# DELPHIN 6.1 – Tutorial 1

Outside	e Wall with Interior Insulation
1	Part 1: Simulation of Previous Construction
1	Project Setup and Modeling of the Construction
2	Assigning materials and sizing layers
3	Interfaces – boundary conditions
4	Outputs
5	Discretization
6	Initial and Simulation Conditions
7	Run Simulation11
2	Part 2: Adding a Capillary Active Inside Insulation
3	Appendix A – climate and boundary conditions
1	Cretae Interfaces – boundary conditions
2	Creating Climate Conditions
3	Creating Boundary Conditions
4	Creating and Assigning Interface Definitions
4	Appendix B – Outputs
1	Output Grids and Frequency
2	Output Files

## **Outside Wall with Interior Insulation**

In this tutorial a typical old masonry will be used analyzed that is modeled and simulated before and after mounting an inside insulation.



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

## *1 Part 1: Simulation of Previous Construction.*

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

## 1 Project Setup and Modeling of the Construction

After starting DELPHIN one can see different buttons for the project setup and control in the left top area of the main window. The first step is generating a new project. The *Create project*-button (or File  $\rightarrow$  New...) opens an assistant for creating a new project:

	200 Herrin Oject Adamstant	
Import Delphin 5 project	Introduction This wizard guides you through	h the basic step
DEI DHIN Varden 6.0.16	In setting up a DELPHIN simula	ition project.
News Version 6.0.16 was released on 01/31 2018. The n report.	ш	
	$\Box$	
	DELPHIN Version 6.0.16 News Version 6.0.16 was released on 01/31 2018. The n report.	

This assistant guides you through the first steps. After confirmation with click on *Next* a dialog opens where one can add some project information (see next picture). Here the current project can be described. This description will be shown in the project selection list next time you start DELPHIN.



The next dialog allows you to select a location. One can select a location from a list with hourly weather data from the DELPHIN climate database. For this tutorial project the location TRY 2010 Potsdam, Germany should be selected.



After a click on *Next* the geometry setup will be shown.



Here one can select the basic geometry type and the main construction grid. We want to calculate a one-dimensional wall structure therefore 1D Vertical (Wall) should be selected. The type of construction can be changed at any point in time later. The existing historic construction has 3 layer (columns). The thicknesses of these layers can be edited later. In the next dialog the boundary conditions can be set.

C New Project Assistant		×
Setup predefined interfaces Select the boundary conditions, interfa	ces. For 1D constructions, the interfaces wi	ill be already assigned to the sides of the construction.
NTREST	in of profektional interfaces table in ended CM (2013) From 2014 direct control interfaces and control interfaces from control interfaces and c	d dpla 3% d val 200 v 90 v
		<back next=""> Cancel</back>

Checking the topmost selection skips this process. The boundary conditions must then be defined manually. In the second selection box you can define where the outside should be. If this box is checked, outside is on the left side, otherwise it is on the right. In this tutorial the left should be inside, so the hook must be removed. Afterwards the indoor climate and the outdoor climate are set. Here the following is selected:

- For the indoor climate, a climate calculated according to WTA leaflet 6.2 with a normal humidity load +5% is selected. This climate is also recommended according to DIN 4108-3 as a design climate for residential spaces.

- For the outdoor climate, the data of the previously set climate location is used. The inclination is 90°, i.e. a vertical wall. This wall is oriented to the west (270°). In Germany this orientation is usually associated with the highest rain loads.

By clicking on Next, the selected surfaces are generated and assigned to the construction and a new dialog will be shown.

The next dialog allows to select a small amount of predefined outputs and the basic output schedule.

on New Project	t Assistant	
Setup prede Select the geometry	fined outputs outputs you want to create and the associated output grids. The profile outputs will be already assigned to spu	an the initial
DELPHIN	Skip generation of predefined outputs           Predefined outputs (assigned to entire construction)           PT Temporature profile           PT Relative humidity profile           PT Relative context profile           PT Relative humidity profile           PT Integral moisture mass           Predefined outputs for 1D constructions (assigned to boundaries)           PT Surface temperature           PT Surface temperature           PT Surface temperature that the time is th	
Q	Output grids Define output frequency for Fields/profiles; 1.5 Scillar values: 1 1	ld •
	< Back Finish	Cancel

With the chosen settings above all field and profile outputs will be created each 1.5 days. For all other outputs an hourly schedule will be used.

This is the last dialog of the new project assistant. After clicking on *Finish* the main window will be shown and one can proceed with the selection of the materials.



The main window shows the basic construction and a message box which let you start the material database selection. The displayed question can be skipped for further projects.

D	Name	Category	Producer	Rho [kq/m3]	cp [J/kgK]	Lambda [W/mK]	р[]	W80 [kg/m3]	WSat [kg/m3]	Aw [kg/m2s0
551	Lime Sand Stone Xella Ytong 2004	Building brick	Xella Internatio	1743.8	868.3	0.854/	27.9	37.6	333.1	0.0
320	Lime sand stone	Building brick		1900.0	1000.0	1.000/	28.0	25.0	290.0	0.0
321	Lime sand stone - replaced by material with I	Building brick	Xella Internatio	1740.0	870.0	0.850/	4.8	39.0	360.0	0.
685	Lime-Sandbrick	Building brick		1704.6	890.9	1.188/	18.7	10.5	238.5	0.
791	Loam brick green	Building brick		1787.1	760.2	0.826/	36.0	58.0	412.6	0.
349	Loam stone	Building brick		1040.0	1000.0	0.400/	7.1	41.0	460.0	0.
512	Normal Brick	Building brick	Wienerberger AG	1786.3	888.7	0.548/	18.0	13.4	319.4	0.
506	Normal Brick - replaced by material with ID 507	Building brick	Wienerberger AG	1786.3	1000.0	0.801/	18.8	13.4	319,4	0.
507	Normal Brick - replaced by material with ID 512	Building brick	Wienerberger AG	1786.3	1000.0	0.801/	18.8	13.4	319.4	0.
264	Normal Brick - replaced by material with ID 512	Building brick	Wienerberger AG	1400.0	1000.0	0.550/	19.0	11.4	319.0	0.
540	Old Building Brick Rote Kaserne Potsdam (ou	Building brick		1842.5	772.2	0.797/	37.6	2.0	284.0	0.
529	Old Building Brick Am Weinberg Berlin	Building brick		1966.5	878.9	0.928/	26.5	1.0	240.1	0.
530	Old Building Brick Am Weinberg Berlin inside	Building brick		1674.2	933.0	0.685/	12.8	4.9	357.0	0.
838	Old Building Brick Bologna 3enCult	Building brick		1731.2	1092.2	0.624/	24.6	8.5	318.4	0.
532	Old Building Brick Bologna 3enCult - replace	Building brick		1759.0	1092.2	0.624/	24.5	7.5	318.4	0.
531	Old Building Brick Bornstätter Feld	Building brick		1853.9	788.0	0.722/	42.5	4.4	192.9	0.
490	Old Building Brick Dresden ZA	Building brick		1835.6	814.6	0.528/	15.7	5.4	282.9	0.
533	Old Building Brick Dresden ZB	Building brick		1769.2	1000.0	0.789/	8.6	9.0	334.0	0.
491	Old Building Brick Dresden ZC	Building brick		1904.4	847.4	0.808/	22.9	3.1	212.4	0.
492	Old Building Brick Dresden ZD	Building brick		1619.5	953.1	0.403/	10.5	3.7	361.0	0.
493	Old Building Brick Dresden ZE	Building brick		1657.2	907.0	0.574/	12.8	4.9	357.0	0.
494	Old Building Brick Dresden ZF	Building brick		1975.7	846.0	1.012/	41.1	7.4	169.4	0.
495	Old Building Brick Dresden ZG	Building brick		1715.2	920.2	0.543/	22.2	6.9	322.1	0.
496	Old Building Brick Dresden 7H	Building brick		1851.9	794.1	0.659/	12.0	2.3	282.5	0.

For the original construction we need three materials:

- Masonry: Normal Brick with ID 512
- Inside plaster: Lime plaster (historical) with ID 148
- Outside plaster: Lime cement plaster with ID 145

The material database view shows material names, producers (if known) and some basic parameters. Also important is the material ID (see first column). This is a unique ID that facilitates the identification of a material. Furthermore there is a filter for the material category and a name filter. As example for selecting the brick masonry one can use the category 'Building brick'. A click on *Import* adds the material to the project. Proceed in the same way with the two plasters. The picture below shows the main window after this process. One can see in the right top edge the three materials gray and italic. This means that the material are imported into the project but are not assigned to the construction yet. In the next steps the materials should be assigned and the thickness of the layers must be changed.

e Edit View Window Tools Help	
)	Materials #
	Lime Plaster (historical) [148]
	Ume Coment Plaster [145]



Once you have selected a material, either **double-click** the material, or click on the "**Import**" button (3). Then the import dialog opens allowing you to select some properties of the material reference.

Note, that <u>importing a material</u> actually means <u>creating a reference to a material file</u>, which contains all parameters of a material used in the project. The dialog gives some options on how to reference materials:

Material refe	rence			x
Name:	Old Building Brick Rote Kaserne Potsdam (outer brick 1) [540] Edit	Base parameter	Value	Unit
		ρ	1842.5	kg/m <sup>3</sup>
Color:	2 1 Open in text editor	ср	772	J/kgK
		λ	0.797	W/mK
Material data file:	DB_materials/AltbauziegelRoteKasernePotsdamAussenziegel1_540.m6	μ	37.56	•
Reference:	\${Material Database}/AltbauziegeRoteKasernePotsdamAussenziegel1 540.m6 Duplicate material file	Wsat	284.0	kg/m³
		W80	2.0	kg/m³
	Reference with file path relative to project file	Aw	0.07	kg/m <sup>2</sup> s <sup>1</sup> / <sub>2</sub>
	Reference with file path relative to material database directory	Kleff	2e-08	s
	<ul> <li>Reference with file path relative to user material database</li> </ul>			
	O Reference with abcolute file nath			
3				
				OK Cancel

The name (1) is just an identification for this material and can be changed to any descriptive text. In addition, the color (2) can be adjusted so that materials are easily distinguished. Lastly, you can choose how the file path should be stored within the project (3). This is important when project files are copied around or moved across the network, because of which relative paths should be preferred normally. On the right side of dialogue (4) a selection of material parameters is shown. In this tutorial, you can simply keep the defaults in this dialog.

Once done with all three materials, you can close the material database view. You will now see the central modeling window:



The layout of the windows can be adjusted at will. The various definition lists can be shown/hidden via the Windowmenu on the upper left. Also, they can be dragged around and put together in tabs. For this and the following tutorials, we will use the standard layout.

## 2 Assigning materials and sizing layers

For changing the thickness of the left layer click on the left column and change the size in the edit field left of 'Dimensions' to 15 mm.



After changing the thicknesses of the other columns assign the materials to the construction.

Materials 6 X
Named Bask (512)
Line Plaster (long) [130]
3

To do this, you need to select a layer (or several layers) with the mouse (1). Then select the desired material in the material list (2) and finally press the green assignment button (3). The order of the actions can be reversed as well.

After assigning all materials the construction view should show the layers of the construction with the corresponding material colors. Furthermore the material names in the list will be shown bold. 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.

In general all assignments can be checked in detail in assignment lists. The material assignment list is opened by pulling the lower margin of the material view to the top.

Next you have to create the interfaces (boundary conditions) for inner (left) and outer (right) side.

#### 3 Interfaces – boundary conditions

The boundary conditions are already created and assigned by the new project assistant.



On the bottom right in DELPHIN (1), the list of existing surfaces is shown. A surface is a summary of different boundary conditions and the corresponding climate data. Here, there are surfaces for inside and outside. The coloured markings show which boundary conditions are used:

- Red heat transfer
- Blue Vapour diffusion
- Yellow short wave solar radiation
- Brown long wave radiation balance
- Green driving rain
- Dark blue water contact
- Light blue Air flow

If a surface is marked in the list (click on it) then the assignment to the construction is shown as a thick dotted line in the view (2). A double click on a surface opens the settings dialogue.

Specification							
Name: Outside							
Type: Standard Interface for outdoor climate [EngineeringOutdoor] v							
Surface Properties							
Orientation [0360 Deg]: 270				~			
Inclination [0180 Deg]: 90							
Outside Conditions							
User-defined outdoor climate [OutdoorUse	'Data]						
Heat conduction	h_c - Convective heat conduction exchange coefficient [W/m2K]:	12	v	]			
	h_r - Radiant heat conduction exchange coefficient [W/m2K]:	5					
	Effective heat conduction exchange coefficient [W/m2K]:	12		ſ			
Vapor diffusion	Vapor diffusion mass transfer coefficient [s/m]:	7.5e-08	Y	Compute with Lewis relation			
	sd-value of painting / surface coating [m]:	0	×				
Short-wave solar radiation	Solar adsorption coefficient [-]:	0.7	~	J			
<ul> <li>Long-wave radiation exchange</li> </ul>	Long-wave emissivity [-]:	0.9	Y				
Wind driven rain (DIN EN ISO 15927-3)	Reduction/splash coefficient [-]:	0.7	v				
Convert to detailed medal							

For more information regarding boundary conditions and climate see appendix A.

#### 4 Outputs

Now outputs can be set. For this tutorial the following outputs will be used:

- Temperature profile temperatures in the whole construction over time
- Relative humidity profile
   relative humidities in the whole construction over time
- Moisture content profile volumetric moisture content in the whole construction over time
- Moisture content integral moisture mass in the total construction
- Surface temperature left side temperature on the inner surface
- Surface relative humidity left side relative humidity on the inner surface

All outputs are defined from the 'create project' assistant but only the first four are already assigned to the (whole) construction. This is shown by the 'Output files' dialog. All assigned outputs are written with bold characters not assigned constructions italic.

Temperature profile	
Relative humidity profile	
Moisture content profile	
Moisture content integral	
Surface temperature - left side	
Surface temperature - right side	
Surface relative humidity - left side	
Surface relative humidity - right side	

Now the two surface related outputs shall be assigned. Click on the most left column with the surface (1), click on the output file in the list (2) and then on the assignment button (3), or in reversed order.

	1	2	Output Files Temperature profile Relative humidby profile Moisture content profile Moisture content integral Surface homperature - Jeff side Surface homperature - Jeff side Surface relative humidby - Left side Surface relative humidby - Left side	0 ×
Olmensions: 15 1000 mm	Selection: 0,0 0,0 Elements (sel/used/grid): 1/3/3	Coordinate Range: 000) (0.015;1.000)		

After clicking on the assignment button a dialog will occur. Here one can decide if the output should be related to a volume element or to a position. For a surface output a position related assignment should be used. The coordinate

system for the geometry runs from left to right. That means the left most side has the x coordinate 0. Please fill out the dialog as the picture below shows.



After clicking on *Ok* the assignment will be performed and a small red dot will mark the position in the construction view. As next step assign the surface relative humidity in the same way. The 'Output files' dialog will show all assigned outputs bold. One can add, remove or change the outputs by using the buttons on the top of the dialog. The other tab shows the 'Output grids' that means the time schedules. If this (or another) tab is not shown check it on the upper left in the >> Main menu >> Window item



#### 5 Discretization

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).

C Dephin 5.11 - unsamed mp*	Automatic grid generation	<u>?</u> ×
	- Auto-Discretization Options P X-Direction V-Direction I Z-Direction (only for 3D grids)	Grid statistics           Grid elements (total/used):         47/47           Smallest grid dimension in (m) (v/y):         0.001/1           Largest grid dimension in (m) (v/y):         0.0447499/1           Grid Preview         0.0447499/1
	Variable Grid Options Minimum element size:  Stretch factor:  Stretch factor:  1.3	
		OK Cancel Apply

In the Automatic Discretization dialog we will only discretize the construction in X-direction - as we have to deal with a wall with horizontal transport direction only. 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 (1 mm and 50 mm are good default values), and the stretch factor. 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 to 1.4 is suitable for 1D-constructions. For 2D-constructions a higher stretch factor of approx. 1.5 is advisable to avoid long simulation durations.

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 thin or boundary elements.



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

## 6 Initial and Simulation Conditions

After specifying the boundary conditions it is time to select the simulation and modeling options. Open the modeling dialog by click on the selected button in the tool bar (or >> Main menu >> View, or F7):



This dialog contains three different tabs to change options. The first tab allows to select the basic properties of the physical model. For this example you need to enable the balance equations for heat and moisture transport only:

Model Options   Solver Options   Performance Options	
The settings on this page control basic properties of the physical mo	del.
P Energy Balance Equation	Additional Modeling Options
Default initial temperature: 20 C	Use anisotropic material transport model
C Use thermal conductivity of dry material (LAMBDA)	Prevent overfilling
C Use design value of thermal conductivity (LAMBDA_DESIGN)	Output options
Use moisture-dependent thermal conductivity	Output time unit: h
Default initial relative humidity: 50 %	Over-hygroscopic moisture limit (condensate): 95
P Moisture Balance Equation	Write binary output files
Default initial relative humidity: 50 %	Number predsion in ASCII files: -1 🚔
Suse Kirchhoff potential for liquid flux calculation	Simulation Time Frame
Use gravity	Start date 01/01/2007 00:00:00
Use equilibrium ice model	End date 01/01/2008 00:00:00
E the Marco Delacara Devention	Duration 1 a
Air Mass balance Equation	1
1 Ose gravity	
Salt Balance Equations	
Salt Simulation Options	
Pollutant Balance Equations	

Other important settings in this dialog are the duration and the starting time of simulations. Here, we use only one year as duration of the simulation. The starting time is not important for a design simulation. However, the initial conditions need to be specified. They can be set globally in the "Defaults" section of the balance equation

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

Model Options   Solver Options   Perform	nance Options
The settings on this page control basic propert	lies of the physical m
F Energy Balance Equation	
Default initial temperature:	20 0
Dse thermal conductivity of dry material	(LAMBDA)
Use design value of thermal conductivity	(LAMBDA_DESIGN)
Use moisture-dependent thermal conduct	tivity
Deput initial relative humidity:	50 %
Moistule Balance Equation	
Default initial relative humidity:	50 %
Use Kirchhoff potential for liquid flux cale	sulation
Use gravity	
Use equilibrium ice model	
Air Mass Balance Equation	
E the weath	

## 7 Run Simulation

In the same dialog as before one can also start the simulation.



In this dialog you can choose between a solver with graphical output during the calculation process and an external solver. The latter is faster and particularly for 2D simulations the recommended choice. In this example, and generally for 1D simulations, the internal solver is advisable. It is also possible to set the *Number of parallel threads*. Especially for 2D simulations this can speed up the simulation many times over. The simulation is started with "Start simulation" and the simulation window opens:

The simulation window shows the current temperature, relative humidity and moisture profiles as well as the total moisture content over time which can be used to quickly check the results. Also, the current solver performance statistics can be shown (left vertical tab 'Solver statistics').



Once the simulation is complete

miast files	F	storial	1.1.450												
oject me:		utonal	1_1.00p	Palakia i	Duluble C	malat	laf kank	male	hindida	-				I dfa	In all A
immand line arg	uments: p.	ove C:/	Programmi	ng/Delphiny	Delphino	rbeipr	uno_uunk	Det	ninoyoc	qman	03/1	projects/1	utonal1	_1.dopverbosity-	sever=1 -b=
364.792 d	12/31/	00 1	9:00:00	8.582	s. 4	12.50	05 d/s	4	2.403	d/s	-	0.005	3		
364.833 d	12/31/	00 2	0:00:00	8.583	s 4	12.50	)5 d/s	- 43	2.407	d/s		0.004	3		
364.875 d	12/31/	00 2	1:00:00	8.584	8 4	2.50	04 d/s	4	2.407	d/a		0.003	3		
364.917 d	12/31/	00 2	2:00:00	8.586	5 4	2.50	)3 d/s	-4.	2.406	d/s		0.002	5		
364.958 d	12/31/	00 2	3:00:00	8.586	5 4	2.50	4 d/s	4	2.412	d/s		0.001	.5		
1 000 a	01/01/	e	0.00.00	o for	e 8	tean:	speed		arrent	cape:	sa i	o ooo			
1.000 a	01/01/	01	0.00.00	0.201	e	2.35	/3 U/S		6.412	an a		0.000	5		
Solver stat	tistics														
Wall clock	time					-	8.586	8					-		
Framework:	Output	writ	ing				1.055		(1)	2.29					
Integrator	: Steps					-			0.00		0.55	12	9794		
Integrator	Newton	ite	rations			-						18	6784		
Integrator:	Newton	con	vergence	e failur	es	-							6		
Integrator	: Error	test	failure	12		-						1	0158		
Integrator	: Functi	on e	valuatio	on (Newt	on)	-	2.600	5	(3)	0.29	6)	18	6785		
Integrator	LES se	tup				-	0.337	-		3.92	÷)	2	4499		
Integrator	LES SC	TAG				-	0.304	3	0	\$.54	1)	18	0784		
LES: Jacob:	ian mati	IX C	valuatio	ons		-	0 117	12	12.1		5.1		4400		
LES: Funct	on eval	nati	on Litar	shian ne		2	0 180	-	12	2 10	21		9481		
mont thurst	eon eval	time to a	ou toact	waan ye		_	0.100	- C			-1	*	a a a a		

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



There are currently two different post processors: the original one from DELPHIN 5 and the newly developed version (PostProc2). The new postprocessing contains the most important features of the old version, so it can be used in most cases. It also has new features that facilitate the evaluation and especially the comparison of data.

PostProc 2 is used for this tutorial. PostProc 2 has to be installed before because it is not included in the DELPHIN 6 setup. You can find the installer for it on the download page (http://www.bauklimatik-dresden.de/downloads.php) under PostProc. The default settings of the post processor can be changed in >> Edit >> Settings >> External programs.

And then this on tools thep			 
Undo Simulation time frame changed Redo Open project in text editor Open project Run simulation	Ctrl+Z Ctrl+Shift+Z F2 Ctrl+Shift+C F9	副軍王王	2
Preferences			
		9. – E.	

Choose the tab 'External tools' and select which kind of post-processing shall be used.

Executable:		
C:/Program Files/IBK/PostProc 2.0/Post	tProcApp.exe	
Select DELPHIN 5 PostProc C:/Progr	am Files/IBK/Delphin 6.0\D5PostProc\Postproc.exe	
Select PostProc 2 c:\Progra	im Files\JBK\PostProc 2.0\PostProcApp.exe	
fext editor		
Executable:		
C:/Program Files (x86)/PSPad editor/P	SPad.exe	

Click on the corresponding button of the post-processor version to select it. After opening the PostProc 2, the data of the current project should normally already be loaded. If not, these data must be loaded from the project directory.

For further information regarding the postprocessor: see tutorial 3 (for PostProc 2, PDF, english) on the website http://bauklimatik-dresden.de/delphin/documentation.php. There is also an online help which can be accessed via the following link: <u>http://www.bauklimatik-dresden.de/postproc/help/en/index.html</u>

## 2 Part 2: Adding a Capillary Active Inside Insulation

Since the thermal resistance of the existing wall does not meet modern requirements of buildings, an additional insulation is added. In this tutorial we assume that adding an insulation on the outside is not possible (perhaps due to a historical facade, etc.), therefore a capillary active insulation system with calcium silicate will be used. The CaSi insulation is fixed with a glue mortar.

In order not to delete the project with the historic, uninsulated wall, save the project with different file name first. That way the previous results remain for comparison.

At first, the materials "Calsitherm KP Glue Mortar" (ID 705) and "Calsitherm Climate Board F" (ID 706) 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, left ellipse) with the according thickness (5mm glue mortar and 80mm insulation). To do this, select the innermost element and then click on the button for Add layer. Then the following dialogue opens:



In the input line above you can specify the thickness of the new layer. Below it there are three graphic buttons that show how to use this thickness:

- Left thickness corresponds to the whole new layer
- Centre thickness corresponds to the construction width plus the new layer
- Right thickness corresponds to the marked column plus the new layer

If you just want to add a new layer of the appropriate thickness, click on the left button.

Then you can assign the materials to the respective layers (see construction sketch at the very beginning 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, right ellipse).



Whenever the construction has changed it is advisable to check that all boundary conditions and outputs are still assigned to the correct layers. Interfaces of 1D constructions are automatically set to the right boundary. In this case the original surface outputs keeps in the old inside plaster (see small red dot in picture above). In order to set the outputs again at the inner surface one have to change the assignment. Assignment lists normally are hidden. In order to show all lists click in the >> Main menu >> Window: Toggle visibility of all assignment windows.



The red arrow marks the position of the assignment parameter for the (former) surface outputs. As you can see the x-coordinate now is set 0.085 m. This has to be changed to 0.0 m: double-clicking on the assignment allows you to edit values.

remperature pr	ofile		
Relative humidit	ty profile		
Moisture conten	t profile		
Moisture conten	t integral		
Surface tempera	sture - left s	ide	
Surface tempera	ature - right	t side	
Surface relative	humidity -	left side	
Confirm autott	According .	and a state	
Surrace resouve	turning a	right side	
Surrace relative	mannuny -	right side	
	numuny -	right side	
• • • • 0	Element	Temperature profile	_
0 0 70 0 0 0 70 0	Element	Temperature profile Relative humidity profile	
0 0	Element Element	Temperature profile Relative humidity profile Moisture content profile	_
O 70 0	Element Element Element	Temperature profile Relative humidity profile Moisture content profile Moisture content integral	_
O 70 0     O 70 0	Element Element Element Element Coordin	Temperature profile Relative humidity profile Moisture content profile Moisture content integral Surface temperature - left side	

Now the simulation can be repeated.

 $\dots$  End of 1<sup>st</sup> Tutorial  $\dots$ 

## *3* Appendix A – climate and boundary conditions

## 1 Cretae Interfaces – boundary conditions

First one has to create the interfaces for inside and outside. Click on the green plus button in the 'Surface/Boundaries' dialog. For this tutorial standard interfaces will be used.

Contract Theorem	Interface/Boundary condition		? ×
	Spedification		
	Name: Outside		
	Type: Standard interface for outdoor dima	te [EngineeringOutdoor]	
	Surface Properties		
1	Orientation [0360 Deg]: 270		
	Indination [0180 Deg]: 90		
	Outside Conditions		
	User-defined outdoor climate (OutdoorUser	Dəta]	2
	P" Heat conduction	h_c - Convective heat conduction exchange coefficient [W/m2K]:	12 -
		h_r - Radiant heat conduction exchange coefficient [W/m2K]:	5
		Effective heat conduction exchange coefficient [W/m2K]:	12
	P <sup>-</sup> Vapor diffusion	Vapor diffusion mass transfer coefficient [s/m]:	7.5e-08 Compute with Lewis relation
		sd-value of painting / surface coating [m]:	0 -
	Short-wave solar radiation	Solar adsorption coefficient [-]:	0.7
	F Long-wave radiation exchange	Long-wave emissivity [-]:	0.9
	Wind driven rain (DIN EN ISO 15927-3)	Reduction/splash coefficient [-]:	0.7 .

The right picture shows the dialog with the settings for an outside interface. The orientation is set to 270 Deg (West) and the inclination to 90 Deg (vertical wall). The selected type 'Standard interface for outdoor climate' will use the climate data from the selected location (see create project assistant). The outside conditions should be set to 'User-defined'. Here one can set which boundary condition is used and other basic settings. Next an interface for the inside boundary will be created. Use the settings shown in the next picture.

Interface/Boundary condition	?
Specification	
Name: Inside	
Type: Standard interface for indoor climate [EngineeringIndoor]	2
Indoor Conditions	
User-defined indoor climate (IndoorUserData)	
Temperature [C]:	20
Relative humidity [%]:	50
Surface heat transfer coefficient (convective+radiative) [W/m2K]:	8
Code and the second state to be the	50.06

After creating the interfaces they need to be assigned to the boundaries. This works similar to the material assignment.

Delphin 6.0.16 - Tutorial1_1.d6p*		_ & ×
File Edit View Window Tools Help	International Action of the second	
	H II II 🛍 🕷 🖪 🔁	Materials **
		Normal Brick [507]
1		Lime Plaster (historical) [148]
60		
1		3
		Circlesis Brinshills
1 M		
8		2 Untilde Inside
		2

The inside climate will be located to the left. Select a column (1), select the interface to assign (2) and last click on the button for assignment to a left side (3). Proceed with the right side for the outside interface. One can check if the interface is assigned correctly by clicking on an interface in the 'Surface/Boundaries' list. The interface will be marked in the construction view with a dashed line (see picture below).



#### 2 Creating Climate Conditions

As mentioned before, the indoor/outdoor conditions or sides of the construction are not fixed. Instead, they are specified by a selection climate data and boundary conditions. This chapter shows how to create definitions manually. This allows more options.

For hygrothermal analysis, we want to use winter climate. That means indoors a temperature of **20** °C, and relative humidity of **50** %, and outside **-5** °C and **80** % R.H. throughout a period of **90 days**.

The climate is represented by a total of four climate conditions, which we will create first. To create a climate condition, go to the **Climate Conditions** list window. As with all definition windows, pressing the plus button (1) creates a new definition.



Doing this for Climate Conditions open the dialog for climate data. There you can specify the climate data, basically a single climatic data component: identification name (1), **Temperature** as type of quantity (2), variability in (3) where we use **Constant** and finally the value in (4).

Climate condition	
Specification	
Name: Indoor temperature	
Type: Temperature [Temperature]	Kind: [Constant value [Constant]
Constant parameters	
Constant value:	24
``	
Note Z	3 4
instead of creating constant climate data definitions, you can alway	is specify constant dimate parameters directly in the climate condition selection field
research of the and the card of the card of the card and a	a sherra consider compare haraneses a mercal an and compare constrator belocitor under

The following table lists names, types and values for the four conditions to be created:

Descriptive name	Condition type	Value (without unit)
Indoor temperature	Temperature	20
Outdoor temperature	Temperature	-5
Indoor relative humidity	Relative Humidity	50
Delphin 6.1	Page 18	http://www.bauklimatik-dresden.de

Outdoor relative humidity Relative Humidity 80
--

Once all of these are created, the list of conditions should look as follows:

Climate Conditions	8	×
🕂 🥒 问 🗕		
Indoor temperature		
Outdoor temperature		
Indoor relative humidity		
Outdoor relative humidity		

Note, all definitions are shown in gray italic, which means they are not being used in the model. In the next step, we'll change that.

## 3 Creating Boundary Conditions

The climate data is needed and referenced in boundary condition definitions. Boundary conditions effectively describe how the construction surface interacts with the climate, resulting in heat and moisture fluxes across the construction surface. Different physical effects such as heat conduction, vapor diffusion, air infiltration, water contact etc. are all described in different boundary conditions.

Similarly as for climate conditions, select the Boundary Conditions window and create in total four boundary conditions. Start with the first boundary condition by clicking on the plus-button in the **Boundary Conditions** list window.

Boundary	condition		
Specificatio	n	_1	
Name:	Indoor heat condu	ton	
Type:	Heat conduction (	eatConduction] 📃 💌 Kind: Exchange	e coefficient (Exchange)
Schedule:	<no alw<="" schedule="" td=""><td>ys enabled&gt;</td><td>Create new</td></no>	ys enabled>	Create new
Climate dat	ta	2	3′
Temperatu	re [Temperature]	Indoor temperature	• Edt
Imposed h	eat flux (HeatFlux)	<select create="" new="" or=""></select>	- Create new.
Wind veloc	ity [WindVelocity]	<select create="" new="" or=""></select>	Create new
Parameter		4	
Exchange	coefficient for still a	[W/m2K]:	-
Slope coef	ficient for moving ai	[]:	
Exponent f	for moving air []:		5
			OK Cancel

Within the dialog enter a descriptive name (1), select a boundary condition type (2), select a calculation model (3), specify requires climate conditions (4) and finally set the model parameters. For heat conduction, the relevant parameter is the heat transfer coefficient. In this tutorial we do not model long wave radiation exchange separately, so the heat transfer coefficient is an effective heat transfer coefficient including convection and mean long wave radiation exchange (8 =  $3 + 5 \text{ W/m}^2\text{K}$ ). Also, the heat conduction boundary condition needs the temperature adjacent to the wall surface, which is in our case the "Indoor temperature", created in the last step.

The following table lists the names, types, parameters and climate condition dependencies of all four boundary conditions to be created:

Descriptive name	Туре	Kind (model)	Parameter	Dependent Climate Conditions
Indoor heat condition	Heat Conduction	Exchange Coefficient	8 W/m <sup>2</sup> K	Indoor temperature
Outdoor heat condition	Heat Conduction	Exchange Coefficient	17 W/m <sup>2</sup> K	Outdoor temperature
Indoor vapor diffusion	Vapor Diffusion	Exchange Coefficient	25e-9 s/m	Indoor temperature Indoor relative humidity

Outdoor vapor diffusion	Vapor Diffusion	Exchange Coefficient	75e-9 s/m	Outdoor temperature
				Outdoor relative numidity

Note how the descriptive name of each climate condition is used to reference it within other definitions. The Boundary Condition list window should now show all four boundary conditions as unused definitions. The climate conditions, however, are now shown as used (black).

Boundary Conditions 🗗	× Climate Conditions & X
🕂 🥒 问 —	🕂 🥒 问 —
Indoor heat conduction	Indoor temperature
Outdoor heat conduction	Outdoor temperature
Indoor vapor diffusion	Indoor relative humidity
Outdoor vapor diffusion	Outdoor relative humidity

## 4 Creating and Assigning Interface Definitions

Boundary conditions are grouped together in Surface/Boundary definitions. These represent a surface to some space, for example a room or the outside.

In the next step, we create two surfaces/boundaries, one for the inside and one for the outside. Go to the Surfaces/Boundaries definition list window (1) and press the + button (2):



As with all definitions in DELPHIN, give the new interface a unique description name, for example "Inside". Then, select the type of surface/boundary. In this tutorial, we have created boundary conditions manually, so we use **Detailed/scientific interface**. Afterwards check the boundary conditions that shall be associated with this interface/surface. For the inside surface, check "Indoor heat conduction" and "Indoor vapor diffusion".

lame: Inside			
ype: Detailed/scientifi	c interface defined by severa	boundary conditions [Detailed]	
urface Properties			
vientation [0, 260 Deal	0		
nentation [0500 beg]	U		
nclination [0180 Deg]:	90		
oundary Conditions			
oundary Conditions	Name	Туре	Name: Indoor heat conduction
oundary Conditions	Name	Type Heat conduction	Name: Indoor heat conduction Type: Heat conduction [HeatConduction] Kind: Exchange coefficient [Exchange]
oundary Conditions           Indoor heat conduct           Outdoor heat conduct	Name ion	Type Heat conduction Heat conduction	Name: Indoor heat conduction Type: Heat conduction [HeatConduction] Kind: Exchange coefficient [Exchange] Used hv:
Indoor heat conduct         Outdoor heat conduct         Utdoor heat conduct         Indoor of the state conduct         Indoor of the state conduct	Name ion ction	Type Heat conduction Heat conduction Vapor diffusion	Name: Indoor heat conduction Type: Heat conduction [HeatConduction] Kind: Exchange coefficient [Exchange] Used by:
Undoor heat conduct Undoor heat conduct Undoor heat conduct Undoor vapor diffusio Undoor vapor diffusio Undoor vapor diffusio	Name ion ction n sion	Type Heat conduction Heat conduction Vapor diffusion Vapor diffusion	Name: Indoor heat conduction Type: Heat conduction [HeatConduction] Kind: Exchange coefficient [Exchange] Used by: • Inside
Undoor heat conduct Undoor heat conduct Undoor heat conduct Undoor vapor diffust Undoor vapor diffust Undoor vapor diffust	Name clion n n	Type Heat conduction Heat conduction Vapor diffusion Vapor diffusion	Name: Indoor heat conduction Type: Heat conduction [HeatConduction] Kind: Exchange coefficient [Exchange] Used by: •Inside
Indoor heat conduct         Outdoor heat conduct         Outdoor heat conduct         Indoor vapor diffusit         Outdoor vapor diffusit         Outdoor vapor diffusit	Name Ion ction n	Type           Heat conduction           Heat conduction           Vapor diffusion           Vapor diffusion	Name: Indoor heat conduction Type: Heat conduction [HeatConduction] Kind: Exchange coefficient [Exchange] Used by: •Inside

Similarly, create an interface "Outside" which uses the outdoor boundary conditions.

## 4 Appendix B – Outputs

## 1 Output Grids and Frequency

Before defining the output quantities we would like to monitor, we need to define the frequency of outputs. This is done in output grid definitions. It is meaningful to limit outputs for profiles, since such output files tend to become large. A good procedure is to create two output grids, one for profiles, i.e. outputs that write values for all elements, and one for scalars, for example sensor values (temperatures, relative humidities at certain positions) or integral values (i.e. total moisture content).

In the **Output Grids** list window, create a new definition (plus-button) and give it an identification name (1). The output frequency need not be constant throughout the simulation, but can be different in separate intervals of the simulation. So next, you need to select the number of such intervals (2). For this tutorial, leave the interval number to 1.

Edit output	grid	8
pecification		
lame:	Profiles	
lumber of in	tervals: 1 🗢	
ntervals		
efinition of i	intervals 2 1	
ormat: Ente	er time offsets or intervals as " <value> <unit>" strings,</unit></value>	
lote: A dura	tion of 0 in the last interval will let this interval run forever.	
	Interval #1	
Start time	0 d 🤇 🤇	
Duration	od 🖌 🗸	
End time		
Step size	1h 4	
Computed in	tervals (computed time offsets are converted to the unit of the start time offset)	
	Interval #1	
Start time	0 d	
Duration	infinite	
End time		
Step size	1h	
		OV Cancel

For each interval, you need to select the duration of the interval (3) and the output step size (4). Alternatively, you can specify start or end times of intervals, whereas <u>time</u> means <u>relative time since start of the simulation</u>.

The last interval can be given a duration of 0 d (don't forget the time unit!). This means that the interval is running as long as the simulation runs.

Create now output grids named "Profiles" and "Scalars" with one interval each, and output time steps of 1 h for "Profiles" and 10 min for "Scalars".

## 2 Output Files

Now outputs can be defined. This is done via defining output files. In the **Output Files** window, create a new output file definition (use the plus-button again). Start as usual with an identification name (1), but mind, that for output files the name is also used for the file name. That means you must not use characters that are not allowed for filenames (no slashes, colons etc.). In this tutorial, use "Temperature Profile" for the first output file definition.

Output	8 ×
Specification	
Flename (without path): Temperature Profile	
Quantity selection	
Quantity: Temperature	
Conversion/calculation options 2	
Average or integrate values of several selected elements/sides or store each individual value?	
Individual values of each selected element or side [Single]	•
Integration/averaging in time?	
Write values as calculated at output times [None]	•]
Output value unit:	c -
Output frequency 3	
Output grid: Profiles	• Edit
	OK Cancel

Next you have to select the physical quantity for the output (2). In the quantity selection list that is opened with the associated button, you see all the quantities that can be calculated with DELPHIN.

Туре	Name	Unit	Description
Element-based source/sink	MoistureLoadWTAConvection	kg/m3s	Source: Moisture load from WTA convective source)
Boundary flux	FluxShortWaveRadiationGlobal	W/m2	Global Short Wave Radiation absorbed from surface
Flux between elements	FluxAirConvection	kg/m2s	Dry air convection mass flux
Flux between elements	TotalFluxHeat	W/m2	Summation flux for energy balance equation
State variable or related quantity	Temperature	С	Temperature
State variable or related quantity	ThermalConductivity	W/mK	Thermal conductivity
State variable or related quantity	ThermalConductivity_Y	W/mK	Thermal conductivity in simulation direction Y
State variable or related quantity	ThermalConductivity_Z	W/mK	Thermal conductivity in simulation direction Z
State variable or related quantity	MoistureMassDensity	kg/m3	Total mass density of liquid water, water vapor and ice
State variable or related quantity	<b>OverhygroscopicWaterMassDensity</b>	kg/m3	Mass density of overhygroscopic liquid water (condensate) with respect to REV
State variable or related quantity	IceMassDensity	kg/m3	Mass density of ice with respect to REV
State variable or related quantity	IceVolumeRatio	%	Ratio of ice phase volume to effective saturation
State variable or related quantity	RelativeHumidity	%	Relative humidity
State variable or related quantity	GasPressureOffset	Pa	Gas pressure offset to atmospheric pressure
Element-based source/sink	ThermalLoad	W/m3	Source: Thermal load/Heat gain
Element-based source/sink	LongWaveRadiationLoad	W/m3	Source: Thermal load from longwave radiation exchange
Element-based source/sink	ThermalLoadAirChange	W/m3	Source: Thermal load from air change
Element-based source/sink	MoistureLoadAirChange	kg/m3s	Source: Moisture load (vapor) from air change
Element-based source/sink	MoistureEnthalpyAirChange	W/m3	Source: Enthalpy associated with moisture load from air change
Element-based source/sink	MoistureEnthalpyWTAConvection	W/m3	Source: Enthalpy associated with moisture load from WTA convective source
Element-based source/sink	MoistureLoad	kg/m3s	Source: Moisture load/Liquid water gain
Element-based source/sink	MoistureLoadEnthalpy	W/m3	Source: Enthalpy associated with moisture load/Liquid water gain
Element-based source/sink	SaltProductionRateBoundWater	kg/m3s	Source: Mass production rate of bound water from dissolved salt
Element-based source/sink	SaltProductionRateEnthalpy	W/m3	Source: Enthalpy associated with isothermal dissolution/crystallization of salt
Element-based source/sink	ThermalLoadGroundWaterFlow	W/m3	Source: Thermal load/heat gain due to imposed ground water flow
Element-based source/sink	VOCAdsorptionRate	kg/m3s	Source: Mass production rate of solid phase VOC from condensation out of gas phase
Element-based source/sink	VOCDesorptionRate	kg/m3s	Source: Mass production rate of gas phase VOC from emission of solid phase
Element-based source/sink	VOCLoadAirChange	kg/m3s	Source: VOC mass concentration gain due to air change
Element-based source/sink	VOCSource	mg/m3s	Source: VOC mass production rate due to predefined emission source

In this list select the quantity "Temperature" and close the dialog.



To navigate quickly the table and quickly select the quantity you need, simply click on a table cell in the keyword or description column and start typing the name/keyword. The cursor will then jump to the cell. Also, you can sort quantities based on type and name. Furthermore, all frequently used quantities are displayed in bold at the top (can be switched off with a hook at the bottom left).

The selected quantity also defines the default output unit and whether it is an element or side-based quantity. Finally, we select the output grid (3). This defines how frequent outputs are written.

Now the **Output Files** list window should hold a single output "Temperature Profile". In the grid generation step the total geometry was discretized into many elements. Now we apply the created temperature output to all elements, which means we are obtaining the temperatures in all elements at each output time. This allows us to create temperature profiles across the geometry for each output time (see post-processing steps in part 2 of the tutorial).

To assign an output select all elements (either per mouse, or keyboard holding the Shift key, or by pressing Ctrl+A). Then select the output file definition and use the assignment button **E**. Again, you should see the output file definition to change from gray italic to normal black, indicating its use in the project.

It is also possible to merge the results from several elements into a single number. For example, you can compute a weighted average value ("Mean" value), or an integral value ("Integral"), for example the integral moisture content. These spatial operations are termed **space type** of the output and can be selected in the output file definition dialog. When the values in each element shall be monitored (the default), the space type is "Single".

Simulation outputs are time dependent quantities. There are some time-related calculations (termed **time type**) built into DELPHIN. For example, you can compute an average value in time (keyword "Mean"). The averaging always starts at the begin of the simulation. More advanced averaging like gliding average calculation is done in the post-processing. Fluxes or sources can be integrated in time ("Integral"). If the value shall be written to file as it is calculated at the output time, the time type is "None". You can set this parameter also in the output file definition dialog.

For analysis of the simulation it is meaningful to create the following outputs:

Output name	Quantity	Schedule	Space type	Time type
Temperature Profile	Temperature	Profiles	Single	None
Relative Humidity Profile	RelativeHumidity	Profiles	Single	None
Moisture Content Profile	MoistureMassDensity	Profiles	Single	None
Overhygroscopic Moisture Profile	OverhygroscopicWaterMassDensity	Profiles	Single	None
Moisture Mass Integral	MoistureMassDensity	Scalars	Integral	None
Condensate	OverhygroscopicWaterMassDensity	Scalars	Integral	None
Accumulated Moisture Flux	TotalFluxMoisture	Scalars	Integral	Integral



If you have to create many definitions with similar properties, use the "Copy definition" function via the button  $\mathcal{P}$ .

All output files except the last have to be assigned to the entire geometry. The flux output is used to obtain the accumulated moisture mass in the construction, which changes dynamically due to moisture fluxes into and out of the construction. If done correctly, the results obtained with this output should match that of the moisture mass integral (actually the difference of the moisture mass integral to the initial moisture mass integral).

Boundary flux outputs are generally assigned on the surfaces/boundaries of the geometry. In this tutorial, we will assign this output to the leftmost and rightmost element. First select (only) the leftmost element, then select the "Accumulated Moisture Flux" output and click on the assign-from-left button \*\*. Then do the same for the rightmost element and assign the flux from right. When you now select the flux output, the construction should highlight both boundary sides.





Due to the clustering of the grid near the boundary, it may be difficult to select the boundary elements (or you have to zoom in quite a bit). Alternatively, you can display the construction differently with all elements being <u>shown</u> with the same size. You can switch between these display modes with the button IIII.

Now with the output definitions in place, we are ready to start the simulation.