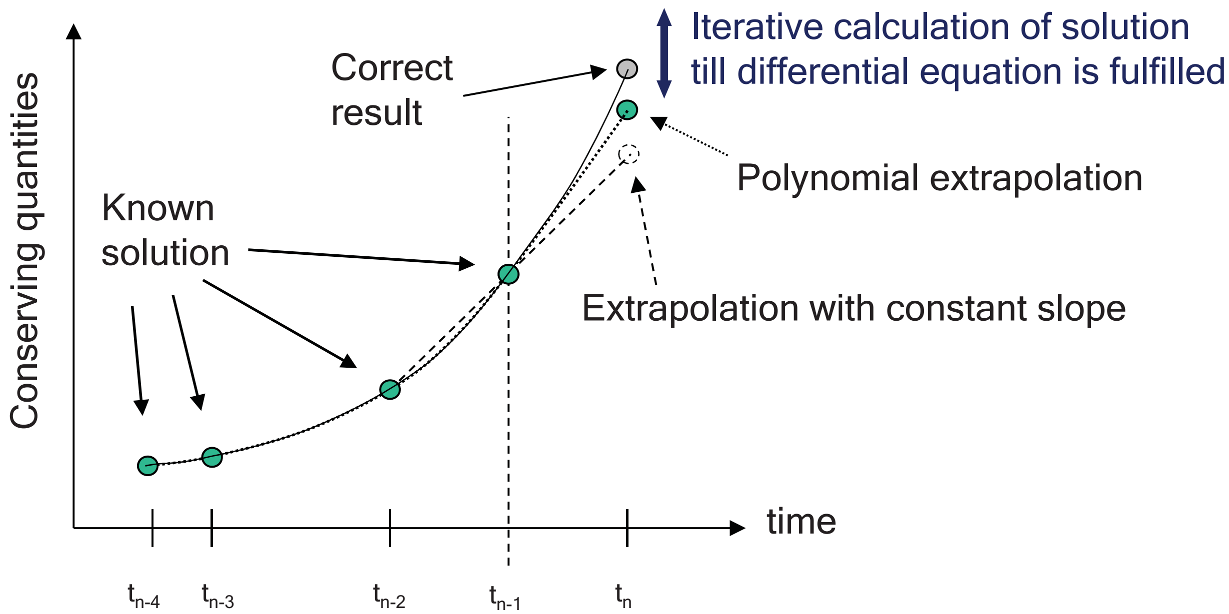


Numerical solution

Discretised differential equation (example: moisture mass balance)

$$\frac{\partial w}{\partial t} = -\frac{1}{\Delta V}\left[\sum_A \left(g_{vapour} + g_{liquid}\right)_A\right]$$

Numerical Integration



Numerical solving methods at a glance

2 balance equations * n elements = number of equations & unknowns

Vector with unknowns: $\mathbf{y} = \{Q_i, w_i\}$

System of differential equations: $\frac{\partial \mathbf{y}}{\partial t} = \mathbf{f}(t, \mathbf{y})$

Solution of equation systems by time integration:

$$\mathbf{y}(t) = \mathbf{y}_0 + \int_t \mathbf{f}(t, \mathbf{y}) \, dt$$

→ Simulation software DELPHIN

Part 2

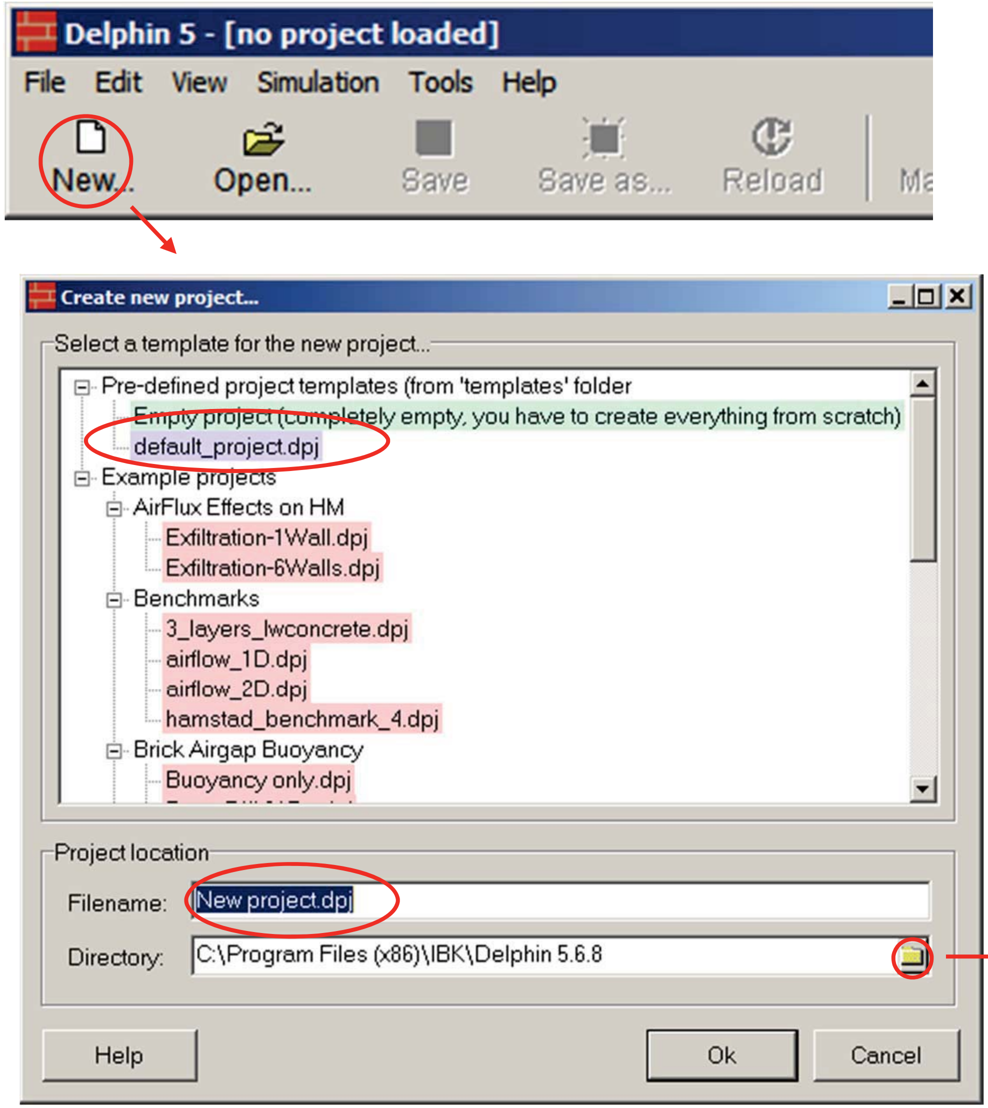
The Application of DELPHIN

DELPHIN: Program operation

Steps - New Project

- Open new project template
- Choose memory location of the project
- Delphin project name

DELPHIN opens a standard template (.dpj).*



Choose folder

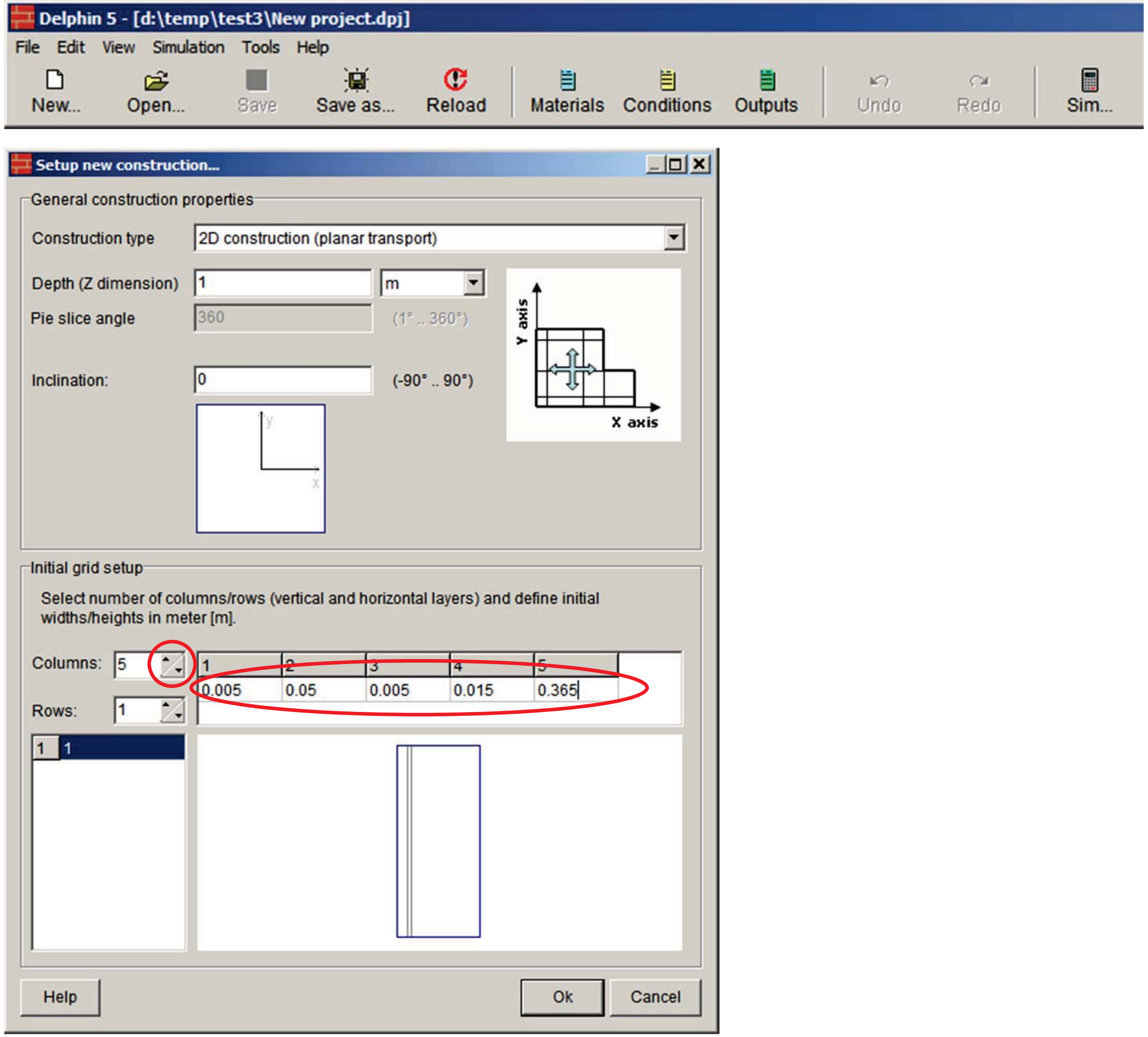
Steps - Construction

- Define type of construction, here: 1D horizontal
- Adjust number of material layers in x-direction
- Adjust thickness of different layer in [m]

Only transport in x-direction:

The height (y-direction) und depth (z-direction) should be 1 m to calculate a wall area of 1m².

DELPHIN then opens the construction view and shows the succession of layers – initially without materials.

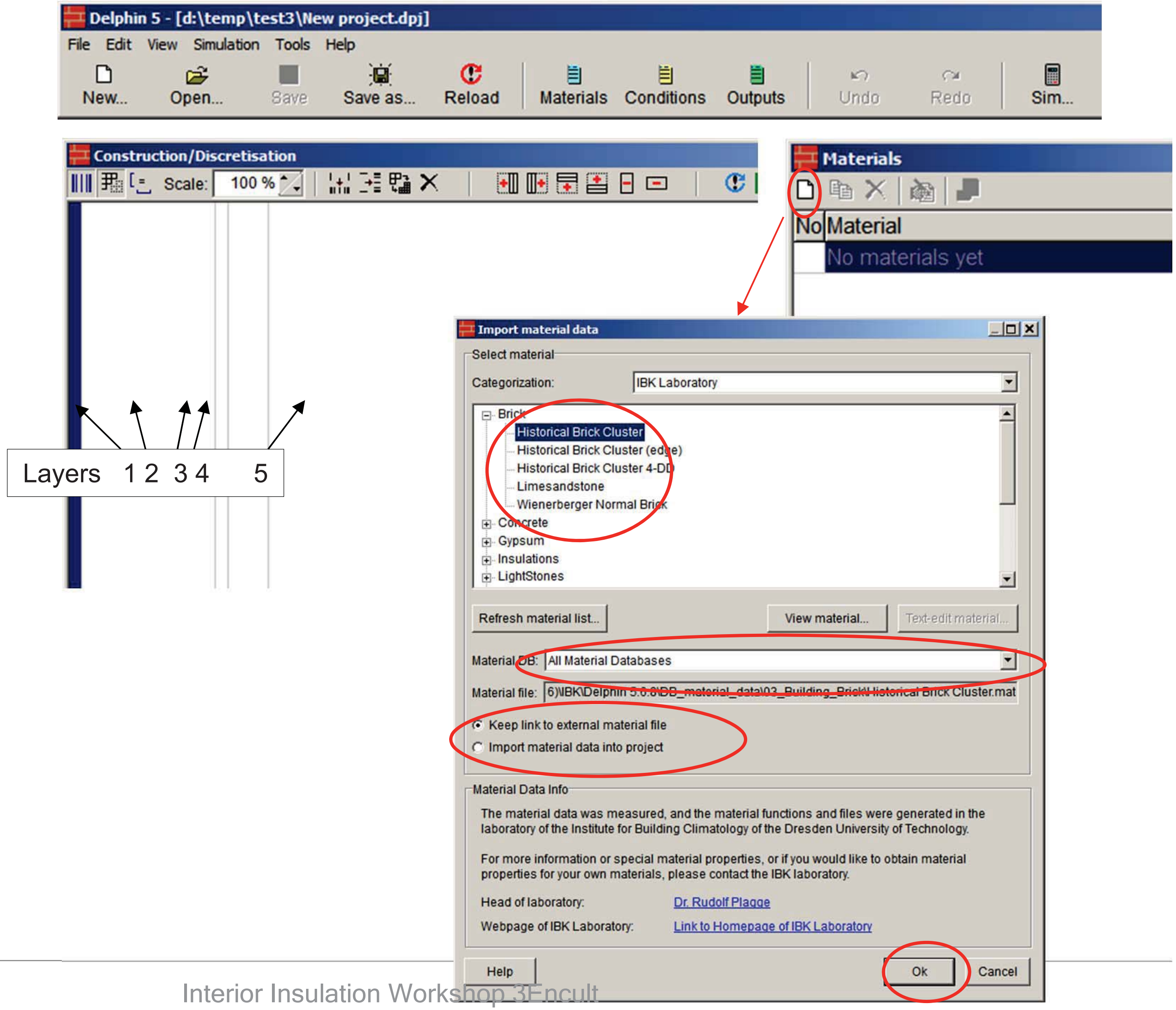


DELPHIN: Program operation

Steps - Material

- Import material
- Choose program or user data base
- Choose import modus
- Choose material and import it

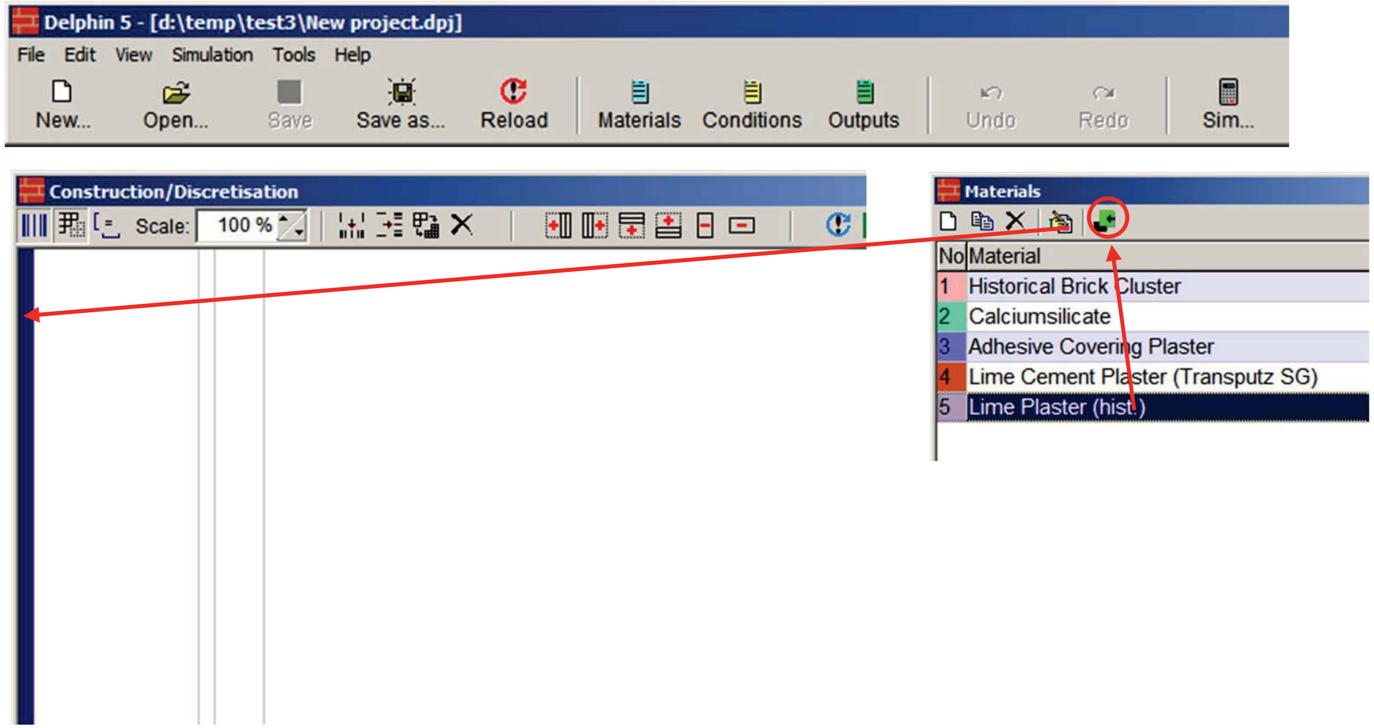
DELPHIN shows the imported materials in the material list.



DELPHIN: Program operation

Steps

- Mark material and favoured layer
- Click on green assign button



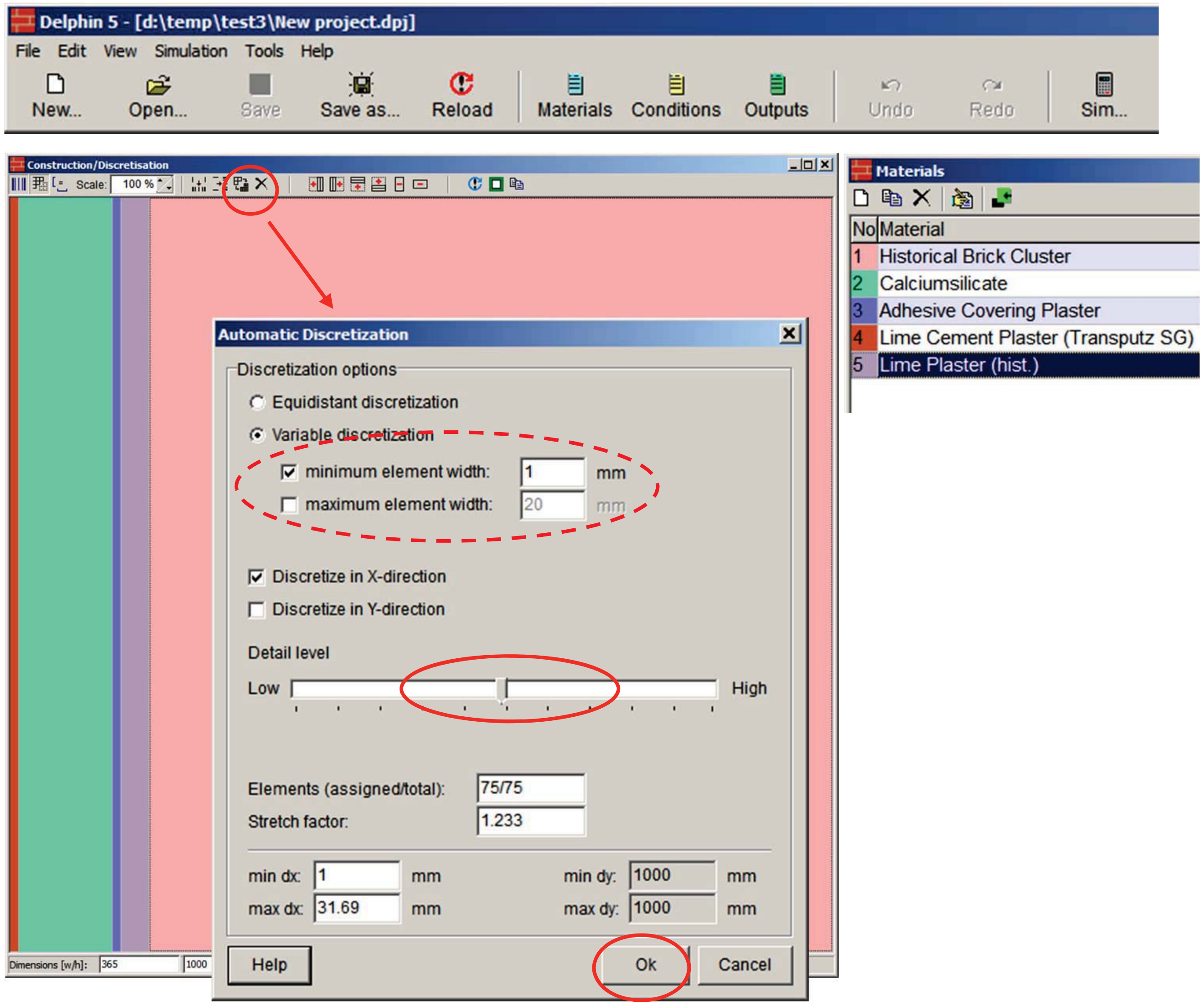
DELPHIN generates a material assignment data set and colours layers with material assignment corresponding to the colour of the material.

DELPHIN: Program operation

Steps

- Call discretisation dialogue
- Set grade of refinement (higher = more refined discretisation)
- Set minimal/maximal element thickness eventually
- Start discretisation (>> Ok)

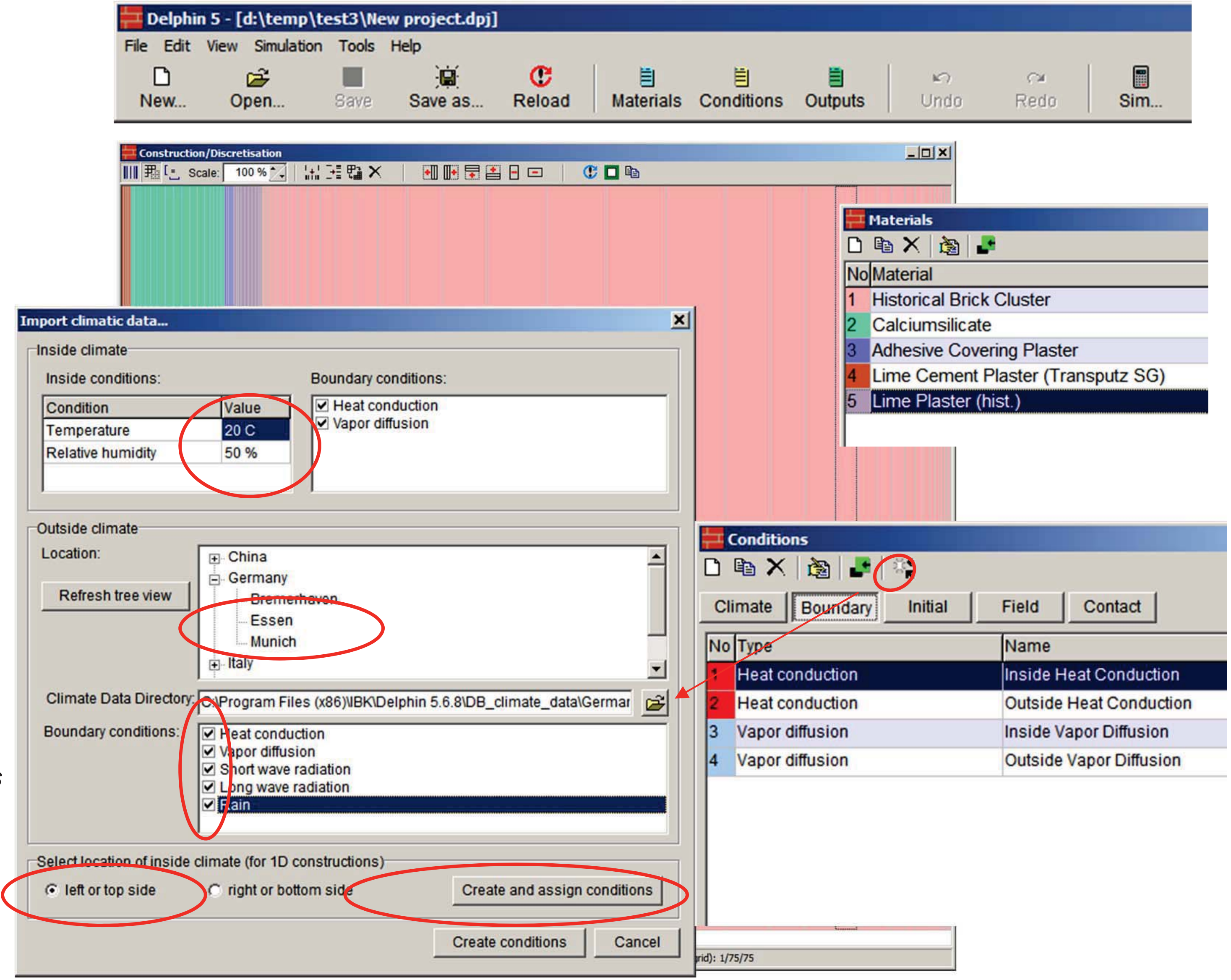
DELPHIN divides material layers in discrete volume elements.



Steps - Climate

- Import climate conditions
- Adjust internal and external climate data
- Import and assign boundary conditions data sets

DELPHIN shows the imported climate and boundary conditions in the conditions windows and enables the assignment to the construction.

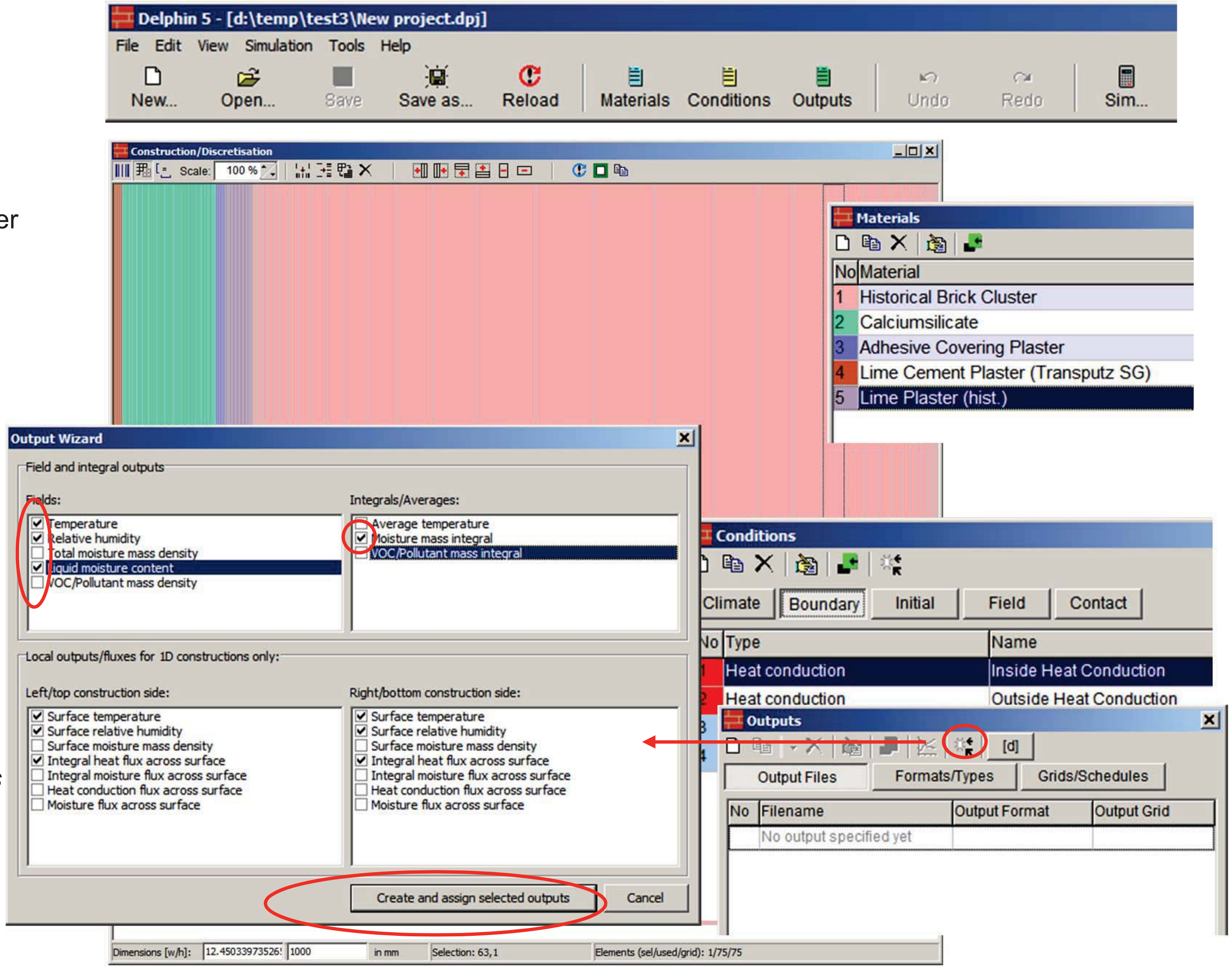


DELPHIN: Program operation

Steps - Outputs

- Start Outputs-Wizard
- Deactivate VOC-outputs, activate water content
- Generate and assign output files

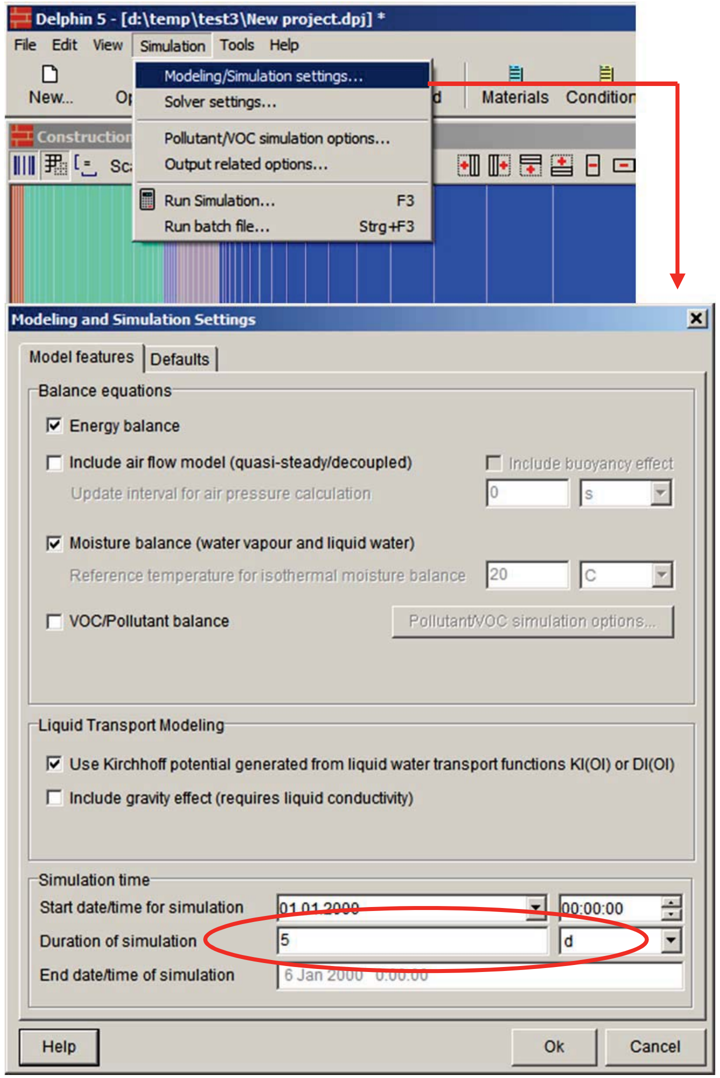
DELPHIN generates output files and enables the assignment to the geometry.



DELPHIN: Program operation

Steps - Simulation

- Open modelling and simulation properties
- Define starting point and total duration of simulation

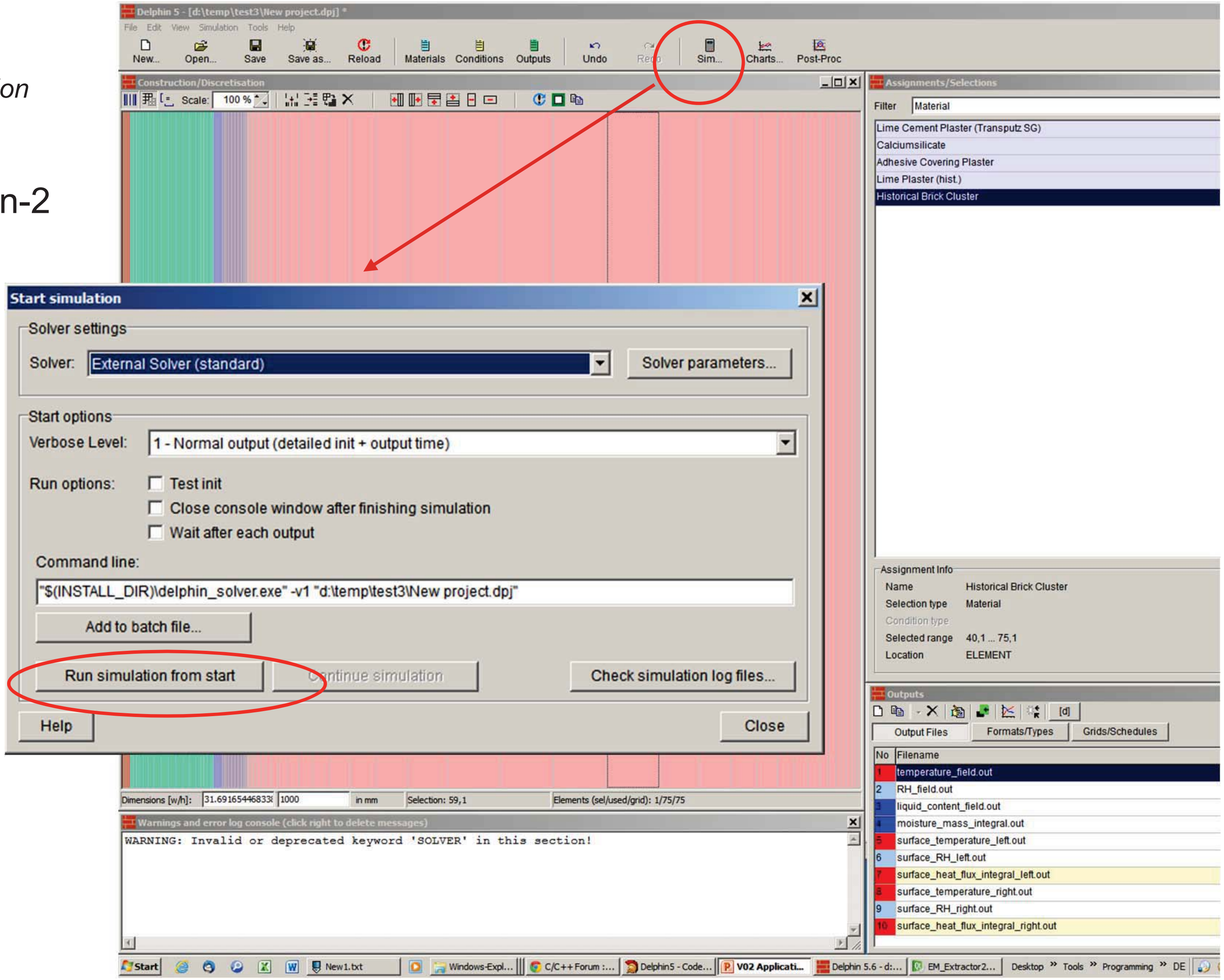


DELPHIN: Program operation

Start DELPHIN simulation

Steps – Simulation-2

- Start solver dialogue
- Start simulation



DELPHIN starts the numeric solver in a separate windox.

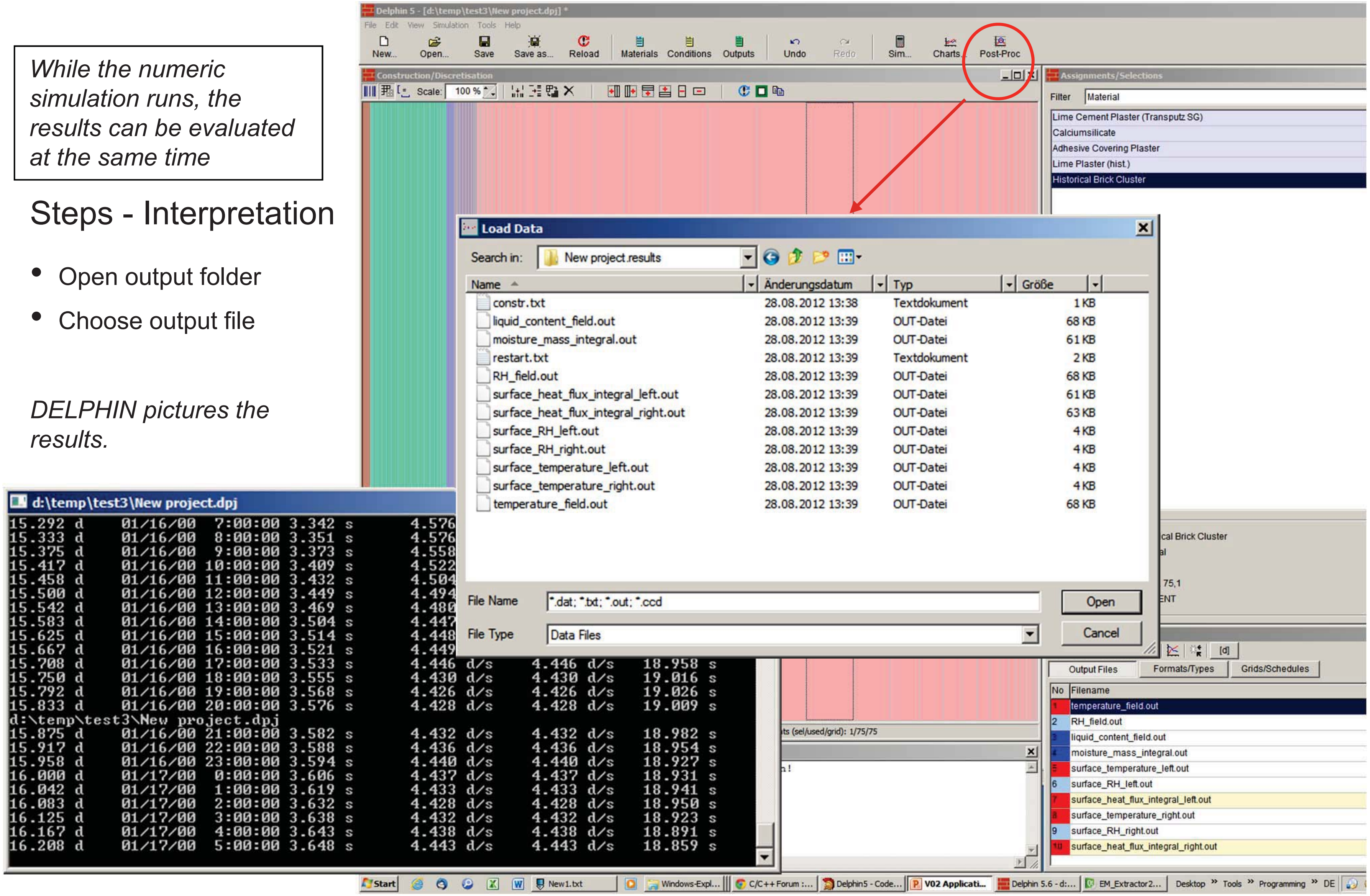
DELPHIN: Program operation

While the numeric simulation runs, the results can be evaluated at the same time

Steps - Interpretation

- Open output folder
- Choose output file

DELPHIN pictures the results.

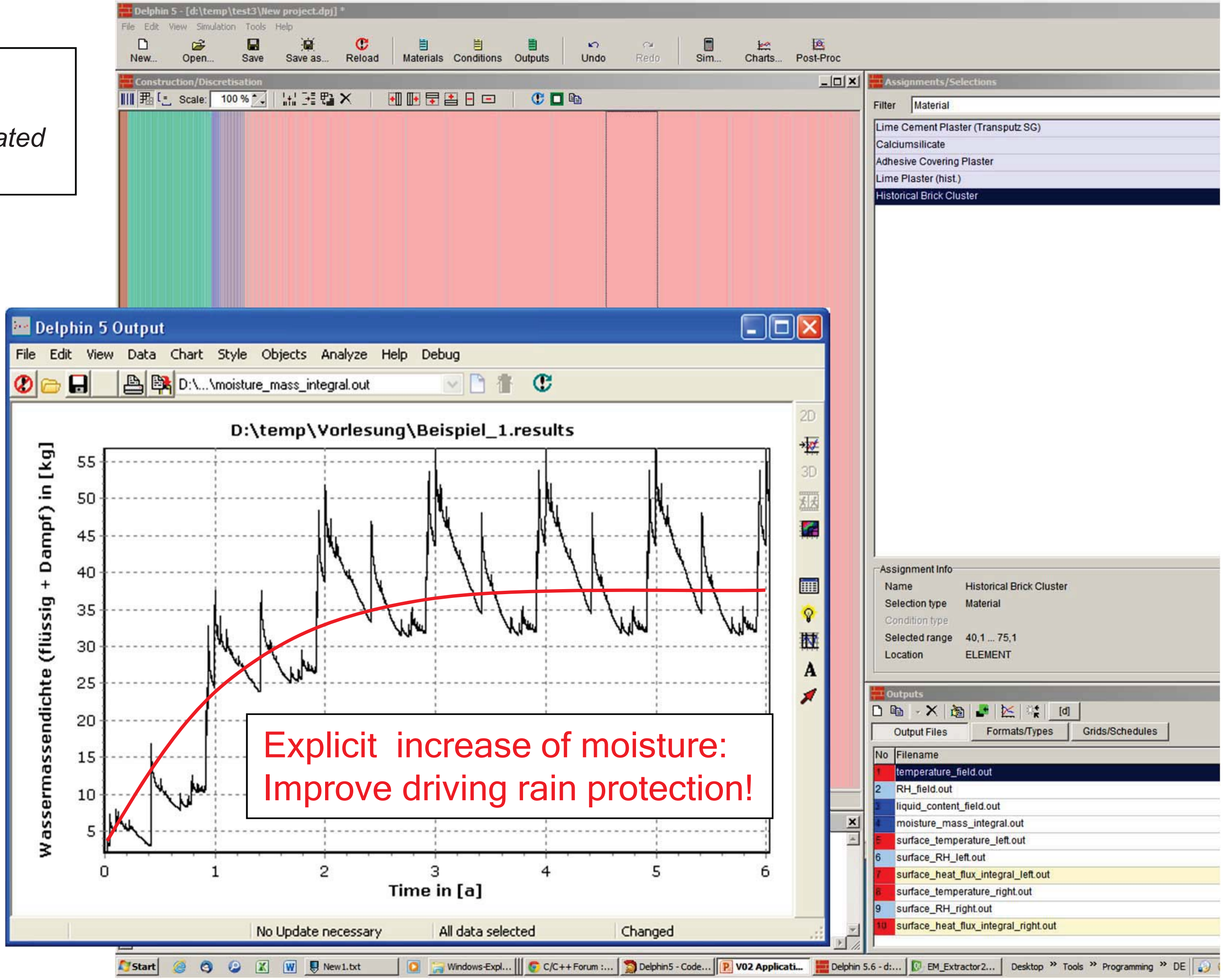


DELPHIN: Program operation

While the numeric simulation runs, the results can be evaluated at the same time

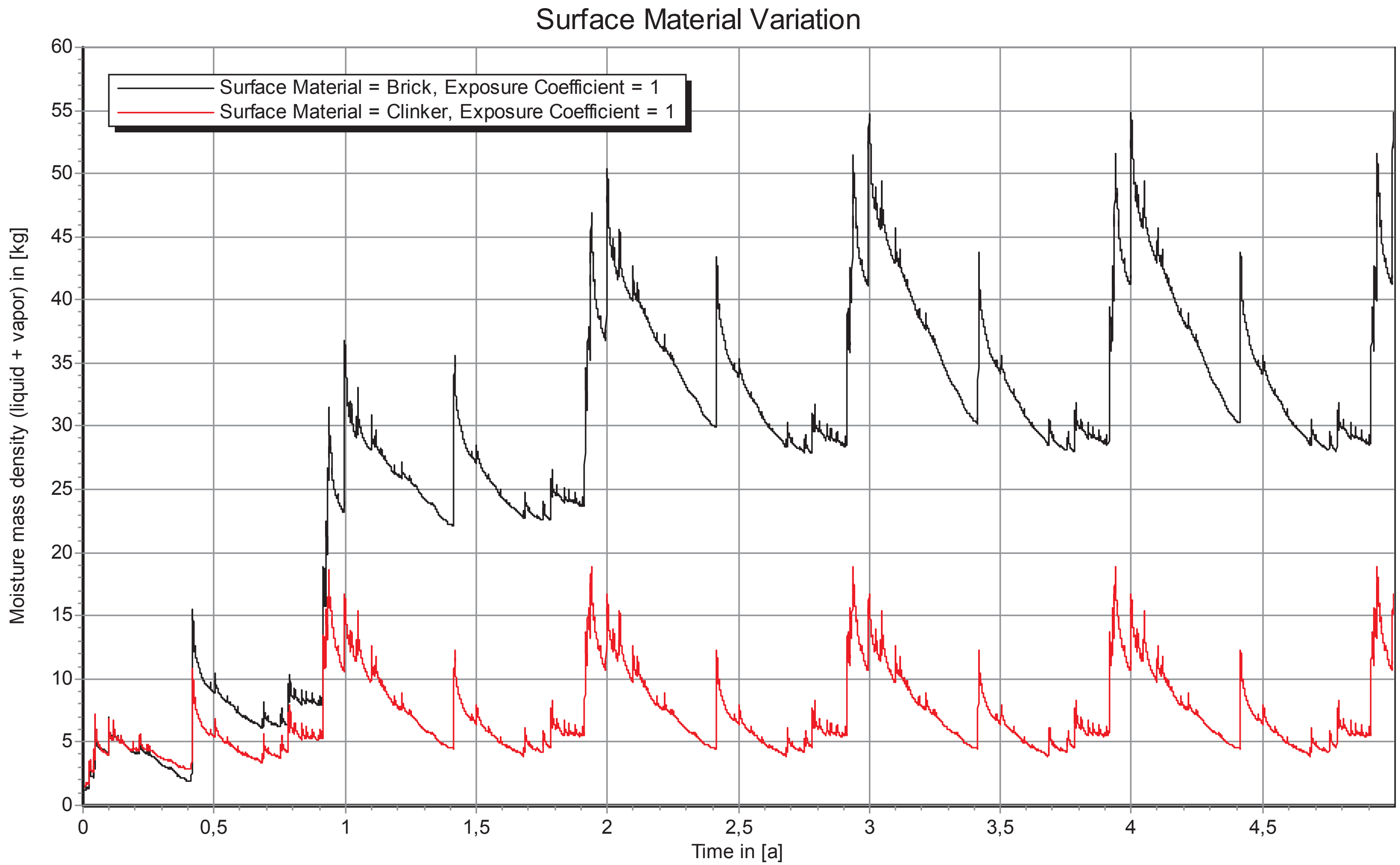
Steps - interpretation

- Updating while calculation possible
- Manifold adjustment of the charts
- Export into other software via clipboard

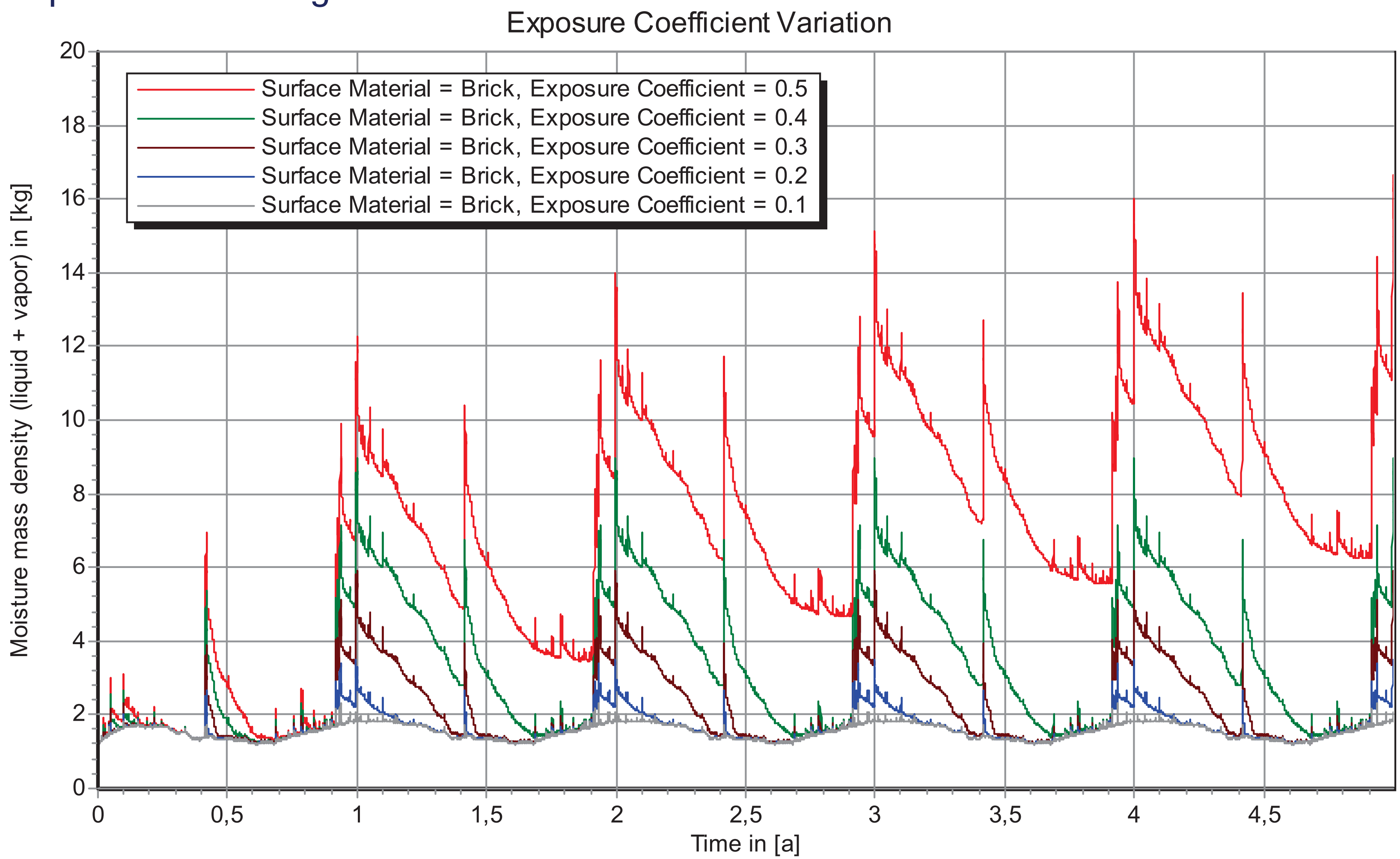


DELPHIN: Results – Variation studies

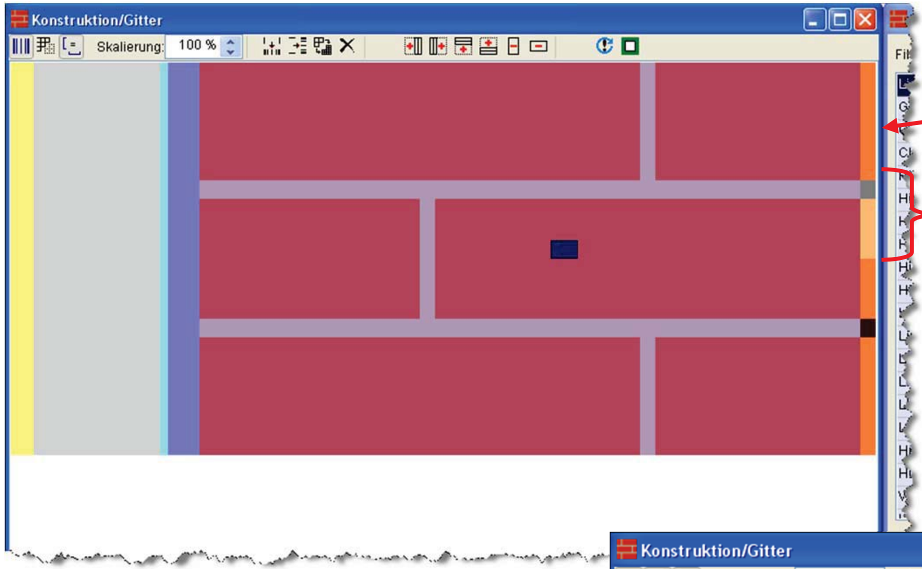
Different materials



Rain protection/roofing:

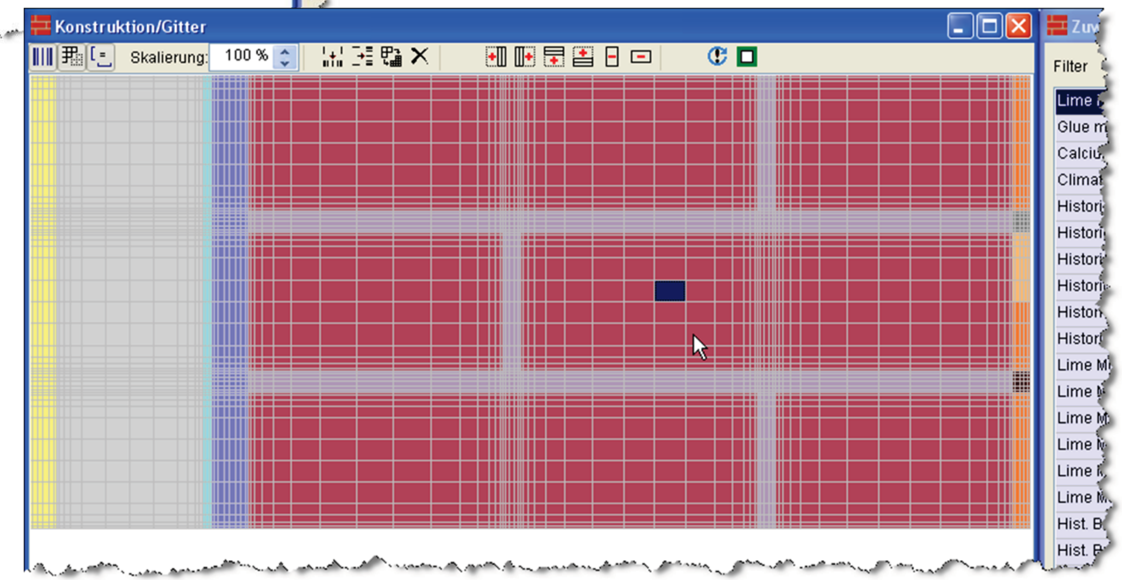


DELPHIN – Analysis of 2D problems



Wall with hydrophobisated surface

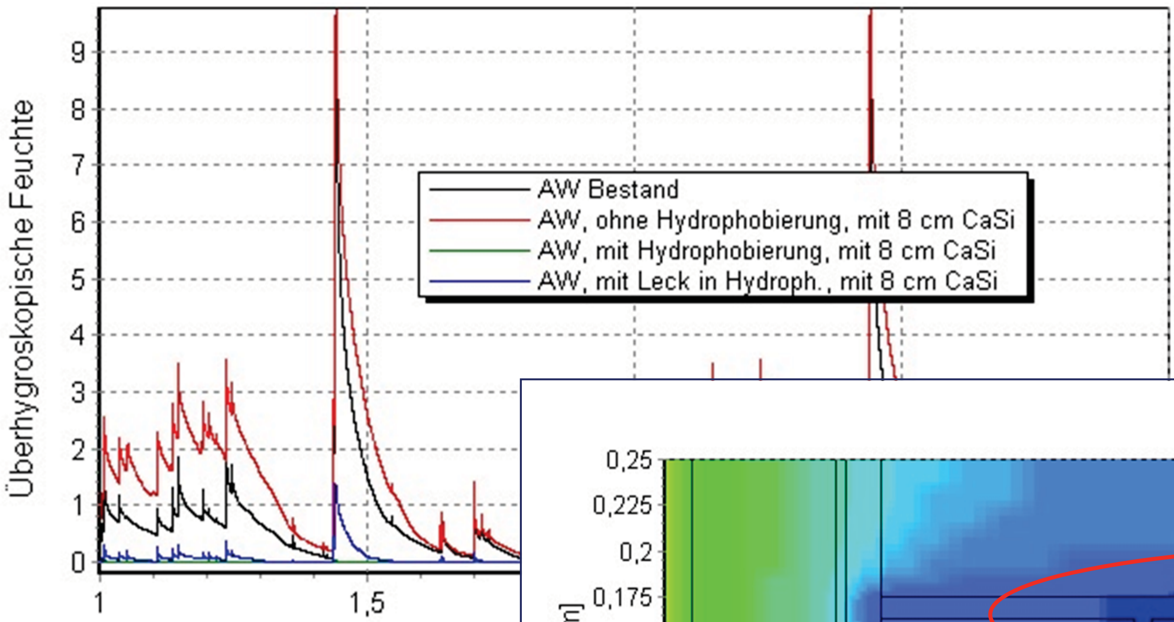
Area with faulty hydrophobisation

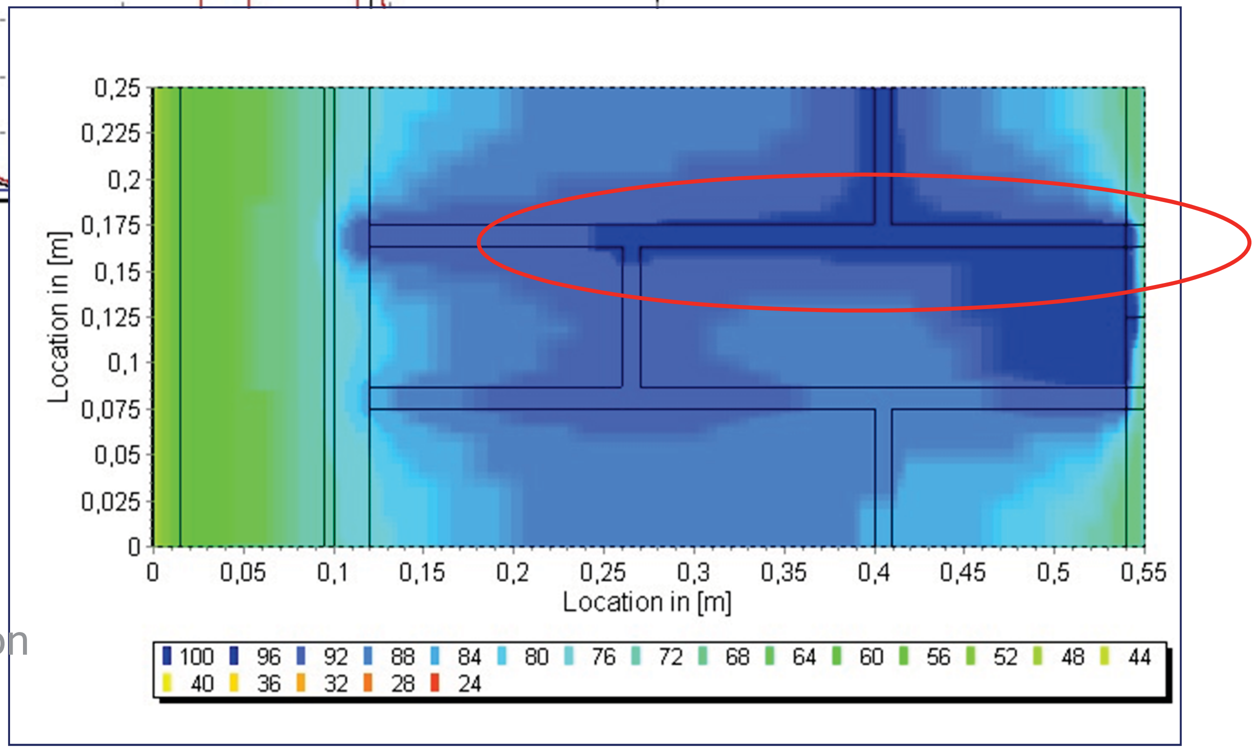


2D grid after discretisation

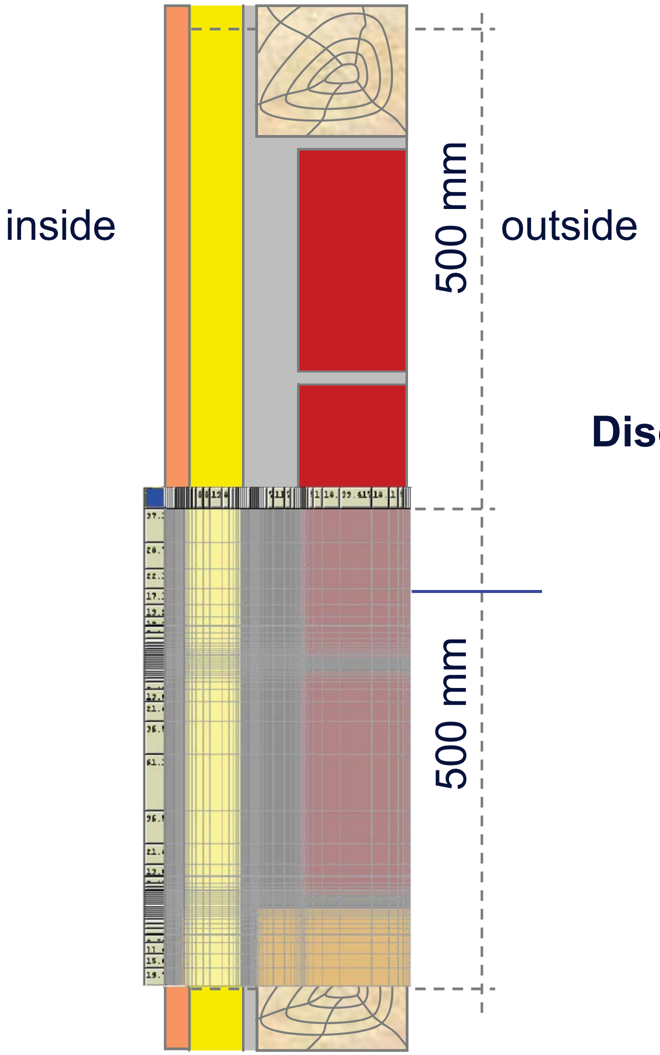
DELPHIN – Analysis of 2D problems

Analysis of variants

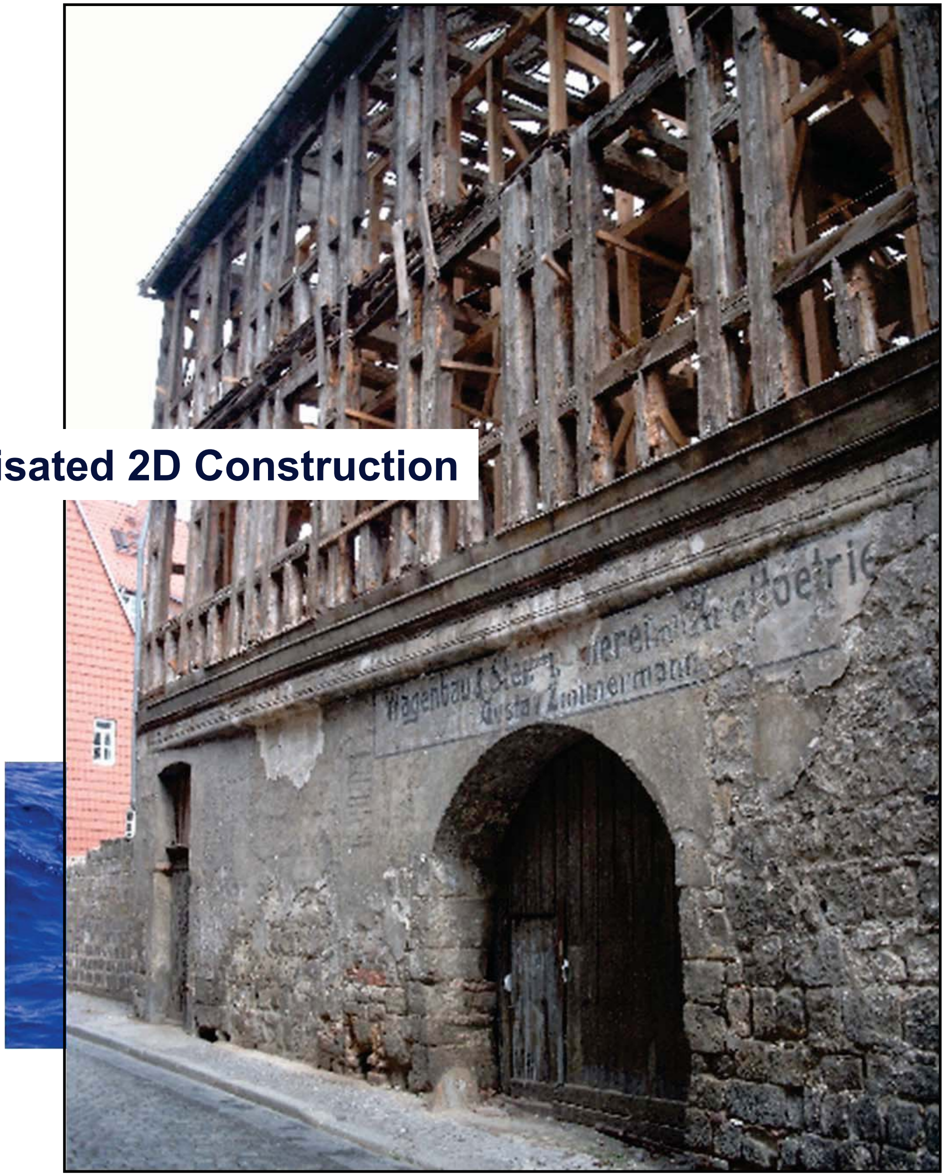




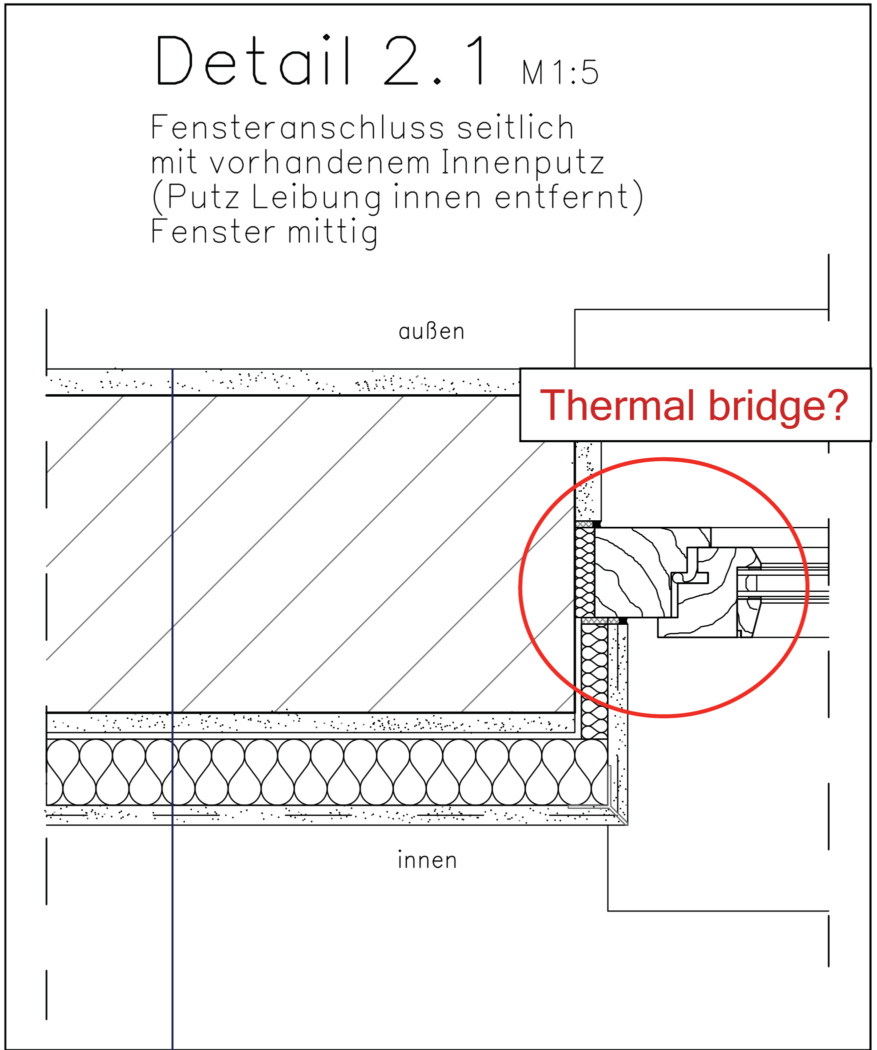
Field of humidity with faulty hydrophobisation

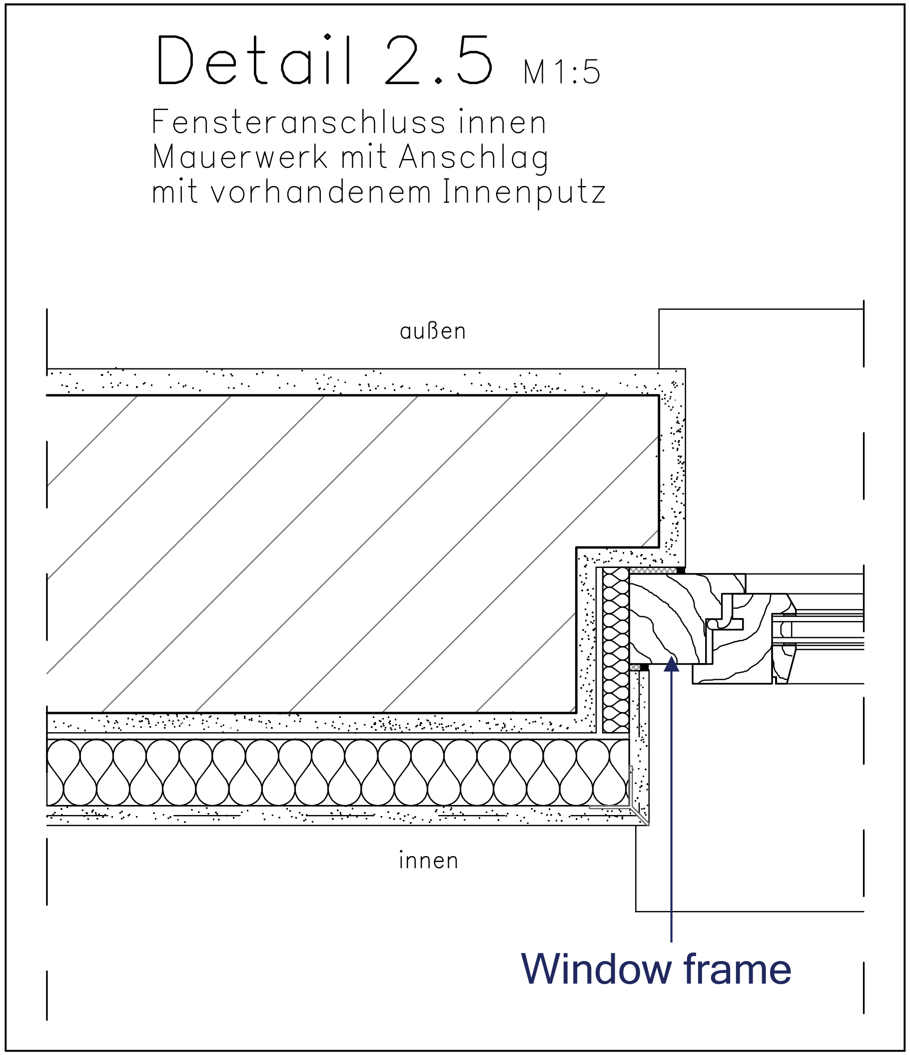


Discretisated 2D Construction



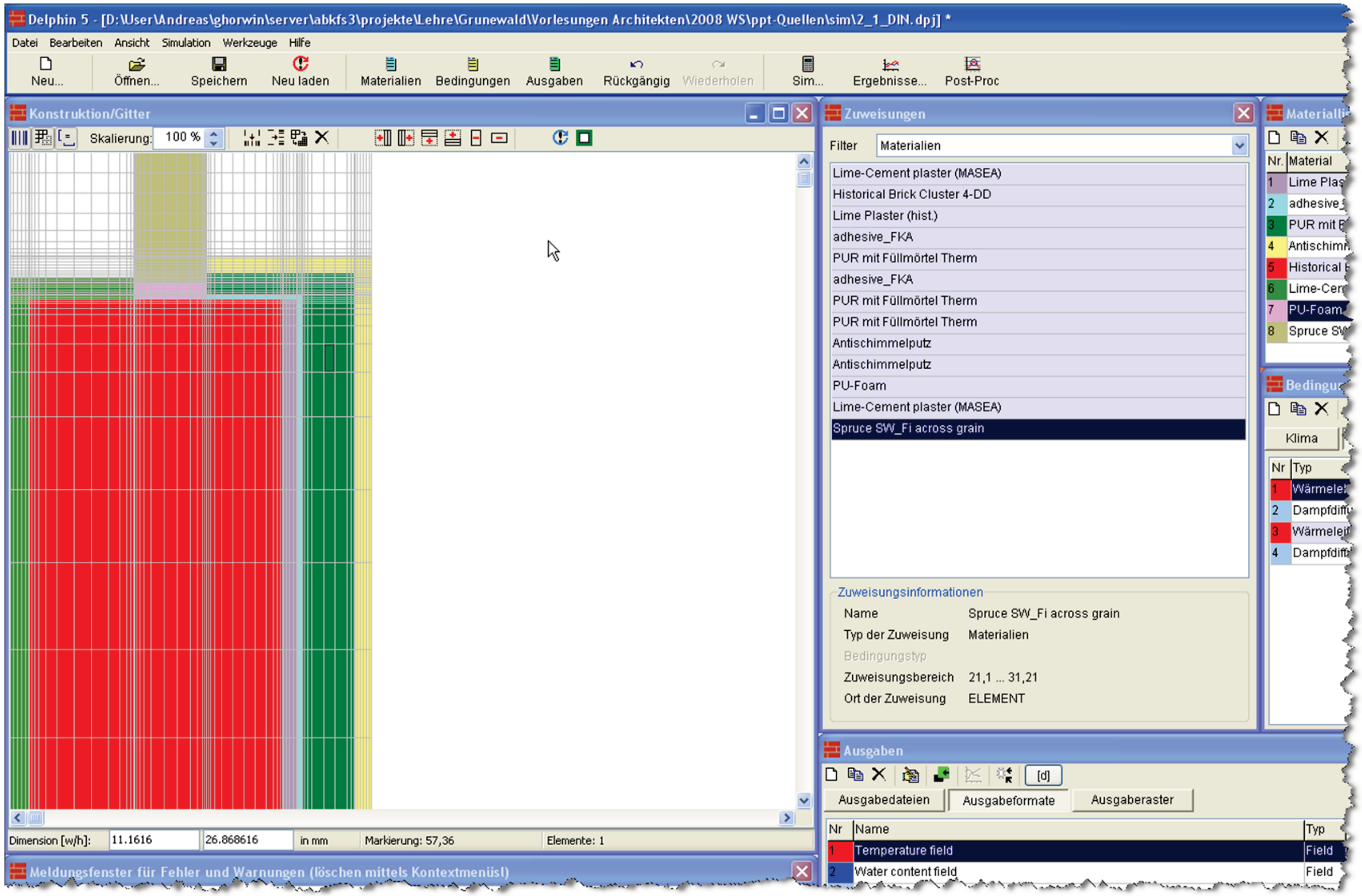
DELPHIN – Analysis of constructional details





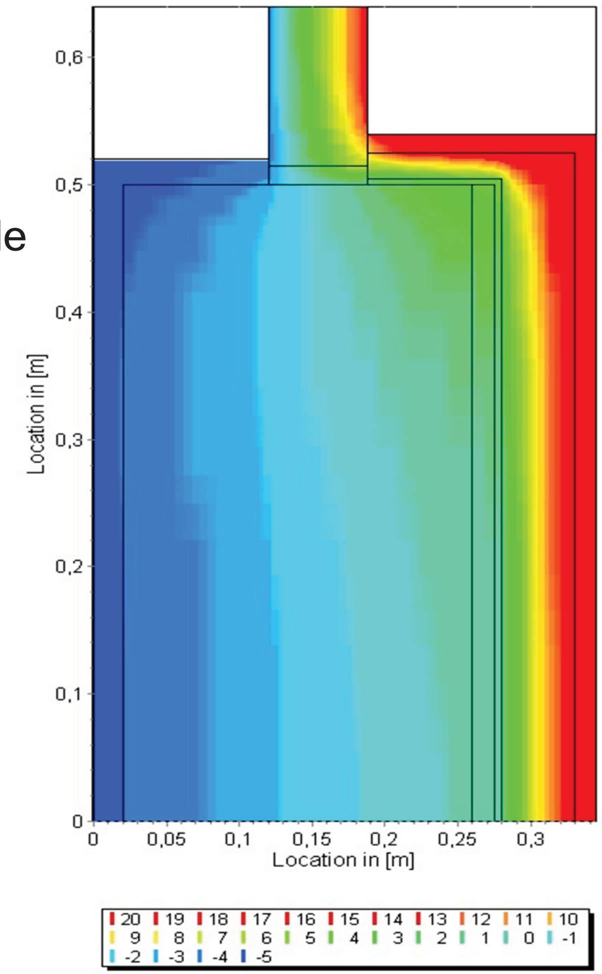
Plaster, Brick, Plaster, Glue mortar, Insulation, Plaster

- Analysis with DELPHIN



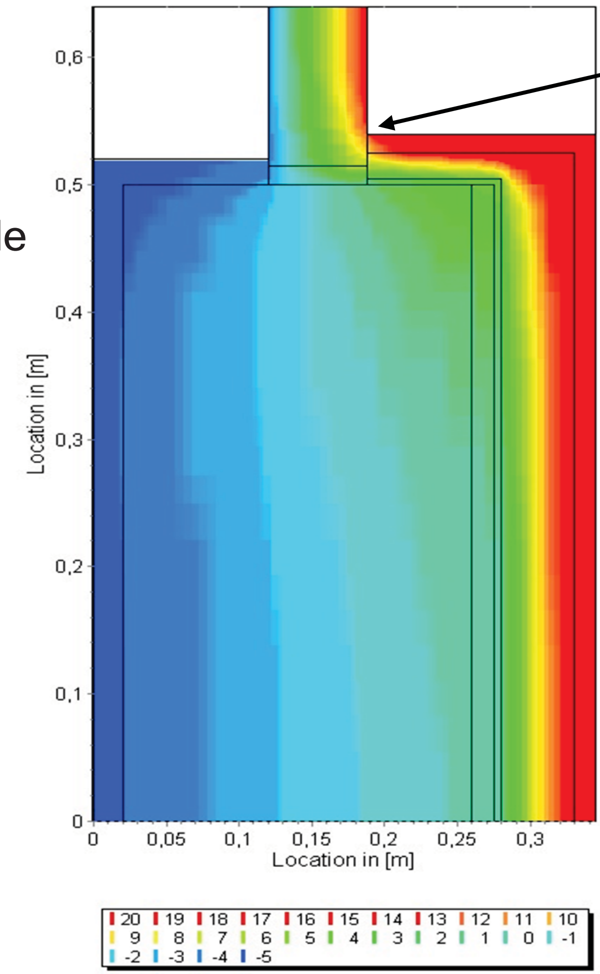
- Detail 2.1 - with window rabbet

outside



inside

- Detail 2.1 - without window rabbet



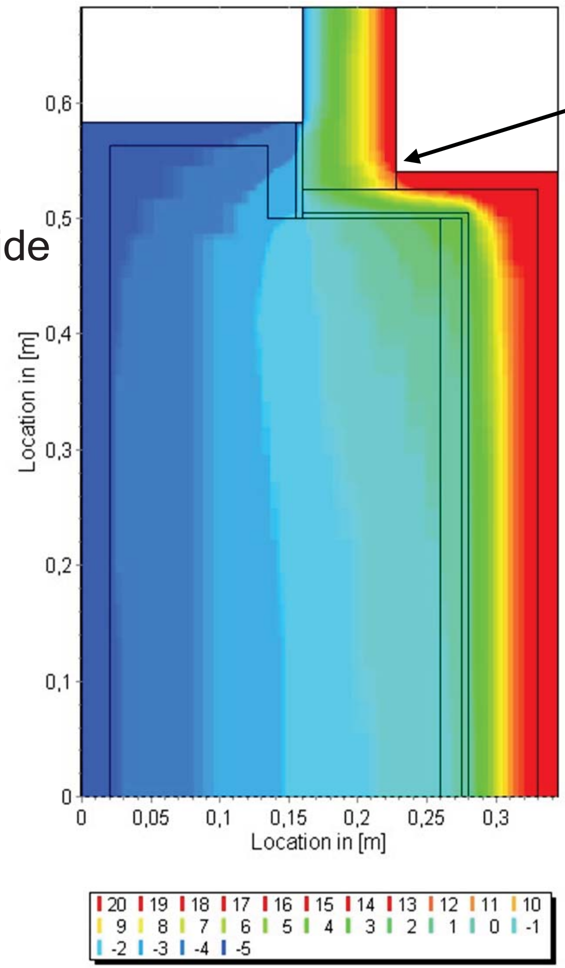
$$\vartheta_{oi} = 12{,}1°C$$

Incorrect! DIN 4108-2 demands:

$$\vartheta_{oi} > 12{,}6°C$$

- Detail 2.5 - with window rabbet

outside



inside

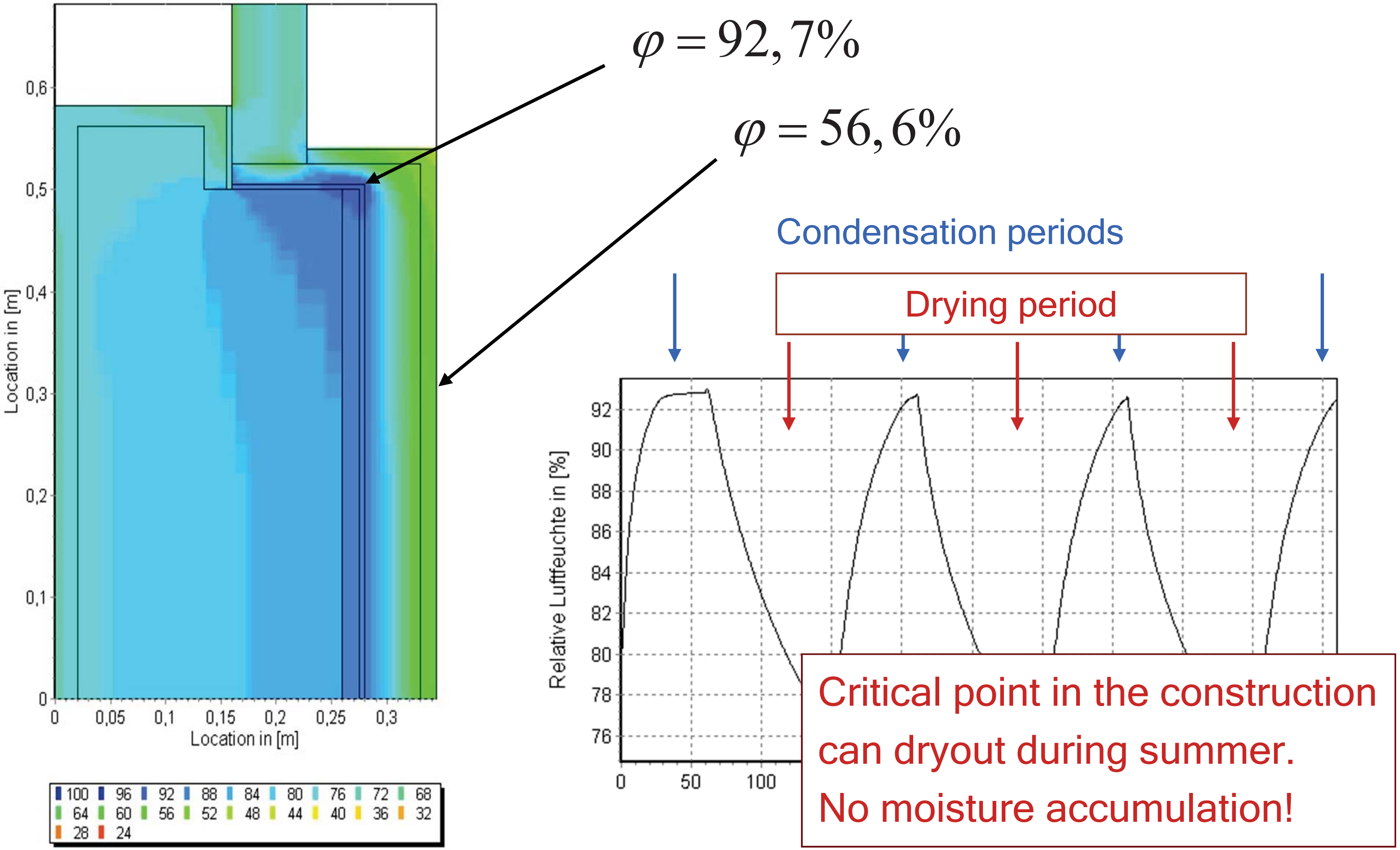
$$\vartheta_{oi} = 13,8°C$$

Ok, thermal protection fulfilled.

Minimum thermal protection of the guideline ensures, that the dew point temperature resp. minimum pretensions for mould are not reached or exceeded (e.g. on wallpaper).

What about the internal condensate?

- Critical moisture content: Condensation and drying behaviour



Moisture Damages

Overview

Introduction

Damages due to Moisture
Current state of damage assessment

Surface effects

Mould

Algae

Moisture inside building elements

Introduction
Ice forming
Insect infestation
mechanical damages

Introduction

Possible moisture damages:

- Destruction of moisture-sensitive materials (e.g. rotting, degradation)
- Biological damages due to mould, fungi, insects
- Corrosion of metals
- Penetration of frost boundary into the condensation zone (Freeze/Thaw-cycle, crystallation pressure)
- High vapour pressure due to high temperatures in moist areas
- Shrinkage and swelling (cracks)
- Salt transport with moisture transport (efflorescence, spalling).

Current state of moisture assessment in Germany

Standard compliant assessment for different constructions according DIN 4108-2, DIN 4108-3 und DIN EN ISO 13788:

Avoidance of…

- Moisture damages at surface (Condensation, Mould)
- Interstitial condensation
- (Avoidance/Reduction of driving rain penetration)

Standard compliant assessment for different constructions according DIN 4108-2, DIN 4108-3 und DIN EN ISO 13788:

Methods partially old, with many simplifications and restrictions.
New numerical methods only mentioned.

↓ ↓ ↓

Which climatical influences, damage potentials, boundary conditions (e.g. climate, initial conditions) or physical processes are considered?

Are the results meaningful for the damage or will be the calculation methods of the current procedures leads to excessively critical or wrong declarations?

Properties and restrictions of calculation methods

DIN 4108-2, DIN 4108-3 and DIN EN ISO 13788:

- All methods: steady state
- Thermal methods: 1D - (Minimum heat protection R), 2D or 3D (f_{Rsi}-factor)
- Hygrothermal methods („*Glaser*“): only heat and vapor transport

More analytical methods (COND) :

- Steady state, one-dimensional, heat, vapor and liquid water transport as well as moisture storage

Numerical methods (Delphin, Wufi):

- Arbitrary climate conditions (e.g. rain) and initial conditions
- one- and two-dimensional (in future also 3D)
- heat, vapor and liquid water transport and more physical phenomena (e.g. air flow, pollutant)
- Detailed information from every position in the construction

Feuchteschäden an der Oberfläche (Red. der Wärmeenergieverluste):

Mindestwärmeschutz
(Außenwand, DIN 4108-2)

$$R_{min} = 1{,}2\, m^2 \cdot K / W \leq R = \sum_{i=1}^{n} \frac{\lambda_i}{s_i}$$

Mikroklima auf der Innenoberfläche
(DIN 4108-2, DIN EN ISO 13788)

$$f_{Rsi} = \frac{\theta_{si} - \theta_e}{\theta_i - \theta_e} \geq 0{,}7$$

Feuchteschäden in der Konstruktion:

Verdunstung des Tauwassers
(DIN 4108-3, DIN EN ISO 13788)

$$m_{W,T} \leq m_{W,V}$$

Begrenzung der Tauwassermenge*
(DIN 4108-3, DIN 68800-2)

$$m_{W,T} \leq \ldots kg / m^2$$

* Grenzwert abhängig von Art und Kapillarität des Materials

Mindestwärmeschutz / Heizenergieverluste
(Außenwände, DIN 4108-2)

$$R_{\min} = 1{,}2\, m^2 \cdot K/W \leq R = \sum_{i=1}^{n} \frac{\lambda_i}{s_i}$$

stationär

$$R_{\min} = 1{,}2\, m^2 \cdot K/W \leq R_{trans} = \frac{\overline{q_{si,HP}}}{\overline{\theta_{i,HP}} - \overline{\theta_{e,HP}}}$$

instationär

λ - Wärmeleitfähigkeit
s - Schichtdicke

trans – Transient (instationär)
HP - Heizperiode
q - Wärmefluss
si - Innentemperatur
q - Temperatur

Für Simulation: Definition des Außenklima

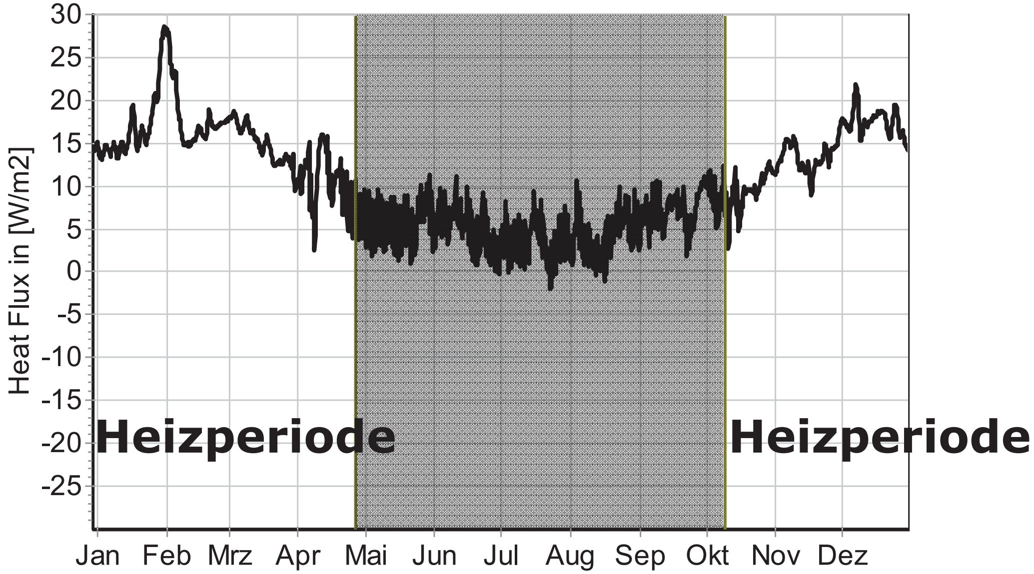
Definition des Innenklima,
(evtl. entwickelt aus dem Außenklima ¯ EN 15026)

Mindestwärmeschutz / Heizenergieverluste

(Außenwände, DIN 4108-2)

$$R_{min} = 1{,}2\, m^2 \cdot K / W \le R_{trans} = \frac{\overline{q_{si,HP}}}{\overline{\theta_{i,HP}} - \overline{\theta_{e,HP}}}$$

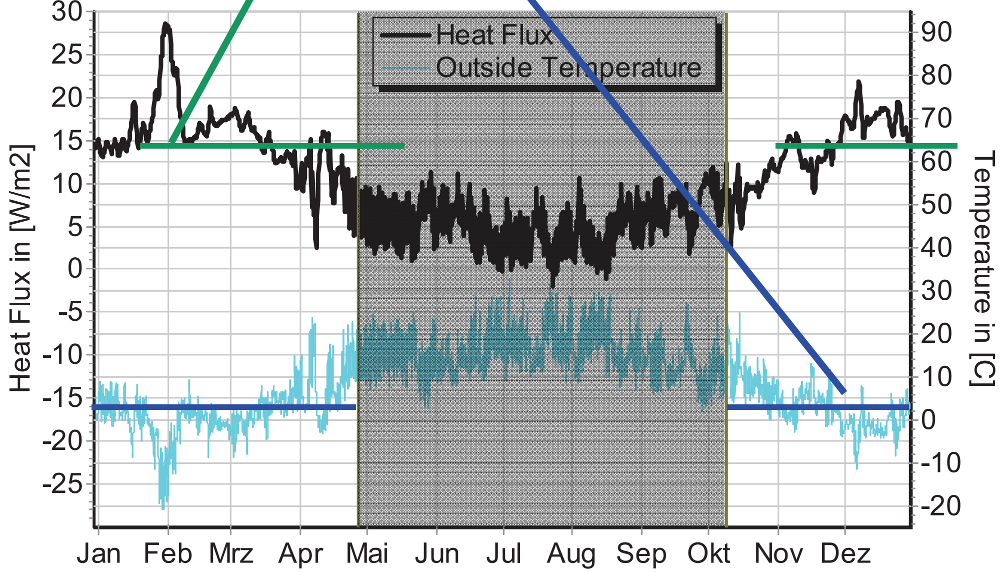
trans - Transient
HP - Heizperiode
q - Wärmefluss
si - Innentemperatur
q - Temperatur



Mindestwärmeschutz / Heizenergieverluste
(Außenwände, DIN 4108-2)

$$R_{min} = 1{,}2\,m^2 \cdot K\,/\,W \leq R_{trans} = \frac{\overline{q_{si,HP}}}{\overline{\theta_{i,HP}} - \overline{\theta_{e,HP}}}$$

trans - Transient
HP - Heizperiode
q - Wärmefluss
si - Innentemperatur
q - Temperatur



Surface effects

Relative humidity at construction surface – steady state

Surface temperature

$$q = \frac{\theta_i - \theta_e}{R_{ges.}} = \frac{\theta_i - \theta_{oi}}{\frac{1}{\alpha_i}} \quad \Rightarrow \quad \theta_{oi} = \theta_i - \frac{\theta_i - \theta_e}{R_{ges.} \cdot \alpha_i}$$

Surface vapor pressure

$$g_v = \frac{p_{vi} - p_{ve}}{r_{v,ges.}} = \frac{p_{vi} - p_{voi}}{\frac{1}{\beta_i}} \quad \Rightarrow \quad p_{voi} = p_{vi} - \frac{p_{vi} - p_{ve}}{r_{v,ges.} \cdot \beta_i}$$

If $(r_{v,ges} \cdot \beta_i) >> 1$

$$p_{voi} \approx p_{vi}$$

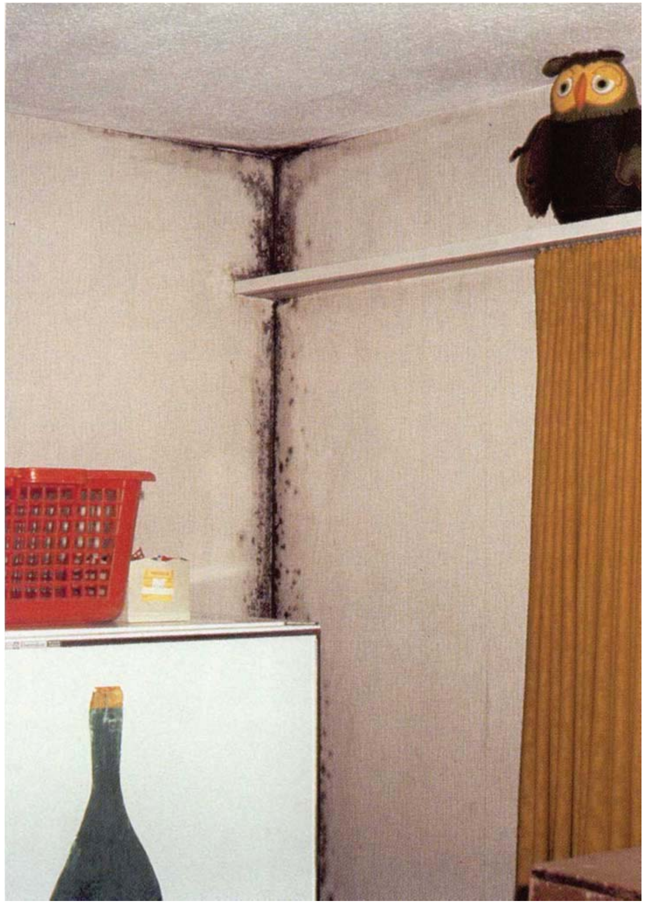
Surface relative humidity

$$\varphi_{oi} = \frac{p_{voi}}{p_s(\theta_{oi})} = \varphi_i \frac{p_s(\theta_i)}{p_s(\theta_{oi})}$$

Surface effects

Mould

Micro climate at inner surface



Moisture assessment - Mould

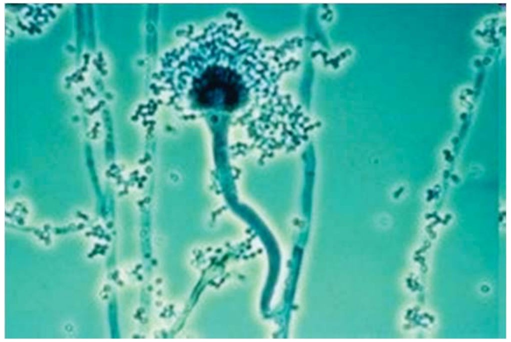
Robert-Sterl-Haus, Naundorf / Pirna

Health hazards due to mould:

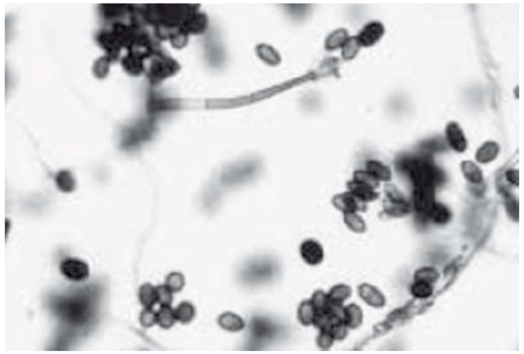
- Allergy
  - Proof is complex (prick test)
  - Tests shows that 5% of the german population has antibodies against mould (contact)
- Toxic effects
  - Some metabolites of mould can be toxic (Myco toxine, Glucans) or carcinogenic (Ochratoxin, Aflatoxine)
  - Plays normally no role for indoor air pollution (more in case of food and work places with high loads)
- Infections
  - Is possible in case of immunodeficient persons
  - Plays normally no role for indoor air pollution (maybe for hospitals)
- Odour nuisance
  - Negative effect on quality of life
  - Sometimes used as proof for mould growth.

Especially proplematic kinds of mould:

- Increase of risk for allergies with high spore formation (Penicillium marneffei, Aspergillus fumigatus)
- Infectios must likely from Aspergillus fumigatus, Aspergillus flavus, Cladophialophora bantiana
- Stachybotrys chartarum can emit a toxin (seldom and very demanding)



Aspergillus fumigatus



Stachybotrys chartarum

Mould - Bacteria

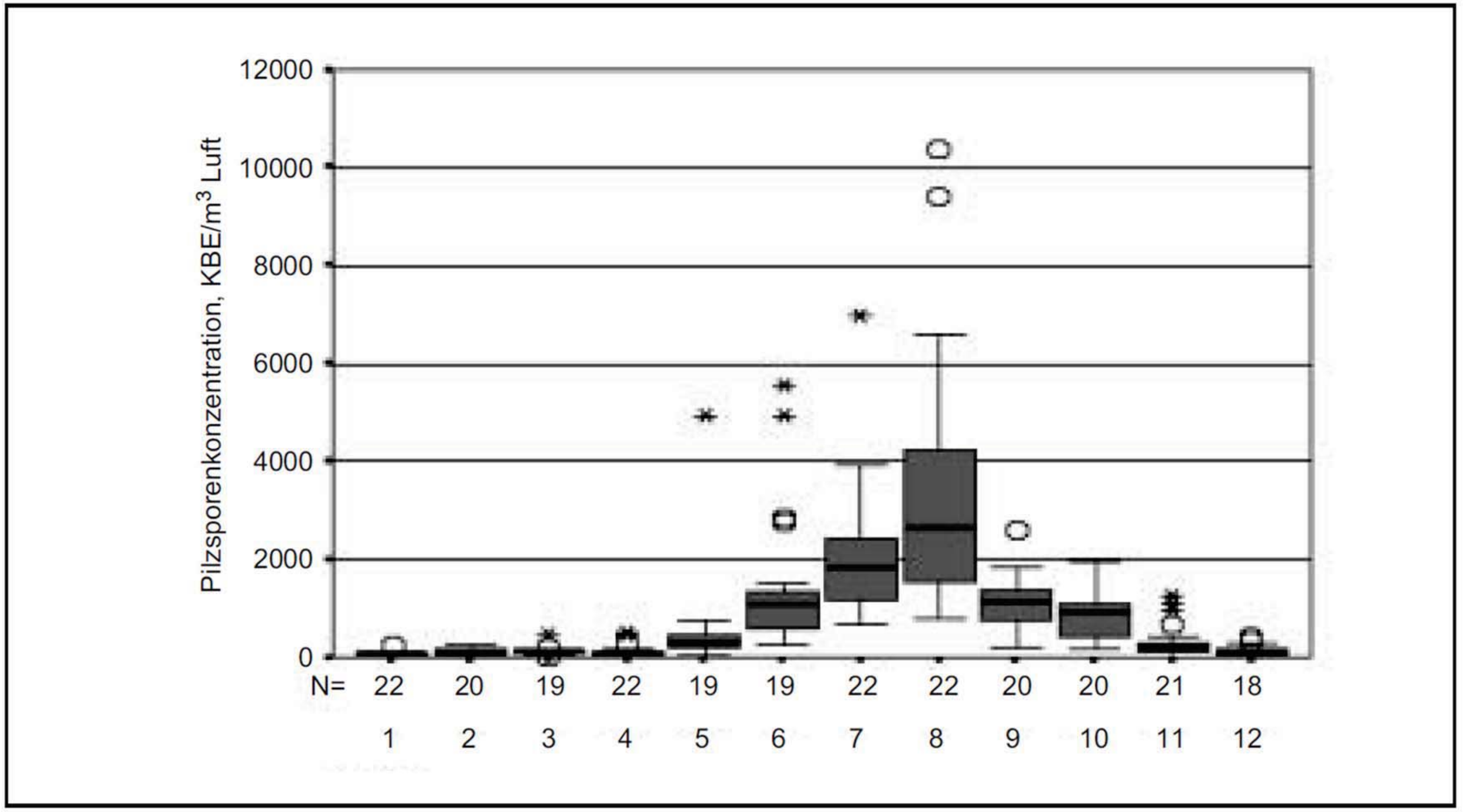
- It can be assumed that a greater growth of bacteria exists if the conditions are favourable for mould growth
- Studies have shown that, especially for high relative humidities, bacteria can growth also without visible mould growth
- Health risks are possible – up to now not studies about this topic exist.

Micro climate at the inner surface

Mould growth depends on the following factors:

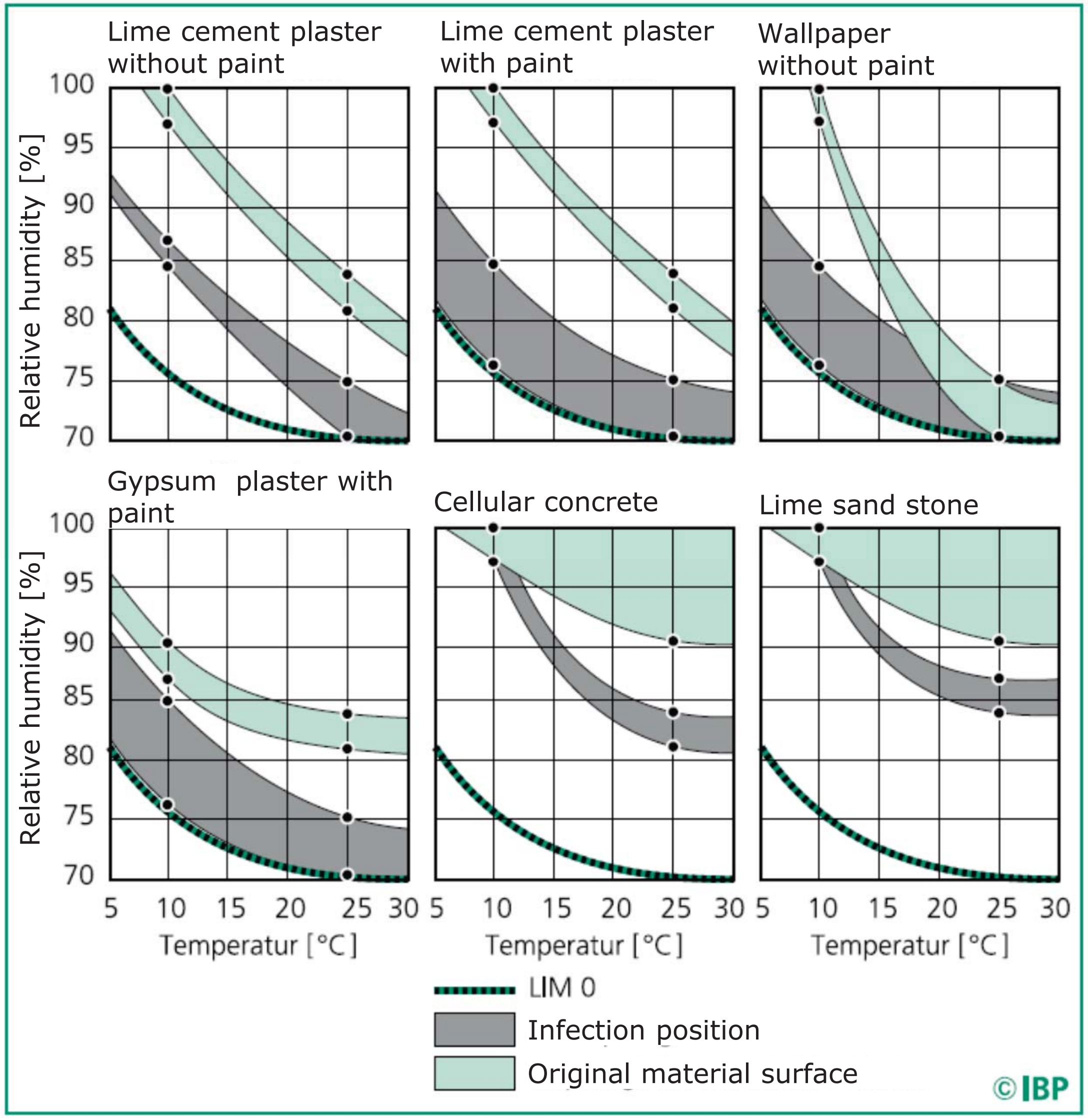
- Combination of temperature and relative humuidity
  - Germination and growth happens only for specific limit values
  - Isoplethes are lines of same growth in a $\varphi(\theta)$-diagram (time dependend)
- Nutrient content
  - Isoplethen differs according nutrient content
  - Mould cannot growth without nutrients
- pH-value
  - Optimal around 5 – 7
  - Tolerated from 2 to 11
  - Especially alkaline surfaces can supress mould growth

Spore concentration in outer air over a year



Quelle: Koch, A., K.J., Heilemann, J., Heinrich, H.E., Wichmann; W., Bischof (2000): Indoor viable mold spores – a comparison between two cities, Erfurt (Eastern Germany) and Hamburg (Western Germany). Allergy 55: 176-180

Moisture assessment - Mould



Isoplethes for different types of materials and surfaces.

Differences between infection postion and normal material surface.

Source: IBP Mitteilung 457, 32 (2005)

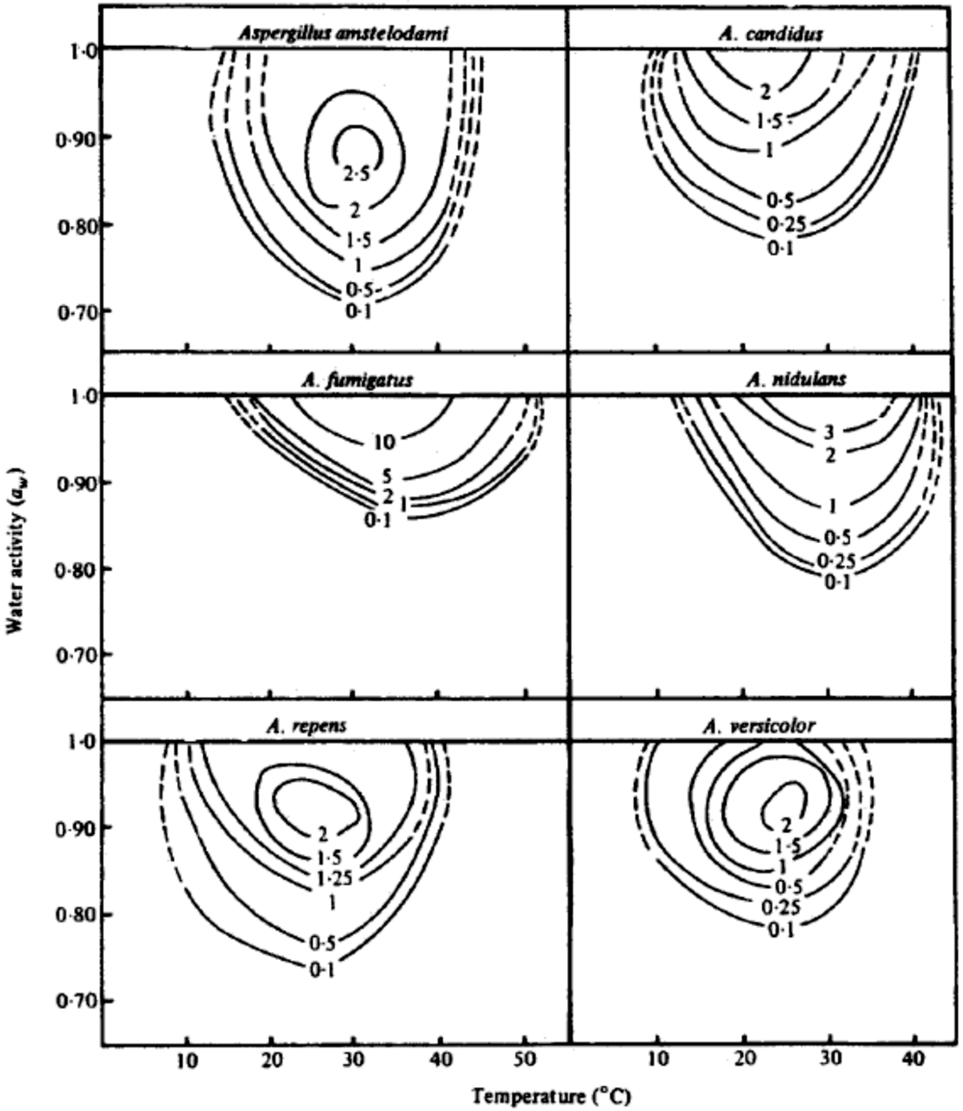
Moisture assessment - Mould

Minimum, optimum and maximum growth requirements of different mould species for temperature, relative humidity and pH according germination and mycelial growth for different risk classes.

Species	Risk class	Growth requirements												
		Temperature [°C]						Rel. Humidity {%]				pH [-]		
		Germination			Mycelial growth			Germination		Mycelial growth				
		min	opt	max	min	opt	max	min	opt	min	opt	min	opt	max
Asp. Flavus	A	10	30	45	6	40	45	80	100	78	98	2.5	7.5	>10
Asp. Fumigatus	A	10	40	50	10	43	57	80	97	82	97	3	6.5	8
Asp. Nidulans	A	10	37	50	6	40	48	75	95	78	97			
Asp. Niger	A	10	35	50	6	37	47	77	98	76	98	1.5		9.8
Asp. Penicillioides	A				5	25	37							
Asp. Versicolor	A	8	30	42	4	30	40	74	91	75	95			
Stachybotris atra	A	5	25	40	2	23	37	85	97	89	98			
Risk class A	A	5	33	50	2	40	57	74	96	75	97	2	7	10

Source: Dissertation Dipl.-Ing. Klaus Sedlbauer, Universität Stuttgart

Moisture assessment - Mould



Isoplethes for different species of Aspergillus on optimal nutrients.

Quelle: Nielsen, K. F.; Nielsen, P. A.; Holm, G.:

Growth of moulds on building materials under different humidities.

Proceedings of Healthy Buildings (2000),

Vol. 3, S. 283 - 288

Micro climate at inner suface:

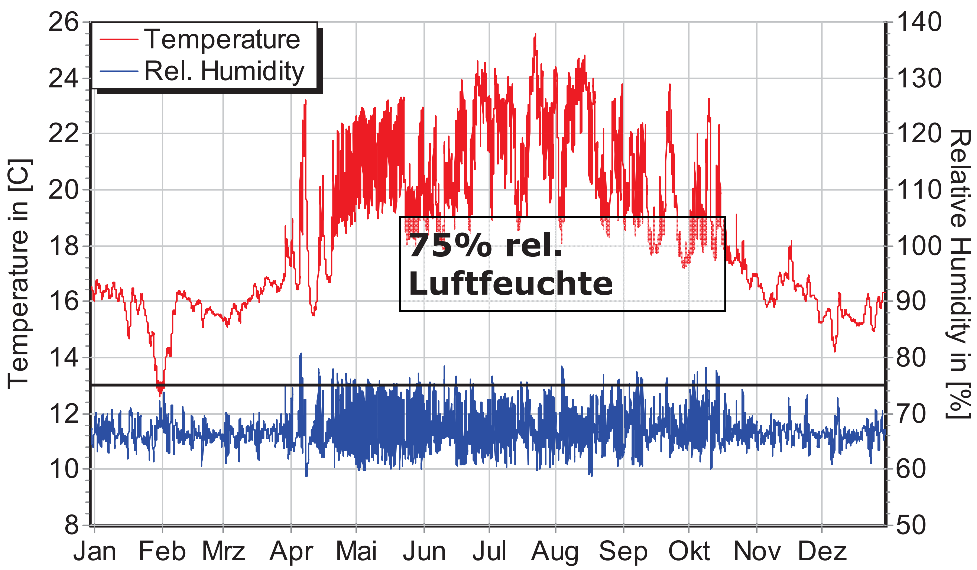
Temperature factor f_{Rsi} (according DIN 4108-3 S. 17f)

$$f_{Rsi} = \frac{\theta_{si} - \theta_e}{\theta_i - \theta_e} \geq 0{,}7$$

Corresponds for θ_i=20°C and θ_e=-5°C → θ_{si}>12,5°C

This corresponds to φ_i=50% K φ_{si}≈80%

Mould growth:
Under very favourable conditions from
75% rel. humidity



Micro climate at inner suface:

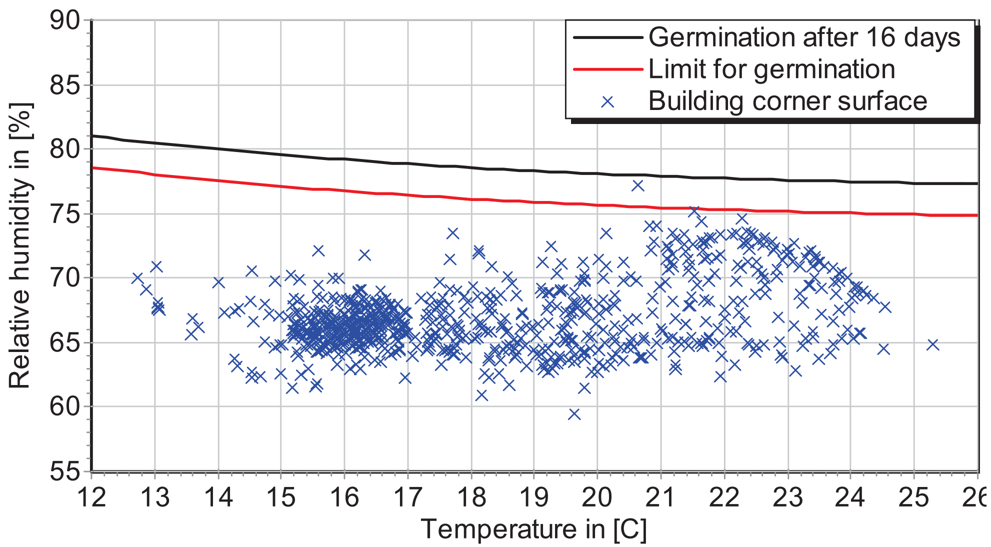
Mould growth prediction

→ Isopleth-model (Sedlbauer, WTA-Merkblatt 6-3-05)

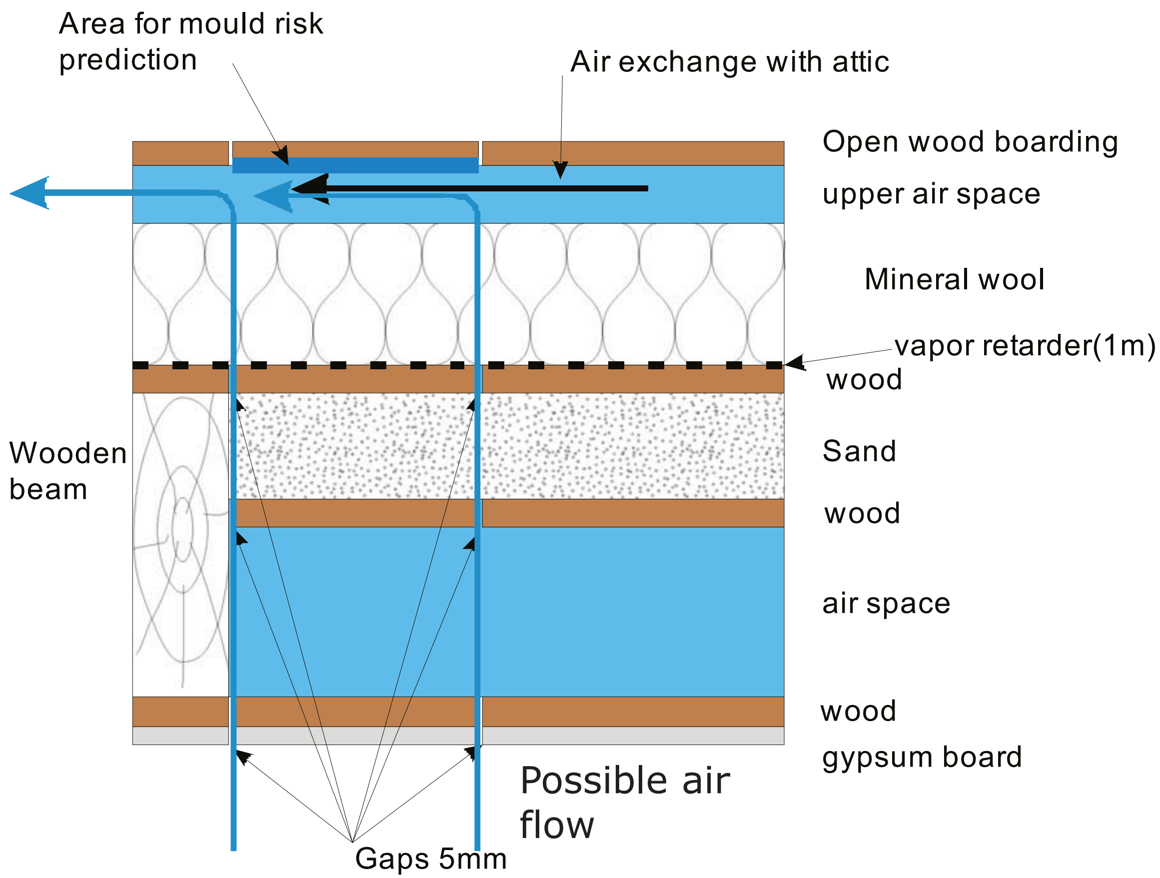
→ Viitanen et al. (dynamic model)

(e.g. 1997. Modelling the time factor in the development of mould fungi in wood—The effect of critical humidity and temperature conditions. *Holzforschung* 51(1):6–14.)

Isoplethes for spore germination (lines) with averaged data φ(θ) at a inner surface of a building corner



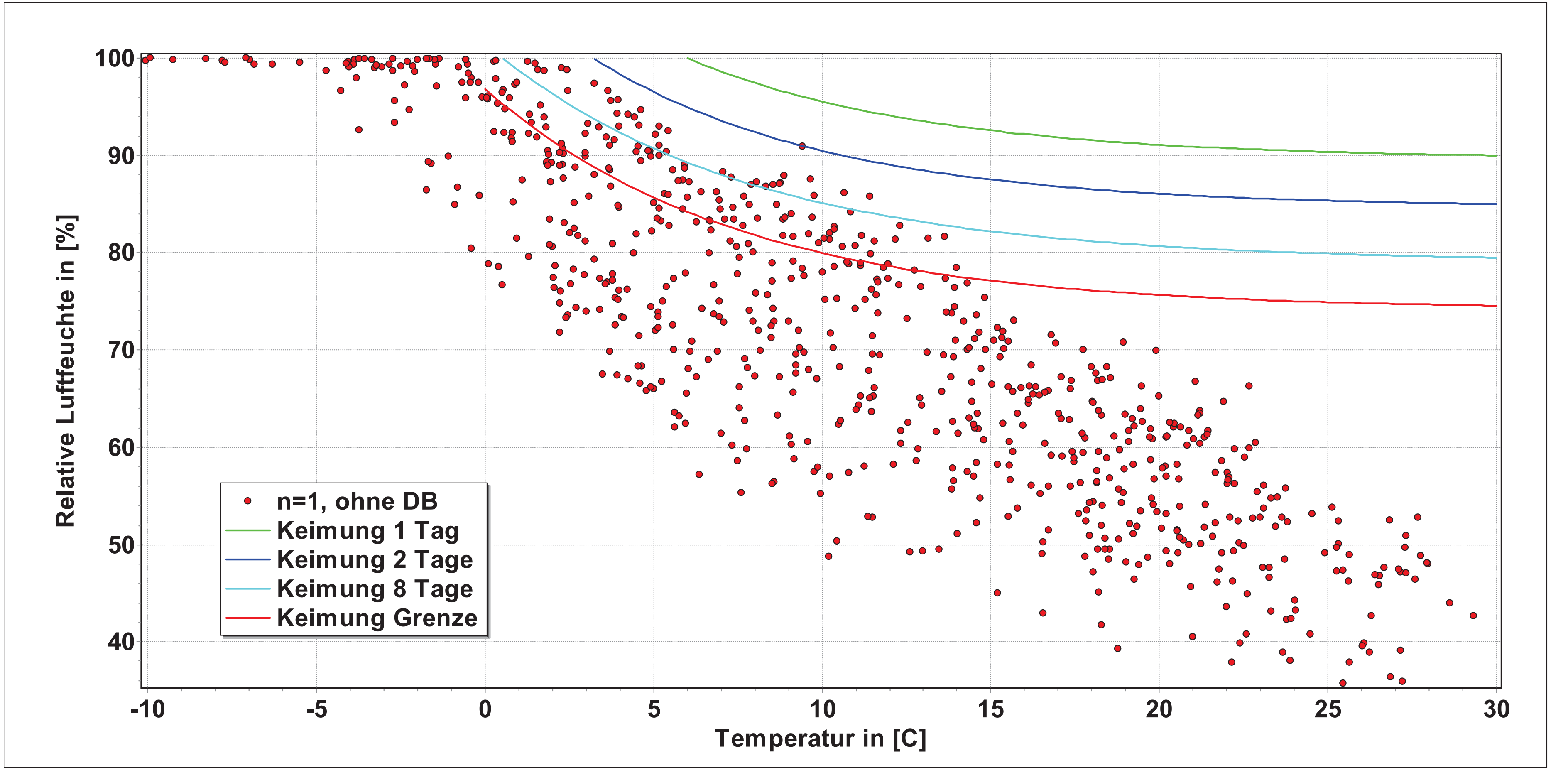
Example – storey ceiling between normal room and unheated attic



This construction is calculated with Delphin in different variants /with and without air flow through the construction)

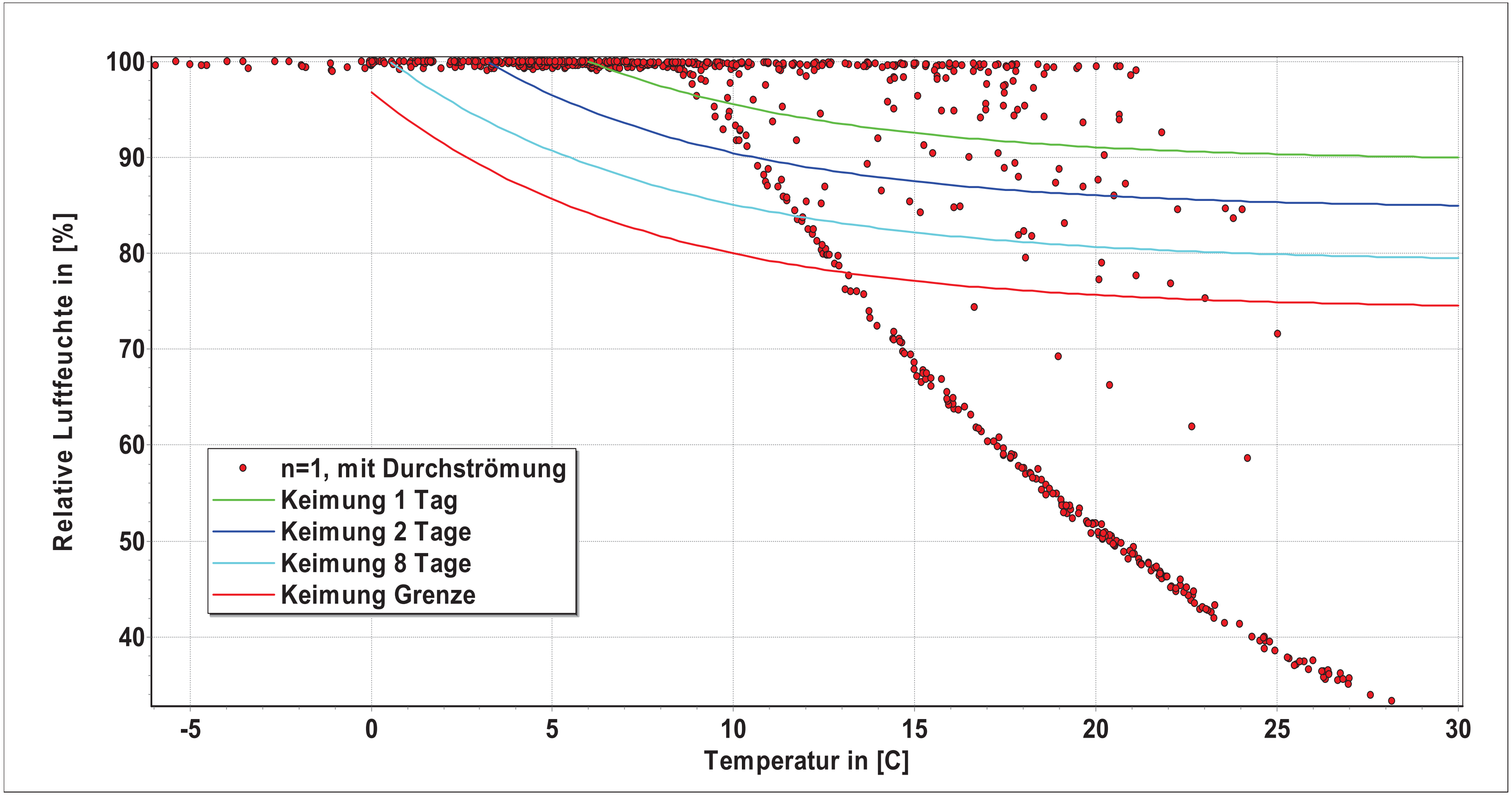
Construction

Example – storey ceiling between normal room and unheated attic



Isopleth diagram, without vapor retarder, airtight construction

Example – storey ceiling between normal room and unheated attic



Isopleth diagram, without vapor retarder, with airflow

Mould– Viitanen Model

- The mould prediction model from Viitanen is an empirical dynamic modell
- Basis for calculation are the hourly values of temperature and relative humidity at the building surface
- Calculation procedure:
  - control of temperature limits (between 0 and 50°C)
  - Calculation of critical relative humidity for germination
  - If humidity is higher – calculation of mould growth intensity (depends on temperature, moisture, material, surface)
  - From this calculation of a cumulative mould index
  - If humidity falls below the critical value – calculaion of a declining intensity
    - *Will be subtracted from mould index*
- Result is a time depending mould index

Mould– Viitanen Model

Mould index	Growth	Description
0	No growth	Spores not activated
1	Small amount of mould on surface (microscop)	Initial stages of growth
2	<10% coverage (microscop)	-
3	<10% coverage (visual)	New spores produced
4	10-50% coverage (visual)	Moderate growth
5	>50% coverage (visual)	Plenty of growth
6	100% coverage (visual)	Very heavy and tight growth

Calculation approach

critical relative humidity:

$$RH_{crit} = \begin{cases} T > 20 \rightarrow -0.00267 \cdot T^3 + 0.16 \cdot T^2 - 3.13 \cdot T + 100.0 \\ T \leq 20 \rightarrow 80\% \end{cases}$$

Mould growth intensity ($\varphi > RH_{crit}$):

$$\frac{dM}{dt} = \frac{1}{7 \cdot \exp\left(-0.68\ln\left(T\right) - 13.9\ln\left(RH\right) + 0.14W - 0.33Q + 66.02\right)} \cdot k_1 \cdot k_2$$

M – Mould index
Q – Surface quality factor
W – Material factor
k_1, k_2 – correction coefficients

Calculation approach

Correction factor 1:

$$k_1 = \begin{cases} M < 1 \rightarrow 1.0 \\ M > 1 \rightarrow \dfrac{2}{t_v / t_m - 1} \end{cases}$$

t_v – time for germination until the first visible appearance
t_m – time for germination

$$t_m = \exp\left(-0.68\ln(T) - 13.9\ln(RH) + 0.14W - 0.33Q + 66.02\right)$$

$$t_v = \exp\left(-0.74\ln(T) - 12.72\ln(RH) + 0.06W + 61.5\right)$$

Correction factor 2:

$$k_2 = 1 - \exp\left(2.3\left(M - M_{\max}\right)\right)$$

M_{max} – biggest possible mould index under given conditions

$$M_{\max} = 1 + 7\frac{RH_{crit} - RH}{RH_{crit} - 100} - 2\left(\frac{RH_{crit} - RH}{RH_{crit} - 100}\right)^2$$

Calculation approach

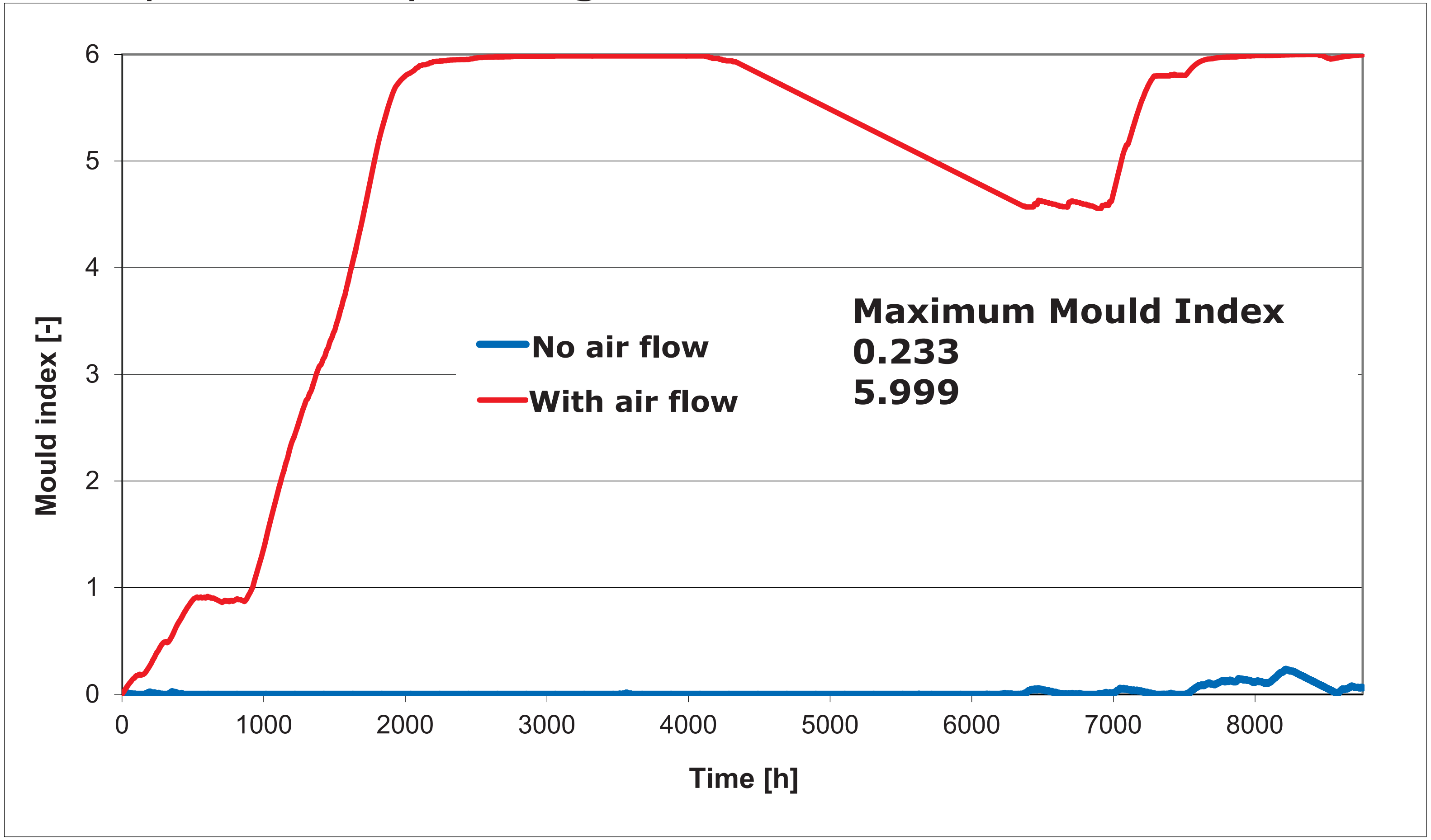
Declining rate:

$$\frac{dM}{dt} = \begin{cases} t - t_1 \leq 6h & -0.032 \\ 6h \leq t - t_1 \leq 24h & 0.0 \\ t - t_1 > 24h & -0.016 \end{cases}$$

t-t_1- Lenght of dry period

These value will be added to the mould index and can diminish this under poor conditions.

Example – storey ceiling between normal room and unheated attic



Mould index acc. Viitanen, without vapor retarder, with and without air flow

Surface effects

Algae

Algae growth at outer building surfaces

Up to now no model exists for algae growth at building surfaces.

Moisture assessment - Algae

View of a facade retrofitted with a insulation composite system.
The differences of algae growth is related to thermal bridges from mounting elements.

Source: Helmut Künzel, Angewandte Ökologie, ARCONIS 2/03

Algae growth at outer building surfaces

Up to now no model exists which can be used for calulating algae growth based on hygrothermal state variables like the mould models.

Temperature and (high) relative humidities are the dominating factors

Possible model:

→

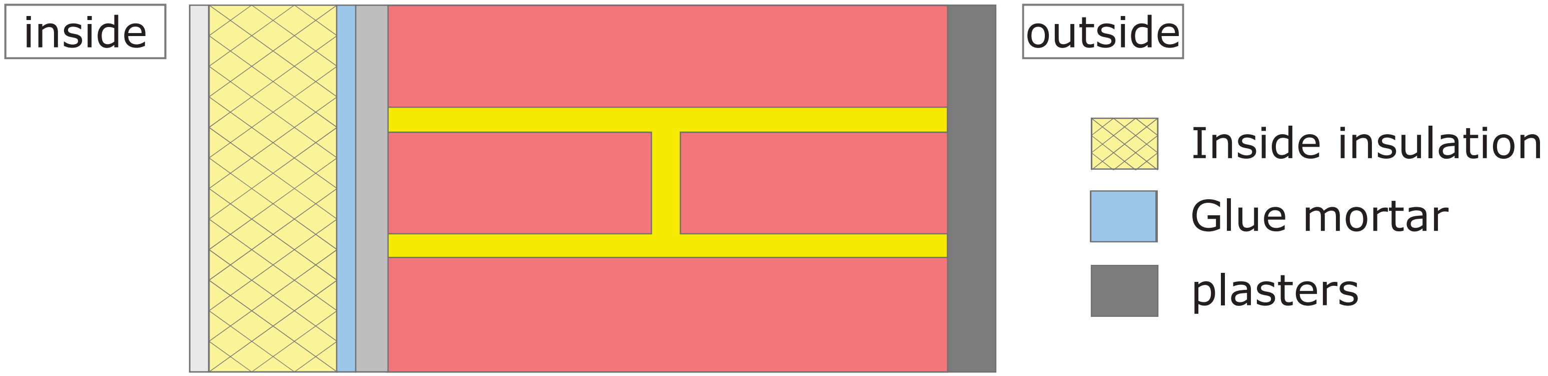
$t_{cond,e}$: yearly period with moisture content near condensation (> 98% r.L.) <u>and</u> temperatures over 0°C at the outer surface

Moisture inside the construction

Condensation: DIN 4108-3 (*Glaser-Method*),
DIN EN ISO 13788

Limits:

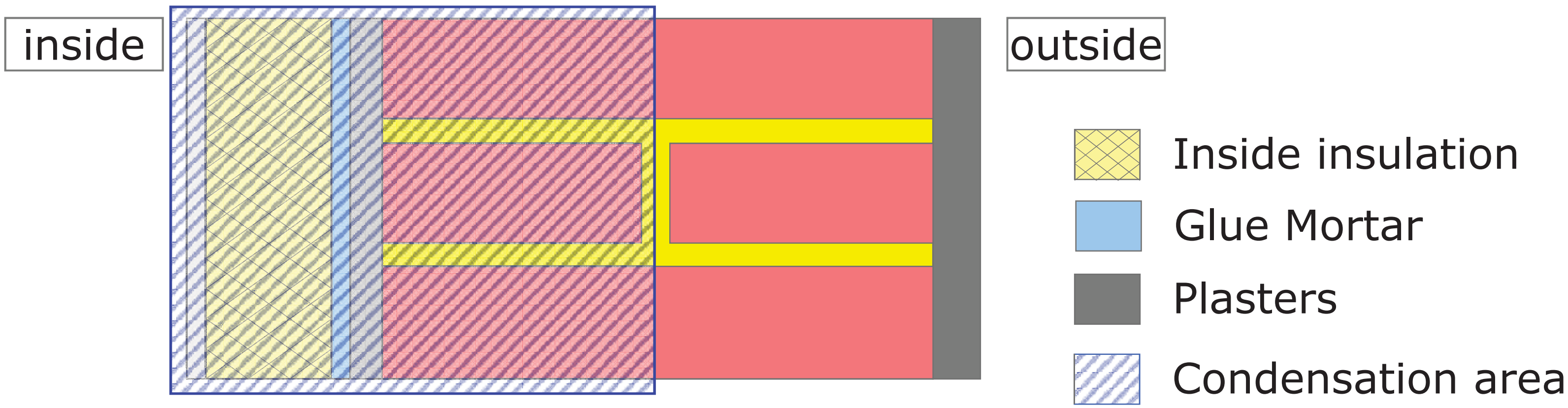
- Only heat and vapor transport, no moisture storage,
- (constant climate conditions,
- Material properties constant (not mousture or temperature dependend),...



Moisture assessment - Condensation

Condensation : Calculation with sophisticated methods

- "Reale" climate,
- Moisture storage,
- Vapor and liquid water transport,
- Material properties as functions of moisture content,...

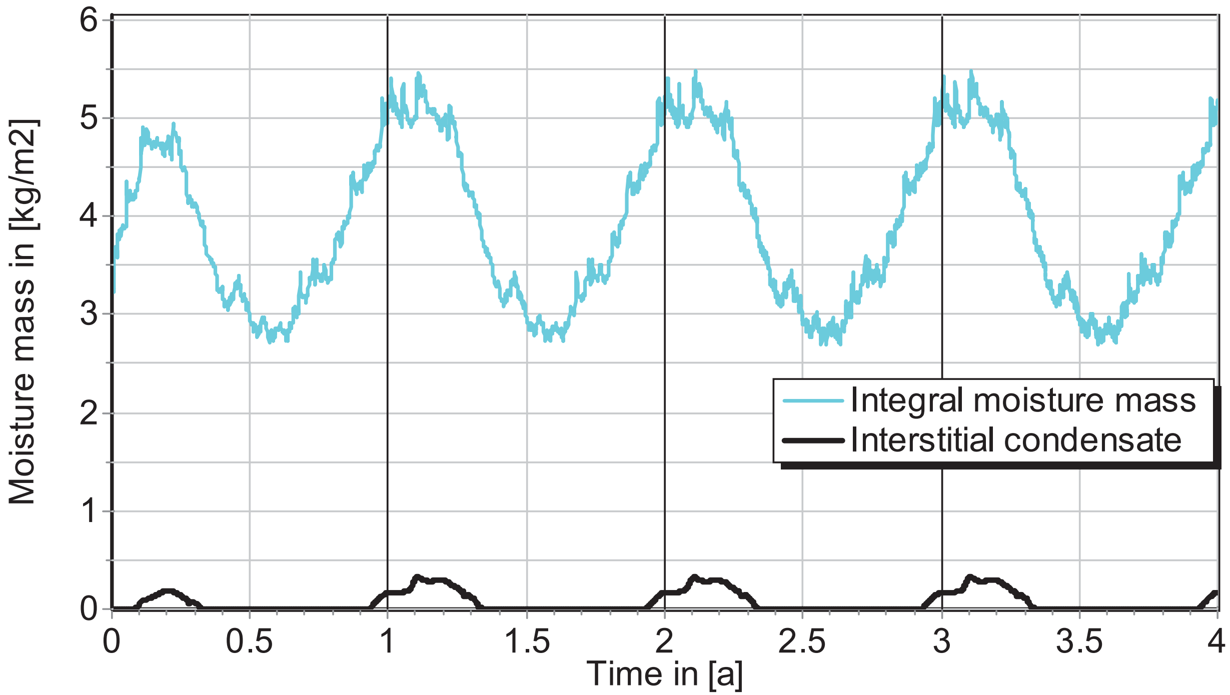


→ No rain, no condensation at outer surface

→ Comparison with standard: Only condensation at inside insulation and inner half of wall

Condensation : Calculation with sophisticated method

- "Reale" climate,
- Moisture storage,
- Vapor and liquid water transport,
- Material properties as functions of moisture content,...



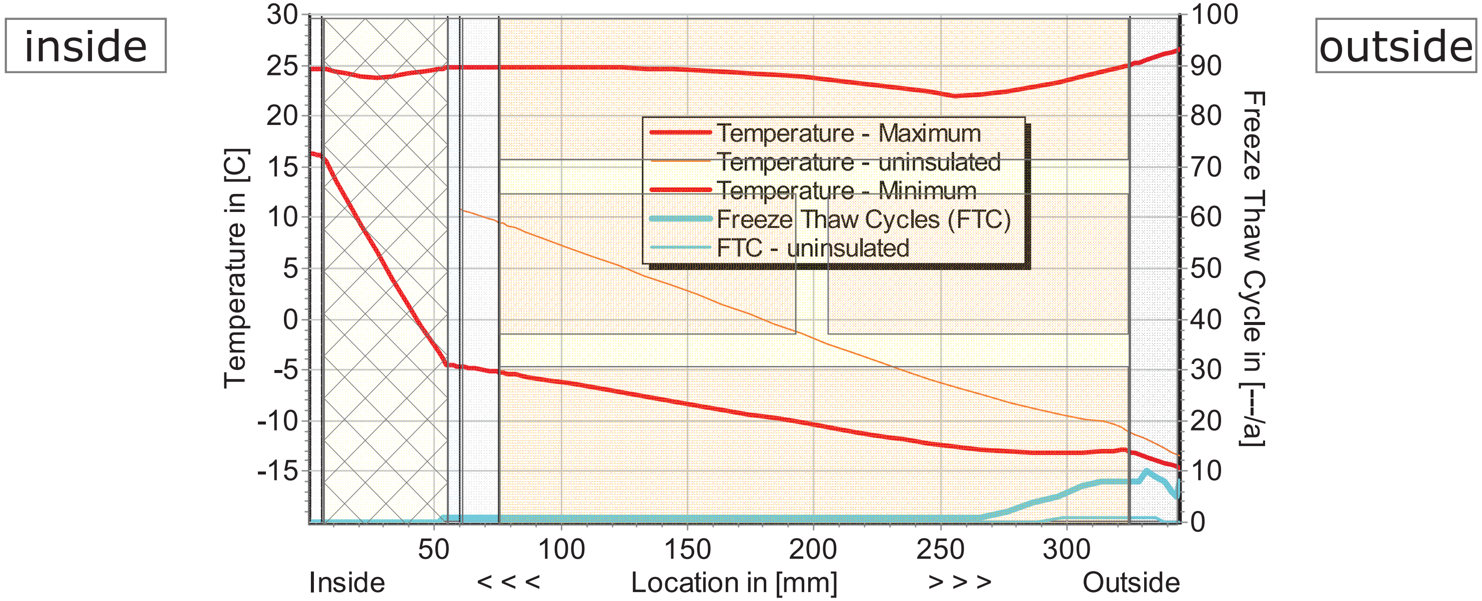
Hygrothermal load indicators

- Freeze/Thaw-cycles (Temperature and moisture profile)
- Biological vermin growth inside the construction
- Weathering load can lead to hygrothermal induced damages (swelling, shrinking)

Moisture inside the construction

Ice

Freeze/Thaw-cycles: Modell is based on thermo dynamical considerations: Freeze/Thaw-Cycle (TTC) = f(w,θ) (Xu 1996)

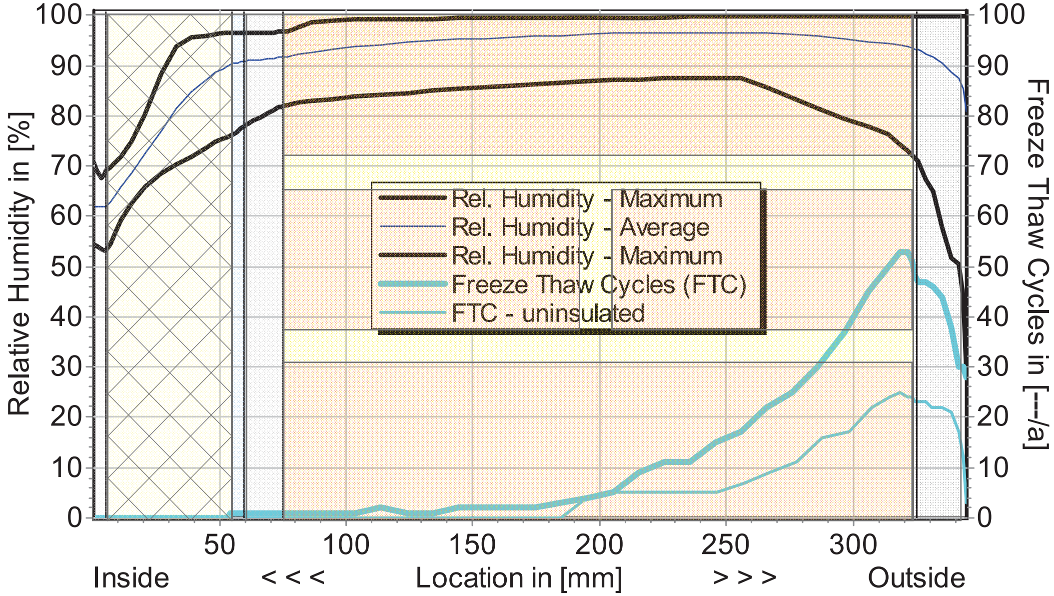


Temperature and Freeze/Thaw-Cacle profiles
(rated risk of FTW, no real damages)

Modell from: P. Häupl, Y. Xu; Numerical Simulation of Freezing and Melting in Porous Materials under the Consideration of the Coupled Heat and Moisture Transport, Thermal Envelope & Building Science, Volume 25 No 1, July 2001, pp. 4-31)

Freeze/Thaw-cycles: Modell is based on thermo dynamical considerations: Freeze/Thaw-Cycle (TTC) = f(w,θ) (Xu 1996)

innen



außen

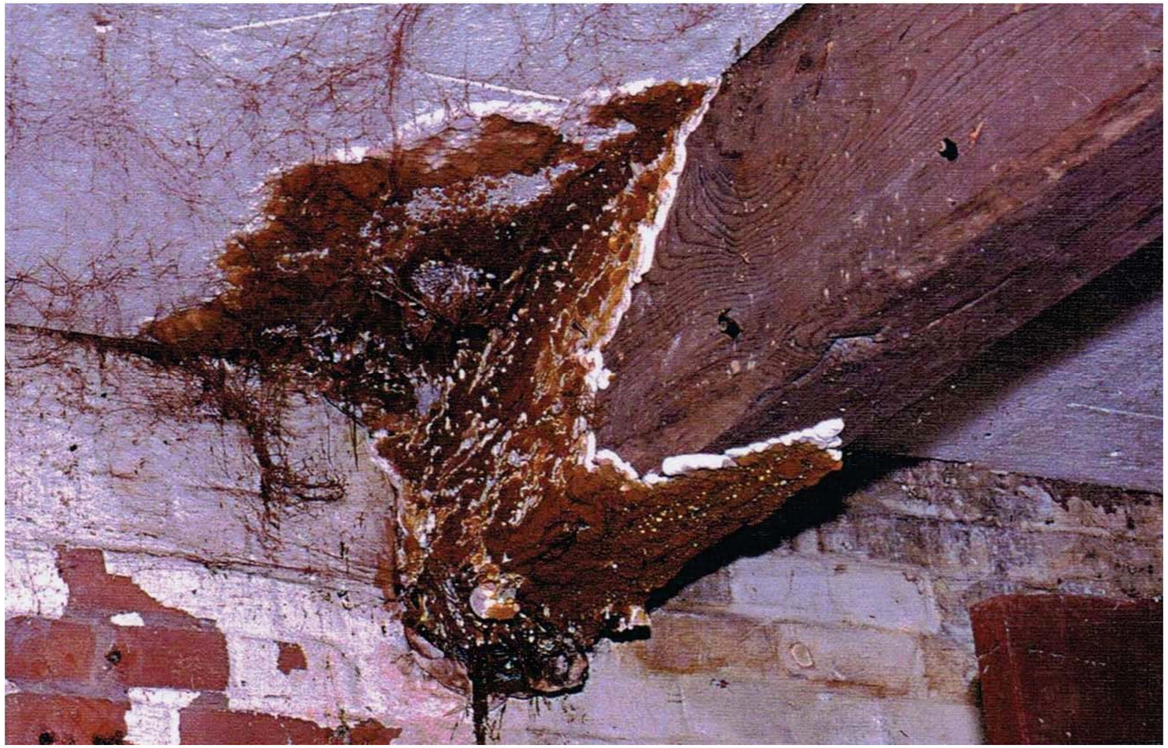
Charakteristische Profile der relativen Luftfeuchte und Frost-Tau Wechsel (bewerten Risiko von FTW, nicht tatsächlich zu erwartende Schäden)

Moisture inside the construction

Inner vermin growth

Vermin growth inside the construction

Aims to common and dangerous biological wood pests: Dry rot (*serpula lacrymans*)



Moisture assessment - Fungi

Historical building in Luckau

Vermin growth inside the construction

Aims to common and dangerous biological wood pests:
Dry rot (*serpula lacrymans*)

Standard (DIN 68800-1): Moisture content > 20 M% only "for short times"

Limiting factors:

- *Temperature*: Germination begins at 3-5°C
- *Germination*: From fibre saturation: 27-30 M% (Spruce)
- *Growth*: From 20 M% only, if fugi has ist own „water supply"

→ Long continually periods with high humidity is positive!

Vermin growth inside the construction

Aims to common and dangerous biological wood pests:
Dry rot (*serpula lacrymans*)

$t_{PGV,20/26}$ Yearly time period with temperatures higher than 2°C and moisture content >20 or 26 M%

$t_{PGV,20/26,max}$: longest continually time period with temperatures higher than 2°C and moisture content >20 or 26 M% (Unit: d)

PGV – Possible Growth of Vermin

Wood damages

- Wood-destroying fungi (dry rot, brown cellar rot)
- Fresh wood insects (bark beetle, shipyard beetle)
- Dry wood insects (old house borer [xylophages], gnawing beetle or wood worm)
- Wood dwelling but not destroying fungi (mould or blue stain fungi)
- Loss of strength of the wood if moisture content is high, for all kinds of strenght; if moisture content is higher than 18M% strenght should be attenuated by 1/6.

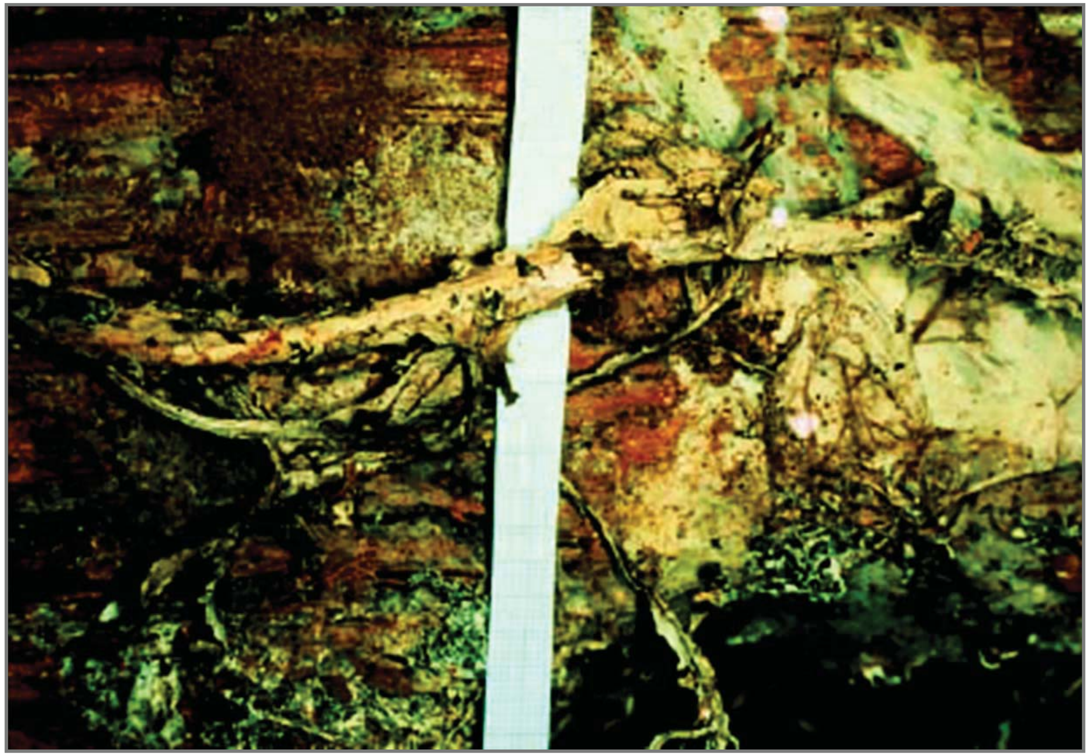
Wood damages

- Wood-destroying fungi (dry rot, brown cellar rot)

Fruiting body of dry rot

Wood damages

- Wood-destroying fungi (dry rot, brown cellar rot)



Mycelium of dry rot

Wood damages

- Wood-destroying fungi (dry rot, brown cellar rot)

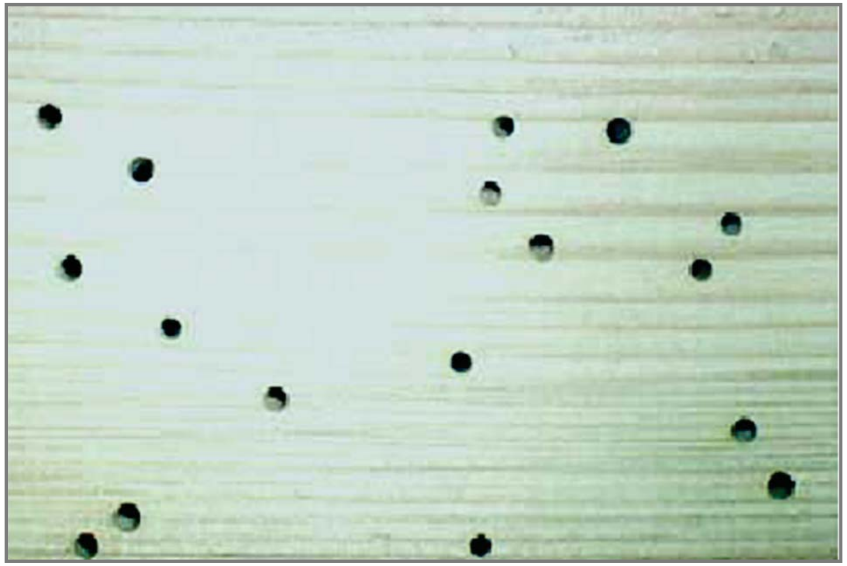
Blight – caused by dry rot or brown cellar rot

- Wood-destroying fungi (dry rot, brown cellar rot)

Fruiting body of brown cellar rot

Wood damages

- Fresh wood insects (bark beetle, shipyard beetle)



Ate holes made by shipyard beetle

Wood damages

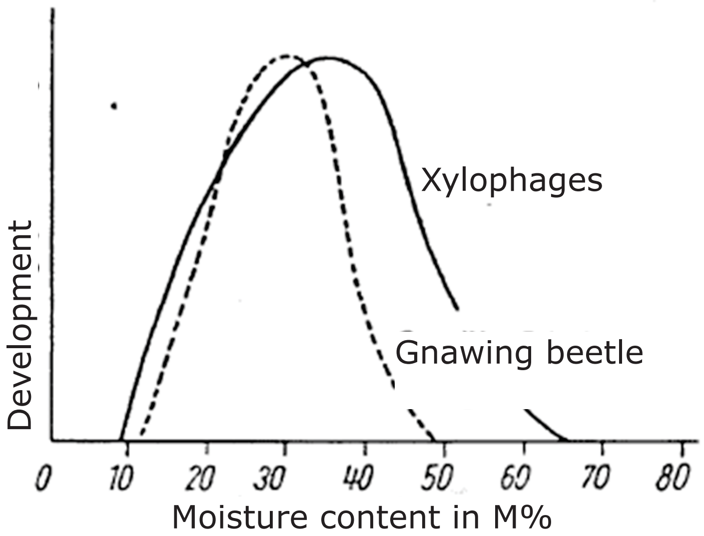
- Dry wood insects (xylophages, gnawing beetle or wood worm)

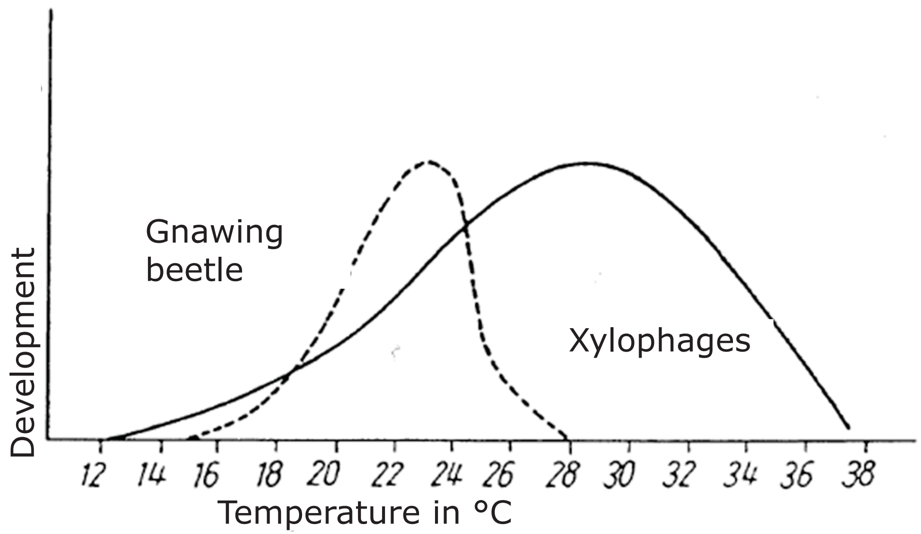
Common gnawing beetle
(larvae are known as wood worms)

Xylophagae

Wood damages

Dry wood insects (xylophages, gnawing beetle or wood worm)





Influence of temperature and moisture content of wood on development of larvae according Becker (from Kempe, K., Holzschädlinge)

For common gnawing beetle:

- Optimum of development around 30M%
- Minimum moisture content 10 – 12M% - below – no development of larvae

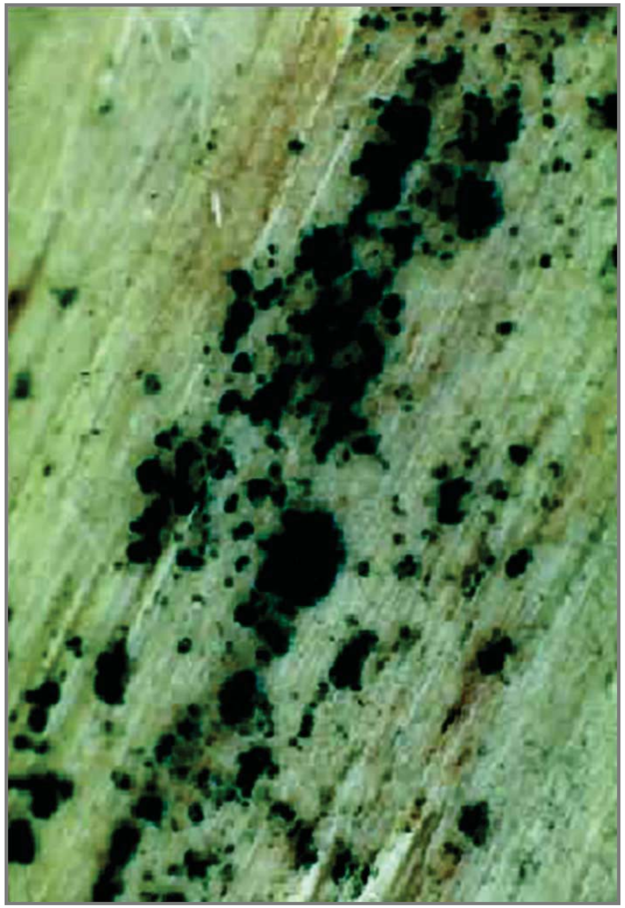
Wood damages

Schäden an Balkenköpfen durch den gewöhnlichen Nagekäfer

Aus: U. Müller, Holzbalkenköpfe in historischen Mauerwerk, Präsentation bei 2. internationaler HolzBauPhysik Kongress in Leipzig, Feb. 2011

Wood damages

- holzbewohnende aber nicht zerstörende Schimmel- oder Bläuepilze



Schimmelpilze auf Holz

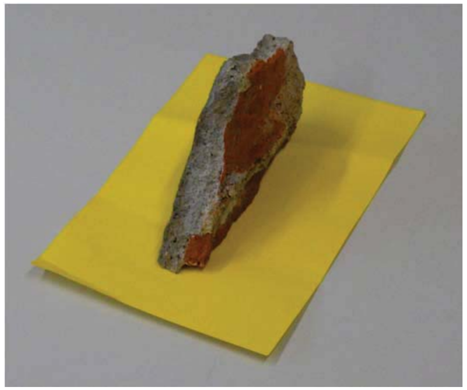
Moisture inside the construction

Mechanical loads

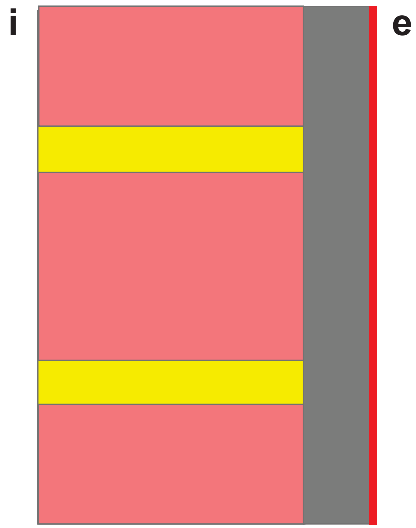
Hygrothermal damages induced by weathering load

Moisture and temperature differences at the building surface can lead to stress and cracks

Moisture and temperature differences between building surface and deeper layers can lead to shear stress → Spalling (flaking, chipping)

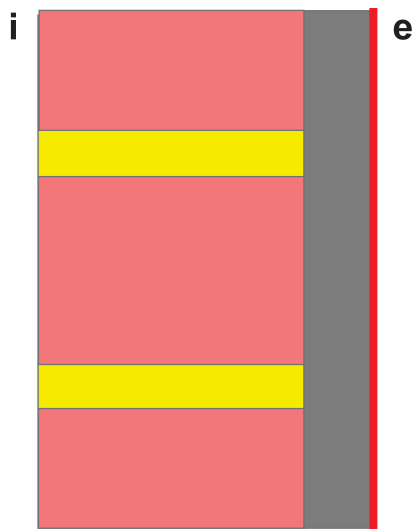


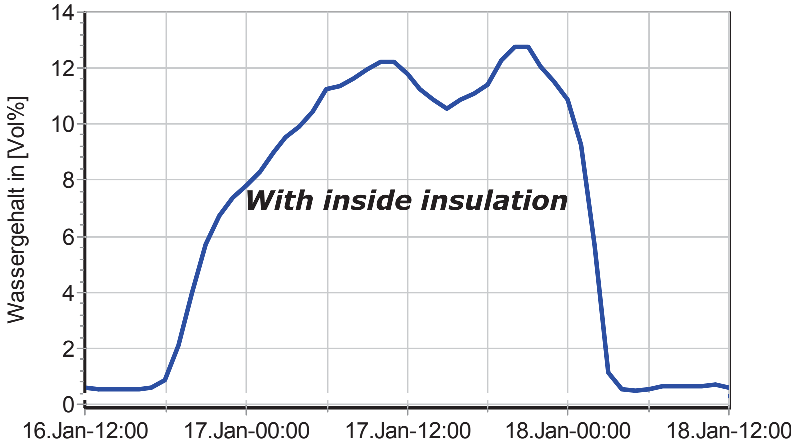
Hygrothermal damages induced by weathering load

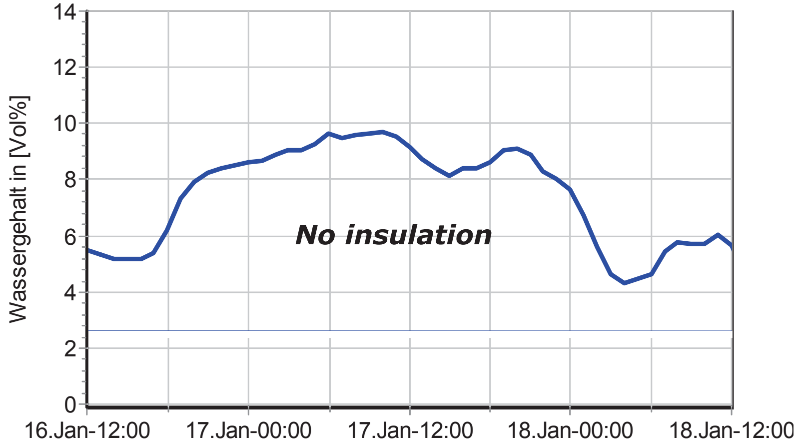


Moisture and temperature differences at the building surface → Cracks

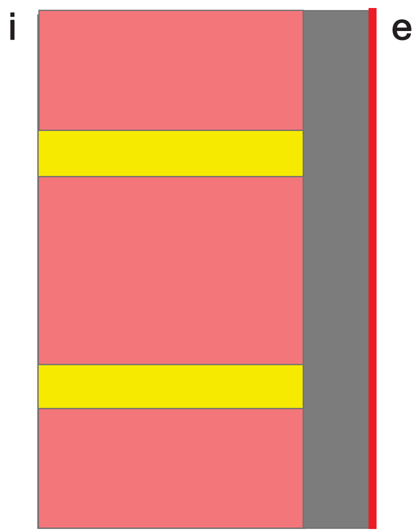
Hygrothermal damages induced by weathering load

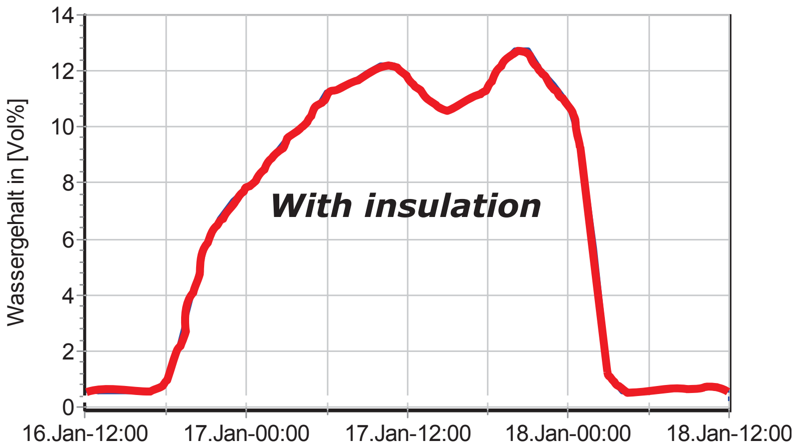


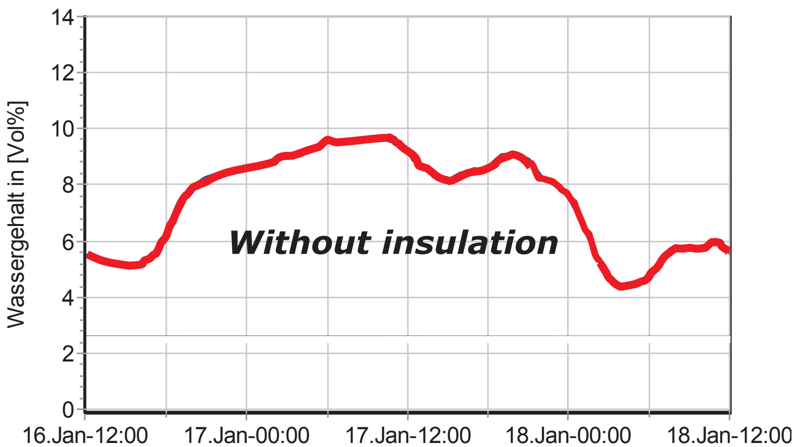




Hygrothermal damages induced by weathering load







here: $WL_{w,\text{with ID}} > WL_{w,\text{without ID}}$

WL_w: Weathering load related to moisture

3 Summary and conclusions

The Institute for Building Climatology at Dresden University of Technology (Faculty of Architecture) investigates the theoretical basis of combined heat, moisture, air, and salt transport in building materials, and also researches other areas of building science. An important goal of our research work is the dissimination of new knowledge to other research institutes, and practitioners. Therefore, we continuously integrate new findings in our user friendly software and calculation tools.

Our software programs shall help other research institutes in their work, assist students in learning fundamentals of builing physics, and support the work of civil engineers, architects, and others working in the field.

The longterm experience in the area of combined heat, moisture, and salt transport processes also benefits expertises and research reports. For instance, our software can be used during the planning phase to estimate the condensation risk of a construction under various environmental conditions, or to investigate the impact of thermal bridges. The software can be used to determine the causes of damage to constructions or materials, or to test new materials for potential application areas and limits and help optimize materials accordingly.

The laboratory measurement technologies and experiments deliver physical HAMT material property data. The experiments are able to provide reliable and secure/safe material information for numerical simulation tools. Classical methods and extended experimental procedures are applied to a large number of building materials.

The methodology is providing the significant increase of quality control in the chain: material porperty - material function - computer simulation - building constructions. The universal performance evaluation methodology can be used to assess the applicability of new materials and constructions, moisture damages and durability problems and restauration methods.

In total TUD have already organized and carried out a teaching and training courses and workshops on the hygrothermal simulation for project partners at the industrial partner location Remmers in Löningen (Germany) between 3rd to 5th September 2012.

The Institute for Building climatology in Dresden University of Technology has delivred Delphin license (the latest Delphin version) for each partner to do hygrothermal simulations for their case studies. TUD organizes also the relevant weather dates and its preparation as input file for the Delphin software. As well as TUD carries out service measurements of characteristic material properties of the existing building components and generation of hygrothermal material functions for numerical simulation tools. Weather data and measurement of characteristic material properties are important to have for the hydrothermal simulation of their case studies.

4 References

[1] Bishara, A.; Plagge, R. Comparison of different interior insulation materials, EWCHP (European Workshop and Training Day on Cultural Heritage) Oslo 2012

[2] Grunewald, J., R. Plagge, M. Bomberg 2001: Introduction to material characterization for input to HAM models, ICBEST 2001 Conference, Ottawa, Canada, and proceedings of the CIB W40 meeting in New Zealand, April 2001a.

[3] Grunewald, J., R. Plagge & P. Häupl 2002: A 2-levelled hygrothermal material database for the numerical simulation program Delphin4. Proceedings of the International Symposium on Building Physics in Nordic Countries, University of Trondheim, Norway, 39-46.

[4] Grunewald, J. & P. Häupl 2002: Ein Modell zur Beschreibung der feuchteabhängigen Dampfleitfähigkeit kapillarporöser Materialien", 11. Bauklimatischen Symposium, Band 1 (S.286-294), TU Dresden, FakultätArchitektur, Institut für Bauklimatik, Eigenverlag September 2002

[5] Grunewald, J. 1997: Diffusiver und konvektiver Stoff- und Energietransport in kapillarporösen Baustoffen. Dissertation, Dresden University of Technology, Faculty of Architecture, Institute of Building Climatology.

[6] Häupl, P., R. Plagge & H. Fechner 2001: Hygrische Materialfunktionen von porösen Baustoffen. Gesundheitsingenieur, Haustechnik - Bauphysik - Umwelttechnik, 122. Jahrgang, 305-316.

[7] Hoffmann, D., K. Niesel & R. Plagge 1996: Relationship between pore structure and other physico-technical characteristics of stone. Congress Proceedings of the 8th. International Congress on Deterioration and Conservation of Stone in Berlin 1996, Vol. 1, 461-472.

[8] Hoffmann, D., K. Niesel & R. Plagge 1995: Water retention and conductivity of porous media. Am. Ceram. Soc. Bull. Vol. 74, No. 11, 48-50.

[9] Plagge R., 1991. Bestimmung der ungesättigten hydraulischen Leitfähigkeit im Boden. Diss. Bodenökologie und Bodengenese, Heft 3, pp 151.

[10] Plagge, R., P. Häupl, J. Grunewald & H. Fechner 2002: Functional description of water storage and transfer in capillary porous building materials. Tagungsband zum 11. Bauklimatischen Symposium an der TU-Dresden, Band 1, ISBN 3-86005-233-0, 338-350.

[11] Plagge, R. 2004: Experimental technique to measure material parameters related to moisture flow. Proceedings of the 2004 Symposium: Study of moisture problems related to the conservation of historical stone monuments and buildings, National Research Institute for Cultural Properties, Tokyo, Japan, 19-32.

[12] Plagge, R., Scheffler, G. & Grunewald, J. 2005: Automatic measurement of water uptake coefficient of building materials. Proceedings of the 7th Symposium on Building Physics in the Nordic Countries, Reykjavik June 2005, 15-22.

[13] Plagge, R., Scheffler, G., Meissner, F., Häupl, P., Fitz, C., Lengfeld, K., Krus, M., Künzel, H. & Klaus, S. 2007: MASEA - Material Property Database of Old and New Building Materials for Software Tools in Building Construction. Proceedings of the 12th Symposium on Building Physics in Dresden, Dresden March 2007.

[14] Bishara, A. Building climatic simulation methods for the solution of design-, planning and renovation projects in hot dry climate and in hot humid climate, Dissertation, Dresden University of Technology 2011

[15] Krus, M. 1995. Feuchtetransport- und Speicherkoeffizienten poröser mineralischer Baustoffe. Theoretische Grundlagen und neue Meßtechniken. Dissertation, Fakultät Bauingenieur und Vermessungswesen Universität Stuttgart

[16] http://bauklimatik-dresden.de/index.php?aLa=en, Software for Building Since, Institute for Building Climatology,Dresden University of Technology

Standards

[17] DIN ISO 11272 Bodenbeschaffenheit - Bestimmung der Trockenrohdichte, DIN Deutsches Institut für Normung e.V., Berlin, Januar 2001

[18] DIN EN ISO 12571 Wärme- und feuchtetechnisches Verhalten von Baustoffen und Bauprodukten - Bestimmung der hygroskopischen Sorptionseigenschaften, DIN Deutsches Institut für Normung e.V., Berlin, April 2000

[19] DIN EN ISO 12572 Wärme- und feuchtetechnisches Verhalten von Baustoffen und Bauprodukten - Bestimmung der Wasserdampfdurchlässigkeit, DIN Deutsches Institut für Normung e.V., Berlin, September 2001

[20] DIN EN ISO 12664 Wärme- und feuchtetechnisches Verhalten von Baustoffen und Bauprodukten - Bestimmung des Wärmedurchlasswiderstandes nach dem Verfahren mit dem Plattengerät und demWärmestrommessplatten-Gerät - Trockene und feuchte Produkte mit mittlerem und niedrigem Wärmedurchlasswiderstand, DIN Deutsches Institut für Normung e.V., Berlin, Mai 2001

[21] DIN EN ISO 15148 Wärme- und feuchtetechnisches Verhalten von Baustoffen und Bauprodukten - Bestimmung desWasseraufnahmekoeffizienten bei teilweisem Eintauchen, DIN Deutsches Institut für Normung e.V., Berlin, März 2003